

A novel wall windcatcher (WWC) natural ventilation system evaluated through CFD and experimental field testing - a solution to single-sided ventilation in multi-story buildings?

Salah Almazmumi^{a,b,*}, Hao Sun^a, Miaomiao Liu^d, John Calautit^a, Carlos Jimenez-bescos^c

^a Department of Architecture and Built Environment, University of Nottingham, UK

^b Architectural Engineering Department, Najran University, Najran, Saudi Arabia

^c Westminster Business School, University of Westminster, UK

^d Department of Mechanical Engineering, University of Bath, UK

ARTICLE INFO

Keywords:

Computational fluid dynamics (CFD)
Field experiment testing
Multi-story and multi-zone building
Natural ventilation
Passive ventilation
Wind tower
Windcatcher
Built environment
High rise building
Single sided ventilation
Indoor environment

ABSTRACT

As cities grow and building densities increase, maintaining indoor air quality and thermal comfort in multi-story buildings presents a significant challenge. Mechanical ventilation systems, though effective, are energy-intensive, costly to maintain, and contribute to emissions. In contrast, natural ventilation offers a more sustainable alternative but can be constrained by building design limitations, particularly in high-rise or multi-story structures where single-sided ventilation (SSV) may not be able to provide adequate airflow across multiple floors or zones, while adopting other strategies such as cross and wind tower ventilation could be challenging due to the layout of indoor spaces. This study proposes an innovative wall windcatcher (WWC) system to enhance passive ventilation in multi-story buildings. The WWC features an external rectangular ventilation duct mounted on the building façade, incorporating separate channels for supplying fresh air and exhausting stale air, utilizing wind-induced pressure differences. A combined experimental and numerical approach evaluated the WWC's performance through computational fluid dynamics (CFD) simulations and field experiment testing of a scaled model. The CFD model demonstrated good agreement with field measurements, confirming its reliability. Compared to SSV, the WWC improved airflow across all floors and wind angles. At 0° wind direction, airflow increased by 1.2 times on the ground floor, 2.2 times on the first floor, and 1.6 times on the second floor. Even at 90° and 180°, where SSV struggled, the WWC maintained improved ventilation. However, the study also identified airflow reversal in the ground-floor exhaust outlet, leading to inefficient ventilation. To resolve this, the dimensions of the exhaust duct branches were optimized by adjusting the cross-sectional areas creating transition zones, successfully redirecting airflow and preventing recirculation. Despite these enhancements, airflow remained non-uniform at certain wind angles, indicating the need for further refinements. The WWC system offers a practical and scalable solution for multi-story building ventilation, particularly in retrofit applications where internal modifications are limited. Due to its modular design, the WWC system can be installed with minimal disruption, making it well-suited for integration into both new buildings and retrofit projects.

1. Introduction

The rapid pace of urbanization and the consequent rise in population density in urban areas have necessitated the construction of multi-story or high-rise buildings [1]. These buildings are designed to accommodate large numbers of people within limited urban spaces. However, they often face challenges in maintaining good indoor environmental quality. One of the primary issues in multi-floor and multi-zone buildings is the

complexity and inefficiency of ventilation systems, which can lead to poor air quality [2]. Densely populated floors with numerous occupants can contribute to high levels of indoor pollutants [3]. In addition, inadequate ventilation, excessive solar heat gain, and the use of heat-retentive construction materials can lead to the accumulation of internal heat in these buildings. This can result in elevated indoor temperatures, causing discomfort, heat stress, and reduced productivity among occupants, particularly in warmer climates [4].

* Corresponding author.

E-mail addresses: Salah.almazmumi@nottingham.ac.uk (S. Almazmumi), John.Calautit1@nottingham.ac.uk (J. Calautit).

<https://doi.org/10.1016/j.buildenv.2025.113307>

Received 5 March 2025; Received in revised form 23 May 2025; Accepted 16 June 2025

Available online 18 June 2025

0360-1323/© 2025 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

To mitigate these issues, most high-rise buildings depend on mechanical ventilation systems such as heating, ventilation, and air conditioning (HVAC) systems. While these systems provide consistent airflow and maintain high indoor air quality, they are energy-intensive and require regular maintenance for optimal performance. This dependence on mechanical systems increases energy consumption and operational costs. It also contributes significantly to carbon emissions [5]. In contrast, natural ventilation offers a more sustainable alternative, particularly for buildings designed with effective airflow strategies that utilize natural forces such as wind and thermal buoyancy.

Natural ventilation strategies in buildings can broadly be categorized into wind-driven ventilation [6] and buoyancy-driven (stack) ventilation [7], both of which utilize passive architectural features to facilitate airflow. Wind-driven ventilation includes methods such as single-sided ventilation [8], cross-ventilation [9], and windcatchers (or wind towers) [10], where airflow is induced primarily by external wind pressure differences across openings or structures. Single-sided ventilation (SSV) typically involves air entering and exiting through the same opening, such as windows or vents, which often results in limited airflow penetration and reduced ventilation effectiveness [11], especially in deep or elongated spaces [12]. In contrast, cross-ventilation relies on airflow passing through openings located on opposite or different sides of a building, generally achieving greater airflow rates and more effective distribution [13,14]. However, the effectiveness of cross-ventilation can be significantly constrained in multi-story and multi-zone buildings due to internal layouts [15], partitioning [16], and external environmental obstructions [17]. Windcatchers, or traditional wind towers (such as "Badgir"), are specialized architectural structures historically employed in hot, arid climates to enhance ventilation through capturing high-altitude airflow and channeling it into buildings [18], thus improving indoor air quality and thermal comfort [19,20].

Buoyancy-driven (stack) ventilation, in contrast, relies on temperature-induced density differences between indoor and outdoor air, creating natural upward airflow movement [7,21]. Architectural elements that facilitate stack ventilation include passive chimneys [22], atriums [23], solar chimneys [24], and double-skin façades [25], which can enhance vertical airflow circulation, especially in taller buildings or structures with internal heat gains. Despite their individual advantages, both wind-driven and buoyancy-driven natural ventilation approaches

face limitations in multi-story structures [15,26], where complex airflow patterns and varying indoor conditions can hinder effective and uniform ventilation across multiple floors [7,27].

Windcatchers, or wind towers, are traditional architectural elements that have been used for centuries in hot, arid regions, particularly in the Middle East [7]. These structures are designed to enhance airflow by channelling wind into buildings and expelling warm, stale air. In Architectural features like vents contexts, adaptations of windcatchers in regions like the UK have been developed to be lighter and more feasible to install on the rooftop of existing buildings [8]. However, these modern windcatchers primarily ventilate the spaces directly beneath them, limiting their effectiveness in multi-story buildings with multiple floors and zones (Fig. 1).

1.1. Novelty, aims and objectives

Given these challenges, innovative solutions are needed to enhance natural ventilation in multi-story buildings. This study proposes the development and implementation of a novel windcatcher system, referred to as the wall windcatcher (WWC), designed to improve natural ventilation in multi-story buildings. The proposed WWC system specifically adapts the traditional windcatcher concept for use in multi-story buildings, where each level consists of separate, isolated spaces requiring independent ventilation. Consequently, several structural modifications were introduced into the conventional windcatcher design to ensure compatibility and effective integration into these building types. The WWC combines the advantages of modern windcatchers, which are easy to install and capable of supplying fresh air and exhausting stale air, with the ability to provide airflow across different floors (Fig. 1), similar to windows or vents. Additionally, the WWC operates without the need for internal ducting and pipework, potentially reducing installation complexity, maintenance requirements, and associated costs. This makes it a practical solution for both new and existing multi-story buildings. For retrofits, this external installation process can minimize disruption to building occupants and operations.

The WWC consists of two main components: a side vent that captures outdoor wind flow and an external tube extending to the top of the building to expel stale air (Fig. 1). Modern windcatchers are typically positioned at roof level and primarily ventilate the uppermost spaces

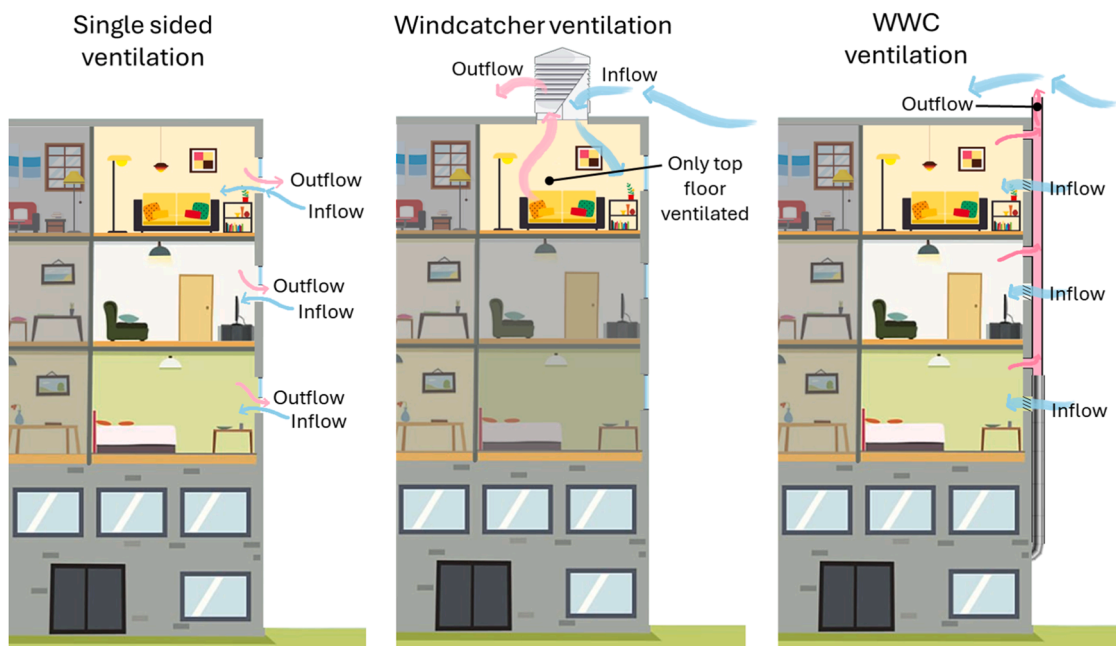


Fig. 1. Application of single sided, windcatcher, and WWC ventilation systems in a multi-story building.

directly beneath their openings, making it challenging to effectively deliver airflow to lower floors in multi-story buildings without additional ducting or modifications (Fig. 1). In contrast, the WWC uses a rectangular tubing system mounted on the building's exterior. One end of the tube protrudes into the outdoor environment, exposed to prevailing winds, while the other end extends into the interior spaces to be ventilated (Fig. 2). High-level winds flowing across the top opening of the tube create a lower pressure zone inside, generating a pressure gradient between the top of the tube and the indoor environment. This gradient promotes the upward expulsion of stale indoor air through an exhaust channel, while fresh air is simultaneously drawn into the building through a separate supply channel positioned on the same wall or side as the exhaust. The separation of supply and exhaust channels also allows for the potential future integration of pre-cooling or heat recovery strategies. Designed to function regardless of wind direction, this system ensures continuous natural ventilation. By utilizing wind-induced pressure differences, the WWC ensures continuous airflow, thereby enhancing natural ventilation efficiency.

