



Optimal design of green buildings using computational fluid dynamics and climate simulation tools

M. Aram¹ · O. Abessi²

Received: 4 July 2018 / Revised: 18 April 2019 / Accepted: 8 May 2019 / Published online: 13 May 2019
© Islamic Azad University (IAU) 2019

Abstract

Natural ventilation or self-ventilation is an economical way to improve indoor air quality. Climate factors such as temperature, received radiation, humidity, and wind flow can significantly affect natural ventilation of the buildings. This study aimed to investigate the performance of some commonly known strategies such as building orientation, aperture area, and coating color to promote natural ventilation on the hot and humid climate of southern Caspian Sea, north of Iran. These techniques are traditionally used without any specific knowledge on their individual importance and mutual effects. As an advanced modeling tool for energy assessment, Design-Builder was used along with computational fluid dynamics tab to numerically evaluate the results of each change. Among the many possible cases, seven scenarios were investigated by developing a comprehensive computer simulation model. The results show that thermal comfort can be increased up to 30% and suitable airflow can be achieved by increasing the space on the windward side, roof window on the leeward side, and employing light-colored coating for the rooftop. Applying these factors decreased the apparent temperature up to 10 °C in the summer design week.

Keywords Natural ventilation · Energy assessment · Green building · Thermal comfort · Climate

Introduction

There is a global trend toward using clean and natural energy resources such as solar and wind energy (Oropeza-Perez 2015). As a part of this trend, revision of energy-saving standards is mainly influencing the rules and regulations in terms of energy consumption and emission rate (Nejat et al. 2015). According to statistics, heating, ventilation, and air-conditioning (HVAC) systems consume up to two-third of the energy supply of cities (Elshafei et al. 2017; Orme 2001). Thus, increasing the energy efficiency of buildings became a priority in many parts of the world by passing regulations and new standards (Roetzel et al. 2010). Green building is looking for a process in which building construction is compatible with the environment and conservation of resources

during its lifetime. The concept of preliminary engineering design for sustainable construction seems to perfectly match this idea. As a mechanism helping to improve indoor air quality, cooling processes, and optimal energy consumption, natural ventilation has become a popular trend recently (Li et al. 2018; Zhou et al. 2014). For buildings employing natural ventilation, in addition to a suitable air quality, the energy consumption has also been reported to decrease by up to 30–40% in comparison with mechanically ventilated buildings (Yu et al. 2014; Liu et al. 2015; Zhang et al. 2015; Hu and Chen 2015; Gratia and Herde 2004; Schulze and Eicker 2013). However, due to the complexity of air flow inside and outside of buildings, designing a naturally ventilated house is considered a challenging task (Chen 2004; Seifert et al. 2006; Hu et al. 2008). Among natural factors, climatic conditions including temperature, wind, precipitation, radiation, and direction of the sun are known to play more important roles. The ventilation performance is influenced by a combination of various design features such as opening type and ratios, building area, site location and the number of floors. Other factors include wind velocity and direction, external barriers and coat color, opening patterns, and architecture of the internal walls (Gao and Lee 2011). Computational analysis of indoor airflow via different approaches enables us

Editorial responsibility: Parveen Fatemeh Rupani.

✉ O. Abessi
OAbessi@nit.ac.ir

¹ Department of Civil Engineering, Tabari University, Babol, Iran

² Department of Civil Engineering, Babol Noshirvani University of Technology, Babol, Iran

to simulate wind velocity and flow direction inside the house (Guo et al. 2015). Computational fluid dynamics (CFD) is a set of numerical methods developed to calculate the pressure, velocity, temperature, and other properties of flows in a specific region of space. By solving the governing equations of the flow and applying the boundary conditions, the CFD models can provide detailed information on the speed, direction, and pressure of airflow and the amount of energy and heat transferred in a building. The specified boundary conditions such as climate conditions, energy absorption and heating, natural ventilation, and forced ventilation systems or HVAC can be used to calculate the temperature of the indoor space. A review of the literature revealed that a number of studies investigated the natural ventilation process in the buildings.

Hatamipour and Abedi (2008) reviewed the traditional and efficient technologies of natural ventilation that are currently used in the hot and humid climate of Iran southern coasts. Inspired by these technologies, they proposed a set of engineering methods for contemporary structures to enhance natural ventilation. Construction of buildings along East–West direction, separation of some spaces from the other parts, and design of building shading to prevent obstruction of air circulation are examples of passive cooling schemes proposed by Hatamipour and Abedi. Yıldız and Durmus (2011) investigated a building in Izmir and studied the parameters affecting its ventilation performance in the warm and wet climate of Turkey. Moreover, by conducting a sensitivity analysis, they argued that the total area of the openings (windows), thermal transfer coefficient (U), and glass absorption coefficient are probably the most effective factors in this regard. Gao and Lee (2011) investigated the influence of different openings configurations on natural ventilation performance using the concept of mean age of air (AOA) and the measurements from gas detectors in a typical residential house in Hong Kong. Evaluation results indicated that natural ventilation performance can be enhanced by installing two groups of openings perpendicular to each other and properly positioning building orientation and doors. Prajongsan and Sharples (2012) used computational fluid dynamics (CFD) analysis for a residential building in Bangkok to study the possibility of using ventilation shafts to enhance natural ventilation and thermal comfort. To this end, using the CFD package offered in Design-Builder software, the air velocities in the room with and without the ventilation shaft were calculated. Significant differences were found in the air flow velocity and cooling potential of the rooms equipped with a ventilation duct on the one side and an opening on the other. Adamu et al. (2012) investigated the performance of buoyancy-driven airflows in four selected natural ventilation strategies for a single-bed hospital ward. The strategies were single window opening, same-side dual-opening, inlet, and stack co-works, and the

new concept of ceiling-based natural ventilation (CBNV). The effect of these strategies was investigated using CFD and thermal simulation of air flow in a hospital ward in the center of London as a case study. The results showed that at least 25% of the ventilation opening fraction is needed to provide the required airflow and thermal comfort in the winter. Other approaches found to minimize summer overheating to different extents, except window-based design. The CBNV concept was also efficient to deliver fresh air directly over patients in isolated parts of wards.

Hanan (2015) also explored the possibilities of energy savings in a villa house in the United Arab Emirates by integrating the natural ventilation and mechanical air-conditioning systems. A detailed analysis of the annual energy cost of the house was carried out through numerical simulations. The results were then compared with the actual bills throughout a sample year. The energy consumption was also compared with the results of various natural ventilation strategies, obtained through CFD analysis of wind flow in the internal spaces. Baghaei et al. (2016) studied bilateral opening and one-way ventilation on the efficiency of maximum wind circulation in the interior space of a residential apartment house. They used CFD tools offered by the Design-Builder model to simulate four different scenarios in which the internal separator walls of a house were slightly changed. It was observed that in the optimal design of the indoor space, installation of windows facing the windward and leeward directions along the flow path provides adequate air flow, which significantly reduces the humidity of the residential space.

Elshafei et al. (2017) conducted an experimental and numerical study to explore the impacts of natural ventilation on the indoor thermal comfort of a residential building in Alexandria, Egypt. The CFD tools offered in Design-Builder were adopted for the steady-state condition, and Reynolds-averaged Navier–Stokes model was used to estimate wind flow field within the study domain. Experimental measurements were then used to validate the simulations. Air velocity, relative humidity, and temperature field were considered for the quantification of thermal comfort. To improve indoor air quality and ventilation, modifications were made to the window design parameters such as windows size, placement, and shades, consequently decreasing air temperature by 2.5% while increasing the air velocity by six times.

Li et al. (2018) used ANSYS FLUENT 16.2 as a powerful computational fluid dynamics (CFD) software to estimate the potential of wind-driven natural ventilation during the early stages of building design. A steady-state Reynolds-averaged Navier–Stokes (RANS) standard $k-\epsilon$ turbulence model was used and validated for different ventilation scenarios. To help architects and engineers, the numerical results were utilized to develop empirical equations for fast and accurate evaluation of the ventilation rate. Carrying out



a set of numerical simulations and experimental tests, Aflaki et al. (2019) also explored the efficiency of single-sided ventilation and cross-ventilation on indoor air velocity and flow patterns of a residential unit in a high-rise building. As a full-featured 3D-CFD solution tool, the FloEFD software was used for numerical analysis. This software is known to be a robust and reliable simulation model capable of quickly solving Navier–Stokes equations. RANS turbulence models along with the transport equations were used to measure the turbulent kinetic energy (k) and the dissipation rate (ϵ). Their results show that the enhancement of natural ventilation can increase indoor air velocity up to four times and average air changes per hour up to 27%.

