Analysis of thermal comfort and indoor air flow characteristics for a residential building room under generalized window opening position at the adjacent walls

D. Prakash a,*, P. Ravikumar b

a SASTRA University, Thanjavur, Tamil Nadu, India
b Mechanical Engineering Department, Mahendhra Engineering College, Thiruchengodu, Tamil Nadu, India

Received 26 June 2013; accepted 2 February 2015

Abstract

Thermal comfort is an imperative factor that determines the health and productivity of the occupants living in residential buildings. The growing health related symptoms and demand for the electrical energy encourage the occupants to switch over to natural ventilation. Thermal comfort for naturally ventilated buildings mainly depends on the size and orientation of window openings. Even though most research works include the study on indoor thermal comfort for various positions of window opening it was limited to single sided and cross ventilated buildings. In real situation most of the rooms attached to the residential buildings are having window openings at their adjacent wall and hence this paper was focused to study the occupants’ thermal comfort and indoor air flow characteristics for a room with adjacent window openings under generalized approach. Computational fluid dynamics (CFD) technique is employed to study the indoor air flow for a three-dimensional room model. The CFD simulation is checked for grid independence and having good validation with experimental measurements on the reduced scale model at wind tunnel and with the network model with the $k$–$\varepsilon$ turbulence model. Air temperatures along various midlines, planes, areas occupied by low temperature zone and predicted mean vote (PMV) contours are presented in this paper. From this study a new set of strategies are identified to locate the window openings and the best location improves percentage of low temperature by 50%, reduces the PMV and PPD by 0.12% and 3.51%, respectively with reference to the worst window opening position.

© 2015 The Gulf Organisation for Research and Development. Production and hosting by Elsevier B.V. All rights reserved.

Keywords: Thermal comfort; CFD; Ventilation; Network model; Wind tunnel

1. Introduction

People living in hot and arid climatic zones have been affected by many building related health symptoms like head ache, heat stroke, dehydration, frostbite, lung diseases etc due to lack of thermal comfort. These health symptoms ultimately reduce the productivity level of the occupants and hence occupants are paying more attention...
to keep their indoor environment under good thermal comfort. Thermal comfort is the state of mind that expresses the satisfaction with the thermal environment. Thermal comfort can be maintained by dissipating the heat generated by the body due to metabolic activity to the environment and thereby the thermal equilibrium was maintained with the surrounding. The indoor thermal comfort can be maintained by providing proper ventilation which in turn depends on many factors like wind force, outdoor temperature, surrounding building topography, height, shape, orientation, window opening type, size and its position. Among the above factors, window opening can be designed easily by the engineer for good ventilation and thermal comfort and rest of the factors could not be easily controlled.

In this context, Gao and Lee (2011) evaluated the influence of opening configuration on natural ventilation performance of residential unit at Honk Kong and stated that relative position of the two window opening groups (bed room windows and living room windows) was the most affecting parameter. Also better natural ventilation performance can be achieved when the two openings are positioned in opposite direction or perpendicular to each other. Hassana et al. (2007) investigates the effect of window combinations on ventilation characteristics for thermal comfort in buildings. The author also stated that for single sided ventilation with two non adjacent openings gives better ventilation than adjacent openings. Seifert et al. (2006) studied the airflow for the building envelop having window opening nearer to the roof and floor level, windward opening nearer to the roof and leeward opening starts from the mid height of the building and both the window openings nearer to the roof level. Stavrakakis et al. (2012) optimized the window opening design for thermal comfort in naturally ventilated building by the artificial neural network method. Ramponi and Bloken (2012) studied the physical diffusion of indoor air for the building envelope with both openings near to the floor and at the mid height of the building under 5% and 10% of wall porosity. Favarolo and Manz (2005) also analyzed the influence of opening configuration on natural ventilation performance by flow visualization and CFD technique and identified that ventilation performance is mainly affected by its vertical position and width of the opening. However, all the above studies are focused to investigate the effect of window opening orientation and its size for the building envelope with single sided ventilation or cross ventilation. Similarly other studies on building ventilation performance are also made on sided ventilation or cross ventilation. Allocca et al. (2003) analysed the single sided natural ventilation to study the effect of buoyancy on the ventilation rate. Straw et al. (2000) presented the results of experimental, theoretical and computational investigations of the wind driven ventilation with openings on the opposite walls of the room. Caciolo et al. (2012) developed a new set of empirical relations in leeward conditions based on the CFD simulation and full scale experiments. These correlations have been set up to assess the airflow rate due to the combination of stack and wind effects. Kato et al. (1992) investigated the mechanism of cross ventilation by the LES model in a room with opening at their opposite walls. Karava et al. (2011) made an experimental study of basic cross ventilation flow characteristics that are essential for accurate natural ventilation modeling and design. The authors also mentioned that the airflow pattern in room with cross ventilation is complex and cannot be predicted by simplified macroscopic models such as orifice equation. Liang ji et al. (2011) investigated the influence of fluctuating wind direction on cross ventilation using wind tunnel experiments with the aim of improving the evaluation accuracy for natural ventilation. Similarly, the recent research works are also pertained to study the air flow in a room with single sided ventilation (Zhen and Kato, 2011; Caciolo et al., 2011) and cross ventilation (windows at the opposite walls) (Hu et al., 2008; Lo and Novoselac, 2013; Nikolopoulos et al., 2012; Lo and Novoselac, 2012; Lo et al., 2013).

But, in real case, the rooms attached with residential buildings have their openings at the adjacent walls and not at their opposite walls. The air flow for the room with window openings at their adjacent walls is entirely different from single sided and cross ventilated buildings. The air enters through the window openings at the windward side wall and turns 90°, travels along the wall and leaves through the window opening at the adjacent wall. The path of air travel depends upon the relative position of window opening at the adjacent walls. Ravikumar and Prakash (2009) studied the effect of window opening size and its aspect ratio on indoor airflow characteristics for the room with window openings at adjacent walls and the importance of investigating such a type of window orientation is also pointed out. The air flow pattern inside a room is complex and is characterized by multi-flow features such as laminar boundary layers, highly turbulent diffuser jets and low turbulence flow.

