RESEARCH ARTICLE

CFD analysis for airflow distribution of a conventional building plan for different wind directions

Obula Reddy Kummitha1,*, R. Vijay Kumar2 and V. Murali Krishna1

1Department of Mechanical Engineering, B V Raju Institute of Technology, Narsapur 502313, Telangana, India
and 2Department of Civil Engineering, B V Raju Institute of Technology, Narsapur 502313, Telangana, India

*Corresponding author. E-mail: obulareddy.bec10@gmail.com http://orcid.org/0000-0002-5088-3329

Abstract

Computational fluid dynamics analysis of a building plan has been investigated with predominant wind velocity for different wind directions. The flow properties’ variation in the computational domain has been modeled by solving the Reynolds-Averaged Navier–Stokes (RANS) equations with the finite volume second-order discretization scheme. The turbulence of airflow distribution in and around the building has been modeled with the Shear Stress Transport (SST) $k$–$\omega$ turbulence model from the analysis of different turbulence models. Numerical results are analysed by evaluating and comparing the various flow properties at different building plan locations with different wind directions. The development of pressure coefficients, wind-driven driving force, and air change per hour are studied for different wind directions. From the analysis of numerical results, it is identified that better ventilation with sufficient airflow distribution has existed when the wind is coming from the west direction.

Keywords: computational fluid dynamics; ventilation; airflow distribution; air change rate

Nomenclature

\begin{itemize}
\item $P$: Pressure (Pa)
\item $T$: Temperature (K)
\item $u_x$: Velocity in the $x$-direction (m/s)
\item $u_y$: Velocity in the $y$-direction (m/s)
\item $\tau_{ij}$: Shear stress (N/m²)
\item $q_i$: Heat flux in the $x$-direction (W/m²)
\item $h_t$: Total enthalpy (J)
\item $e_t$: Total energy (J)
\item $k$: Turbulent kinetic energy (J/kg)
\item $\omega$: Specific turbulent dissipation rate (1/s)
\item $\epsilon$: Rate of dissipation of turbulent kinetic energy (J/kg.s)
\item $\Gamma_k$: Effective diffusivity of $k$ (m²/s)
\item $G_k$: Generation of turbulent kinetic energy, $k$
\item $Y_k$: Dissipation of $k$
\item $S_k$: Source term, $k$
\item $\Gamma_\omega$: Effective diffusivity of $\omega$
\item $G_\omega$: Generation of turbulent kinetic energy, $\omega$
\item $Y_\omega$: Dissipation of $\omega$
\item $S_\omega$: Source term, $\omega$
\item $D_{k\omega}$: Cross-diffusion term
\item ACH: Number of air changes per hour (1 per hour)
\item $C_{\text{pww}}$: Wind-induced pressure coefficient on the windward side
\item $C_{\text{plw}}$: Wind-induced pressure coefficient on the leeward side
\item $Q$: Volumetric flow rate of air (ft³/min)
\item $V$: The volume of the space (house or room) (ft³)
\end{itemize}

1. Introduction

The design and optimization of natural ventilation for a conventional building plan are a challenging task, and they can be achieved with the help of computational fluid dynamics (CFD) techniques (Lomas, 2007; Chen, 2009; Gilani, Montazeri,
Conventional building plan CFD analysis for airflow distribution

The pressure difference is the driving force for the flow of air from one location to another; the same investigation of different wind directions and visualized the development of flow and nonflow regions in and around the building. The rate of airflow through openings strongly depends on the pressure difference between inside and outside the building. The concept of windcatcher plays a more significant role in enhancing the effectiveness of natural ventilation. Montazeri and Montazeri (2018) investigated the rooftop wind catcher concept with different outlet openings using CFD studies. The numerical analysis identified that better indoor air quality did not exist when outlet openings are very close to the windcatcher. Evola and Popov (2006) performed computational fluid dynamic analysis for the cross-ventilation room investigated experimentally by Jiang et al., (2003) using a wind tunnel. A comparison was made between experimental and numerical results and observed good agreement for CFD and experimental values, both qualitatively and quantitatively.

Jones and Whittle (1992) investigated the current status and capability of CFD to predict the airflow distribution inside and around the buildings. This research paper was given a useful review on the application of CFD capability to predict airflow distribution. CFD modeling has been done with the basic principles of conservation of mass, momentum, and energy. All these fluid flow governing equations are partial differential equations that are not easy to solve using the analytical method because of their nonlinear terms. In this paper, the authors explained the concept of discretization. It is the process of converting partial differential equations into a set of linear algebraic equations using discretization techniques, which are defined as the primary finite difference, finite element, and advanced finite volume method. All these linear equations are solved using numerical iterative techniques until it reaches a converged solution. Finally, the authors identified the credibility of CFD for the design of building ventilation.