A review of the previous works indicated that investigations are more concentrated on the effects of external boundaries and house interior architecture. Environmental conditions and the effect of climate variables such as temperature, humidity, wind direction, and solar radiation on airflow pattern together with the site orientation, shading, comfort conditions, and resident satisfaction are the other determining factors that need to be considered in building operating conditions. Hydrodynamic CFD models are well-designed to simulate airflow and circulation but are computationally demanding and limited to consider only a few climatic parameters in boundaries. This study pursues the idea of green building by emphasizing the influences of design variables such as house opening, coating color, and orientation on the preliminary engineering design process prior to the implementation of plots. The changes in these factors and their mutual effects on indoor air quality are discussed through a set of computer simulations. The wind-driven ventilations and their interaction with other climate factors were investigated. The combination that provides the highest natural ventilation will be presented as the optimal design, by which air residence time is reduced and indoor air quality is enhanced. Finding the significance of each variable will help engineers to better invest in-house features during the early design stage.

Materials and methods

General framework

In this study, a computational fluid dynamics tool is used together with an energy simulation program to investigate buildings indoor airflow and thermal condition. As an advanced multi-discipline modeling tool for energy-efficient design, Design-Builder is used for the simulation of air properties. The software is a powerful 3D model for the visualization, offering flexible tools for the analysis of energy consumption and resident comfort in the early stages of building design. This software provides a graphical interface

for thermal simulation and hydrodynamic modeling of airflow. The CFD engine incorporated into the Design-Builder has been developed based on the SIMPLER algorithm. Generally, this software is easier to use compared to sophisticated CFD packages such as Fluent, CFX, and OpenFOAM. Additionally, it requires less data to set up the geometry and boundary conditions. This software allows engineers and energy assessors to use all modules together with the hydrodynamic module for the simulation of realistic conditions with the accuracy level required in engineering applications.

The approach proposed in this paper comprises airflow properties, temperature field, relative humidity, shading, and solar radiation on a house architecturally engineered for the humid subtropical climatic conditions of Babolsar, a city in northern Iran. Based on the architectural design guidelines found in the literature, different scenarios for natural ventilation enhancement are developed and the performance of each is investigated. Architectural elements such as orientation in the site, aperture area, and coating color were considered as the main variables. The effectiveness of each and their combined impacts were investigated. Figure 1 shows the design variables and the approach used in the research to evaluate the ventilation performance and thermal comfort. Climate variables such as dry bulb temperature ($^{\circ}\text{C}$), wet bulb temperature ($^{\circ}\text{C}$), atmospheric pressure (hPa), global solar irradiation (W h/m^2), diffuse solar irradiation (W h/m^2), cloud cover (oktas), wind speed (knots), and wind direction (degrees clockwise from north) were used as the weather data inputs to Design-Builder. The values for these variables were obtained from a reference weather station in the region. The hot weather conditions of August, in which the cooling systems are most demanded, were considered for simulations as the worstcase scenario in terms of comfort conditions. Monthly wind roses were generated, and the prevailing wind direction and magnitude for the August were used in simulations. Three pairs of scenarios were investigated, from each of which the better scenario was used in an optimal combined scenario.

Case study

This study was carried out in an unoccupied land with the area of 16 hectares in Babolsar, Iran. The site was located at 36.70° latitudes and 52.62° longitudes with a semi-subtropical climate, characterized by warm humid summers and mild winters. A complex town with 123 sections (1000 m^2 each) was designed with a duplex residential house in each. The weather data were collected over a 60-year period, recorded by the synoptic meteorological station of Babolsar (latitude: 36.72 , longitude: 52.65 , and altitude: -21 ms). The dataset is the typical data of hourly observations of synoptic weather stations. The regional statistics on wind data are extracted and used to develop the monthly wind roses of the region

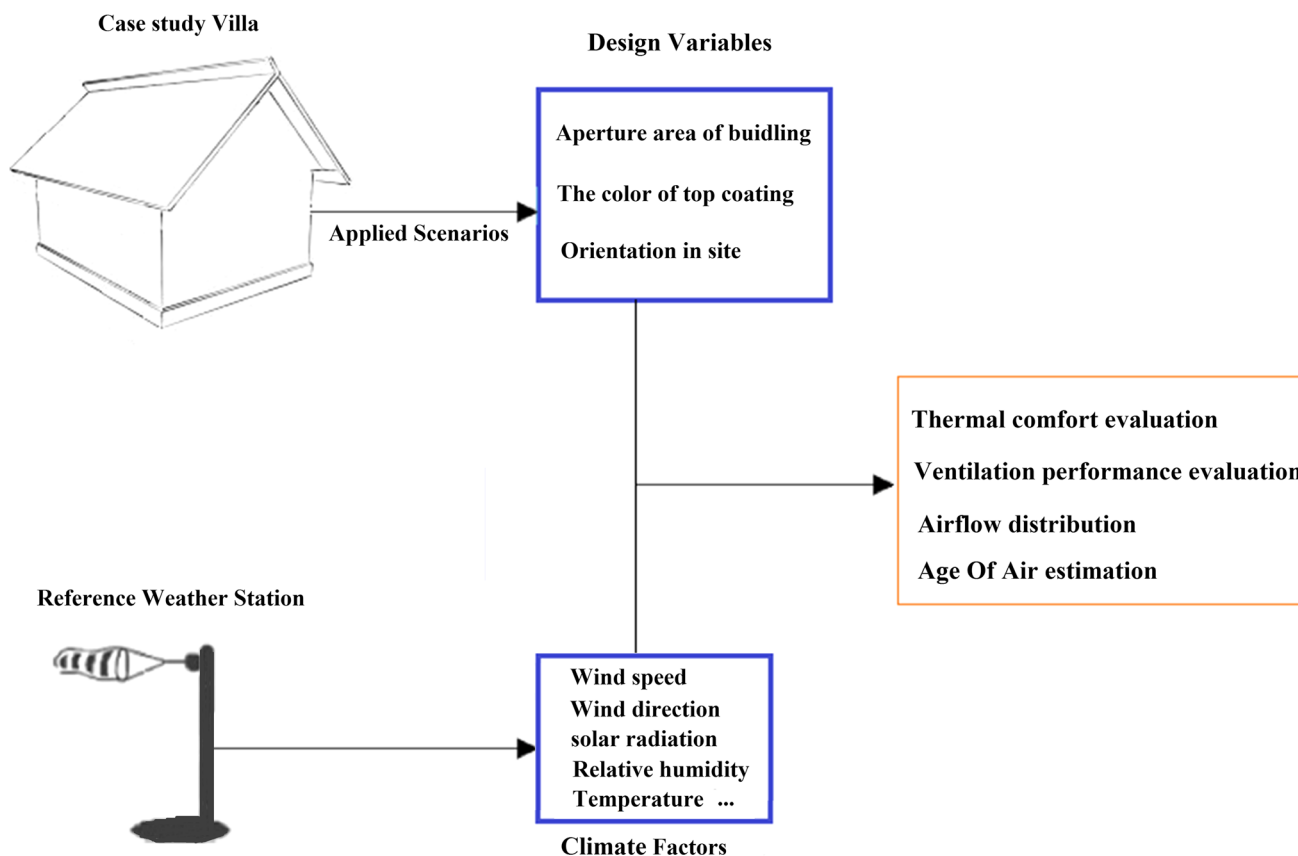


Fig. 1 Schematic of the applied methodology, design variables, and climate factors

by a computer program called WindrosePlot. The meteorological data are also used in energy simulation processes. Design-Builder uses hourly weather data to define the buildings external conditions and utilize them in the energy and CFD simulation processes through the interconnections established.

The relative humidity varied from 70 to 85%, and mean daily temperature ranges from 7 °C (December) to 30 °C (August). Wind velocity lies between 4 and 5.3 m/s blowing dominantly from west or northwest. As exhibited in Fig. 2, the direction of the prevailing wind is from west to east (270°) with an average speed of 6.76 Knots (3.48 m/s). Recorded in the synoptic meteorological station of Babolsar, the mean temperature, wind speed, wind direction, and relative humidity are given in Table 1.

Figure 3 represents the site layout (left) and building plan (right) of the study area. Each house includes a 1000 m² garden with a building with two floors, four bedrooms, and two living rooms. Provided in an open space, we were able to apply different designing scenarios. To examine the natural ventilation performance, the doors (except the gateway) and windows were assumed to stay open while no heat sources or HVAC systems were operating during the modeling process. A duplex villa

was designed with wall openings and a continuous roof window to speed up the ventilation process and allow the airflow to escape from the upper zone. The roof window, on the leeward side, works like the traditional wind tower to create a low-pressure space and accelerate the airflow outside. To evaluate the changes in natural ventilation, based on Jamaludin et al. (2014), several scenarios were developed. The total area of the inlet and outlet openings determines the minimum and maximum allowable building area, which were 4 and 40%, respectively. The directions of the pitched roof aperture were also investigated in two scenarios: (1) the roof window on leeward side (wall openings windward) (2) the roof window along the prevailing wind direction. At the end, the effect of roof and walls coating color, i.e., its brightness, on the indoor temperature and airflow was investigated. Additionally, the amount of received light was also compared.