This study **aims** to evaluate the feasibility of the WWC system in enhancing natural ventilation in multi-story buildings. A computational fluid dynamics (CFD) model of the WWC device integrated into a multi-story building is developed to assess ventilation performance. The model employs the 3D steady Reynolds-averaged Navier–Stokes (RANS) and the standard k- ω turbulence model. Additionally, an atmospheric boundary layer (ABL) wind profile is used to simulate the approach wind. Its performance is compared against single-sided ventilation. A small-scale prototype was constructed, and field experiments were

conducted in Nottingham, UK. This helped assess its ventilation performance under real-world conditions, and the experimental data was used to validate the CFD modelling. The study evaluates the impact of varying wind speeds and directions, external duct geometry, and variations in duct branch sizing to address challenges such as reverse airflow or uneven ventilation distribution, focusing particularly on airflow and pressure distribution.

The study is guided by key research questions: How effectively does the WWC system improve natural ventilation across multiple floors compared to conventional single-sided ventilation? What is the impact of varying wind speeds and directions on the system's airflow and pressure distribution? How do different external duct geometries affect the uniformity of air distribution across different building rooms? The research also seeks to enhance the design to ensure consistent airflow distribution across each floor, thereby enhancing the overall performance of the WWC system in a multi-story context.

2. Literature review

Multi-story buildings require efficient ventilation systems due to their complex internal layouts and challenges associated with distributing airflow uniformly across multiple floors [15]. Ineffective ventilation in these buildings can result in poor indoor air quality, overheating, discomfort, and reduced occupant productivity [4]. Mechanical ventilation systems, though commonly used, are energy-intensive, costly to maintain, and contribute significantly to carbon emissions [5,28]. Natural ventilation provides a sustainable alternative, utilizing passive

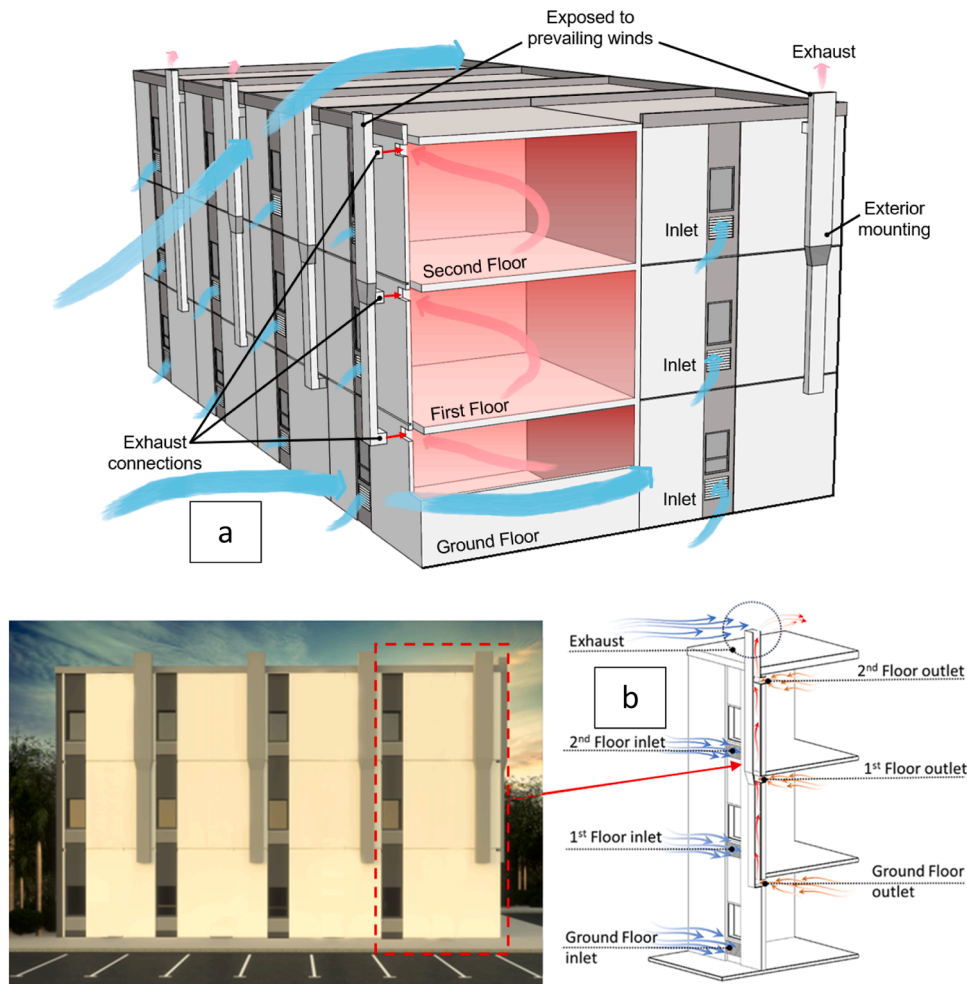


Fig. 2. (a) Example of the installation of the WWC system into a multi-story building, and (b) the working principle.

elements such as windows, vents, chimneys, and windcatchers to harness wind [6] and buoyancy-driven airflow [7].

Single-sided ventilation (SSV), commonly employed in multi-story buildings due to compact unit arrangements, involves airflow entering and exiting through openings located on only one façade [29], typically driven by wind-induced pressures [6]. Despite being simple and cost-effective, the effectiveness of SSV is generally limited, particularly for deeper indoor spaces or elongated rooms, due to restricted airflow penetration and limited internal mixing [12]. Additionally, multi-story buildings, especially those in dense urban environments, experience complex airflow interactions significantly influenced by local turbulence and sheltering from surrounding structures. Gough et al. [8] noted that single-sided ventilation models frequently underestimate ventilation rates in sheltered scenarios, with discrepancies up to a factor of ten. They emphasized that incorporating local turbulence intensity significantly improves model accuracy for urban multi-story buildings.

Recent experimental investigations have demonstrated that the performance of SSV can be enhanced through specialized airflow-guiding components. For example, while louvres generally reduce airflow due to reduced effective opening areas, strategically positioned guide vanes can markedly enhance airflow rates by minimizing recirculation and directing airflow more effectively, potentially achieving ventilation rates up to five times greater than simple openings [11]. Additionally, techniques such as wing walls, which generate differential pressures around openings, can further augment SSV effectiveness by boosting airflow rates and improving indoor air circulation [6].

Advanced studies have explored the phenomenon of "pumping ventilation," a specialized form of single-sided ventilation driven by periodic vortex shedding, which has shown promise for multi-story applications [15,30]. CFD analyses indicate that asymmetric placement of openings and internal partition configurations significantly affect airflow distribution and ventilation efficiency in multi-story structures. Zhong et al. [15] revealed that simultaneous pumping ventilation across multiple floors can enhance airflow rates, particularly benefiting intermediate floors, though careful consideration of aperture separation and positioning remains critical to maximizing benefits [30].

In contrast, cross-ventilation characterized by airflow through openings on opposite or adjacent building façades generally offers higher ventilation rates compared to SSV, quickly dissipating internal heat and pollutants [9,13]. Chu [13] demonstrated that cross-ventilation could achieve airflow rates approximately 20 times greater than SSV under identical wind speeds and openings. However, implementing cross-ventilation effectively in multi-story buildings is challenging due to shared walls, internal partitions, limited external façade exposure, and dependence on building orientation and local wind conditions [14]. Cheung and Liu [27] emphasized that closely spaced multi-story buildings substantially reduce cross-ventilation effectiveness due to sheltering effects and disrupted airflow patterns. Their findings indicated that optimal airflow performance in urban building clusters could be significantly enhanced through careful consideration of building spacing and staggered arrangements.

Moreover, environmental elements such as trees can further complicate airflow dynamics, potentially reducing cross-ventilation effectiveness. Zhang et al. [17] observed that tree placement, foliage density, and shape significantly impact cross-ventilation rates, underscoring the importance of strategic landscape planning to maintain adequate natural ventilation in urban settings. Therefore, effective natural ventilation strategies for multi-story buildings require careful design consideration of aperture configurations, building geometry, environmental context, and local wind patterns to ensure robust ventilation performance across varying conditions [16,31,32].

Other passive ventilation approaches, such as vertical ventilation shafts and passive chimneys, employ buoyancy-driven (stack) effects to achieve airflow in multi-story buildings. These methods utilize temperature-induced pressure differences: warmer indoor air rises through vertical shafts and exits the building, drawing cooler outdoor

air into lower openings. Shi et al. [7] showed that a well-designed vertical shaft could distribute uniform airflow rates across multiple floors, even when internal heat gains varied between stories, emphasizing the importance of balancing buoyancy forces. Additionally, Prajongsan and Sharples [33] demonstrated that integrating vertical shafts significantly improved airflow in single-sided ventilated spaces, increasing occupant comfort hours from 38 % to 56 % during the hot season in a 25-story residential building. However, while effective under notable indoor-outdoor temperature differences, these methods can be less reliable during mild weather when temperature gradients diminish, leading to reduced airflow and inconsistent ventilation rates.

Windcatchers or wind towers represent another effective method of natural ventilation [34], which can deliver higher ventilation rates compared to conventional single-sided or cross-ventilation techniques, particularly in dense urban environments [35]. Traditional window-based methods can become compromised in built-up areas due to disrupted wind patterns caused by surrounding structures, leading to reduced airflow effectiveness. In contrast, windcatchers can overcome these challenges by capturing wind at higher altitudes where airflow tends to be stronger, steadier, and less obstructed, subsequently channeling this airflow into building interiors to enhance ventilation effectiveness [36,37]. For example, Sadeghi et al. [38] reported an average indoor air velocity of 0.53 m/s using a windcatcher configuration, which was approximately three times greater than typical window ventilation. Historically utilized in hot, arid climates such as those in the Middle East and North Africa, windcatchers have recently been adapted for modern use in temperate regions including the UK [39]. Numerous studies have examined how variations in structural parameters, including height, opening size, geometry, and orientation, affect windcatcher ventilation performance [40,41].

However, it is crucial to acknowledge that the effectiveness of natural ventilation methods such as windcatchers is dependent on ambient temperature conditions. This dependency poses challenges, particularly during periods of high ambient temperatures, where reliance solely on wind-driven ventilation may lead to overheating. Consequently, recent studies have incorporated supplementary cooling strategies into modern windcatchers such as evaporative cooling [34] and heat pipes [42]. While rotary thermal wheels [43], and tube-based heat recovery systems [44] are integrated with roof top windcatcher to enhance the year-round effectiveness and thermal performance of windcatcher-based natural ventilation systems in cold climates.