Air flow analysis can be performed by methods like theoretical models, experimental testing and Computation Fluid Dynamics (CFD) technique. In order to obtain reliable information concerning the air flow and the pressure distribution around and inside a naturally ventilated building, full-scale measurements can be performed (Straw et al., 2000; Koinaklis, 2005). However, wind tunnel tests on small-scale models are usually preferred, as they allow the control of wind speed and direction, as well as the study on different configurations (Eftekhari et al., 2003; Wong and Heryanto, 2004), Walker and White (1992), Nielsen and Olsen (1993), Dascalaki et al. (1995), Zeidler and Fitzner (1997) used the tracer gas measurement technique for their air flow studies. The study of complex flow patterns by experimental approach is highly infeasible and it provides the flow pattern details only at the specified locations. An alternative approach is to rely on CFD, based on
the numerical solution of the set of governing equations which describe the flow field. CFD-based programs are widespread and provide a detailed description of the air flow. In recent years, their application has become more and more popular, thanks to the increase of computational power and to the improvements in turbulence modeling (Chen, 2004). CFD technique is very powerful and spans a wide range of industrial and non-industrial application areas. It provides the capability to investigate the complex flow structures and provide detailed results at every point in the flow domain. With the aid of CFD, Hoang et al. (2000), Rouaud and Havet (2002) and Song and Kato (2003) performed air flow analysis in various buildings. With all these information, present article made an attempt to implement CFD technique in the analysis of air flow in a room by changing the position of its windows at the adjacent walls.

2. Room with adjacent window opening: CFD simulation and Validation

The office room under investigation is of size 5 m × 5 m × 5 m (W × H × L) and having two similar window openings of size 1 m × 1 m located at the adjacent walls as shown in Fig. 1a. The building model may be created either in 2-dimensional or 3-dimensional geometry. Visagavel and Srinivasan (2009), Evalo and Popov (2006) conducted the air flow simulation with a two dimensional geometry. However, the two dimensional model does not give realistic simulation of airflow, as it does not consider airflow separation over building sharp edges. Also in the two dimensional analysis, the flow characteristics can be predicted only for the limited locations. Such a 2-dimensional building model provides realistic solution if the model, and the applied boundary conditions are symmetrical to any plane. Hence, three dimensional modeling of geometry is the most appropriate choice for this study. However, the disadvantage in the three dimensional simulation is that it consumes more time for solving the fluid domain to the required convergence level. The external flow over the building envelope is also simulated by modeling the external atmospheric zone. After performing many simulation cases for identifying the external atmospheric zone size, it was predicted finally as 30 × 30 × 20 m (W × L × H). In this external atmospheric zone, the test room under research is located centrally as shown in Fig. 1b.

The major assumptions involved in this analysis are as follows:

i. Ventilation due to wind force is only considered.
ii. An isolated room is analyzed.
iii. Air properties are assumed to have a constant value with reference to atmospheric temperature.
iv. Air is entering in a perpendicular direction to the window opening.
v. Fluctuation of wind velocity is assumed to be constant and is neglected.

Based on these assumptions, the governing equations to be solved are as follows:

Continuity equation

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho v_x)}{\partial x} + \frac{\partial (\rho v_y)}{\partial y} + \frac{\partial (\rho v_z)}{\partial z} = 0 \]  \hspace{1cm} (1)

Momentum equation

\[ \frac{\partial \rho v_x}{\partial t} + \frac{\partial (\rho v_x v_x)}{\partial x} + \frac{\partial (\rho v_x v_y)}{\partial y} + \frac{\partial (\rho v_x v_z)}{\partial z} = \rho g_x - \frac{\partial P}{\partial x} + R_x + \frac{\partial}{\partial x} \left( \mu_x \frac{\partial v_x}{\partial x} + \frac{\partial v_x}{\partial y} \frac{\partial v_x}{\partial y} \right) + \frac{\partial}{\partial y} \left( \mu_x \frac{\partial v_x}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_x \frac{\partial v_x}{\partial z} \right) + \tau_x \] \hspace{1cm} (2)

\[ \frac{\partial \rho v_y}{\partial t} + \frac{\partial (\rho v_x v_y)}{\partial x} + \frac{\partial (\rho v_y v_y)}{\partial y} + \frac{\partial (\rho v_y v_z)}{\partial z} = \rho g_y - \frac{\partial P}{\partial y} + R_y + \frac{\partial}{\partial x} \left( \mu_x \frac{\partial v_y}{\partial x} + \frac{\partial v_y}{\partial y} \frac{\partial v_y}{\partial y} \right) + \frac{\partial}{\partial y} \left( \mu_x \frac{\partial v_y}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_x \frac{\partial v_y}{\partial z} \right) + \tau_y \] \hspace{1cm} (3)
\[
\frac{\partial \rho v_x}{\partial t} + \frac{\partial (\rho v_x v_z)}{\partial x} + \frac{\partial (\rho v_x v_y)}{\partial y} + \frac{\partial (\rho v_x v_z)}{\partial z} = \rho g_y - \frac{\partial P}{\partial y} + R_e + \frac{\partial}{\partial y} \left( \mu_e \frac{\partial v_y}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_e \frac{\partial v_y}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_e \frac{\partial v_z}{\partial z} \right) + \tau_z
\]

where \( v_x, v_y, \) and \( v_z \) are the components of velocity in \( x, y \) and \( z \) directions, \( \rho \) is the density, \( t \) is the time, \( g \) is the gravity, \( \mu_e \) is the effective viscosity, \( P \) is the pressure, \( R_e \) is the source term for distributed resistance (suffix \( i \) is \( x, y \) and \( z \)) and \( \tau \) the viscous stress.