The natural ventilation, therefore, was evaluated through seven different scenarios. Each pair of them was compared, and design sensitivity analysis was carried out as follows:

1. Aperture area of building (maximum and minimum openings)
2. The color of top coating on roof and walls (brightness)

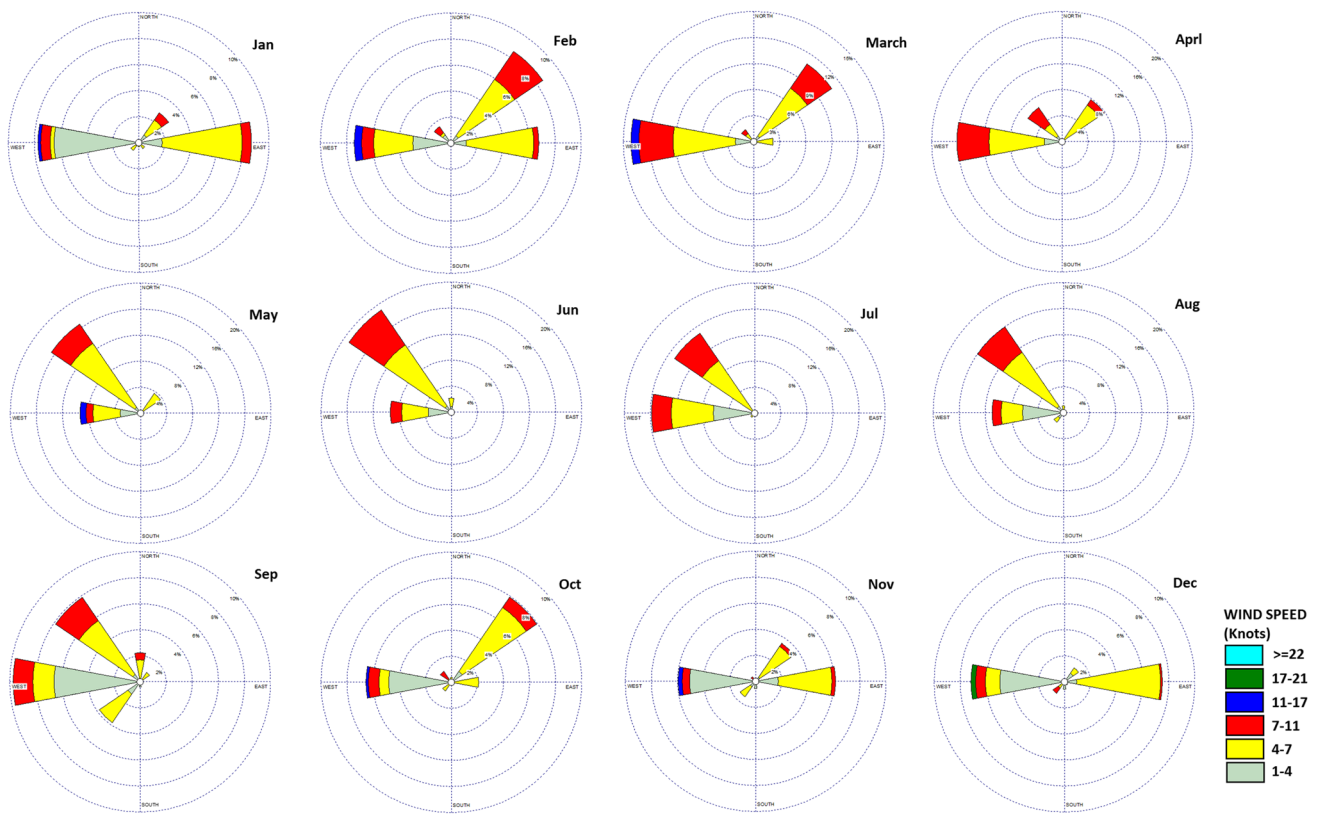


Fig. 2 Monthly wind roses based on Babolsar synoptic meteorological station data, over 60 years

Table 1 Mean temperature, wind speed, wind direction, and relative humidity recorded in Babolsar synoptic meteorological station

Year	Month												
	Jan.	Feb.	Mar.	Apr.	May.	Jun.	Jul.	Aug.	Sep.	Oct.	Nov.	Dec.	
2002	4.5	4.4	4.7	5.2	4.8	5.3	3.9	4.5	3.5	3.7	3.5	4	Average of wind speed in Knots
2002	84	81	78	82	78	73	76	77	79	82	84	83	Average of relative humidity in percent
2002	180	45	45	270	315	315	315	315	360	45	225	90	Prevailing wind direction
2002	9.3	10.2	13.4	15	18.9	24.6	27.5	27.7	26.1	22.7	16	7.6	Average of mean daily temperature in C

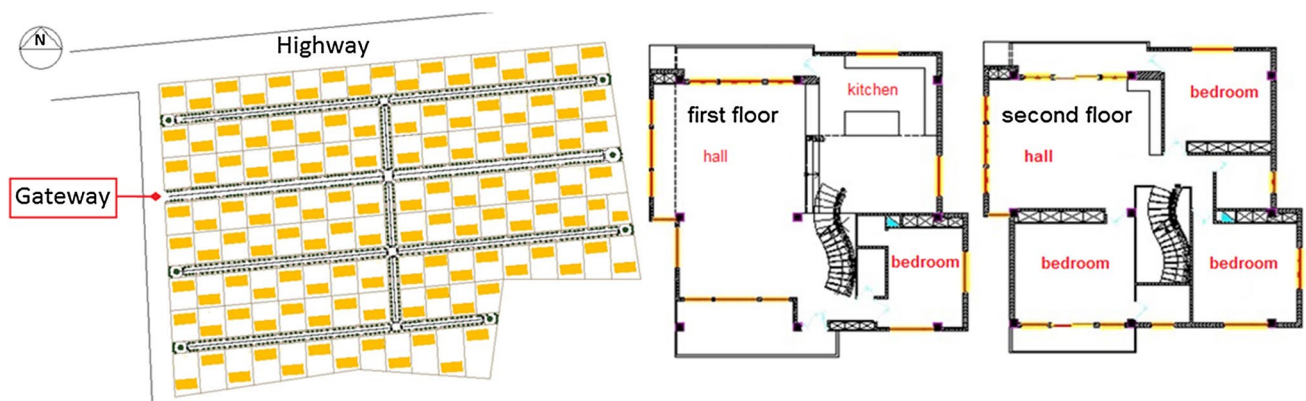


Fig. 3 Site layout (left) and building plan (right) of the case study

Fig. 4 The minimum and maximum area of opening surfaces

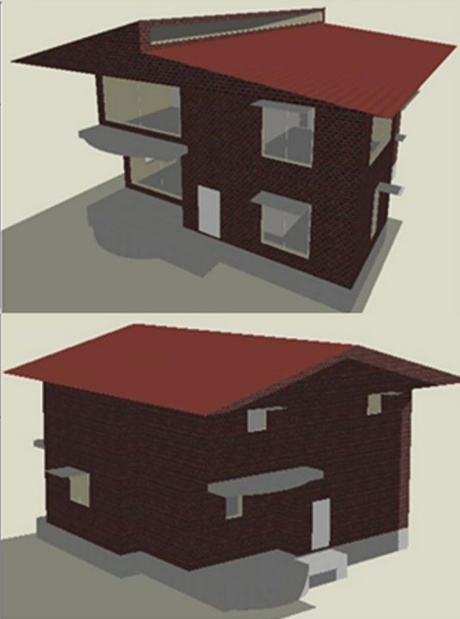
$144.255 \times 0.4 = 57.7 \text{ m}^2$	Hall & Kitchen	First Floor		
$25 \times 0.4 = 10 \text{ m}^2$	Bedroom			
$75 \times 0.4 = 30 \text{ m}^2$	Hall	Second Floor		
$27 \times 0.4 = 10.8 \text{ m}^2$	Bedroom			
$24.5 \times 0.4 = 9.8 \text{ m}^2$	Bedroom			
$23.5 \times 0.4 = 9.4 \text{ m}^2$	Bedroom			
$144.255 \times 0.04 = 5.77 \text{ m}^2$	Hall & Kitchen	First Floor		Second Floor
$25 \times 0.04 = 1 \text{ m}^2$	Bedroom			
$75 \times 0.04 = 3 \text{ m}^2$	Hall			
$27 \times 0.04 = 1.08 \text{ m}^2$	Bedroom			
$24.5 \times 0.04 = 0.98 \text{ m}^2$	Bedroom			
$23.5 \times 0.04 = 0.94 \text{ m}^2$	Bedroom			

Table 2 Characteristics of external coating materials

Surface	Solar absorptance	Visible absorptance	Conductivity (W/m K)	Specific heat (J/kg K)	Density (kg/m ³)
Dark wall	0.7	0.7	0.84	800	1700
Light wall	0.6	0.6	0.5	1000	1300
Dark roof	0.7	0.7	1	800	2000
Light roof	0.4	0.4	0.29	1000	1250

- Orientation of the site (toward the prevailing wind direction or arbitrarily oriented)
- The combination of scenarios

Aperture area

To enhance the natural ventilation in the buildings, Jamaludin et al. (2014) suggested that:

- Rectangular plans are more efficient than a square plan
- Elevated floor allows the wind to flow into the interior spaces more easily

Tantasavasdi et al. (2001) also suggested that the optimal total area for inlet and outlet openings is about 40% of the floor area in order to achieve the desirable natural ventilation. To increase the indoor airflow, the O.K.B¹ of windows was proposed to be set proportional to the residents' lifestyle. Based on the height at which residents spend most of their time, it can be argued that a window height of less

than 1 m should be selected to allow the residents to be more exposed to air flows. These factors were considered for the optimal design in the scenarios planned for this study. According to Iranian National Building code (2013), the opening area of each space must be at least 4% of the floor area to allow minimum air exchange with the surrounding environment. Therefore, the minimum and maximum areas of building openings are calculated between 4 and 40% as follows (Fig. 4):

The color of top coatings (roof and walls)

Depending on the amount of radiation that a building receives or absorbs, its surface temperature increases after sunrise. For a light-colored surface, the absorbed heat is lower and vice versa. The effect of sunlight on building surface temperature may be sometimes greater than the ambient air temperature. According to the principles of climatic design, bright colors reflect up to 90% of the solar energy, while dark colors reflect only 15% or less. To investigate the effect of the color of exterior walls on the in-house air temperature and indoor air flow rate, two models were simulated, one of which used bright white top coating, and the

¹ Occupancy Kingston Benchmark.