Turbulence is the critical parameter to visualize the flow field of the airstream in and around the building. Yang, Wright, Etheridge, and Quinn (2006) conducted both the experimental and numerical study of natural ventilation. Numerical analysis was conducted with different turbulence models, and a comparison has been made. The numerical analysis identified that with an increase in fluctuating velocity, the Reynolds-averaged Navier-Stokes (RANS) model could not predict the total ventilation rate. Numerical analysis was extended for expected improved results with complex turbulence models such as large eddy simulation (LES). Chen and Srebric (2002) reported a detailed procedure for analysing indoor environments and verifying, validating, and predicting required results. CFDs have been used to determine heat transfer, fluid flow, and transport of chemical species; furthermore, a wide range of analyses for heating ventilation components – air conditioning and refrigeration (HVAC&R) applications (Ladeinde & Nearon, 1997). The latest developments and improvements in recent computing tools (CFD) make it possible to use these tools to analyse buildings' indoor environment. However, the decision to use CFD purely depends on the cost of computation, its time, and realistic modeling of the problem and its results (Martin, 1999). It was also reported that using CFD and the results might not be accurate (Baker & Gordon, 1997; Chen, 1997). Hence, the engineering justification (Post, 1994) is required, and it is achieved by introducing verification and validation of numerical results, which will increase the credibility of the CFD tool.

The energy demand for buildings and their constructional sectors combined is accountable for 38% of total global final energy (Saadatian, Haw, Sopian, & Sulaiman, 2012; Ghardiri, Lukman, Ibrahim, & Mohamad, 2013). These are also responsible for developing CO₂ emissions, around 40% of total emissions (Calautit & Hughes, 2014; Aflaki, Mahyuddin, Awad, & Baharum, 2015). All these emissions will create a terrible problem of global warming. From the amount of total global electricity, building sectors consume around 60% of electricity using all electrical appliances like HVAC equipment (Moosavi, Mahyuddin, Ab Ghafer, & Azzam Ismail, 2014; Manzano-Agugliaro, Montoya, Sabio-Ortega, & García-Cruz, 2015; Chenari, Días Carrilho, & Gameiro da Silva, 2016). The increase in demand for building energy consumption can be decreased with various passive cooling techniques. Jomehzadeh et al., (2017) reported a review of different passive cooling techniques like windcatcher and natural ventilation optimization. Compared to mechanical ventilation, the concept of natural ventilation plays a vital role in reducing buildings’ energy consumption (Daghf, 2015). Natural ventilation provides good indoor air quality compared to mechanical because of its better interaction with the outside environment (Liu et al., 2020). The capability of energy-saving with natural ventilation creates more enthusiasm to introduce various passive techniques, which are classified as cooling and heating techniques using various construction materials, technologies, and solar techniques (Kwok & Grondzik, 2011; Geeta & Velraj, 2012; Santamouris & Kolokotsa, 2013; Waqas & Ud, 2013).

CFD can also be applied for the assessment of greenhouse building ventilation design. Mistriotis, Bot, Picuno, and Scarascia-Mugnozza (1997) investigated the efficiency of greenhouse ventilation using CFD. Before selecting CFD for their respective application, researchers conducted a validation study and understood the CFD tool’s capability to predict indoor airflow distribution and compared the same with experimental results and identified good matching of results. The authors have carried out the numerical analysis with varying greenhouse length, reflecting on the ventilation design and visualizing its influence on ventilation design efficiency. Finally, researchers demonstrated that CFD is a powerful tool for predicting indoor airflow distribution with different ventilation designs.

Wind-induced pressure difference and stack effect are the two critical parameters acting as driving forces for buildings' natural ventilation. Straw, Baker, and Roberston (2000) performed both the experimental and numerical study to analyse the effect of wind-induced pressure difference on a cubic structure's ventilation performance. In this paper, the surface pressure coefficient was considered a measurement parameter to visualize the ventilation performance difference, while two openings were considered in both parallel and normal to the wind direction. These measurement results were compared with different measuring techniques used for mean and fluctuating ventilation rates. For the parallel configuration case, most ventilation is taken by the fluctuating component compared to the mean ventilation, whereas in normal cases, mean ventilation is greater than the fluctuating component.