Energy equation

\[
\frac{\partial}{\partial t} (\rho C_p T_o) + \frac{\partial}{\partial x} (\rho v_x C_p T_o) + \frac{\partial}{\partial y} (\rho v_y C_p T_o) + \frac{\partial}{\partial z} (\rho v_z C_p T_o) = \frac{\partial}{\partial x} \left( k \frac{\partial T_o}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T_o}{\partial y} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T_o}{\partial z} \right) + W^e + E_k + Q_v + \phi + \frac{\partial P}{\partial t}
\]

where \( C_p \) is the specific heat, \( T_o \) is the total temperature, \( K \) is the thermal conductivity of air, \( W^e \) is the viscous work term, \( Q_v \) is the volumetric heat source term, \( \phi \) is the viscous heat generation term and \( E_k \) is the kinetic energy.

3. Boundary conditions and validation of the CFD simulation

In general, velocity of air flow varies with respect to building height. This variation is specified either by logarithmic profile (Evalo and Papov, 2006) or by dividing the velocity inlet into a number of sub inlet zones (Asfour and Gadi, 2007). In this paper, the wind entering zone is divided into 4 sub zones. The wind velocity at these sub zones are predicted from the following Eq. (6) (Asfour and Gadi, 2007).

\[
V = V_t c H^a
\]

where, \( V \) is the wind speed at ground level (m/s), \( V_t \) is the reference wind speed measured experimentally, \( H \) is the height of the building, \( c \) is the parameter relating wind speed to terrain nature (0.68 in the open country terrain), \( a \) is the exponent relating wind speed to the height above the ground (0.17 in the open country terrain). Ambient temperature is specified as 306 K. Free slip boundary conditions are employed to the wall surfaces. The temperature value at the side walls, floor and roof is specified as 312, 303 and 325 K, respectively. These values are based on midday measurements conducted in an actual building during the summer. Since the room analyzed in this paper is an office room some electrical appliances used by the occupants are considered and their heat generation value is assumed to be 25 W/m². This generated heat is uniformly applied to the floor as a boundary condition. T grid scheme of meshing is employed and this scheme uses tetrahedral shaped element for meshing the flow domain. The grid size in meshing must be independent to the results obtained from CFD simulation and hence a grid independence check is required to conduct. For this grid independence check, grid sizes of 0.9, 0.8, 0.7, 0.6 and 0.5 m are used in meshing and solved with the above mentioned boundary conditions. The air temperature and velocity at the midline \( Y_1 Y_2 \) are determined for different mesh sizes and shown in Fig. 2. Three-dimensional, double-precision, segregated solver is used to solve the governing equations sequentially. Standard \( k-\varepsilon \) turbulence model is employed with standard wall function (Launer and Spalding, 1974). It is a two equation model in which the solutions of two separate transport equations allowing the turbulence velocity and length scale are to be independently determined. This model was widely used in industrial flow and heat transfer simulation with reasonable accuracy and robustness. For this \( k-\varepsilon \), the values of turbulent kinetic energy, \( K \) and turbulent kinetic energy dissipation rate, \( \varepsilon \) at the inlet region are calculated from Eqs. (7) and (8), respectively (Versteeg and Malalasekera, 1995).

\[
K = \frac{3}{2} \left( V_{avg} \times T_i \right)^2
\]

\[
\varepsilon = C_\mu k^2 \frac{\varepsilon}{T_i}
\]

where \( V_{avg} \) is the average flow velocity, \( T_i \) the turbulence intensity and \( l_i \) is the turbulence scale length. \( T_i \) and \( l_i \) are taken as 4% (Posner et al., 2003) and 0.4 m (AIRPAK, 2001) respectively and \( C_\mu = 0.09 \). The \( K \) and \( \varepsilon \) values at the velocity inlet boundary condition are calculated as 0.0024 and
4.82 × 10⁻⁵, respectively. All the cases are iterated up to the convergence level of 10⁻⁶. In the solution control, the second order upwind method is specified.

From Fig. 2, it is clearly identified that mesh of 0.6 and 0.5 are result independent since both the mesh sizes reproduce same air temperature and velocity along the midline Y₁Y₂ with negligible difference. Approximately 42 cells in the x direction, 33 cells in the y direction and 42 cells in the z direction are required for meshing the model with size of 0.6 m. Based on the above methodology the window opening position on the adjacent walls are changed as given in the Table 1 and the indoor air flow characteristics are analyzed and predicted at the locations shown in Fig. 3. In Fig. 3, the position of the mid lines X₁X₂, Y₁Y₂ and Z₁Z₂ and planes P₁, P₂, P₃ and P₄ are shown. Also in this study, low temperature zone was predicted which refers to the zone having the temperature in the range of 306–307 K.

3.1. Validation of CFD simulation with wind tunnel experiment

Wind tunnel experiments are generally considered the most reliable source of pressure data for building in the design phase. Wind tunnel measurements were performed to analyze the flow for a simplified model (Yang et al., 2006; Karava and Stathopoulos, 2011). In this study, the open circuit Boundary layer wind tunnel was used to predict the pressure coefficient over the cube surface which refers to the room model without window openings. The wind tunnel is of 4 m long and has cross sections of 0.3 × 0.3 m² which is enough to conduct the test for the reduced scale model. The measurement of pressure over the surface of the model was performed in Pitot tube manometer setup attached in front of the wind tunnel setup. The building model at a scale of 1:35 was built from a wooden cardboard and had a dimension of L/W/H = 0.15 × 0.15 × 0.115 m³ (5 × 5 × 4 m³ in full scale). Surface pressure was measured at 12 locations on the vertical facade and 7 locations at the top surface. Small tubes of diameter 3 mm are inserted in the holes of the test cube and the other end of the tubes are connected to the Pitot tube manometer. The pressure coefficient along the cube surface is predicted from Eq. (2.22).

\[
C_p = \frac{P - P_o}{(0.5 \times \rho \times v^2)} \quad (2.22)
\]

where \(P\) is the static pressure at the tested location

\(P_o\) – Free stream static pressure
\(\rho\) – Free stream density
\(v\) – Free stream velocity

The same cubic model was created in a full scale in the GAMBIT software and the flow over the cube surface is simulated in the Fluent software. The pressure coefficients along the cube surface are predicted, compared with the wind tunnel data and shown in Fig. 4.