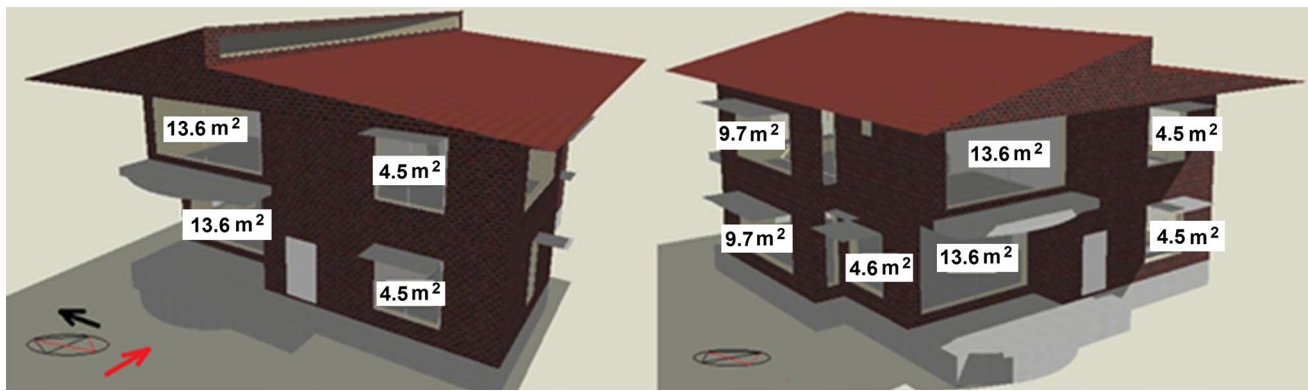


Fig. 5 Representation of two scenarios of orientation in site

other brick materials on the ceiling and walls. The details of the external coatings used for the two aforementioned scenarios are shown in Table 2. In this table, the solar absorption is the absorption coefficient of solar waves including visible, ultraviolet, and infrared waves. Visible absorbance indicates the absorption of visible waves, which is only 50% of solar radiation. Conductivity or thermal conductivity is the property of a material to conduct or transfer heat. The specific heat is the amount of heat per unit mass required to increase the temperature by one degree Celsius, and density in this table determines the density of material for each surface. As exhibited, these values can vary in two selected scenarios for bright-to-dark brick-colored surfaces.

Site orientation

The general configuration of the buildings in each of the two scenarios is visualized in Fig. 5. The black arrow represents the north direction, and the red arrow shows the prevailing wind direction. The area of apertures of each facade is also shown in the figures. In the first case, the roof window is perpendicular to the direction of prevailing wind flow, and the bedrooms are located on the windward side. In the second case, the roof window is on the leeward and the living room is located on the windward face.

Residence time (age of air)

The average age of air (AOA), according to the definition, is the length of time during which a specific volume of air completely leaves a particular zone. AOA is the average time for air to travel from an inlet to any point of the room and is assumed an important parameter in assessing indoor air quality, in addition to air temperature, air velocity, and pollutant concentrations. For simplification purposes, by assuming a stable state for the building space, the age of air can be calculated from the following equation:

$$\text{Residence time (AOA)} = V/Q \quad (1)$$

In this equation, the AOA, V , and Q represent, age of air (s), the volume of space (m^3), and the volumetric flow rate (m^3/s), respectively. Given the volume of each space, the AOA can be calculated by measuring the air flow rate from each of the inputs and outputs.

Modeling procedure

Energy Plus, the powerful engine of Design-Builder for building energy simulation, was used to quantify the house performance in this study. Energy Plus is a console-based program developed by the US Department of Energy's (DOE) Building Technologies Office (BTO) to model heating, cooling, ventilation, lighting, and plug and process loads in buildings. As free and open-source software, Energy Plus is well-integrated within the Design-Builder, empowering it for advanced dynamic thermal simulations at the smallest time steps. The Design-Builder is a front-end interface for the Energy Plus simulations and links energy simulations to other modules dynamically. A combination of control measures was adopted in Design-Builder to model natural ventilation. Design-Builder uses a built-in coupling approach to get the output from Energy Plus as an input to its CFD module. The wind-induced pressure/speed on each surface can also be fed into the CFD module to estimate the natural ventilation within a certain domain. Energy Plus is a collection of many program modules that work together using a variety of systems and energy sources. The core of the simulation is a model of the building that is coded in FORTRAN and works based on fundamental heat balance principles. The model is well described in engineering references and Energy Plus documentations on its Web site at energyplus.net.

In this study, using the Energy Plus program, climate-based modeling was developed which included heat and moisture simulation, radiant surface heating/cooling, and energy and heat exchange simulations. The capabilities of the model in coupling the Energy Plus outputs with the CFD module were used to optimize the combination of design

variables by knowing the importance of each. The process involved in CFD simulation is described in the following section.

Governing equations

The movement of air can be due to density differences with the surrounding (buoyancy flux), the initial momentum (momentum flux), and the speed of ambient air. Soon after flow formation, the initial momentum and buoyancy fluxes will form a layer of shear stress in the flow boundaries, consequently causing turbulence in flow. As the laminar flow at the engineering scale is generally unlikely to happen, the modeling of turbulence is essential for the analysis of actual air flows in the building. The CFD module of Design-Builder has been applied for the steady-state modeling of the air flow in this study. CFD is the art of solving a set of algebraic equations using digital computers. Using this technique, several pieces of software were developed with advanced solver and graphical tools. It was also embedded in numerous software packages as the simulation engine. In Design-Builder, CFD modules are developed for both external and internal analysis. These modules numerically solve the partial differential equations governing the fluid motion, i.e., the conservation laws for the mass, momentum, energy, and concentration. The equations comprise nonlinear second-order partial differential equations as follows:

$$\frac{\partial}{\partial t}(\rho\varphi) + \text{div}(\rho u\varphi) = \text{div}(\Gamma \cdot \text{grad}\varphi) + S \quad (2)$$

where φ represents the dependent variables which is a scalar field $\varphi = \varphi(x, y, z)$, in which the terms from left to right $\frac{\partial}{\partial t}(\rho\varphi)$, $\text{div}(\rho u\varphi)$, $\text{div}(\Gamma \cdot \text{grad}\varphi)$ and S are transient (rate of change), convection, diffusion, and source terms, respectively. In this equation, t is time, ρ is mass density, u is the velocity of flow, the gradient operator defines as $\text{grad}\varphi = i\frac{\partial\varphi}{\partial x} + j\frac{\partial\varphi}{\partial y} + k\frac{\partial\varphi}{\partial z}$, the div is the divergence of a vector, and Γ or Γ is the scalar diffusivity.

The dynamic simulation engine solves all the governing equations, i.e., mass conservation equations (continuity equation), the conservation of the momentum (Navier–Stokes equations), and advection–diffusion equation for temperature and concentration evolutions (if required). The terms in these equations assume instantaneous values at time t , which based on Reynolds decomposition, can be decomposed into its time-averaged (\bar{u}) and fluctuating quantities u'_i , ($u_i = \bar{u}_i + u'_i$). Similarly, φ as a scalar quantity can be $\varphi_i = \bar{\varphi}_i + \varphi'_i$, where φ'_i is a fluctuating quantity. Substituting these expressions for the flow variables into the instantaneous equations and take the time average, the differential forms of the equations can be formulated as (Li et al. 2018):

The continuity equation:

$$\frac{\partial\rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (3)$$

Momentum conservation equation:

$$\rho\left(\frac{\partial u_i}{\partial t} + u_j u_{ij}\right) = -P_i + (\rho - \rho_0)g_i + \left[\mu(u_{ij} + u_{ji}) - \overline{\rho u'_i u'_j}\right]_j \quad (4)$$

where P_i is pressure, g is the gravity acceleration, μ is dynamic viscosity, and $\overline{\rho u'_i u'_j}$ is apparent stress.