Knowledge of wind-induced pressure development on building walls and equipment and the parameters that affect the pressure development is essential for optimizing building natural ventilation (Letchford & Mehta, 1993; Hunt & Linden, 1999; Linden, 1999; Uematsu & Isyumov 1999; Karava et al., 2004; Montazeri & Azizian, 2008; Cóstola, Blokken, & Hensen, 2009; Montazeri, Montazeri, Azizian, & Mostafavi, 2010; Fatemeh, Dilshan, & Mohd, 2020; Zheng, Qiuhua, & Li, 2020). Whenever the wind is coming from different directions, two zones will be created concerning the building's wind coming direction and position. Windward and leeward are the two zones defined as
high-pressure and low-pressure zones, respectively, which will develop the required pressure difference for airflow through the building’s openings. The development of pressure difference depends on many parameters like approaching wind flow conditions (Stathopoulos, 1997; Wu, Sarkar, Mehta, & Zhao, 2001), design of the building (Uematsu & Isyumov, 1999), wind direction (Levitan, Mehta, Vann, & Holmes, 1991), building surroundings (Kim, Yoshida, & Tamura, 2012). The literature found that many researchers investigated the building ventilation but they have not accounted for the air change per hour (ACH) concerning different wind directions. It is also observed that the authors have not considered different turbulence models and their effect on airflow distribution. Hence, this research considered the various turbulence models, ACH, and pressure-induced driving forces for different wind directions. In this research paper, numerical analysis has been performed with different turbulence models to visualize the airflow distribution for a conventional building plan concerning different wind directions. Wind-induced pressure coefficients and driving force are calculated for different wind directions. As we considered wind-induced ventilation analysis, the buoyancy effect has been neglected and considered only wind speed’s effect to influence the airflow distribution in and around the building. Ventilation for the house’s different locations has been assessed by analysing the ACH.

2. The Conventional Building Plan and Computational Domain Modeling

The conventional building plan has been designed for 437 square feet with a single bedroom, living room, kitchen, and toilet (Fig. 1). The building plan consists of six windows and one main door that has interaction with the surrounding air. The living area consists of two windows; one is located on the north wall and another is placed on the east wall. The bedroom has two windows; one is located on the south wall and another on the west wall. The kitchen area has two openings to exchange cooking fumes with the outside atmosphere. For the designed conventional plan, the computational domain has been modeled by incorporating the building’s surrounding area. It is considered four times the length and width on the building’s respective side to visualize the effect of airflow wind direction on airflow distribution inside and around the building (VDI, 2000; Evola & Popov, 2006). The blockage ratio for the given dimensions is 1.2% (Frank et al., 2004).

3. Mathematical and Numerical Modeling

In CFD problems, numerical results’ accuracy strongly depends on the respective flow governing equations. To identify the optimum airflow distribution results, the numerical analysis has been considered with the RANS governing equations along with finite volume second-order discretization scheme. The pressure correction equation has been solved by using the SIMPLE algorithm of pressure–velocity coupling. The numerical simulation’s convergence criteria were assumed to be obtained when all the scaled residuals reach a minimum of $10^{-6}$ level of convergence. The flow governing equations were considered as follows:

mass balance (the continuity equation): \[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0; \quad (1)
\]
momentum balance: \[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_k} (\rho u_i u_k) = -\frac{\partial P}{\partial x_k} + \frac{\partial}{\partial x_k} (\tau_{ij} u_j); \quad (2)
\]
energy equation: \[
\frac{\partial}{\partial t} (\rho e) + \frac{\partial}{\partial x_i} (\rho h_i u_i) = \frac{\partial}{\partial x_j} (\tau_{ij} u_j - q_i). \quad (3)
\]

3.1 Turbulence modeling

The selection of a suitable turbulence model is the major part of numerical modeling to visualize eddies’ formation and dissipation and its effect on flow parameters. If all the RANS governing equations are solved without considering any approximations of turbulence modeling, it is called direct numerical simulation, and it will not give the effect of turbulence in the flow stream. From the literature review (Teodosiu et al., 2014; Wang & Chen, 2009), it is identified that many researchers investigated the selection of turbulence models for their respective applications, and comparison has been made among different turbulence models and experimental results to identify the turbulence model that gives closer matching results to that of experimental. In this research paper, different turbulent models were considered to identify the best suitable turbulence model for accurate airflow distribution for the conventional building.
Conventional building plan CFD analysis for airflow distribution