Despite their advantages, the effectiveness of windcatcher systems in multi-story buildings remains underexplored compared to their application in single-story or low-rise structures [40]. Modern windcatchers are generally designed to capture wind at the roof level and redirect it downwards into the building [10]. This design effectively ventilates the space directly beneath the windcatcher, typically the top floor, but its ability to distribute fresh air across multiple floors remains limited. The downward airflow generated by the windcatcher may lose momentum as it travels through the building, making it less effective for ventilating lower floors. Additionally, the lack of a mechanism to uniformly distribute air across different levels can result in uneven ventilation, where only the uppermost areas benefit from improved airflow, while lower floors remain inadequately ventilated. This limitation highlights the need for alternative designs or enhancements that can extend the benefits of windcatcher systems to all floors in a multi-story building.

Numerous studies [22] have explored the performance of windcatcher ventilation devices; however, the majority have concentrated on single-zone or single-floor applications, limiting their relevance for more complex, multi-story buildings. For example, one investigation [42] numerically and experimentally assessed a windcatcher integrated with heat pipes to improve thermal comfort in hot climates. Although the findings confirmed enhanced performance due to the heat pipes, the scope was limited to a single-room scenario, leaving potential multi-floor applications unexplored. Similarly, other studies [43,44] have aimed to enhance windcatcher effectiveness in colder climates

through the integration of heat recovery systems, yet these efforts were also restricted to single-floor setups and did not consider the complexities of multi-story ventilation challenges. Additionally, research efforts [27], such as those involving the addition of wing walls [45] or other design modifications to enhance ventilation performance, have primarily focused on optimizing windcatchers for single-floor use. These improvements have not been adequately tested or validated in the context of multi-floor spaces, where airflow dynamics and inter-floor interactions present additional complexity, highlighting a significant gap in the current literature.

This study introduces a windcatcher system designed specifically for multi-story buildings, characterized by its multidirectional functionality, simplicity, and ease of retrofit. Fig. 2 illustrates a schematic diagram of the proposed device. A rectangular tube is mounted externally on the building wall, with one end exposed to the outdoor environment and prevailing winds, and the other end extending into the interior space for ventilation. The Bernoulli effect, fundamental to the WWC system's design, describes how an increase in airflow velocity corresponds to a decrease in static pressure [46,47]. The WWC system employs this principle via its external exhaust tube, which is exposed to higher wind velocities at elevated heights. As wind passes horizontally across the tube's upper opening, the high-velocity airflow creates a region of lower static pressure inside the exhaust tube (see Fig. A.1). This pressure difference actively draws stale air from within the building upward and out through the exhaust channel, simultaneously pulling fresh outdoor air inward through inlets positioned on the building's side walls. This natural mechanism enhances ventilation efficiency without requiring additional energy or mechanical intervention.

In contrast, the stack effect relies primarily on temperature-induced buoyancy, occurring when indoor air is warmer and lighter than cooler outdoor air, causing warmer air to rise naturally and exit from higher-level openings. Cooler outdoor air is then drawn in through lower-level openings due to the resulting pressure differential. While effective, the stack effect is predominantly temperature-driven and can be less predictable or effective during mild weather or in buildings without significant internal temperature gradients. The WWC system, by primarily utilizing the Bernoulli effect, offers notable advantages, particularly for multi-story buildings. It does not depend strictly on internal temperature gradients or thermal buoyancy, making it capable of providing effective ventilation irrespective of indoor-outdoor temperature differences. This is particularly beneficial during mild weather conditions, when stack-driven ventilation may be inadequate. Furthermore, as the WWC's operation relies predominantly on external wind flow rather than internal thermal differences, it can offer more predictable and consistent ventilation performance across a broader range of conditions. It should be noted that this study primarily focuses on evaluating the wind-driven airflow performance of the WWC system.

This study investigates the ventilation effectiveness of the proposed WWC system in multi-story buildings. Field testing was conducted to measure airflow velocities at various points within ventilated spaces. A CFD model was developed to simulate the WWC system's performance, and the results were validated against experimental field data. The influence of design parameters and different wind conditions was analyzed using the CFD model. Key metrics such as airflow speed and air distribution across each floor were evaluated.

3. Method

This section outlines the integrated experimental and computational approaches undertaken to evaluate the performance of the proposed WWC system. The methodology consists of three main components: the operational principles and geometric configuration of the WWC system (Section 3.1), the development and implementation of a computational fluid dynamics (CFD) model (Section 3.2), and a field experiment designed to validate the CFD simulations under real-world conditions (Section 3.3). The WWC system was tested against a conventional single-

sided ventilation (SSV) strategy to assess improvements in airflow distribution and ventilation effectiveness in multi-story buildings. Both the simulation and experimental procedures were designed to capture airflow behaviour, optimize duct performance, and ensure the system's applicability in real-world building contexts.

3.1. System operation and model geometry

The WWC features strategically placed inlets at the lower corners of rooms for fresh air intake and outlets at the upper corners connected to a central exhaust tube. This tube extends through all floors and reaches the building's rooftop, effectively removing stale air. The inlets were initially positioned in the corners to maximize the spatial separation between the air intake and exhaust points, thereby preventing air short-circuiting and enhancing internal airflow mixing to ensure effective ventilation throughout the building.

It is important to clarify that in our WWC system, the openings on each floor (inlets and outlets) are components integrated into the overall ventilation strategy facilitated by the external windcatcher cavity itself, rather than integral parts of the windcatcher's structure. These openings were strategically placed to enhance airflow distribution and minimize short-circuiting within each individual floor, specifically addressing the isolated ventilation requirements unique to multi-story buildings.

Fresh air enters the building through the lower inlets and flows into the indoor space, naturally displacing stale air. As the fresh air enters, the displaced stale air rises and exits the building through a dedicated outlet at the top of the interior space. This process is further enhanced as outdoor air flows across the top of the tube, which extends to the building's rooftop, creating a lower pressure inside the tube. This pressure difference increases the airflow within the inlet spaces, promoting ventilation throughout the building (Fig. 2). For comparison, the performance of the WWC system will be evaluated against a similar-sized building and rooms with single-sided ventilation on the same side, as shown in Fig. 3(a). This comparison aims to assess the relative effectiveness of the WWC in improving ventilation and airflow distribution.

Fig. 3 illustrates the geometry dimensions of the multi-story building CFD model, showing the single-sided ventilation configuration and the WWC system configuration. The model represents a simplified, idealized three-story building with the proposed system mounted on one side. It is important to note that this building model is not intended to replicate a real building, as it excludes various internal and external features that could influence airflow. The geometry was modelled in CAD and then transferred to ANSYS Workbench for CFD meshing and simulations.

The selection of a rectangular cross-section for the WWC system's ductwork was based on practical considerations for building integration and airflow optimization. Initially, the system was conceptualized without specific constraints on the cross-sectional geometry. However, experimental validation and iterative optimization indicated that the exhaust duct required a gradual increase in cross-sectional area with height to sustain effective airflow and pressure balance. A circular cross-section was initially considered but found impractical, as accommodating the necessary diameter increase would significantly protrude from the building envelope, complicating retrofitting and integration. Conversely, a rectangular cross-section offered greater adaptability by allowing width adjustments without substantially increasing depth, making it more suitable for integration within the wall structure of both existing and new buildings. This design choice enhances the WWC system's applicability and scalability across diverse architectural contexts.

3.2. CFD modelling approach

3.2.1. CFD theory

This study utilized the control volume approach, employing ANSYS FLUENT 2022 R1 software to conduct steady-state Reynolds-Averaged Navier-Stokes (RANS) simulations for analyzing fluid flow dynamics.

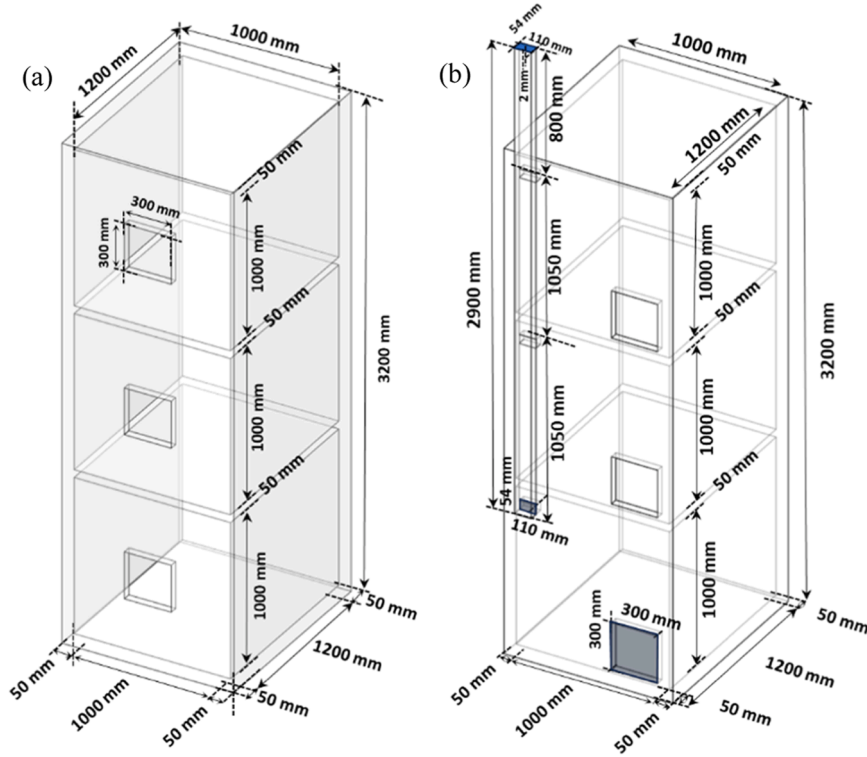


Fig. 3. Dimensions of the multi-story building CFD model with (a) single sided ventilation and, (b) WWC system.

The three-dimensional Computational Fluid Dynamics (CFD) simulations specifically focused on turbulent and incompressible airflow characteristics within and around the proposed windcatcher system. The standard $k-\omega$ turbulence model was selected to resolve the RANS equations. This choice was informed by existing literature on modeling windcatchers and similar ventilation systems [35,48], complemented by a comparative analysis (Fig. A.2) carried out in this study to assess the suitability of different turbulence models. The analysis considered five widely-used turbulence models: Standard $k-\epsilon$, RNG $k-\epsilon$, Realizable $k-\epsilon$, Standard $k-\omega$, and $k-\omega$ SST [49]. Each model was evaluated based on airflow predictions at critical measurement locations (ground and first-floor inlets and outlets), across multiple wind speeds, and subsequently validated against experimental data.