In Fig. 4, the CFD predicted pressure coefficient values along the cube surface have the trend similar to the pressure coefficient obtained from the scaled model testing at wind tunnel. However, the CFD predicted \(C_p\) values are having a marginal deviation at the wind ward side and having comparatively good agreement on the roof and leeward side. Since the deviation is in the negligible level, the CFD simulation is considered to have good agreement with the experimental predictions. Montazeri and Blocken (2013), Hoof et al. (2012), Ramponi and Blocken (2012) also used the reduced scale model for the analysis of airflow through wind tunnel test rig and their corresponding CFD simulations are validated with the experimental predictions.

3.2. Validation of CFD simulation with network model

The CFD simulation is also compared with the network model for the cubic model with window openings at their adjacent wall. This comparison adds the validation of CFD simulation. In this validation process, the mass flow rate of air passing through the window openings is

Table 1
<table>
<thead>
<tr>
<th>Test case details for generalized window opening position.</th>
<th>Equations</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Changing the position of window in y direction</td>
<td>(h^* = \Delta H/H)</td>
<td>0.1, 0.2, 0.3, 0.4, 0.5, 0.6 and 0.7</td>
</tr>
<tr>
<td>Changing the position of window in x and z directions at same height</td>
<td>(x^* = \Delta W/W), (z^* = \Delta L/L)</td>
<td>0.1, 0.2, 0.3, 0.4, 0.5, 0.6 and 0.7</td>
</tr>
<tr>
<td></td>
<td>(s = x = z)</td>
<td></td>
</tr>
</tbody>
</table>

Figure 3. Identified locations of test case room for result prediction.
determined from the CFD simulation and it is compared with the mass flow rate predicted from the network model. This method could be useful for studies that have no access to laboratory or full-scale testing facilities. Test cases in the Table 1 are simulated through CFD technique and the mass flow rate obtained at the windward side opening is compared with the mass flow rate predicted from the network model. Larsen et al. (2011), Stavarakakis et al. (2008) also reported to use the network model for validating the CFD simulation if the required experimental results are not available. The procedure adopted to determine the mass flow rate of air due to wind pressure difference by the network model is as follows:

\[
Q_{\text{network}} = A_{\text{Effective}} \sqrt{2 \Delta p / \rho}
\]  

(9)

where \(Q_{\text{network}}\) is the mass airflow rate, \(A_{\text{Effective}}\) is the effective area of the opening, \(\rho\) is the air density and \(\Delta p\) is the pressure difference calculated from Eq. (10) (Asfour and Gadi, 2007).

\[
\Delta p = 0.5 \rho V^2 |C_{pn} - C_{pi}|
\]  

(10)

where \(V\) is the wind velocity, \(C_{pn}\) and \(C_{pi}\) are the pressure coefficients at the opening \(n\) and at the inside space respectively.

By substituting Eq. (10) in Eq. (9), the resulted \(Q_{\text{network}}\) is given in Eq. (11).

\[
Q_{\text{network}} = A_{\text{Effective}} V (C_{pn} - C_{pi}) (|C_{pn} - C_{pi}|)^{-1/2}
\]  

(11)

By Law of Conservation of mass, the above equation can be written as Eq. (12) (Asfour and Gadi, 2007).

\[
\sum_{n=1}^{N} A_{\text{Effective}} V |(C_{pn} - C_{pi})^{-0.5} + |C_{p(n+1)} - C_{pi}|^{-0.5} = 0
\]  

(12)

In the present study, pressure coefficients (\(C_{pn}\)) at the windward and leeward sides window openings are 0.7 and −0.5, respectively. By using the \(C_{pn}\) values in Eq. (12), the internal pressure coefficient (\(C_{pi}\)) is calculated as 0.1. From this \(C_{pn}\) value, \(\Delta p\) is predicted and in turn it is substituted in equation 9 for the determination of the \(Q_{\text{network}}\) as 0.76 kg/s. Asfour and Gadi (2007) used the same method for validating the CFD simulation. Discrepancy between the CFD results and network model is provided in Table 2. The maximum discrepancy is obtained as 7.12% for the case of \(h^* = 0.1\) (\(h^*\) is nondimensional which refers the location of window opening from the ground surface). Hence, the CFD model is in reasonable agreement with the network model. Such a method can be recommended as a validation method for studies that have no access to laboratory or full-scale testing facilities (Asfour and Gadi, 2007).

4. Effect of window opening position from the ground surface on indoor air flow characteristics

The position of window opening with reference to the ground level is studied by changing the \(h^*\) values from 0.1 to 0.7 with an increment of 0.1. The temperature variations along the midlines \(X_1X_2\), \(Y_1Y_2\) and \(Z_1Z_2\) for various \(h^*\) values are shown in Fig. 5.

The temperature variation along the midline \(X_1X_2\), decreases gradually from \(X = 0.5\) to 4.5 m for all the \(h^*\) cases. Among the analyzed cases, \(h^* = 0.7\) and 0.1 have the maximum temperature along the \(X_1X_2\) midline. For \(h^* = 0.2\), 0.3 and 0.4, the temperature variation is very similar and especially in between the distance \(X = 1.5\) to 4 m, the temperature is comparatively low. In the midline \(Y_1Y_2\), for \(h^* = 0.1\), the temperature is minimum as 306.5 K up to 1 m from the ground surface and a drastic rise in temperature is identified as 307.125 K between 1 and 3 m. However, for \(h^* = 0.2\), the temperature is almost constant up to \(Y = 3.5\) m and above 3.5 m the temperature rises drastically.