The different turbulence models are described along with their transport governing equations as given below:

**Standard k-ε model:**

\[
\frac{\partial}{\partial t} \left( \rho k \right) + \frac{\partial}{\partial x_i} \left( \rho \bar{u_i} k \right) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M + S \tag{4}\n\]

\[
\frac{\partial}{\partial t} \left( \rho \varepsilon \right) + \frac{\partial}{\partial x_i} \left( \rho \bar{u_i} \varepsilon \right) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + G_\varepsilon - Y_\varepsilon + D_\omega + S_{\varepsilon} \tag{5}\n\]

The standard k-ε model comes under the category of the two-equation turbulence model. In these transport equations, both \(G_k\) and \(G_\varepsilon\) represent the generation of turbulent kinetic energy due to buoyancy and fluctuation velocity, respectively. \(Y_M\) represents the compressible turbulence dilatation function. \(S\) refers to the source term, and model constants are defined with default values.

**SST k-ω model:**

\[
\frac{\partial}{\partial t} \left( \rho k \right) + \frac{\partial}{\partial x_i} \left( \rho \bar{u_i} k \right) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G_k - Y_k \tag{6}\n\]

\[
\frac{\partial}{\partial t} \left( \rho \omega \right) + \frac{\partial}{\partial x_i} \left( \rho \bar{u_i} \omega \right) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_\omega} \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_{\varepsilon}. \tag{7}\n\]

This turbulence model also comes under the category of two-equation turbulence models. This model is mostly suitable for near-wall modeling because of its perfect \(y^+\) values. The terminologies of parameters used in transport equations are defined as follows: Both \(G_k\) and \(G_\varepsilon\) stand for generation turbulent kinetic energy concerning fluctuation velocity and generation of \(\omega\). \(Y\) terms stand for dissipation concerning \(k\) and \(\omega\). \(D_\omega\) is the representation of cross-diffusion term, which is defined as shown below:

\[
D_\omega = 2 \left( 1 - F_1 \right) \rho \sigma_\omega \frac{\varepsilon^2}{k} \left[ \frac{1}{\lambda} \frac{\partial}{\partial x_j} \left( \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \right) \right]. \tag{8}\n\]

\(F_1\) stands for blending function

\[
F_1 = \tanh \left( \arg_1 \right). \tag{9}\n\]

where \(\arg_1\) is the identification of minimum for the function of the positive portion of the cross-diffusion term,

\[
\arg_1 = \min \left[ \max \left( \sqrt{\frac{2}{C_{\mu k} \sigma_k}}, \frac{500 \nu}{\nu^{\omega}}, \frac{4 \rho \sigma_{k,\omega} \varepsilon}{C_{D_{k,\omega}}^2} \right) \right]. \tag{10}\n\]

**LES model theory:**

The LES model is the latest and advanced model for the accurate prediction of eddies and its diffusion in the flow field. The fluid flow governing equations under LES modeling have been obtained by filtering the primary partial differential governing equations.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho \bar{u_i}) = 0 \tag{10}\n\]

\[
\frac{\partial}{\partial t} \left( \rho \bar{u_i} \bar{u_j} \right) + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \bar{u_i}}{\partial x_j} \right) - \frac{\partial}{\partial x_i} \left( \mu \frac{\partial \bar{u_j}}{\partial x_i} \right) - \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \bar{u_i}}{\partial x_j} \right) = \frac{\partial}{\partial x_i} \left( \mu \frac{\partial \bar{u_j}}{\partial x_i} \right) - \frac{\partial}{\partial x_i} \left( \mu \frac{\partial \bar{u_j}}{\partial x_i} \right). \tag{11}\n\]
Figure 4: Computational domain and its results for validation.
where $\sigma_{ij}$ is the stress tensor and it is defined from the molecular viscosity as follows:

$$\sigma_{ij} = \left[ \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] - \frac{2}{3} \mu \frac{\partial \bar{u}_l}{\partial x_l} \delta_{ij},$$

(12)

4. Boundary Conditions and Grid Independence Study

The inflow boundary conditions are defined as Dirichlet boundary conditions with the known pressure and velocity of incoming wind (airflow). The flow parameters at the outlet boundary conditions are extrapolated from the physics of flow governing equations and its discretization techniques. All the walls of the domain are defined with no-slip and smooth surface conditions. In CFD problems, computational results’ accuracy and stability will vary with mesh elements’ size and quality. The mesh’s cell quality has been controlled by limiting orthogonal quality, aspect ratio, and skewness within the acceptable range with hexahedral and tetrahedral elements (Fig. 2). Near-wall modeling has been refined with the combination of tetrahedral and hexahedral elements (Sanyal & Dalui, 2020). The free stream region (away from the walls) has been defined with hexahedral elements.