Mass conservation equation:

$$\rho\left(\frac{\partial C}{\partial t} + u_j C_j\right) = \left(\rho\alpha C_j - \overline{\rho u'_j C'}\right)_j + q_c + R \quad (5)$$

where C is the (scalar) concentration and C' is fluctuating concentration, $\overline{\rho u'_j C'}$ is Reynolds mass flux, q_c is mass decay, and R is mass source.

Energy conservation equations:

$$\rho C_p\left(\frac{\partial T}{\partial t} + u_j T_j\right) = \left(\lambda T_j - \rho c_p \overline{u'_j T'}\right)_j + H \quad (6)$$

where T and T' are temperature and fluctuating temperature, λ is the thermal conductivity, H is all the volumetric heat sources, and $\rho c_p \overline{u'_j T'}$ is Reynolds thermal flux.

To solve these equations, Design-Builder uses the Reynolds-averaged method, which is based on mathematical averaging of instantaneous variables in the solution of Navier–Stokes equations. For most engineering applications, resolving the details of the turbulent fluctuations in flow dynamics is unnecessary. The modeler only needs to know how turbulence affected the mean flow. As a computational procedure to close the system of mean flow equations, turbulence models are developed to calculate the mean flow without calculating the full time-dependent flow field. Reynolds-averaged Navier–Stokes (RANS) equations are explicit with some assumptions to simplify the problem. In these equations, the number of uncertainties is greater than the known factors and a set of simultaneous algebraic equations are formed to be solved by a numerical method. There are classical models developed based on the RANS equations, e.g., zero-equation model (like mixing length model), one-equation model (Spalart–Almaraz), two-equation models (k – ε style models, k – ω model), and seven-equation models. The k – ε model is the most widely used turbulence model and is generally accepted for engineering purposes. This method is based on the solving of two independent transition equations for estimation of turbulence kinetic energy (k) and energy dissipation rate of turbulence kinetic energy (ε) to calculate the effective viscosity of flow (μ_t) as follows:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad (7)$$



where C_μ is a constant number and k and ε are both derived from partial differential equations, which themselves are derived from a manipulation of the Navier–Stokes equations. The k and ε equations are defined as follows:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (8)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S_\varepsilon - \rho C_2 \times \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon \quad (9)$$

where C_1 , C_2 , C_3 are experimental coefficients, σ_ε and σ_k are turbulent Prandtl and Schmidt numbers, respectively. In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, and S_k and S_ε are user-defined source terms.

The coefficients are as follows:

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right], \eta = S \frac{k}{\varepsilon}, S = \sqrt{2S_{ij}S_{ij}} \quad (10)$$

The constants in the above equations are derived from experimental observations and previous studies and are proposed as follows (Elshafei et al. 2017):

$$C_{1\varepsilon} = 1.44, C_2 = 1.9, \sigma_\varepsilon = 1.2, \sigma_k = 1 \quad (11)$$

This model is one of the most widely used and tested of all turbulence models. Considering its high efficiency and performance in simulation of complex flows, the k - ε model was implemented for the simulation of turbulence under steady-state airflow conditions in this study. For the assumed scenarios, the house was exposed to local winds and the performance of indoor ventilation was predicted.

Model validation

CFD is a modeling technique to solve the equations of motion and predict an event or series of phenomena. The computed results should be ideally consistent with the real-world observations, but there are sources of errors and uncertainties in the simulations which may induce errors in the results. For airflow simulation, use of turbulence models leads to uncertainties in the computed results and there is not a universal agreement on using a validated model for all flow regimes and spaces. Therefore, it is essential to validate models using appropriate experimental data. However, it is not possible to validate the entire CFD model, rather it can only be validated for the application domain. In this study, the results of a full-scale measurement were used to validate

our CFD model, and then the validated model was employed to evaluate further alternatives. To obtain quality results, the experiment requires an environmental chamber with a controlled indoor and outdoor environment using well-equipped sensors and data acquisition systems to measure air velocity, air temperature, and relative humidity. The chamber should also have the flexibility to change from one spatial configuration to another.

There are experimental data available in the literature, but most of which cannot be used for the validations as detailed information concerning flow and the thermal conditions is required. Additionally, the airflow and its spatial characteristics must be similar to those investigated. In this study, an experiment reported by Kobayashi and Chen (2003) was used to validate the model for the performance of a floor-supply displacement ventilation system. The validated model was then used to study the aforementioned scenarios. In their study, an environmental chamber for a two-person office was investigated. The chamber was a well-insulated box to simulate indoor environment on different weather conditions. The study chamber was 5.16 m long, 3.65 m wide, and 2.27 m high. The room included some internally heated objects (computers, lamps, and residents) and two round-shaped floor diffusers with a diameter of 200 mm. The SF6 (Sulfur hexafluoride) tracer gas source with a rate of 348 mL/min was introduced at the top of Occupant 2. Air velocity, air temperature, and the SF6 concentration were measured under the steady-state conditions at nine different points. At these points, six anemometers, six sampling tubes for SF6, and two thermocouples were installed from the floor to the ceiling to carry out the measurements.

A model with similar specifications knowing the size and the location of the heated objects in the test chamber was developed in Design-Builder. Modeling of air diffusers was a challenging task, for which a similar method as described in Kobayashi and Chen (2003) was used. Finally, the model was run under the steady-state conditions defined for the geometry, based on the details mentioned above. The details of the boundary conditions are the same as listed for the reference case in Table 2 of Kobayashi and Chen (2003). Supply air temperature was set to 19 °C and the airflow rate per diffuser at 0.052 m³/s. Moreover, the number of diffusers was 2 and activity of two persons was considered. Mesh-dependency test was conducted, and the k - ε model was utilized to quantify turbulence in RANS model.

Performance of the ventilation system was evaluated in terms of thermal comfort and indoor air quality by comparing the computed results with the observations. In the

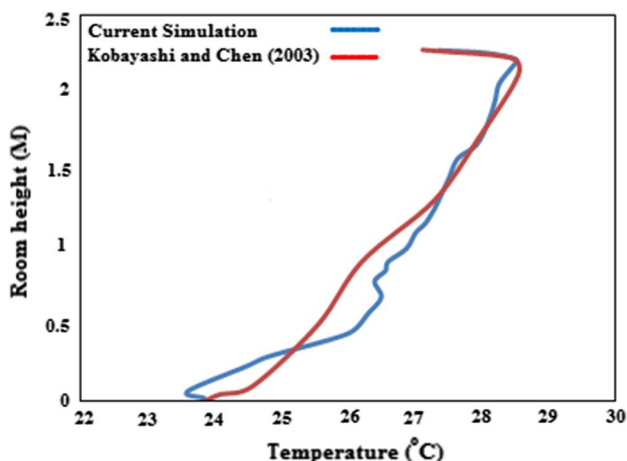


Fig. 6 Comparing the temperature variation along the room height for $X = -0.86$ and $Y = 1.825$

validation process, the temperature gradient, air velocity, and ventilation effectiveness were considered. As shown in the following figure, the computed temperature gradient at $X = -0.86$ and $Y = 1.825$ along the room height generally follows the pattern observed in the experiments under the same conditions. Air velocity and ventilation effectiveness were also checked in the validation process. In Fig. 6, the result of simulations for the velocity field and air circulation in the study domain are exhibited. Comparing the calculated results with those reported by Kobayashi and Chen (2003) for the reference case shows a discrepancy of lower than 10%. It provides a basis to use the validated

model for the simulation of the ventilation scenarios with few changes in geometries and boundary conditions.

Results and discussion

Overall findings

The simulation results are obtained under the climatic conditions of the study area in summer hottest design week (Table 1). Isothermal and isovelocity contours are shown in the following figures. Simulations were conducted under steady-state conditions, and the wind and hourly weather data of the climate zone were manually imported. To model the airflow, as previously mentioned, a model based on the Reynolds-averaged Navier–Stokes (RANS) known as $k-\epsilon$ model was employed. A PC equipped with an 8-core processor at a frequency of 3.2 GHz and 32 GB RAM was used for the simulations, taking about 12–36 h to complete each run.

The influences of the opening or aperture area were investigated first by simulating the two extreme cases of maximum and minimum recommended area for the buildings external windows. The results show that for the case of minimum aperture, internal air velocity was about 0.01–0.2 m/s and the temperature ranged from 25.86 to 27.54 °C (see Fig. 7).