For \(h^* = 0.3\), the temperature magnitude varies gradually from 306.5 to 308 K along \(Y_1Y_2\). Temperature variation is found similar for the cases \(h^* = 0.3\) and 0.4. For the cases, \(h^* = 0.3\) and 0.4, the temperature rises slightly up to 1.75 m and between 1.75 and 3 m, a drastic fall in temperature is identified and above 3 m the rise in temperature is drastic. In the cases of \(h^* = 0.6\) and 0.7, the temperature is almost constant at about 307 K up to a height of 3.5 m and above 3.5 m a drastic rise in temperature is noticed. For all the \(h^*\) cases, the temperature is raised in a drastic manner at the height of 4 m above the ground surface. The temperature trend along \(Z_1Z_2\) for all \(h^*\) values are almost constant between 1.5 and 4.5 m in the Z axis. In the \(Z_1Z_2\) midline also, \(h^* = 0.2, 0.3,\) and 0.4 yield the low temperature in comparison with rest of the \(h^*\) cases.

The velocity variations along the midlines \(X_1X_2\), \(Y_1Y_2\) and \(Z_1Z_2\) for the analyzed \(h^*\) values are shown in Fig. 6.

In the \(X_1X_2\) midline, for all the \(h^*\) values, the velocity increases from \(X = 0.5\) to 1 m and between 1 and 4 m the velocity is almost constant and between 4 and 5 m, a slight drop in velocity is noticed. Among the analyzed \(h^*\) cases, \(h^* = 0.2, 0.3, 0.4\) and 0.5 follow similar pattern of velocity.
variations along $X_1X_2$. The velocity magnitude raises negligibly from the cases $h^* = 0.2$–0.5. For $h^* = 0.1$ and 0.6, the air velocity between $X = 1$ and 4 m has low magnitude and it is almost constant as 0.125 m/s. However, this low magnitude velocity is 0.25 m/s lesser than other $h^*$ cases except for $h^* = 0.7$. For $h^* = 0.7$, the velocity variation follows a unique trend and it is having a least magnitude of air velocity along $X_1X_2$.

In the $Y_1Y_2$ midline, all the $h^*$ values yield different types of trend. For $h^* = 0.1$, the indoor air velocity is maximum up to $Y = 1$ m, and above 1 m, the velocity gradually reduced to 0.15 m/s at $Y = 4.5$ m. For $h^* = 0.2$, the maximum velocity is obtained between 1 and 1.75 m as 0.35 m/s and above 1.75 m, the velocity gradually reduced to 0.5 m/s at $Y = 4.5$ m. Also for the cases $h^* = 0.3$ and 0.4, a similar trend in velocity variation is identified along $Y_1Y_2$.

### Table 2
Discrepancy of CFD simulation with network model for generalized window opening positions.

<table>
<thead>
<tr>
<th>$h^*$</th>
<th>0.1</th>
<th>0.2</th>
<th>0.3</th>
<th>0.4</th>
<th>0.5</th>
<th>0.6</th>
<th>0.7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Predicted mass flow rate form CFD code (kg/s)</td>
<td>0.698</td>
<td>0.704</td>
<td>0.706</td>
<td>0.713</td>
<td>0.724</td>
<td>0.732</td>
<td>0.739</td>
</tr>
<tr>
<td>Discrepancy with network model (%)</td>
<td>7.12</td>
<td>6.3</td>
<td>6.17</td>
<td>5.35</td>
<td>4.055</td>
<td>3.11</td>
<td>2.28</td>
</tr>
<tr>
<td>$x^* = x^* = z^*$</td>
<td>0.1</td>
<td>0.2</td>
<td>0.3</td>
<td>0.4</td>
<td>0.5</td>
<td>0.6</td>
<td>0.7</td>
</tr>
<tr>
<td>Predicted mass flow rate form CFD code (kg/s)</td>
<td>0.715</td>
<td>0.712</td>
<td>0.7007</td>
<td>0.713</td>
<td>0.700</td>
<td>0.710</td>
<td>0.711</td>
</tr>
<tr>
<td>Discrepancy with network model (%)</td>
<td>5.06</td>
<td>5.43</td>
<td>6.8</td>
<td>5.35</td>
<td>6.8</td>
<td>5.68</td>
<td>5.58</td>
</tr>
</tbody>
</table>

---

Figure 5. Indoor temperature variation along the midline for various $h^*$ values.
and for these cases the maximum velocity of 0.4 m/s is obtained at $Y = 2$ and 3 m, respectively. For $h^* = 0.5$ and 0.6, the velocity was reduced gradually from $Y = 0.5$ to 2 m and increased suddenly to a maximum magnitude of 0.3 m/s at a height of $Y = 3.25$ and 3.75 m, respectively. For $h^* = 0.7$, the indoor air velocity is reduced gradually up to 2.75 m and an increase in velocity is noticed up to 3.75 m from the ground surface and above which, a slight fall is identified.

In the $Z_1Z_2$ midline, all the $h^*$ values show a rise in indoor air velocity up to $Z = 1$ m and beyond 1 m, the velocity is reduced gradually up to a distance of $Z = 3$ m. The velocity variation for the cases $h^* = 0.2$, 0.3, 0.4 and 0.5 shows similar pattern of trend. For these $h^*$ values, the velocity reduces from $Z = 1$ to 4 m however, the variation in velocity among the $h^*$ cases is 0.05 m/s. The velocity magnitude increased slightly by changing the $h^*$ values from 0.2 to 0.5 along $Z_1Z_2$ midline. For rest of the $h^*$ values (0.1, 0.6, 0.7), the velocity reduces significantly from $Z = 1$ to 3 m and this change in velocity is about 0.125 m/s.