A grid independence study has been carried out to investigate the variation of results using mesh elements and optimum grid size. It is observed that with an increase in the number of elements (optimum grid size), the accuracy also increases (Fig. 3). Based on computational time and cost, the medium mesh has been considered in this research work.

4.1 Validation

In any CFD problem, validation is the crucial task to understand the CFD tool’s credibility for applying the respective problem. In this paper, the CFD tool’s validation has been carried out by investigating the numerical results for the open literature’s experimental data. From Fig. 4, it is identified that both the numerical and experimental results are in good agreement qualitatively and quantitatively.

4.2 Analysis and identification of turbulence model

Numerical analysis has been conducted on the experimental model (Jiang et al., 2003; Evola & Popov, 2006) similar to this type of research work with different turbulence models to identify the best suitable turbulence model for this research work. In the atmosphere, there is a chance of developing large eddies around the building envelope. Hence, LES is also considered to analyse how best it is among all the models. However, we finally found that the SST $k$–$\omega$ model results are closely matched with the experimental. A comparison has been made among different models (Fig. 5) with the experimental results and we identified that the $k$–$\omega$ turbulence model gives closer results to the experimental results. It is also observed that with an increase in mesh elements, the $k$–$\omega$ model gives the closer approximate results at near-wall boundaries.
5. Results and Discussion

For the prediction of local wind velocity and its direction, weather data have been monitored for May 2019 in the local wind station at BVRIT Narsapur, India. For the accurate prediction and finalization of wind velocity and its direction, it also considered the data from the online source of timeand-date (Fig. 6; www.timeanddate.com). From the comparison of experimental and online weather reports, it is observed that the average velocity of wind is 5.15 m/s at a 10-m height from the ground level, and the wind is moving in the west direction for most of the time of the day. The wind rose has been generated for the 1-month data of wind velocity and its direction (Fig. 7).

The prediction of airflow distribution and identifying its effect on comfort conditions are the most critical task while designing a conventional building plan. Numerical analysis of the conventional building plan has been carried out with predominant wind velocity (1 m/s) taken from the weather report’s experimental results for 1 month. From the visualization of wind direction for that particular month, four wind directions have been considered and we analysed the airflow distribution inside and around the building. The numerical results of airflow distributions are shown in Fig. 8.

From the visualization of airflow distributions, it is identified that insufficient airflow has existed in the bedroom when the air is coming from either east or north direction. Many contaminant gases are generated in the house from the kitchen area in the form of cooking fumes. Well-designed ventilation for the kitchen can reduce the level of contaminants. Better control of cooking fumes is viable when the wind is coming from either south or east direction, it leads to the carrying of cooking fumes into the region of the living area and creates discomfort.

ACH is the critical parameter to be considered for the evaluation of sufficient ventilation. ACH gives information on how many times the air volume has been replaced within 1 hour. The ACH values are calculated with the following expression:

\[
ACH = \frac{60Q}{V}. \tag{13}
\]

ACH strongly depends on wind velocity, wind direction, and building design. In this research, ACH values have been investigated for the whole area and different locations (hall, bedroom, and kitchen) of the house concerning different wind directions and tabulated as shown below (Table 1).

From the analysis of ACH values for different wind directions, it is observed that many air exchanges are not taking place for the bedroom area when the wind is coming from either east or north direction. When the wind is coming from the west direction, the kitchen area has registered less air exchange. The ventilation effectiveness has been finalized based on the ACH values of the bedroom and hall, as we are spending most of the time in these areas compared to the kitchen. Comparatively, east coming wind direction has better ACH values for the hall, but it has significantly fewer ACH values for the bedroom, so not preferable. East and south wind directions have better ACH values for the kitchen area, but kitchen fumes entering into the hall is when the wind is coming from either east or south and creating discomfort. As we give preference to the hall and bedroom areas, the wind coming from the west direction has better ACH values without the mixing of fumes from the kitchen area, as these are exit to the atmosphere directly through the east-side kitchen window. Comparatively, the wind coming from the west direction gives higher ACH values for the house’s whole area.