For the building with maximum aperture area, the internal air velocity and temperature are exhibited in Fig. 8. The simulation results show that the velocity ranged

Fig. 7 Simulation results for indoor air temperature and velocity in the building with the minimum opening

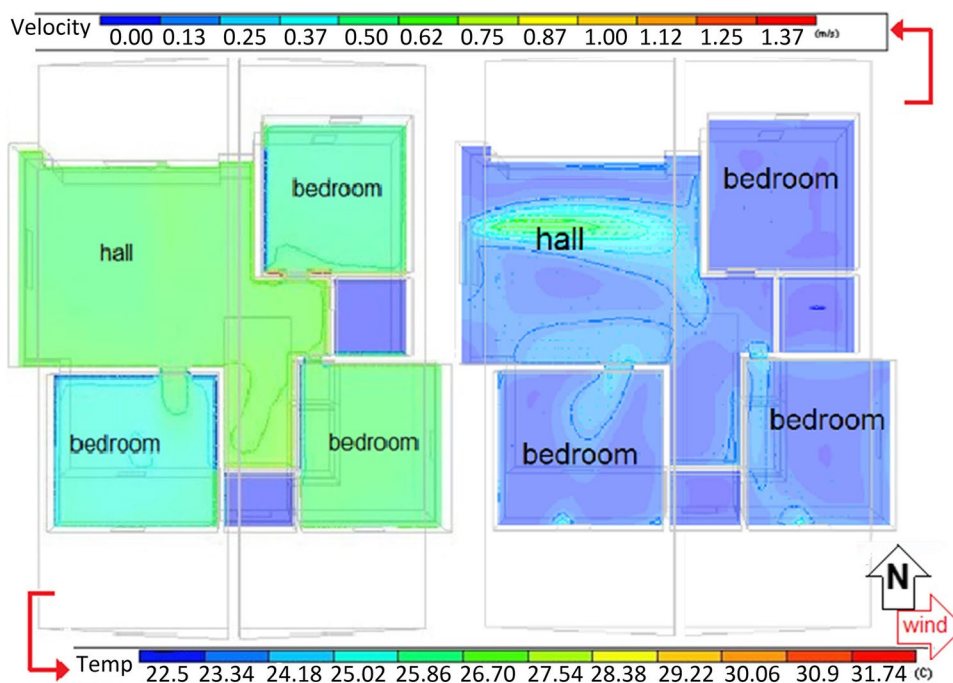
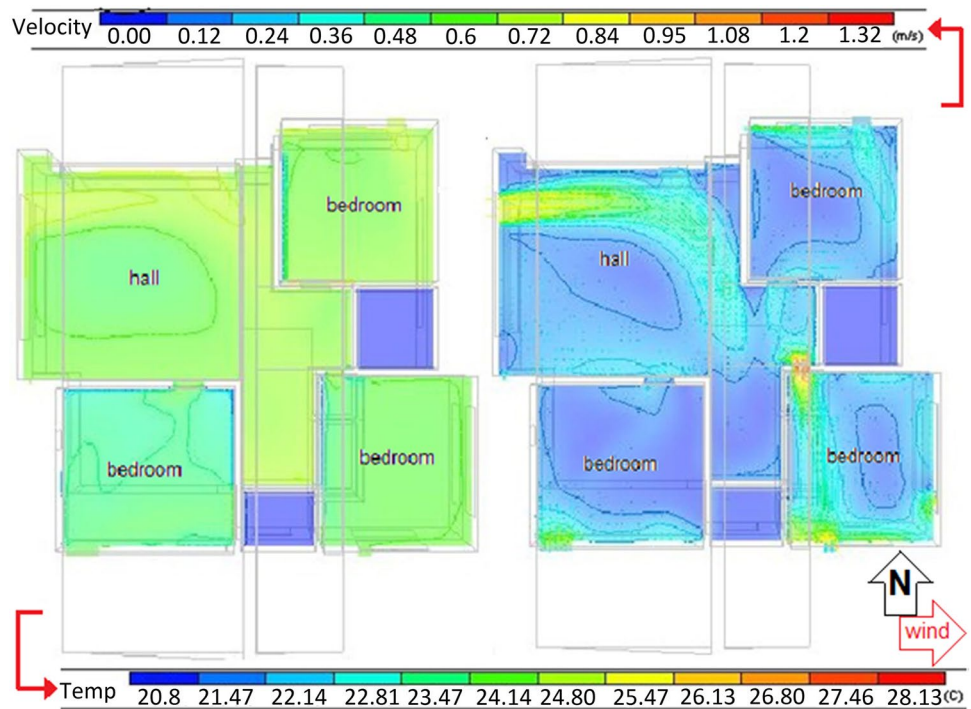


Fig. 8 Simulation results for indoor air temperature and velocity in the building with the maximum opening



from 0.26 to 0.36 m/s and the temperature from 22.81 to 24.88 °C under the same climatic conditions.

To evaluate the results, the comfort conditions of the residents were analyzed quantitatively. According to Candido et al. (2010), under normal conditions, the human body feels comfortable within the air velocity range of 0.2–1.5 m/s. At temperatures above 24 °C, a minimum speed of 1 m/s is preferred, and if the indoor air temperature is greater than 28 °C, the ventilation rate can also be increased. It is worth noting that when the flow speed is greater than 0.8 m/s, in the immediate vicinity of the body, wind flow may lead to dry skin and cause discomfort; thus, for temperatures greater than 24 °C, the appropriate flow velocity was estimated about 0.8–1 m/s, and for temperatures below 24 °C about 0.2–0.8 m/s. The results of simulations for a building with maximum openings show that the indoor air velocity was within the range 0.24 to 0.36 m/s and the corresponding temperature varied between 22.81 and 24.8 °C. For the building with a minimum opening, the indoor air velocity and temperature were within the ranges 0.01–0.2 m/s and 25.86–27.54 °C, respectively. Therefore, for the second scenario, besides the convenient condition for the air flow, a significant difference in temperature (2–3 °C difference) was observed for the simulated scenarios. The decrease in temperature seems to be due to the increases in building opening areas and the better circulation of the incoming airflow together with the improvement in the indoor airflow distribution.

In addition to the simulation of the air flow and heat exchange, the Design-Builder is able to simulate the received light for the climatic and geographic condition of the studied area. For the simulated scenario with minimum opening area, the level of received light was low during the day. Next to the windows on the first floor, it only exceeded 46 LUX and reaches up to 92 LUX in some cases. Due to the same reason, the received radiation on the second floor was also low and did not meet the standards (Table 3).

LUX is the brightness unit obtained by dividing the lumen optical flux unit by area:

Table 3 Rooms required lighting in LUX, Iran National Building codes

Residential zone	Minimum lighting	Suggested lighting
Living room	70	200
Reading room	150	500
Kitchen	100	200
Bedroom		
General	50	100
Bed and Makeup table	200	500
Bathroom		
General	50	100
Mirrors	200	500
Stairway	100	150
Doorway and Elevator	50	150

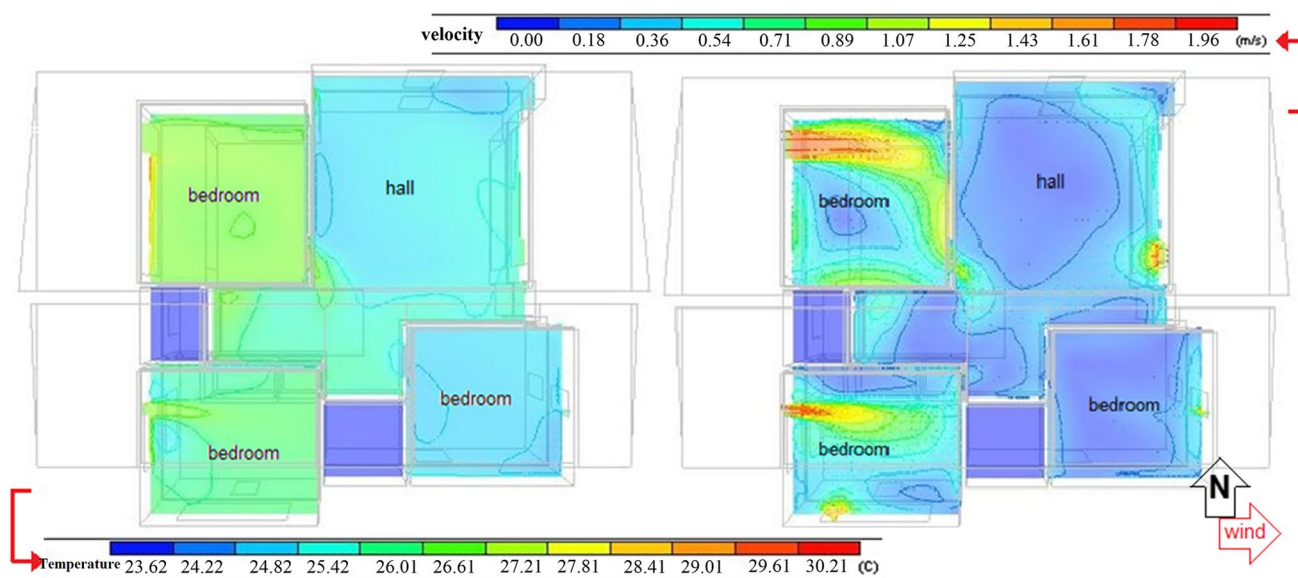


Fig. 9 Simulation results for the indoor air temperature and velocity when the roof window is along the prevailing wind flow

$$LUX = Lm/m^2, Lm : \text{optical flux unit} \quad (12)$$

The *LUX* is also defined as L^{-2}/J , where L is the length and J is the recommended symbol for illumination in the units of the international system (SI), and DF stands for Daylighting Factor, a brightness factor based on percentage. For the second scenario (building with maximum apertures), the first-floor hall received at least 100 *LUX* radiation and the bedroom received about 80 *LUX* for the most of their space. On the second floor, the optical flux per area changes between 70 and 244 *LUX* in the hall and the western bedroom. For the building eastern windows, the estimated lighting level was about 61–122 *LUX*, and this gradually decreases moving away from the windows.