Furthermore, the temperature prevailed in the room is discussed from the temperature plots predicted at the planes P1, P2, P3 and P4 as shown in Fig. 7. For $h^* = 0.1$, at the plane P1, the least temperature is identified nearer to the corner C3 as 307 K and increasing diagonally toward the corner C1. For the plane P2, except the portions nearer to the corner C1 experience low temperature and is less than 307 K. Planes P3 and P4 are having almost similar pattern of temperature contours with the lowest temperature of 307 K that is nearer to the corner C3 and increase diagonally toward the corner C1. For $h^* = 0.2$, all the planes experience low temperature at about 307 K and some portions near the
windward side opening are having comparatively very low temperature of about 306 K. For $h^* = 0.3$, the central region of the plane P1 is having low temperature of about 307 K and for the plane P2 the zone along the path of air travel through the windward window opening is having low temperature. Also in planes P3 and P4 some portions nearer to walls W3 and W4 are having low temperature at about 307 K and portions nearer to wall W1 are having high temperature as above 309 K. For $h^* = 0.4$, plane P3 is having comparatively low temperature and the portions except the corner C1 are experiencing low temperature in the range of 306–307 K. In the plane P4, the portion nearer to wall W3 is having low temperature and the corner C3 is having a temperature of 308.75 K. Plane P1 is having almost uniform temperature at about 307 K except the corner C1. For the plane P2, half of the plane nearer to wall W4 is having comparatively low temperature at about 307 K. For $h^* = 0.5$, 0.6 and 0.7 the temperature pattern at the planes P1 and P2 is almost similar. In these planes some comparatively low temperature zones are noticed nearer to the wall W4 and rest of the portions are having higher temperatures. However for $h^* = 0.5$, some portions along the path of air travel at plane P3 are having low temperature at about 307 K and portions nearer to wall W2 are having elevated temperatures. Similarly for $h^* = 0.6$ and 0.7, the low temperature zone is identified only at the plane P4 and rest of the planes are having uniform high temperature. From these plots, it is inferred that window opening position in the range of $h^* = 0.1$–0.3 offers more low temperature zone at all the planes whereas if it is in the range of $h^* = 0.5$–0.7 only the plane P4 is offering low temperature. Hence positioning the window opening in the range of $h^* = 0.1$–0.3 provides good comfort at the floor level and also up to a height of 3 m from the floor.

For $h^* = 0.1$, plane P1 has some low temperature zones whereas P2, P3 and P4 planes are affected by high temperatures. For $h^* = 0.2$, P1 and P2 have some downs in the temperature level and also the average peak temperature value is 308.325 K which is 0.3 K less than $h^* = 0.1$. For $h^* = 0.3$, P1 and P2 will have the peak temperature value as 307.7 and 308 K, respectively. These values are 0.1 K less than the peak temperature value obtained for the case $h^* = 0.2$. For $h^* = 0.3$, planes P1 and P2 have elevated temperatures in comparison with the case $h^* = 0.2$. For $h^* = 0.4$–0.7, temperature pattern shows some downs at the level of window position and rest of the planes are affected by elevated temperatures.

Figure 7. Indoor temperature plot at planes P1, P2, P3 and P4 for various $h^*$ values.
To identify the best position of window opening from the ground surface, another factor - area occupied by low temperature zone is predicted at the planes P1, P2, P3 and P4 and shown in Fig. 8.

For $h^* = 0.1$, 0.2 and 0.3 the maximum area of low temperature zone is obtained at the plane P1 as 15, 19 and 15.5 m², respectively. Rest of the $h^*$ cases are not having maximum area of low temperature zone at plane P1 and for $h^* = 0.7$ it is completely zero. For the plane P2, the area of low temperature zone starts to increase drastically from the cases $h^* = 0.1$ to 0.2 and almost constant for the $h^* = 0.3$, 0.4 and 0.5 and falls drastically for $h^* = 0.7$. For the plane P3, the area occupied by low temperature zone is less than 10 m² for the cases $h^* = 0.1$, 0.2 and 0.3 and increased gradually as 11.93–15.32 m² for the cases $h^* = 0.4–0.6$, respectively. In the plane P4, the area occupied by the low temperature zone is negligible as 2 m² for the cases $h^* = 0.1–0.4$ and rest of the $h^*$ cases have a slight increase in low temperature zone area as about 5 m². From this figure, it is noticed that, locating the window openings along the line $X_1X_2$, $Y_1Y_2$ and $Z_1Z_2$ is studied by varying the corresponding non dimensional numbers $x^*$ and $z^*$. In this case, the position of both windows is changed uniformly ($x^* = z^* = s^*$) and the $h^*$ value is kept constant at 0.4. The $s^*$ is varied from 0.1 to 0.7 with an increment of 0.1. The temperature along the line $X_1X_2$, $Y_1Y_2$ and $Z_1Z_2$ is shown in Fig. 9.

The temperature trend along $X_1X_2$ for various $s^*$ values reveals three types of trends. For $s^* = 0.1$, 0.2 and 0.3, the temperature decreases gradually along the $X_1X_2$ midline.

However, for $s^* = 0.4$, the temperature decreases from $X = 0.5$ to 2 m and maintains almost constant up to 3 m and increases again up to 4.5 m. For $s^* = 0.5$, the temperature is nearly constant up to 2 m and between $X = 2$ and 4.5 m an increase in temperature is noticed. For $s^* = 0.6$ and 0.7, the temperature steadily raises from $X = 1$ to 4.5 m. In the $Y_1Y_2$ midline, the temperature variation for the cases $s^* = 0.3$, 0.4 and 0.5 shows almost constant up to a height of 3 m from the ground surface and above which a drastic raise in temperature is identified.