The comparative evaluation indicated that the standard $k-\omega$ model consistently provided reliable and accurate predictions across all wind scenarios. It demonstrated good performance, particularly at higher wind speeds, and maintained good numerical stability and convergence during simulations. Thus, due to its overall accuracy, consistency, and computational stability, along with its specific strengths in modeling airflow within confined geometries, the standard $k-\omega$ turbulence model was chosen for detailed analysis in this study. The computational approach employed the finite volume method (FVM) with the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm to couple the velocity and pressure fields. To enhance simulation accuracy, second-order upwind discretization schemes were applied for pressure, momentum, and turbulence parameters. Solution convergence was achieved when the residuals for mass, momentum, and energy equations dropped below the threshold of 10^{-6} , ensuring the robustness and reliability of the simulation outcomes. The steady-state governing equations were implemented according to the standard formulations provided in ANSYS FLUENT [50].

Continuity equation:

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

Where ρ denotes the fluid density, t represents time, and refers to the

$\vec{v}$ velocity vector.

Momentum equation:

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \tau + \rho \vec{g} \quad (2)$$

where p stands for pressure, τ symbolizes the stress tensor, and $\vec{g}$ denotes the gravitational acceleration vector.

Energy equation:

$$\frac{\partial}{\partial t} \left[\rho \left(e + \frac{v^2}{2} \right) \right] + \nabla \cdot \left[\rho v \left(h_f + \frac{v^2}{2} \right) \right] = \nabla \cdot \left(\kappa_{eff} \nabla T - \sum_j h_j \vec{J}_j + \tau_{eff} \cdot \vec{v} \right) + S_h \quad (3)$$

where e represents the internal energy, h_f stands for the specific enthalpy of mass, considering only dry air, κ_{eff} denotes the effective thermal conductivity, T is the air temperature, h_j pertains to the sensible heat of species, $\vec{J}_j$ symbolizes the diffusion flux of species (both of which are not included in this analysis), τ_{eff} denotes the effective stress tensor, and S_h represents the volumetric heat source.

The governing transport equations for the k - ω standard model are specified in Equations (4) and (5):

Turbulence kinetic energy k :

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + G_b \quad (4)$$

Specific dissipation rate ω :

$$\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + G_{ob} \quad (5)$$

In these equations, G_k is indicative of the generation of turbulence kinetic energy due to mean velocity gradients; G_ω signifies the generation term for ω ; Γ_k and Γ_ω represent the effective diffusivities of k and ω , respectively; Y_k and Y_ω account for the dissipation of k and ω resulting

from turbulence; and G_b and G_{ob} are the terms that incorporate the effects of buoyancy.

3.2.2. Computational domain and boundary conditions

The setup illustrated in Fig. 4 includes a velocity inlet, where the wind speed distribution is defined according to an atmospheric boundary layer (ABL) profile. This approach is used to ensure that the airflow simulation within the computational domain reflects realistic outdoor wind conditions. By using the ABL wind profile, the simulation can more realistically capture how wind behaves as it interacts with buildings and the surrounding environment. This wind speed distribution at the inlet is kept consistent across all simulation scenarios, with velocity profiles derived using power-law expressions. The resulting mean velocity profile can be expressed as follows:

$$u(z) = u_{\text{ref}} \left(\frac{z}{z_{\text{ref}}} \right)^{\alpha} \quad (6)$$

In this equation, $u(z)$ represents the mean velocity at a given height z , measured in meters per second (m/s). The power law exponent α is set to 0.30 [51], with the reference height z_{ref} defined as 2.55 m. The velocity at this reference height, u_{ref} , was experimentally determined based on field testing. This wind speed distribution was then applied to the inflow boundary within the computational domain to simulate realistic wind conditions. On the opposite side, a pressure outlet with a static pressure of 0 Pascal (Pa) was established. Symmetry boundary conditions were applied to the side walls and the top wall to simulate an infinite domain in both the horizontal and vertical directions, while the ground was treated as a no-slip wall to account for surface friction. All boundary conditions used in FLUENT are summarized in Table 1.

As shown in Fig. 3, CFD simulations were conducted on two different, idealized model configurations. Fig. 3(a) depicts a three-story benchmark model with single-sided ventilation. This model measures 1200 mm in length, 1300 mm in width, and 3200 mm in height, with a material thickness of 50 mm. Each story has a height of 1000 mm, and there are three ventilation openings located at the center of each level, each measuring 300 mm by 300 mm. In contrast, Fig. 3(b) presents the idealized model with the proposed WWC system. In this model, the ventilation openings on each floor have been repositioned, and placed 100 mm away from the boundaries of each level. The exhaust duct system in this configuration has dimensions of 54 mm in width, 110 mm in depth, and 2900 mm in height, and it is also positioned 100 mm from the boundary.

Fig. 4 illustrates the dimensions and configuration of the computational fluid domain adopted in this study. The domain length is 26.9 m, and the cross-sectional dimensions are 20.3 m $\times$ 12.8 m, with the building height $H = 3.2$ m taken as the reference scale. Tominaga et al. [52] recommend inlet and outlet distances of 5 H and 10 H–15 H,

Table 1

Summary of the CFD model and boundary conditions.

Time	Steady state
Velocity inlet	ABL (m/s)
Wind angle	0°
Pressure outlet	Atmospheric
Gravity	−9.81 m/s ²
Wall conditions	No-slip
Turbulence model	k-omega standard
Mesh type	Tetrahedral mesh

respectively, to accommodate wake development. To improve computational efficiency, we first reduced the distances to 3 H (inlet, side walls, and top) and 5 H (outlet). A subsequent outlet-distance sensitivity study (5 H, 10 H, and 15 H; results in Fig. A.3 showed that: ground-floor velocity changed by only 0.36 %, first-floor velocity by 3.5 %, and second-floor velocity by 2.7 %. No numerical instabilities were observed and all simulations converged smoothly. Furthermore, velocity contours in Fig. A.4 (side and top views) consistently demonstrated stable and uniform airflow patterns for all cases, confirming that the shorter downstream boundary at 5H did not cause numerical inaccuracies or premature truncation of flow structures. Consequently, the outlet location at 5 H was retained, giving an accurate solution while halving the computational cost compared with a 10 H–15 H domain.

3.2.3. Mesh and sensitivity analysis

Mesh quality is a key factor in the accuracy of numerical simulations and the efficiency of computation during the modelling process. In this study, a tetrahedral mesh configuration was generated for the fluid domain using ANSYS Mesh, as depicted in Fig. 5(c). To enhance simulation accuracy, two fluid domains were strategically placed around the model to ensure optimal grid transitioning. The surrounding domain was assigned a mesh size of 0.8 m, with targeted mesh refinement in areas critical to the building's simulation. The building itself was meshed with a size of 0.2 m, while the tube received a finer mesh size of 0.008 m, with further refinement at the tube's top to 0.001 m and at the windows to 0.02 m. These adjustments ensured sufficient mesh density in key areas, thereby enhancing the overall accuracy of the simulation results. To maintain mesh quality, the expansion ratio between adjacent cells was kept below 1.2. The accuracy of the numerical model was validated through a mesh sensitivity analysis, which examined the effects of varying mesh sizes on the results. The grid convergence index (GCI), shown in Fig. 5(b), was employed to assess discretization uncertainty. The methodology and parameters of this analysis are detailed further below.

To initiate the calculation process, establishing the grid dimension, represented by h . In this context, ΔV_i signifies the volume, while N stands for the aggregate count of cells utilized in the computational analysis.

$$h = \left[\frac{1}{N} \sum_{i=1}^N (\Delta V_i) \right]^{1/3} \quad (7)$$

Fig. 5 and Table 2 illustrate the selection process for three distinct grid sizes: fine, medium, and coarse, which were implemented in simulations to ascertain the values of key variables ϕ_1, ϕ_2, ϕ_3 essential for fulfilling the objectives of the study. The refinement factor of the grid, represented as $r = h_{\text{coarse}}/h_{\text{fine}}$, is expected to surpass 1.3 [53]. Given $h_1 < h_2 < h_3$, the respective ratios are defined as $r_{21} = h_2/h_1$, and similarly, $r_{32} = h_3/h_2$. Consequently, the grid sizes were designated as N1 (14,992, 946), N2 (6,171,043), and N3 (2,423,610), resulting in r values of 1.344 and 1.366. The method's effective order, denoted by p , is determined through calculations spanning from Eq. (8) to Eq. (14), involving $\epsilon_{32} = \phi_3 - \phi_2$ and $\epsilon_{21} = \phi_2 - \phi_1$.

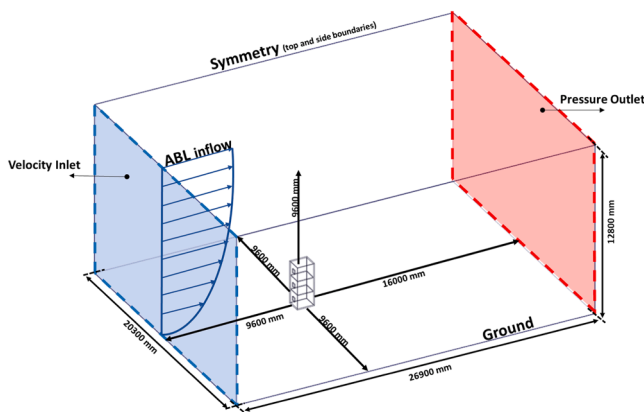


Fig. 4. Dimensions of the computational fluid domain and corresponding boundary conditions.