If the lighting conditions or minimum vision requirements are not met throughout the day, residents have to use the artificial lighting system, which is in contradiction with the objectives of energy optimal use in a green building. Simulation results for the lighting conditions will help to make a better decision to meet both lighting minimum requirement and convenient air flow and heat.

Effect of top coating color (roof and walls)

Based on what was observed above, two new scenarios were simulated for the maximum openings in the climatic condition of Babolsar as follows and the results were evaluated for wind pattern changes and indoor air temperature. Therefore, simulation results on the indoor air temperature and velocity are obtained for both dark- and light-colored building top coatings. For the light-colored scenario, the prevailing temperature was in the range of 22.63–23.83 °C, indicating

a one-degree drop in temperature compared to the building with the dark-colored top coating in the same climate condition. For the airflow, there was no significant variation in flow velocity due to color change in the second scenario. The slight variation observed is probably due to the temperature differences. Although in both scenarios the comfort condition is achieved, in the case of light-colored top coating, the observed temperature was about 1 °C lower, which indicates the lower importance of top coating color for this climate zone compared to our initial expectation.

Effect of site orientation

In this section, as previously mentioned, the thermal and airflow conditions are investigated for two different scenarios: (1) the roof window is perpendicular to the direction of the prevailing wind, and (2) the roof window is on the leeward side of the house. For the first scenario, the flow velocity and indoor air temperature were obtained. As shown in Fig. 9, the overall temperature of the building is obtained between 25.42 and 27.21 °C for the mean height. The lowest observed temperature (25.42 °C) is associated to the living room and the eastern bedroom, and the highest temperature (27.3 °C) is observed in the western bedroom. By checking the results in more details, it can be observed that in many points, the airflow speed is about 0.18 m/s, reaching up to 0.36 m/s in some areas, while in the western (windward) bedrooms, two flow streams with a velocity of 1.43–1.96 m/s are formed.

The simulation results for the second scenario, when the roof window is located on the leeward side away from the prevailing wind direction, are demonstrated in Fig. 10. The roof window is anticipated to accelerate the ventilation

Fig. 10 Simulation results for indoor air temperature and velocity when the roof window is leeward

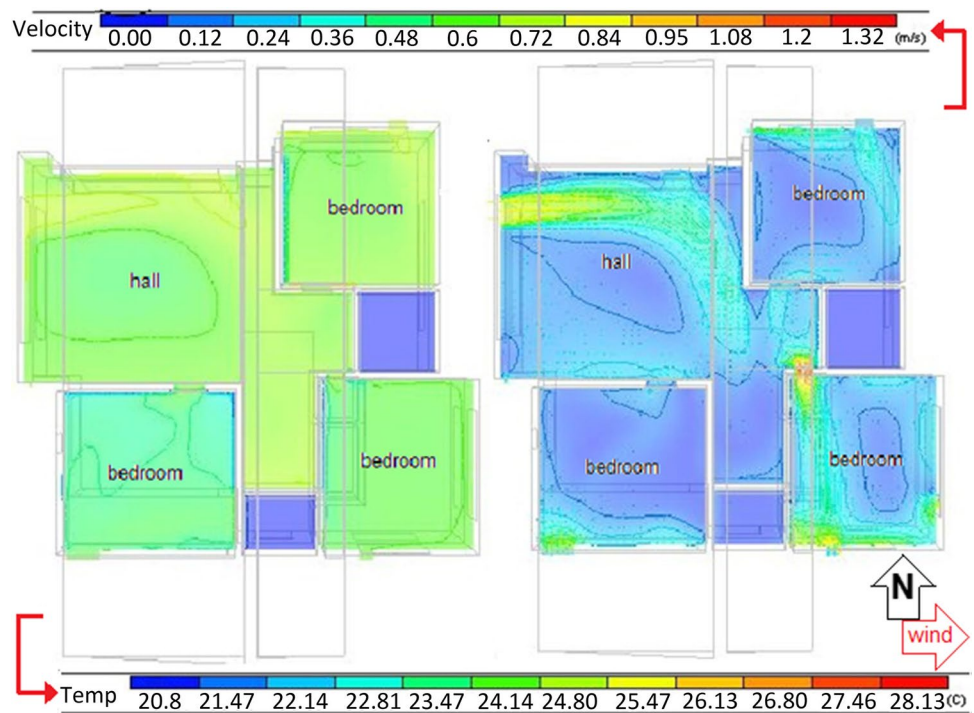


Table 4 Calculation of age of air for each scenario

Scenarios	1st floor			2nd floor		
	Hall	Bedroom	Hall	NE bedroom	SE bedroom	SW bedroom
Maximum aperture windward	53 s	1 min, 24 s	46 s	2 min, 24 s	1 min, 10 s	28 s
Minimum aperture windward	7 min, 24 s	4 min, 25 s	7 min, 52 s	9 min, 20 s	4 min, 46 s	4 min, 20 s
Roof window on the leeward side	53 s	1 min, 24 s	46 s	2 min, 24 s	1 min, 10 s	28 s
Roof window oriented 90° to wind	1 min, 52 s	1 min, 25 s	1 min, 20 s	4 min	1 min, 18 s	22 s

process. For this case, when the West to East wind enters the building, the roof window helps to accelerate the process of indoor ventilation by providing the suction required to discharge the incoming flow out. Roof window is an old technology that has long been used to enhance the indoor air flow and natural ventilation in buildings. This window is subjected to suction and pressure when faced leeward and windward, respectively. Simulations show that for the scenario of leeward roof window, the temperature and air flow speed vary within the ranges of 22.81–24.8 °C and 0.24–0.36 m/s, respectively. Additionally, a flume was formed with a velocity up to 0.84 m/s in the living room (on the west side).

For the first scenario, two bedrooms with an area of 24 and 27 m² were exposed to the western wind. The fast air-flow in such closed space consequently causes discomfort. For the second scenario, when the building is rotated 90° and the living room is on the windward side, the distribution

of air flows inside the house is improved. In the second scenario, a vast area is affected by the wind flow entering from the western living room, which provides suitable ventilation and comfort by 3–4 °C decreases in dry temperature. It is worth noting that in these two scenarios, the first design was only investigated and the interior design was neglected in the simulation, since this has already been investigated by Jamaludin et al. (2014). They proposed that the number of internal walls along the flow path should be reduced down to the minimum in order to achieve better indoor air flow and wind speed along the house.

Changes in age of air

Based on modeling and simulations, it was found that the indoor temperature varies considerably in buildings with the maximum and minimum opening area. Air change effectiveness in buildings, which is known as the age of air (AOA),

also varies drastically in the other scenarios. The mean local age of air is a significant indicator for the freshness of air and effectively shows the building's ventilation efficiency. Using the CFD simulation of the indoor space, the airflow speed was calculated at each point. Given the velocity profile in the openings, the mean velocity of the flow was calculated in the apertures. The residence time, or the AOA, for the scenarios with both maximum and minimum areas of the aperture facing windward is calculated in Table 4. The residence time was also calculated for the case of maximum aperture assuming the roof window is faced both leeward and perpendicular to the leeward direction. The opening of each window was assumed to be similar and 50% of the total window area, as exhibited in Fig. 4. According to the calculations, for the building with maximum aperture, the AOA for the hall decreased up to 6 min compared to the building with minimum aperture. In the northeast bedroom, also the residence time decreased about 6 min. In the south-east and southwest rooms of the second floor, the age of air experienced a roughly 3-min decrease. Comparing the scenarios with the case of maximum aperture in which the roof window is faced leeward and oriented 90° , some changes were observed in the AOA of the living room too (Table 4). When the house is rotated 90° toward the prevailing wind direction, the northeast bedroom (NE) is less exposed to the local wind, consequently increasing the time of residence in the room. The AOA in southeast (SE) bedroom, which was moved southwest (SW) after rotation, only slightly changed, while for the southwest (SW) bedroom, which was moved northwest (NW) after orientation, the AOA experienced an increase.

Discussions

Natural ventilation performance of buildings is influenced by a combination of various design factors. Although the importance of each factor has been previously discussed in the literature, few numerical experiments have been conducted so far. The impact of these design factors may differ from one place to another. Generally, the location of buildings and climatic conditions of the respective region are the variables that affect the influence of factors such as roof coating color, shading and wind. For example, although a considerably important factor in tropical zones, the roof coating color in buildings is assumed to be less important in temperate zones. This study was developed to numerically investigate the performance of three known strategies, namely building orientation, aperture area, and roof coating color, to improve natural ventilation in the hot and humid climate of the northern part of Iran. The strategies are defined based on the architectural design guidelines and

local experiences. Using an engineering software package, the individual effect of each scenario together with their mutual effects was investigated. Such studies only recently became possible after developing advanced architectural, engineering, and energy assessor modeling tools like Design-Builder.