The pressure difference between the windward and leeward regions is the driving force for airflow through the building spaces. Fig. 9 shows the development of high-pressure (positive) and low-pressure (negative) regions on windward and leeward regions for different wind directions. From this contour, it is observed that the pressure difference (ΔP) is more when the wind is coming from the west direction. Hence, wind energy is more
Table 1: ACH concerning different wind directions.

| Wind direction | Hall | Bedroom | Kitchen | Whole house |
|----------------|------|---------|---------|-------------|
| East           | 7    | <1      | 4       | 2           |
| South          | 6    | 3       | 5       | 3           |
| West           | 5    | 4       | <1      | 3           |
| North          | 3    | <1      | 5       | 2           |

for this designed plan while the wind is coming from the west direction. Intermediate/inside walls of the building also play a vital role in developing favorable and unfavorable pressure gradients concerning facing building to the wind direction.

For more visualization and accurate prediction of wind-driven ventilation, the pressure variation among different building regions has been plotted concerning different wind directions. A total of four lines (A, B, C, and D) have been drawn such that each line should cover at least two regions of the building while the wind is moving from the windward zone to the leeward zone.

From the observation of Figs 9 and 10, it can be easily visualized the development of low- and high-pressure regions concerning the position and facing external and internal walls to that of wind coming direction. When the wind moves from east to west or west to east, the partition wall between the kitchen and bedroom acts as an obstacle and creates high pressure on either side of it (Fig. 10). When wind moves from east to west,
sufficient airflow is not created inside the bedroom, leading to suffocation or discomfort. Similarly, when the wind is moving from north to south or south to north, the partition wall in between the hall and bedroom acts as an obstacle.

The wind-driven driving force ($\Delta p$) and pressure coefficients ($C_p$) for different wind directions have been calculated (Fig. 10) and tabulated. It gives maximum and minimum wind-induced driving forces concerning the direction of wind coming over the building. From Table 2, it is identified that the wind-induced pressure coefficient is negative on the leeward side and positive on the windward side, which indirectly gives the direction of airflow movement from high-pressure region to low-pressure region.

5. Conclusion

The ventilation behavior of conventional building plans concerning atmospheric wind direction has been analysed by solving the flow governing equations using the CFD. Airflow distribution within and around the building has been analysed with different wind coming directions. From the analysis of velocity, pressure flow filed, and ACH values, the following key points are identified:

- From the analysis of local weather report and wind rose, it is observed that for most of the time, the wind is moving in the west direction with an average velocity of 0.5 m/s.
- The wind coming from the west direction has obtained better airflow distribution inside the house.
- The wind coming from either west or north direction has achieved the highest ACH value of 3 for the house’s whole area.
- Comparatively, the wind coming from the west direction is feasible because it carries out the cooking fumes generated in the kitchen area.
- The wind-induced driving force ($\Delta P = 1.65$ Pa) for the west direction is higher comparatively to others.

Conflict of interest statement

None declared.
### Table 2: Wind-induced pressure coefficients and driving force.

| Parameter | East   | South  | West   | North  |
|-----------|--------|--------|--------|--------|
| $C_{pww}$ | 0.960  | 0.807  | 0.941  | 0.92   |
| $C_{plw}$ | −0.870 | −0.946 | −1.583 | −0.9055|
| $\Delta P$, (Pa) | 1.14   | 1.09   | 1.65   | 1.13   |

### References

Aflaki, A., Mahyuddin, N., Awad, Z. A.-C. M., & Baharum, M. R. (2015). A review on natural ventilation applications through building façade components and ventilation openings in tropical climates. *Energy and Buildings, 101*, 153–162.

Baker, A. J., & Gordon, E. B. (1997). Computational fluid dynamics - a two-edged sword. *ASHRAE Journal, 39*(8), 51–58.

Calautit, J. K., & Hughes, B. R. (2014). Wind tunnel and CFD study of the natural ventilation performance of a commercial multi-directional wind tower. *Building and Environment, 80*, 71–83.

Chenari, B., Dias Carrilho, J., & Gameiro da Silva, M. (2016). Towards sustainable, energy-efficient and healthy ventilation strategies in buildings: A review. *Renewable and Sustainable Energy Reviews, 59*, 1426–1447.

Chen, Q. (1997). Computational fluid dynamics for HVAC - successes, and failures. *ASHRAE Transactions, 103*(1), 178–187.