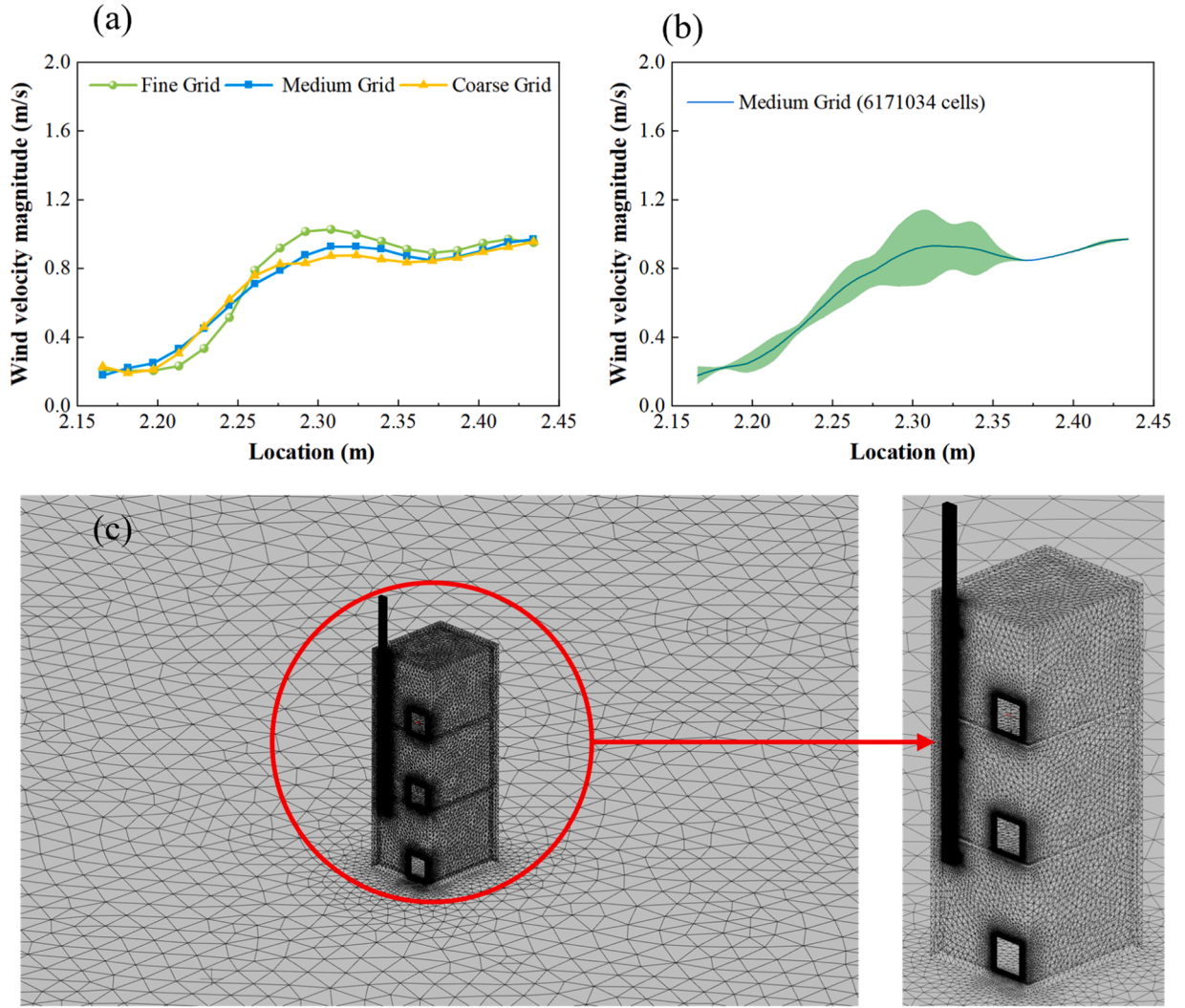


Fig. 5. (a) Results from simulations showing wind velocity magnitude across a horizontal centerline in the model, for assessing grid sensitivity. (b) Analysis of grid sensitivity performed through the application of the GCI method. (c) The tetrahedral mesh configuration.

Table 2
Illustration of computing discretization error using the GCI method.

	Wind velocity magnitude (m/s) at Y = 2.182 m	Wind velocity magnitude (m/s) at Y = 2.229 m	Wind velocity magnitude (m/s) at Y = 2.434 m
N1	14,992,946	14,992,946	14,992,946
N2	6,171,043	6,171,043	6,171,043
N3	2,423,610	2,423,610	2,423,610
r_{21}	1.344	1.344	1.344
r_{32}	1.366	1.366	1.366
ϕ_1	0.205	0.335	0.953
ϕ_2	0.221	0.448	0.972
ϕ_3	0.193	0.461	0.957
p	3.68	6.75	6.09
ϕ_{ext}^{21}	0.197	0.317	0.949
e_a^{21}	7.58 %	34 %	1.99 %
e_{ext}^{21}	3.40 %	5.64 %	0.40 %
$\text{GCI}_{\text{fine}}^{21}$	4.80 %	6.68 %	0.49 %

$$p = \frac{1}{\ln(r_{21})} |\ln |\epsilon_{32}| / \epsilon_{21}| + q(p) \quad (8)$$

$$q(p) = \ln \left(\frac{r_{21}^p - s}{r_{32}^p - s} \right) \quad (9)$$

$$s = 1 \cdot \text{sgn}(\epsilon_{32} / \epsilon_{21}) \quad (10)$$

The values obtained through extrapolation are presented in Equation (13):

$$\phi_{\text{ext}}^{21} = (r_{21}^p \phi_1 - \phi_2) / (r_{21}^p - 1) \quad (11)$$

The calculated relative error, represented by e_a^{21} , the extrapolated relative error, symbolized by e_{ext}^{21} , and the fine grid convergence index, indicated as $\text{GCI}_{\text{fine}}^{21}$, are determined through Equations (15) to (17).

$$e_a^{21} = \left| \frac{\phi_1 - \phi_2}{\phi_1} \right| \quad (12)$$

$$e_{\text{ext}}^{21} = \left| \frac{\phi_{\text{ext}}^{12} - \phi_1}{\phi_{\text{ext}}^{12}} \right| \quad (13)$$

$$\text{GCI}_{\text{fine}}^{21} = \frac{1.25e_a^{21}}{r_{21}^p - 1} \quad (14)$$

This research conducted a comparative analysis using three different grid sizes. The finest grid domain was defined with grid sizes of 1 m, 0.8 m, and 0.6 m for the coarse, medium, and fine meshes, respectively. Furthermore, a coarser grid was developed from the medium grid,

adjusting the grid size in the fluid domain and increasing the building's grid size from 0.1 m to 0.2 m, leading to a total of 2,423,610 cells. For the fine mesh, the building grid size was adjusted to 0.057 m. The tube's grid sizes for coarse and fine meshes were set at 0.008 m and 0.002 m, respectively, while the window grid sizes were established at 0.02 m for coarse and 0.003 m for fine. The tube's top maintained a constant grid size of 0.001 m, akin to the medium mesh. To measure wind velocity (m/s), 18 measurement points were established, with their readings across the three grid sizes displayed in Fig. 5(a). The data in Table 2 highlight points at $Y = 2.18$ m, $Y = 2.23$ m, and $Y = 2.43$ m, showcasing grid

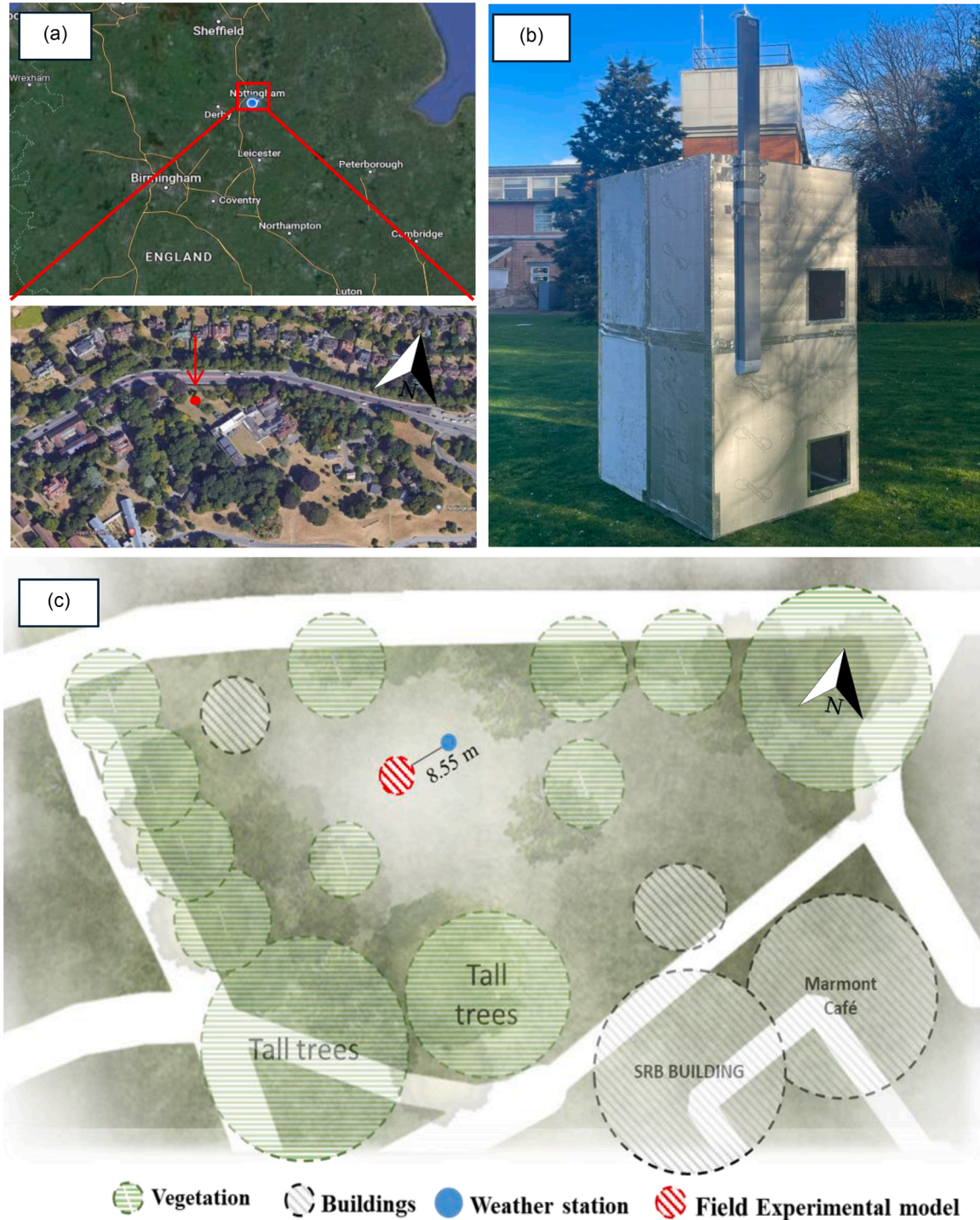


Fig. 6. (a) The location of the experimental site at the University Park Campus, Nottingham, UK. (b) Field experimental model prototype. (c) Site analysis of the field experiment.

convergence indices of 4.80 %, 6.68 %, and 0.49 %, respectively. Given its relatively lower error margin and significantly reduced computational cost, the medium mesh size was chosen for detailed analysis in this study. Fig. 5(b) presents the convergence index and discretization error bars for all 18 points using the medium grid. The highest GCI_{fine}^{21} was observed at $Y = 2.17$ m, registering at 28.70 %, deemed acceptable. Furthermore, the average GCI_{fine}^{21} across all 18 points was calculated as 11.44 %, validating the suitability of the medium mesh size for simulation purposes.