Simulations show that for the hot and humid climate of this region, the wind-induced ventilation is more important than the other factors. Proper opening and orientation designs will guarantee resident's comfort and satisfaction by enhancing the air change rate within the comfortable velocity range. On the other hand, the roof coating color was found to have the least effect. The accuracy of the results from simulations was sufficiently accurate to be practically implemented for the engineering purposes in the field. However, individual studies are needed for each climate zone and strategy. Moreover, the mutual effect of variations in each factor is also needed to be investigated. Finding the significance of each strategy will help engineers to put more effort into the building design features prior to the implementation of the plots.

Recently, novel eco-friendly ideas such as green facades, wall and balcony gardens to incorporate plants and trees in apartment buildings and biomimetic designs such as termite mound lungs ventilation system, the construction of prairie dog burrows and wind catcher have been put forward, in order to control the climate, self-cooling, and self-ventilation in buildings. The functionality of each idea is still a matter of question and needs to be investigated by similar advanced modeling techniques. The present study and all those novel ideas need also to be investigated in the atmosphere of climate change.

Conclusion

Although natural ventilation is preferred over mechanical HVAC systems, there is a lack of study to quantitatively demonstrate its performance for different regions. This study systematically investigated the possibility of using standard strategies to enhance natural ventilation for the climatic conditions of northern Iran. Based on the design guidelines, three pairs of scenarios among many were selected. The considered variables were windows areas, building top coating color, and orientation in site. The results were compared to identify the effectiveness of each in the optimal combined design. Comparing the results for the building with a minimum opening (4% of the building area) and maximum opening (up to 40%) indicated that increases in the opening area will enhance natural ventilation up to 25%. Energy simulation also shows that compared



to dark colors, applying a light color for the building top coating will only lead to a single-degree drop in temperature in the hottest time of summer. Simulations also showed that the best ventilation will not necessarily happen when the house opening is maximum, and orienting the house toward the prevailing wind direction also plays an important role in this regard. It was observed that by orienting roof windows toward the leeward direction, the local wind is able to efficiently accelerate the ventilation process. Enhancing the natural ventilation, therefore, will increase the quality of air and its freshness due to the better air distribution in the interior space.

According to the records of solar radiation in the northern region of Iran, it was observed that the indoor temperature and airflow were not significantly affected by the roof coating color, while changes in the opening area were found to be very effective for the natural ventilation. Moreover, orientation of the site to keep the roof window toward the leeward side was found to be a more important factor than coating color in the studied region. However, this may not be necessarily true for other regions, since the coating color seems to be more important in the central and southern part of Iran where radiation and sunlight are extremely high. The combination of the proposed methods was able to reduce the AOA up to 35% and indoor air temperature about 6 °C without using mechanical ventilation. A 6 °C drop in a dry temperature with 77% relative humidity during the day is actually equal to a 10 °C decrease in the RealFeel temperature (Steadman 1994). In the studied region, it was observed that roof coating color is less important, but the roof window on the leeward side was found to be the most important factor when the prevailing wind blows at one side for a long time.

Acknowledgements The authors acknowledge the funding support of the Babol Noshirvani University of Technology through grant program No. BNUT/390035/97.

References

- Adamu ZA, Price ADF, Cook MJ (2012) Performance evaluation of natural ventilation strategies for hospital wards-A case study of Great Ormond Street Hospital. *Build Environ* 56:211–222
- Aflaki A, Hirbodi K, Mahyuddin N, Yaghoubi M, Esfandiari M (2019) Improving the air change rate in high rise buildings through a transom ventilation panel: a case study. *Build Environ* 147:35–49
- Baghaei DA, Khalatbari Limaki A, Safari H (2016) Opening performance simulation in natural ventilation using design-builder (case study: a residential home in Rasht). *Energy Proc* 100:412–422
- Candido C, deDear R, Lamberts R, Bittencourt L (2010) Air movement acceptability limits and thermal comfort in Brazil's hot humid climate zone. *Build Environ* 45:222–229
- Chen Q (2004) Using computational tools to factor wind into architectural environment design. *Energy Build* 36:1197–1209
- Elshafei G, Negam A, Bady M, Suzuki M, Ibrahim M (2017) Numerical and experimental of the impacts of window parameters on indoor natural ventilation in a residential building. *Energy Build* 141:321–332
- Gao CF, Lee WL (2011) Evaluating the influence of openings configuration on natural ventilation performance of residential units in Hong Kong. *Build Environ* 46:961–969
- Gratia E, Herde A (2004) Natural cooling strategies efficiency in an office building with a double-skin façade. *Energy Build* 36:1139–1152
- Guo W, Liu X, Yuan X (2015) A case study on optimization of building design based on CFD simulation technology of wind environment. *Procedia Eng* 121:225–231
- Hanan MT (2015) Natural ventilation as energy efficient solution for achieving low energy houses in Dubai. *Energy Build* 99:284–291
- Hatamipour MS, Abedi A (2008) Passive cooling systems in buildings: some useful experiences from ancient architecture for natural cooling in a hot and humid region. *Energy Convers Manag* 49:2317–2323
- Hu K, Chen Q (2015) Ventilation optimization for reduction of indoor semi-volatile organic compound concentration based on the variational principle. *Build Environ* 94:676–682
- Hu C, Ohba M, Yoshie R (2008) CFD modelling of unsteady cross ventilation flows using LES. *J Wind Eng Ind Aerod* 96:1692–1706
- Iranian national building code (2013) Iranian code of practice for seismic resistant design of buildings (standard no 2800)/permanent committee for revising the Iranian code of practice for seismic, Ministry of Road and Urban Development. Building & Housing Research center BHRC Publication No. S – 465. ISBN: 978-964-9903-41-5
- Jamaludin AA, Hussein H, Ariffin A, Keumala N (2014) A study on different natural ventilation approaches at a residential college building with the internal courtyard arrangement. *Energy Build* 72:340–352
- Kobayashi N, Chen Q (2003) Floor-supply displacement ventilation in a small office. *Indoor Built Environ* 12:281–291
- Li H, Wang S, Cheung H (2018) Sensitivity analysis of design parameters and optimal design for zero/low energy buildings in subtropical regions. *Appl Energy* 228:1280–1291
- Liu L, Wang L, Zhang Y, Dong B, Song B (2015) Study on ventilation rates at university dormitories in winter. *Procedia Eng* 121:743–748
- Nejat P, Jomehzadeh F, Taheri MM, Gohari M, Majid MZA (2015) A global review of energy consumption, CO₂ emissions and policy in the residential sector (with an overview of the top ten CO₂ emitting countries). *Renew Sustain Energy Rev* 43:843–862
- Orme M (2001) Estimates of the energy impact of ventilation and associated financial expenditures. *Energy Build* 33:199–205
- Oropeza-Perez I (2015) Assessment of natural ventilation within buildings located on hot-temperate and mild-temperate regions. *Energy Proced* 78:1513–1518
- Prajongsan P, Sharples S (2012) Enhancing natural ventilation, thermal comfort and energy savings in high-rise residential buildings in Bangkok through the use of ventilation shafts. *Build Environ* 50:104–113
- Roetzel A, Tsangrassoulis A, Dietrich U, Busching S (2010) A review of occupant control on natural ventilation. *Renew Sustain Energy Rev* 14:1001–1013
- Schulze T, Eicker U (2013) Controlled natural ventilation for energy efficient buildings. *Energy Build* 56:221–232



- Seifert J, Li Y, Axley J, Rosler M (2006) Calculation of wind-driven cross ventilation in buildings with large openings. *J Wind Eng Ind Aerodyn* 94:925–947
- Steadman RG (1994) Norm of apparent temperature in Australia. *Aust Meteorol Mag* 43:1–16
- Tantasavasdi Ch, Srebric J, Chen Q (2001) Natural ventilation design for houses in Thailand. *Energy Build* 33:815–824
- Yıldız Y, Durmus Z (2011) Identification of the building parameters that influence heating and cooling energy loads for apartment buildings in hot-humid climates. *Energy* 36:4287–4296
- Yu CKH, Li M, Chan V, Lai ACK (2014) Influence of mechanical ventilation system on indoor carbon dioxide and particulate matter concentration. *Build Environ* 76:73–80
- Zhang W, Wang L, Ji Z, Ma L, Hui Y (2015) Test on ventilation rates of dormitories and offices in University by the CO₂ tracer gas method. *Procedia Eng* 121:662–666
- Zhou C, Wang Z, Chen Q, Jiang Y, Pei J (2014) Design optimization and field demonstration of natural ventilation for high-rise residential buildings. *Energy Build* 82:457–465