Rest of the $s^*$ cases, have a steady raise in temperature from the ground surface to the roof. In comparison, $s^* = 0.3$, 0.4 and 0.5 cases have indoor air under low temperature in between the height of 1.75 and 4.5 m. In the $Z_1Z_2$ midline, $s^* = 0.3$, 0.4 and 0.5 yield almost constant temperature along the $Z$ direction. Among the analyzed $s^*$ cases, $s^* = 0.4$ yields the lowest temperature along all the midlines. The velocity variation along the same midlines for various $s^*$ values is shown in Fig. 10.

In $X_1X_2$ midline, for $s^* = 0.1$, 0.2 and 0.3, the air velocity is comparatively very low from $X = 0$ to 3.5 m and have a high velocity between $X = 3.5$ and 4.5 m. This shows that the air circulation in $X_1X_2$ midline is predominant only between $X = 3.5$ and 4.5 m for $s^* = 0.1$, 0.2 and 0.3. For $s^* = 0.4$ and 0.5, the highest air velocity is obtained as 0.2 and 0.3 m/s, respectively and it is noticed at $X = 2.25$ m. For $s^* = 0.6$ and 0.7, the maximum velocity of 0.25 m/s is obtained between $X = 1$ m and beyond 1 m. The velocity is drastically reduced to 0.05 m/s at $X = 4.5$ m. This shows that the air circulation is predominant only in between $X = 0.5$ and 1 m for $s^* = 0.6$ and 0.7. In $Y_1Y_2$ midline, the velocity decreases up to 1 m and maintains constant up to 4 m for all the $s^*$ cases except for $s^* = 0.3$, 0.4 and 0.5.

The variations of air velocity for these $s^*$ cases are less than 0.25 m/s. For $s^* = 0.3$, 0.4 and 0.5, the velocity gradually reduces up to a height of 1.5 m and increases drastically to a value of 0.2 m/s at $y = 3.5$ m and above 3.5 m a significant fall in velocity is noticed. However, for $s^* = 0.3$, 0.4 and 0.5, the indoor air velocity is

![Figure 8. Area of low temperature zone at planes P1, P2, P3 and P4 for various $h^*$ values.](image)
comparatively very much high in between $Y = 2.5$ and $4.5$ m. For $Z_1 Z_2$ midline, the air velocity is increased up to $1$ m and later reduced gradually up to $3.5$ m and again increases slightly up to $Z = 4$ m. This trend is common for all the $s^*$ cases. However among the analyzed $s^*$ cases, the indoor velocity is high for $s^* = 0.3, 0.4$ and $0.5$ at all midlines.

The indoor air temperature at the planes $P1, P2, P3$ and $P4$ is predicted for various $s^*$ cases and is shown in Fig. 11. For $s^* = 0.1$, a smaller region nearer to the corner C2 at plane P1 is having comparatively low temperature as $307$ K; in plane P2 a narrow portion along the wall W4 is having low temperature and this portion is having further reduction in temperature at the plane P3 and rest of the portions at the P3 is having elevated temperature. In plane P4 the temperature is increasing diagonally from corner C3 to C1. For $s^* = 0.2$, planes P1, P2 and P3 are having low temperature at one half of the plane nearer to the wall W4 where as the next half that is nearer to wall W2 is having elevated temperature in the range of $308–309$ K. For $s^* = 0.3$ at plane P1, portion nearer to the corner C2 is having comparatively low temperature and less than $307$ K. In plane P2, wide region away from the corner C1 is having low temperature in the range of $306–307$ K and this portion is having further reduction in temperature in plane P3. However for plane P4, the temperature is lower at the corner C3 and increases diagonally toward the other corners. For $s^* = 0.4$, one quadrant of the plane P1 nearer to the corner C1 is having high temperature as $308$ K and rest of the portions are having relatively lower temperature at about $307$ K. In the same case, the planes P2 and P3 are having almost similar pattern of temperature contour however the temperature at the plane P2 is com-
paratively lesser than P3. In both the planes the portions nearer to the corner C1 are having relatively higher temperature. Also in the plane P4, the portions nearer to the wall W3 are having low temperature and rest of the portions are having higher temperature, especially at the region nearer to wall W1 having temperature greater than 308 K. For $s^* = 0.5$, all the planes are having comparatively lesser temperature than in the case $s^* = 0.4$. Also the high temperature region nearer to the corner C1 gets decreased for $s^* = 0.5$ in comparison with 0.4. Specifically the plane P3 in $s^* = 0.5$ is having a large zone under the temperature of 307 K and however in plane P4 certain portions nearer to the corner C1 and C3 show elevated temperature as above 308 K. For $s^* = 0.6$ at plane P1, portion except the zone nearer to wall W3 is having low temperature. But in planes P2 and P3, the low temperature zone moves toward the wall W2 and plane P4 shows most regions at elevated temperature of above 309 K. Finally for $s^* = 0.7$ at plane P1, a marginal zone nearer to the walls W2 and W1 has high temperature, plane P2 is having uniform temperature of about 307 K and planes P3 and P4 experience high temperature at above 309 K. With all these inferences, it is noticed that $s^* = 0.5$ shows almost uniform and lower temperature at all the planes in comparison with other $s^*$ cases.

Area occupied by the low temperature zone is predicted at planes P1, P2, P3 and P4 for the analyzed $s^*$ cases and is shown in Fig. 12.

From Fig. 12, $s^* = 0.3$, 0.4 and 0.5 yield an average low temperature zone area of 12.4, 13.4 and 14.3 m$^2$, respectively. For these $s^*$ cases, around 64% of the room interiors are kept at low temperature. However, for $s^* = 0.1$, 0.2, 0.6 and 0.7, the average low temperature
The zone area is 3.6, 10.01, 10.15 and 9.29 m², respectively. These $s^*$ values keep only 40% of the room interior with low temperature. All these discussions recommend to position the windows in the range of $s^* = 0.3\text{–}0.5$ for reducing indoor air temperature.

6. Thermal comfort index – PMV contour for varying window position

The thermal comfort index – predicted mean vote for the generalized position of window opening studies is deter-
mined by Fanger’s Eq. (13), for a metabolic rate = 70 W/m², work completed = 30 W and thermal resistance of clothing = 0.11 (m²k)/W and relative humidity = 60%.