3.3. Experimental approach

The aim of the experiment was to evaluate the performance of the proposed WWC system under real-world conditions and to validate the CFD modeling. Field experiments were conducted at the University Park campus of the University of Nottingham, UK ($52^{\circ}56'40.3''N$, $1^{\circ}11'45.2''W$), in a suburban environment, as shown in Fig. 6(a). The test location was characterized by two- and three-story buildings and trees ranging from 10 to 20 m in height. Nottingham experiences a temperate oceanic climate (Köppen: Cfb), similar to most of the UK, with warm summers and mild to cool winters. Average annual temperatures range from $2^{\circ}C$ in winter to $20^{\circ}C$ in summer. The prevailing wind direction in Nottingham is influenced by Atlantic Ocean currents, typically coming from the southwest, west, or northwest [54,55] as shown in Fig. A.5 which presents the annual wind rose and monthly average wind speeds for Nottingham. Higher wind speeds typically occur in winter months (December–February) and lower speeds in summer months (June–August). This meteorological information provides context for interpreting the ventilation performance of the WWC system under representative wind conditions.

The field experiment was conducted on January 24th. While acknowledging that ventilation airflow typically requires greater attention in warmer seasons, the primary objective of this test was to validate the CFD modeling rather than conducting a seasonal performance evaluation. The selected date allowed testing under a range of

wind directions and speeds, measured on-site and varying between 0.1 and 1.75 m/s, thereby providing representative conditions suitable for robust model validation. Future work will include additional testing in warmer months to comprehensively assess the WWC system's performance.

3.3.1. Experimental prototype

The experimental model consists of two connected cubes, each with internal dimensions of $1\text{ m} \times 1\text{ m} \times 1.2\text{ m}$ ($W \times H \times L$). These cubes represent the ground floor and first floor, respectively, and are constructed from high-insulated polystyrene foam with a thickness of 50 mm. Each cube has two openings: an inlet, measuring $0.3\text{ m} \times 0.3\text{ m}$ ($W \times H$), located in the lower right corner, and an outlet, measuring $0.11\text{ m} \times 0.054\text{ m}$ ($W \times H$). The outlet in each cube is connected to a vertical exhaust tube that extends to the top of the building. The exhaust tube measures $0.11\text{ m} \times 0.054\text{ m} \times 2\text{ m}$ ($W \times H \times L$) with a wall thickness of 2 mm. It extends vertically from the ground floor outlet, reaching a height of 2.1 m above the building. An additional 0.8 m extension further increases the total height of the exhaust tube, bringing the overall height from the outlet to the top of the extension to 2.9 m (Fig. 7). It should be noted that the experimental setup was limited to two storeys due to practical constraints associated with available resources, construction feasibility, and field-testing safety. An identical two-story CFD model was created for direct validation against experimental data. Following successful validation, a three-story model was used for extended simulations to assess the scalability and broader applicability of the WWC system.

3.3.2. Experimental setup and procedure

To conduct the experiment, probes and a weather station were strategically positioned as follows: Seven Testo 405i hot wire anemometers were used to measure wind speed. These anemometers have an accuracy of $\pm(0.1\text{ m/s} + 5\% \text{ of the measured value})$ for wind speeds of 0–2 m/s and $\pm(0.3\text{ m/s} + 5\% \text{ of the measured value})$ for wind speeds of 2–15 m/s. Two probes were placed in the ground floor cube, two in

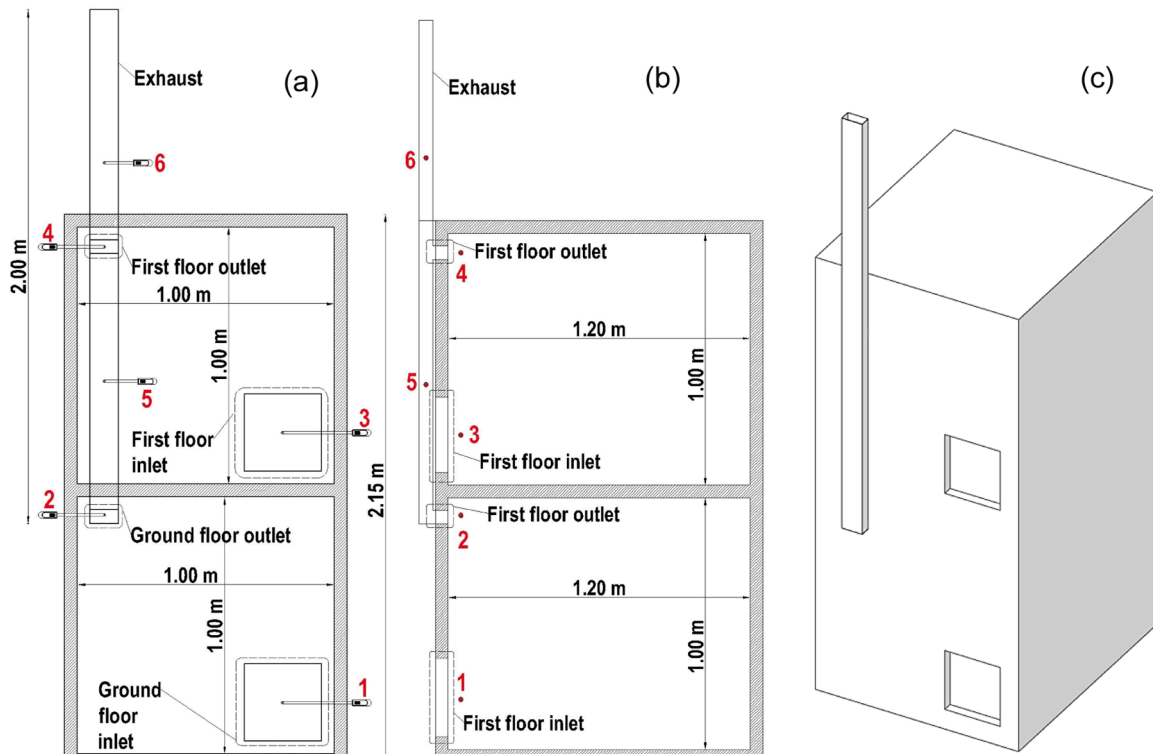


Fig. 7. Field experimental model dimensions and details: (a) elevation, (b) side section, and (c) perspective.

the first-floor cube, two inside the exhaust tube, and the final probe was positioned outside the cubes near the weather station. Weather data, including wind speed, wind direction, and air temperature, was collected using an Ecowitt GW1100 weather station. The wind speed accuracy was ± 1 m/s for speeds between 0 and 5 m/s and ± 10 % for speeds above 5 m/s, while the temperature accuracy was ± 1 °C within a range of -40 °C to 60 °C.

The first probe was placed in the ground floor cube at the center of the inlet, positioned 0.25 m above the floor and 0.25 m from the outer right wall. The second probe was positioned in front of the outlet in the ground floor cube, at a height of 0.928 m from the ground and 0.23 m from the left wall. The third probe was installed in the first-floor cube, centered at the inlet, at a height of 1.350 m from the floor and 0.250 m from the right wall. The fourth probe was placed in front of the outlet on

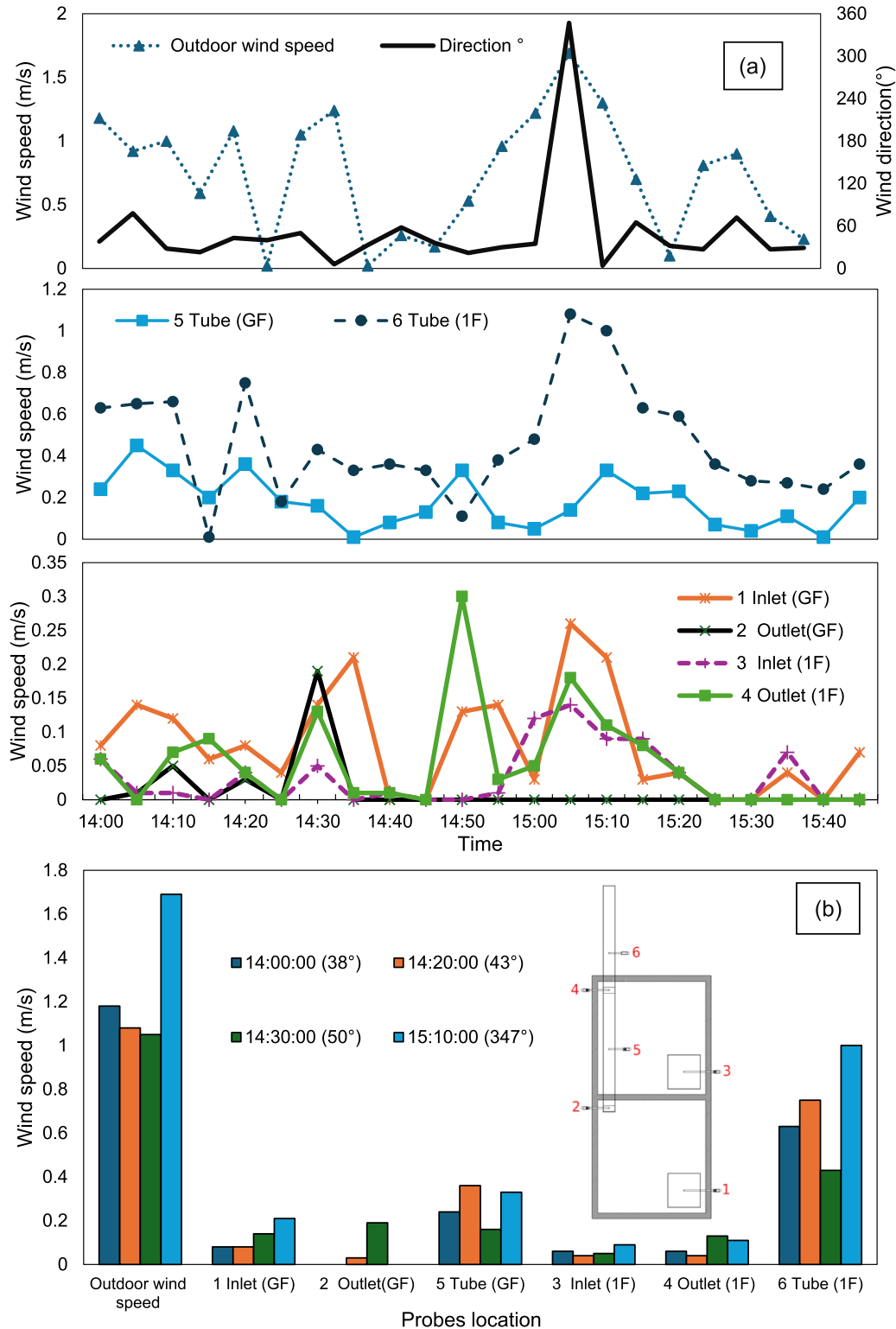


Fig. 8. (a) Outdoor wind speed and direction, tube and inlet/outlet airflow velocity measurements taken during the field experiment from 14:00 to 15:45, and (b) comparison of airflow velocities at various indoor and outdoor measurement locations at selected time intervals.

the first floor, situated at a height of 1.90 m from the ground and 0.23 m from the left wall. All probes inside the cubes were positioned 0.05 m away from the inlet and outlet walls. Additionally, the fifth and sixth probes were installed in the exhaust tube at heights of 1.5 m and 2.35 m, respectively, both located 0.23 m from the left wall (Fig. 7(a)).