Chen, Q. (2009). Ventilation performance prediction for buildings: A method overview and recent applications. *Building and Environment, 44*, 848–858.

Chen, Q., & Srebric, J. (2002). A procedure for verification, validation, and reporting of indoor environment CFD analyses. *HVAC&R Research, 8*(2), 201–216.

Cóstola, D., Blocken, B., & Hensen, J. L. M. (2009). Overview of pressure coefficient data in building energy simulation and airflow network programs. *Building and Environment, 44*, 2027–2036.

Daghigh, R. (2015). Assessing the thermal comfort and ventilation in Malaysia and the surrounding regions. *Renewable and Sustainable Energy Reviews, 48*, 681–691.

Evola, G., & Popov, V. (2006). Computational analysis of wind-driven natural ventilation in buildings. *Energy and Buildings, 38*(5), 491–501.

Fatemeh, M. G., Dilshan, R. O., & Mohd, F. M. (2020). Effects of wing wall at the balcony on the natural ventilation performance in medium-rise residential buildings. *Journal of Building Engineering, 31*, 101316.

Franke, J., Hirsch, C., Jensen, A. G., Krus, H. W., Schatzmann, M., Westbury, P. S., Miles, S. D., Wisse, J. A., & Wright, N. G. (2004). Recommendations on the use of CFD in wind engineering.
In Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics, COST Action C14, (pp. C1.1–C1.11). European Science Foundation COST Office.

Geetha, N. B., & Velraj, R. (2012). Passive cooling methods for energy-efficient buildings with and without thermal energy storage – A review. Energy Education Science and Technology Part A Energy Science and Research, 29, 913–946.

Ghadiri, M. H., Lukman, N., Ibrahim, N., & Mohamad, M. F. (2013). Computational analysis of Wind-Driven natural ventilation in a two sided rectangular wind catcher. International Journal of Ventilation, 12, 51–61. https://www.timeanddate.com/weather/india/hyderabad/historic.

Gilani, S., Montazeri, H., & Blocken, B. (2016). CFD simulation of the stratified indoor environment in displacement ventilation: Validation and sensitivity analysis. Building and Environment, 95, 299–313.

Hunt, G. R., & Linden, P. F. (1999). The fluid mechanics of natural ventilation displacement ventilation by buoyancy-driven flows assisted by wind. Building and Environment, 34, 707–720.

Jiang, Y., Alexander, D., Jenkins, H., Arthur, R., & Chen, Q. (2003). Natural ventilation in buildings: measurement in a wind tunnel and numerical simulation with large-eddy simulation. Journal of Wind Engineering and Industrial Aerodynamics, 91(3), 331–353.

Jomehzadeh, F., Nejat, P., Calautit, J. K., Yusof, M. B. M., Zaki, S. A., Hughes, B. R., & Yazid, M. N. A. W. M. (2017). A review on wind catchers for passive cooling and natural ventilation in buildings, Part 1: Indoor air quality and thermal comfort assessment. Renewable and Sustainable Energy Reviews, 70, 736–756.

Jones, P. J., & Whittle, G. E. (1992). Computational fluid dynamics for building air flow prediction – current status and capabilities. Building and Environment, 27(3), 321–338.

Karava, P., Stathopoulos, T., & Athienitis, A. K. (2004). Wind driven flow through openings a review of discharge coefficients. International Journal of Ventilation, 3, 255–266.

Kim, Y. C., Yoshida, A., & Tamura, Y. (2012). Characteristics of surface wind pressures on low-rise building located among large group of surrounding buildings. Engineering Structures, 35, 18–28.

Kwok, A. G., & Grondzik, W. T. (2011). The green studio handbook: Environmental strategies for schematic design. Elsevier.

Ladeinde, F., & Nearer, M. (1997). CFD applications in the HVAC&R industry. ASHRAE Journal, 39(1), 44.

Letchford, C. W., & Mehta, K. C. (1993). The distribution and correlation of fluctuating pressures on the Texas tech building. Journal of Wind Engineering and Industrial Aerodynamics, 50, 225–234.

Levitan, M. L., Mehta, K. C., Vann, W. P., & Holmes, J. D. (1991). Field measurements of pressures on the Texas tech building. Journal of Wind Engineering and Industrial Aerodynamics, 38, 227–234.

Linden, P. F. (1999). The fluid mechanics of natural ventilation. Annual Review of Fluid Mechanics, 31, 201–238.