PMV = (0.303e⁻⁰.⁰³⁶M + 0.028)((M - W) - 3.05 × 10⁻³(5733
- 6.99(M - W) - p_a) - 0.42(M - W)
- 58.15) - 1.7 × 10⁻³M(5867 - p_a)
- 0.0014M(34 - T_a) - 3.96
× 10⁻⁸f_cl[(T_cl + 273)⁴ - (T_r + 273)⁴]
- f_clh_c(T_cl - T_a))

(13)

where f_cl = factor of clothing, T_cl = clothing temperature, h_c = convective heat transfer coefficient.

In Fig. 13, the predicted mean vote contour at the mid plane of a room parallel to the floor for the analyzed h* cases are shown.

For h* = 0.1, the maximum PMV value is 2.48 and minimum PMV value is 2.25. The PMV value is changing subsequently in all the areas. No single large area is possessed by a unique PMV value. For h* = 0.2, half of the room area has a PMV value of 2.24 and this zone is identified as the most comfort zone in comparison. For h* = 0.3, the central region is affected by a PMV of 2.28 and the portion near the wall face W3 having a PMV of 2.25. Also at the portion along the wall W2, the PMV value is varying drastically from 2.41 to 2.31. For h* = 0.4, major portions of the room are having a PMV value of 2.27 and the region nearer to the wall face W2 has a PMV values of 2.30–2.42. For h* = 0.5, two large areas are affected by a PMV values of 2.26 and 2.29. For h* = 0.6, the central region of the room is affected by PMV value of 2.29 and also the regions nearer to wall face W4 have variations in the PMV values from 2.31 to 2.37. At h* = 0.7, no one large area has a unique PMV value and the PMV value is varying from 2.32 to 2.39 which corresponds to the zone nearer to W4 and W2, respectively. From this PMV contour study, h* = 0.2 is identified as the best position of window openings.

The PMV contours for the analyzed s* cases are shown in Fig. 14.

For s* = 0.1, the window openings at the adjacent walls are very closer to each other. This causes the air to enter and vent out immediately without traveling the interior of the room. Hence, the interior portions of the room have a PMV value of 2.45 and above. For s* = 0.2, the PMV value gets reduced comparatively and the portions nearer to wall face 4 experiences a low PMV value of 2.24. For s* = 0.3, one large zone is affected by a PMV value of 2.25 and the portions nearer to wall face 2 experience PMV values in the range of 2.28–2.41. For s* = 0.4, the windows are positioned at the center of the wall. In this case, the PMV contours are somewhat symmetrical about the mid horizontal axis. Also the central region of the room has a PMV value of 2.23 and the PMV value gets increased toward both the wall faces 2 and 4. For s* = 0.5, one large portion is having a PMV of 2.24 and the zone nearer to wall face 4 is affected by a PMV of 2.27. For s* = 0.6, the portions nearer to the wall face W2 is having a PMV of 2.25 and the subsequent regions toward the wall face W4 have a rise in PMV value. For s* = 0.7, the central zone is having a high PMV of 2.32 and the portions nearer to wall face W2 are having the PMV of 2.27 and the corner C2 is affected by high PMV of 2.47. From all the s* cases,
the room corner C1 is experiencing high PMV value. However, among the analyzed \( s^* \) cases, \( s^* = 0.4 \) provides a better and almost uniform comfort.

7. Conclusion

This paper made an attempt to study the indoor airflow characteristics and thermal comfort of a room with adjacent window openings under generalized position of window openings. The indoor air flow is simulated through CFD technique and the CFD simulation is validated with experimental results from wind tunnel test and with network model. In the first segment, position of window openings from the ground is generalized with the building height as a non-dimensional number (\( h^* \)); while in the second part, the position of window opening in the windward and leeward sides is generalized with the building width and length as \( x^* \) and \( z^* \), respectively. Also the \( x^* \) and \( z^* \) values are equally varied and hence the \( x^* \) and \( z^* \) are commonly referred as \( s^* \). Following conclusions are made from the outcome of CFD simulation under two segments. In the first segment, positioning the windows near to the floor (\( h^* = 0.1 \)) does not provide better ventilation, since at low level the velocity of air entering the room is low. Also the incoming air collects the heat only from the floor and not from other building surfaces. If the windows are positioned near to the roof (\( h^* = 0.7, 0.6, 0.5 \) and 0.4), the air collects heat from the roof surface and not from the floor. This causes a local heating up to height of 2 m from the ground level. The non dimensional number for the best position of window opening from ground surface is identified at \( h^* = 0.2 \), that increases the percentage of low temperature zone by 65%, reduces the PMV and PPD by 0.05% and 1.66%, respectively with reference to the worst window position from the ground surface (\( h^* = 0.7 \)). Further in the second segment, the position of the window opening is changed in \( X \) and \( Z \) directions and their effect on thermal comfort is studied. Placing the window openings nearer to each other (\( s^* = 0.1 \) and 0.2) is not good to provide better thermal comfort. Also for \( s^* = 0.1 \) and 0.2, a low temperature zone is created only near the window openings and rest of the room interiors are affected by high temperature. By providing the window openings at \( s^* = 0.6 \) and 0.7, the incoming air leads to travel for a longer distance along the room and causes recirculation. This recirculation phenomenon allows the air to absorb more amount of heat and creates local heating. However providing the window openings in the range of \( s^* = 0.3 \)–0.5, facilitates the indoor to keep under good thermal comfort. In this study, the best position of window opening is chosen at \( s^* = 0.5 \) and this position increases the percentage of low temperature zone by 50%, reduces the PMV and PPD by 0.12% and 3.51%, respectively with reference to the worst window opening position along the lateral and longitudinal directions (\( s^* = 0.1 \)).

References