It should be noted that the sensor positions were chosen primarily to capture the effectiveness of air delivery at the supply inlets, extraction at the exhaust outlets, and airflow velocities within the exhaust tube for each floor. The collected data were used primarily for validating the CFD model. Recognizing the limitations of the measurement sensors in accurately detecting very low airflow velocities, measurements were focused on areas where airflow reliably exceeded sensor detection thresholds. By targeting these locations, the data provided higher-confidence results, ensuring robust validation of the numerical simulations. Additional locations within the central and peripheral room areas were assessed through complementary numerical analyses, providing comprehensive insights into airflow distribution patterns.

In addition to the indoor probes, one probe was placed outdoors to measure wind speed. This probe was positioned at a height of 2.55 m, matching the height of the weather station. The weather station, located in an open field, collected data on wind speed, direction, and air temperature. This setup adhered to recommended guidelines to minimize airflow disruptions caused by obstructions. The weather station was used exclusively for obtaining wind direction data, while the outdoor wind speed was measured using a Testo hot wire probe, identical to those used inside the model. This ensured consistency in the wind measurement scale across all probes. The data collected from the weather station was wirelessly transmitted using the WSVIEW Plus app and further analysed in Excel. To prevent any interference from the airflow patterns generated by the test model, the distance between the weather station and the test room was set at three times the height of the test room.

4. Results and discussion

This section analyses the results obtained from the field experiment, CFD model validation, and CFD simulations of both the single-sided and proposed system models. The validation of the CFD simulation against the field experiment airflow data was conducted to establish the reliability of the computational modelling method. Additionally, the airflow characteristics of the single-sided and proposed systems under different wind directions were discussed. The optimization was specifically conducted for the proposed system.

4.1. Field experiment results

The field experiment was conducted during a typical winter period in the UK. Outdoor wind speed data collected from the weather station ranged between 0.02 and 1.7 m/s, with wind direction varying between NNW 347° and ENE 78°. This variation allowed us to assess the system's performance under different wind conditions. The model was oriented to the west, meaning that the supply inlet openings did not always face the wind during the test. This setup further helps demonstrate the capability of the system to function effectively even when the wind is not directly facing the supply opening.

Fig. 8(a) shows the indoor wind speed measurements within the rooms and the proposed system's channel, along with the corresponding outdoor wind speed and direction at different times during the test. The data reveal that the outdoor wind speed and direction fluctuated significantly, leading to variations in airflow speed across different

measurement locations in the supply inlets, outlets, and exhaust tube channel. For example, at 14:00, with an outdoor wind speed of 1.18 m/s and a wind direction of 38°, the airflow speed at the ground floor supply inlets was 0.08 m/s, which was 28.6 % higher than that at the first-floor supply inlets. The first floor exhibited lower inlet speeds compared to the ground floor, in contrast to the outlet airflow speeds. Within the exhaust tube channels, the airflow speed on the first floor was 0.63 m/s, significantly higher than the 0.24 m/s recorded on the ground floor. This indicates an unequal flow distribution within the exhaust tube channels, suggesting the need for design adjustments. The measured airflow speeds at the exhaust outlet points on the first floor were significantly higher than those on the ground floor, further highlighting the unequal distribution of flow speeds between the two floors.

At 14:20, with an outdoor wind speed of 1.08 m/s and a wind direction of 43°, the ground floor inlet speed was 0.08 m/s, while the first floor inlet registered a lower speed of 0.04 m/s. The first floor showed significantly higher airflow in the exhaust tube at 0.75 m/s, compared to 0.36 m/s on the ground floor. By 14:30, with slightly altered wind conditions, the ground floor inlet speed increased to 0.14 m/s, while the first floor remained low at 0.05 m/s. Exhaust tube speeds continued to be higher on the first floor. By 15:10, under stronger winds, the ground floor inlet speed decreased to 0.21 m/s, while the first floor improved slightly to 0.09 m/s. The exhaust tube on the first floor reached 1 m/s, further highlighting the uneven airflow distribution between floors, indicating a need for design adjustments.

The data collected during the field experiment revealed notable differences in airflow distribution between the ground and first floors (Fig. 8(b)). The ground floor consistently exhibited higher airflow speeds at the supply inlets compared to the first floor. This discrepancy can be attributed to the airflow dynamics around the building facade, with lower-level inlets being more directly influenced by local pressure variations and turbulence. In contrast, the first floor generally showed higher airflow velocities within the exhaust tube and at the exhaust outlets compared to the ground floor. This outcome likely results from the airflow dynamics within the vertical exhaust channel, where the upward airflow from the lower level contributes to increased airflow velocities at the higher (first floor) level. As a consequence, airflow extraction was higher on the first floor, potentially limiting effective air expulsion from the ground floor.

The system's responsiveness to external wind conditions was evident, as slight changes in wind speed and direction led to significant variations in airflow within the building and air channel. While the WWC system demonstrated effectiveness in supplying and exhausting airflow, the uneven distribution of air across floors and sensitivity to external conditions highlight areas for improvement. Although these experiments did not cover all wind conditions, particularly those with wind directly facing the inlet, further evaluations will be conducted using numerical simulations. First, the modeling will be validated against the experimental data in the following subsection. It is important to note that the CFD validation presented in the following section (Section 4.2) used a numerical model identical to the experimental two-story prototype. The subsequent numerical analyses (Section 4.4–4.5) employing a three-story model were conducted separately to explore the scalability and extended applicability of the WWC system, building upon insights gained from the validation stage.

4.2. CFD modelling validation

Validating the field experimental results is essential before exploring various parameters of the proposed system. This subsection focuses on

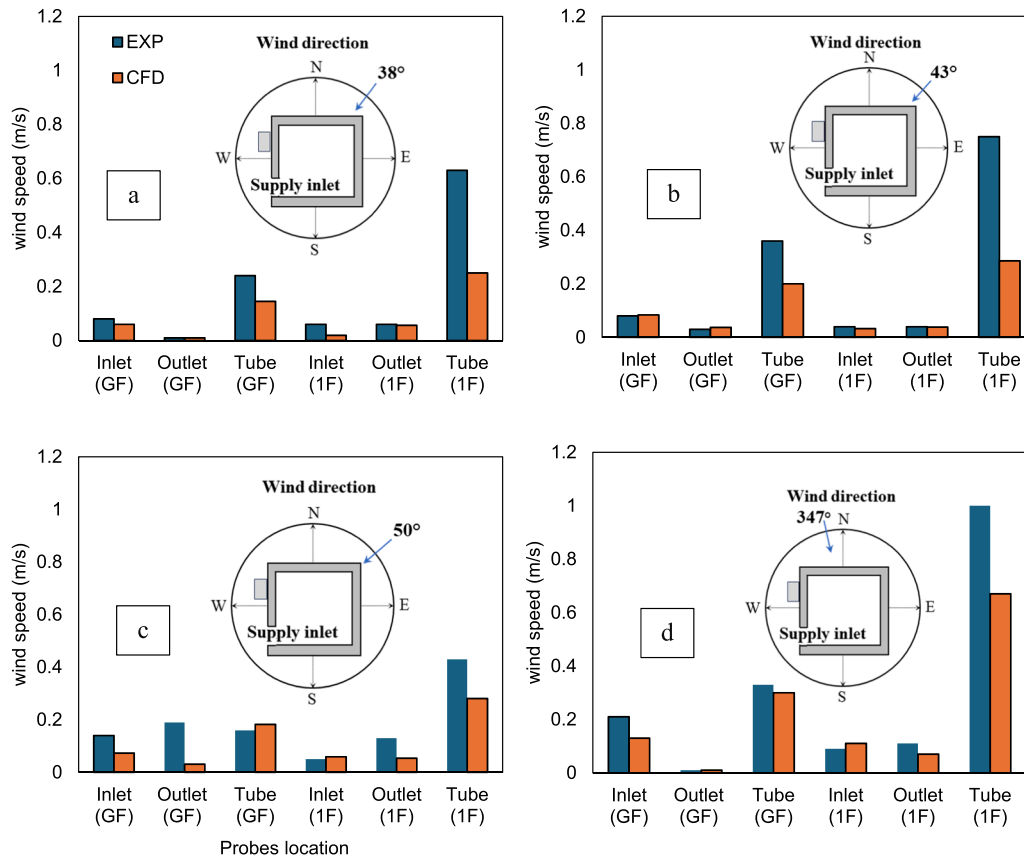


Fig. 9. Validation of CFD model against experimental data for all measurement points: (a) 1.18 m/s at 14:00 with a wind direction of 38°, (b) 1.08 m/s at 14:20 with a wind direction of 43°, (c) 1.05 m/s at 14:30 with a wind direction of 50°, and (d) 1.7 m/s at 15:10 with a wind direction of 347°.

validating the airflow velocity data obtained from the field experiment by comparing it with the CFD model to ensure its credibility and reliability. The validation process reveals a generally consistent alignment between the CFD results and real-world observations across different outdoor wind speeds and directions. Although discrepancies were noted, with the CFD data occasionally showing lower values at specific points compared to the experimental data, the overall trend matches well between the two datasets. These differences may be due to the inherent challenges of replicating complex real-world conditions in a simulated environment, where localized turbulence and environmental variability are difficult to model precisely. Fig. 8(b) presents a comparison of outdoor air velocity measurements, as well as velocities at the supply inlets, exhaust outlets, and within the exhaust tube channel. To improve the accuracy of the airflow simulation and enable a direct comparison, the outdoor wind measurements from the experiment were used as input for the CFD model. These selected data points reflect variations in wind speed and direction.

To quantify these discrepancies, the average error between the experimental and CFD data for each scenario was calculated. Most data points showed good agreement between the experimental and CFD results at the supply inlets and exhaust outlets, with some discrepancies observed in the exhaust tube channel, as illustrated in Fig. 9. For example, at 14:00 in Fig. 9(a), when the outdoor wind speed was 1.18 m/s with a direction of 38°, an average error of 33 % was noted, with some points exhibited deviations of less than 10 %. Similarly, at 14:20, with an outdoor wind speed of 1.08 m/s, the average error was 26 %, as

seen in Fig. 9(b). Although certain points display discrepancies, the overall trend between the experimental and CFD data remains consistent, reinforcing the reliability of the model.

The disparities between the experimental and CFD data could be attributed to several factors. These include the simplifications and assumptions inherent in the CFD model, such as idealized boundary conditions, the representation of complex flow patterns, and the resolution of the mesh, which might not fully capture localized turbulence or subtle variations in airflow. Additionally, inaccuracies in the experimental measurements, such as sensor placement or calibration, could also contribute to these differences. Despite these errors, the systematic comparison indicates that the CFD model is generally reliable and in agreement with the experimental data. This level of accuracy is sufficient to proceed with further simulations, which will explore various parameters of the proposed windcatcher system.