Liu, S., Cao, Q., Zhao, X., Lu, Z., Deng, Z., Dong, J., Lin, X., Qing, K., Zhang, W., & Chen, Q. (2020). Improving indoor air quality and thermal comfort in residential kitchens with a new ventilation system. Building and Environment, 180, 107016.

Lomas, K. J. (2007). Architectural design of an advanced naturally ventilated building form. Energy and Buildings, 39, 166–181.

Manzano-Agugliaro, F., Montoya, F. G., Sabio-Ortega, A., & Garcia-Cruz, A. (2015). Review of bioclimatic architecture strategies for achieving thermal comfort. Renewable and Sustainable Energy Reviews, 49, 736–755.

Martin, P. (1999). Using CFD in the real world. ASHRAE Journal, 41(1), 20–25.

Mistriotis, A., Bot, G. P. A., Picuno, P., & Scarascia-Mugnozza, G. (1997). Analysis of the efficiency of greenhouse ventilation using computational fluid dynamics. Agricultural and Forest Meteorology, 85, 217–228.

Montazeri, H., & Azizian, R. (2008). Experimental study on natural ventilation performance of one-sided wind catcher. Building and Environment, 43, 2193–2202.

Montazeri, H., & Montazeri, F. (2018). CFD simulation of cross-ventilation in buildings using rooftop wind catchers: Impact of outlet openings. Renewable Energy, 118, 502–520.

Montazeri, H., Montazeri, F., Azizian, R., & Mostafavi, S. (2010). Two-sided wind catcher performance evaluation using experimental, numerical and analytical modeling. Renewable Energy, 35, 1424–1435.

Moosavi, I., Mahyuddin, N., Ab Ghafar, N., & Azzam Ismail, M. (2014). Thermal performance of atria: An overview of natural ventilation effective designs. Renewable and Sustainable Energy Reviews, 34, 654–670.

Post, N. M. (1994). Airflow models gaining clout. ENR, 233, 22–25.

Saadatian, O., Haw, L. C., Sopian, K., & Sulaiman, M. Y. (2012). Review of wind catcher technologies. Renewable and Sustainable Energy Reviews, 16, 1477–1495.

Santamouris, M., & Kolokothas, D. (2013). Passive cooling dissipation techniques for buildings and other structures: The state of the art. Energy and Buildings, 57, 74–94.

Sanyal, P., & Dalui, S. K. (2020). Effects of internal angle between limbs of Y plan shaped tall building under wind load. Journal of Building Engineering, 33, 101843.

Stathopoulos, T. (1997). Computational wind engineering: past achievements and future challenges. Journal of Wind Engineering and Industrial Aerodynamics, 67–68, 509–532.

Straw, M. P., Baker, C. J., & Roberston, A. P. (2000). Experimental measurements and computations of the wind-induced ventilation of a cubic structure. Journal of Wind Engineering and Industrial Aerodynamics, 88, 213–230.

Teodosiu, C., Ilie, V., & Teodosiu, R. (2014). Appropriate CFD turbulence model for improving indoor air quality of ventilated spaces. Mathematical Modeling in Civil Engineering, 10(4).

Uematsu, Y., & Isyumov, N. (1999). Wind pressures acting on low-rise buildings. Journal of Wind Engineering and Industrial Aerodynamics, 82, 1–25.

VDI 3783 Part 12. (2000). Environmental meteorology – Physical modelling of flow and dispersion processes in the atmospheric boundary layer – Application of wind tunnels. Beuth Verlag.

Wang, M., & Chen, Q. (2009). Assessment of various turbulence models for transitional flows in an enclosed environment (RP-1271). HVAC&R Research, 15(6), 1099–1119.

Waqas, A., & Ud, Z. (2013). Phase change material (PCM) storage for free cooling of buildings — A review. Renewable and Sustainable Energy Reviews, 18, 607–625.

Wu, F., Sarkar, P. P., Mehta, K. C., & Zhao, Z. (2001). Influence of incident wind turbulence on pressure fluctuations near flat-roof corners. Journal of Wind Engineering and Industrial Aerodynamics, 89, 403–420.

Yang, T., Wright, N. G., Etheridge, D. W., & Quinn, A. D. (2006). A comparison of CFD and full-scale measurements for analysis of natural ventilation. International Journal of Ventilation, 4(4), 337–348.

Zheng, J., Qihua, T., & Li, L. (2020). Wind pressure coefficient on a multi-storey building with external shading louvers. Applied Sciences, 10(3), 1128.