4.3. Multi-story building with single sided ventilation

To assess the performance of the proposed WWC system, a model of single-sided ventilation (SSV) will serve as the benchmark. This section presents an analysis of a CFD model of SSV across different floors. The study examines how natural ventilation varies based on the building's elevation and wind direction. The boundary conditions from the validated model were applied to this simulation to investigate airflow patterns and pressure distribution across all floors. The bar chart in Fig. 10 (c) presents the average indoor airflow velocity of single-sided

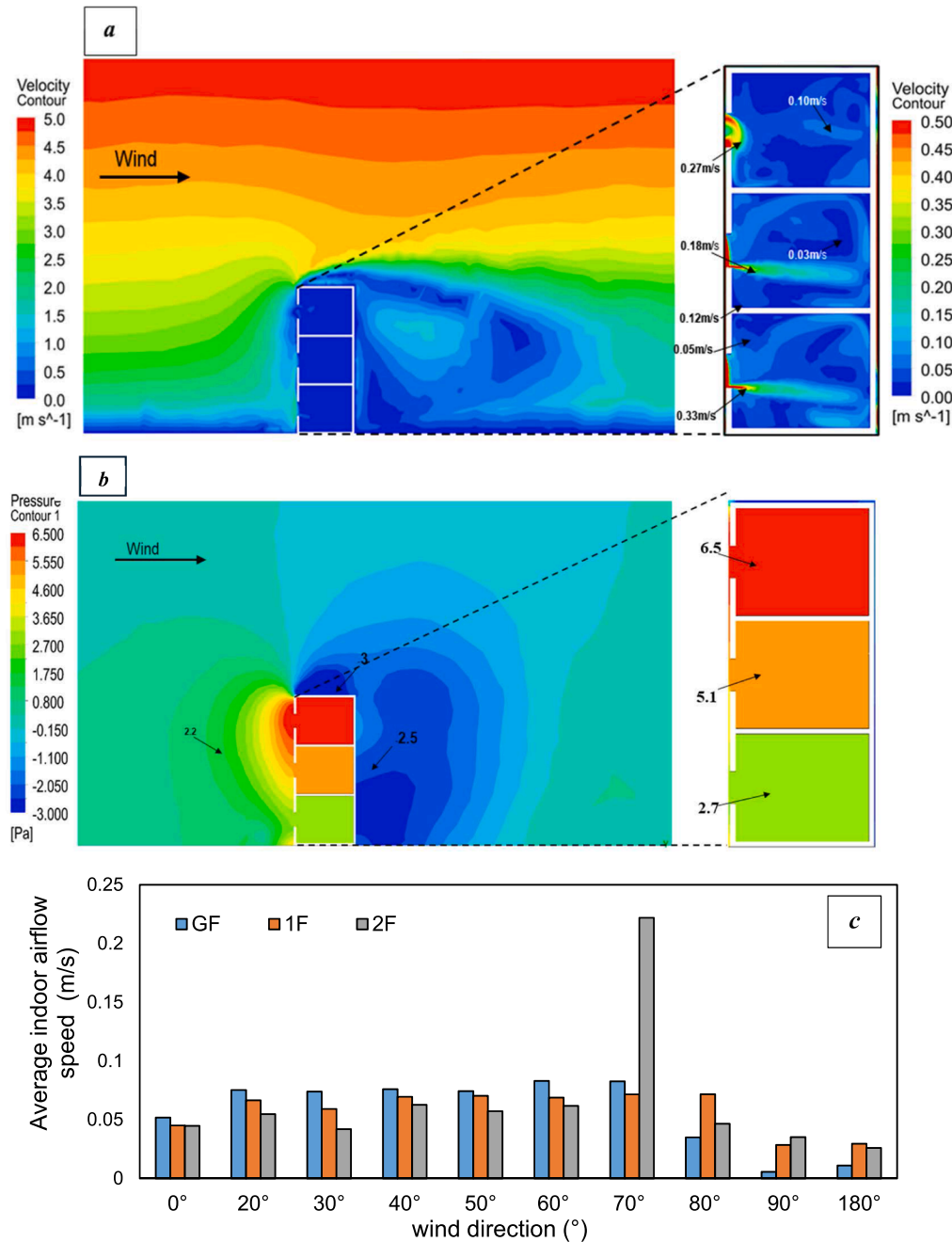


Fig. 10. (a) Cross-sectional plot of the velocity contour in the multi-story building with single-sided ventilation and a zoomed-in view of the velocity distribution in the rooms at 0° wind direction. (b) Cross-sectional plot of the pressure distribution contour in multi-story buildings for single-sided ventilation at 0° wind direction. (c) Average indoor airflow speed of single-sided ventilation for different wind directions.

ventilation across three different floors ground floor (GF), first floor (1F), and second floor (2F) in response to varying wind directions.

Overall, the average airflow velocity on the ground floor was consistently higher than on the first and second floors when wind directions ranged from 0° to 50°, remaining relatively steady within this range. However, starting at 60°, the ground floor airflow velocity began to decrease, continuing its decline through 90° and 180°. Despite having the highest airflow velocity among all floors, it never exceeded 0.1 m/s. The highest airflow velocity on the ground floor was recorded at angles of 60° and 70° (0.08 m/s), while the lowest occurred at angles of 90° (0.005 m/s) and 180° (0.01 m/s).

The airflow velocity on the first floor (1F) remained mostly constant with very slight variations, showing values comparable to those on the ground floor up to 50°. However, it experienced a slight increase around

70° before beginning to decline. The highest airflow velocity on the first floor was recorded at angles of 70° and 80° (0.071 m/s), while the lowest airflow velocity was observed at angles of 90° and 180° (0.028 m/s and 0.029 m/s, respectively). The second floor exhibited a similar trend and fluctuation to those observed on the ground and first floors, but with generally lower values from 0° to 60°. The average airflow velocity on the second floor fluctuated between 0.04 m/s and 0.06 m/s during this range. However, at 70°, the airflow velocity peaked significantly higher than on both the ground and first floors, reaching 0.22 m/s. After this peak, the airflow velocity quickly decreased, following the downward trend observed on the other floors.

Further analysis can be conducted by examining the CFD contours in Fig. 10(a), which depict the velocity distribution in and around the multi-story building with single-sided ventilation at a wind angle of

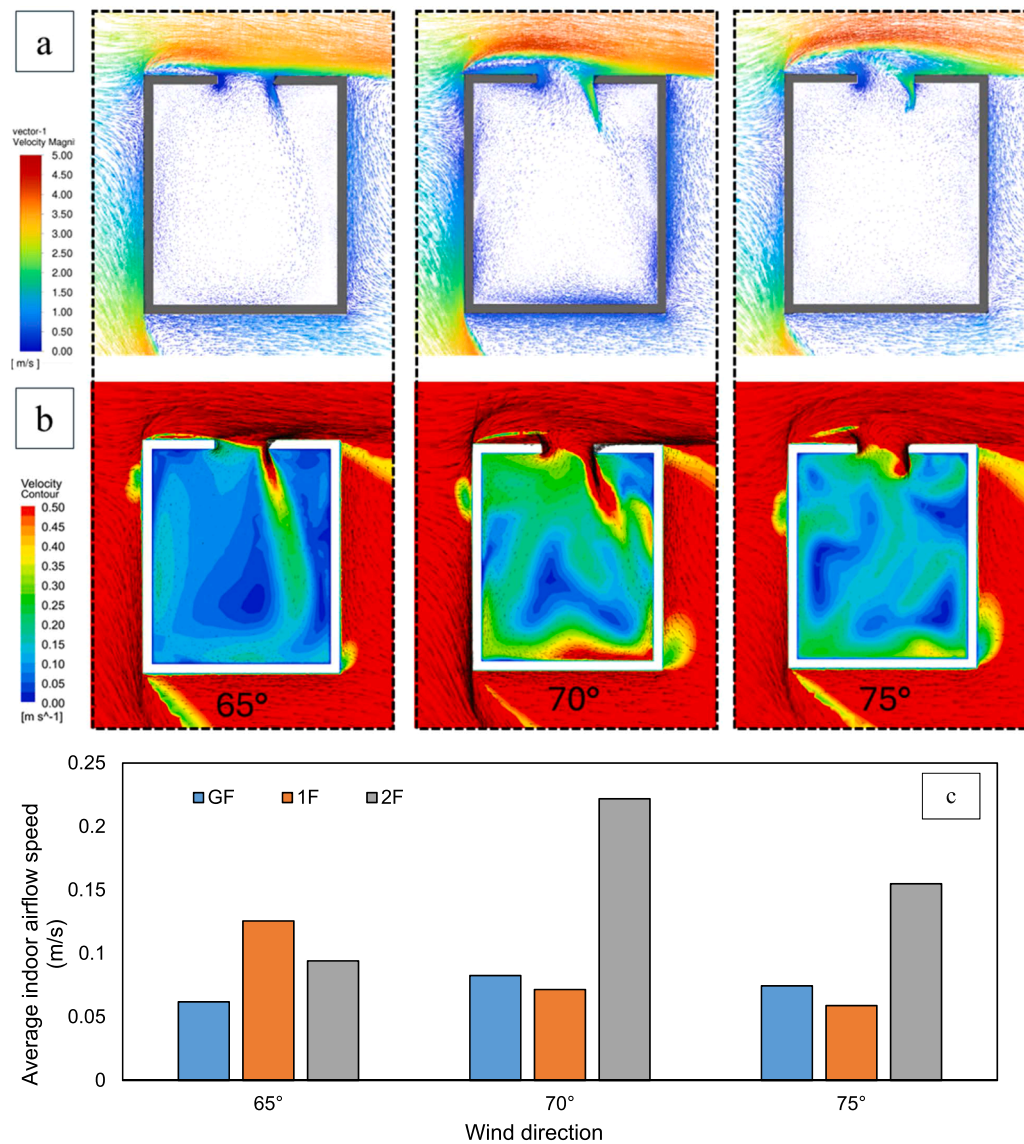


Fig. 11. Horizontal cross-sectional plot of the velocity (a) vectors and (b) contours in the multi-story building with single-sided ventilation at 65°, 70°, and 75° wind direction. (c) Average indoor airflow speed of the wind direction 65°, 70° and 75°.

0° The diagram shows that the ABL wind flow enters the inlet boundary on the left, with the airflow splitting some entering the building's openings, while most passes over or shears to the pressure outlet on the right. The wind flow approaching the windward side of the building shows an upward lift, with some air moving downward, which influences the airflow in and out of the windows. Recirculation zones are observed at the top and rear of the building. In single-sided ventilation, the airflow primarily depends on a combination of wind forces and natural buoyancy to drive air in and out of the openings, but it is less effective than cross-ventilation, which benefits more from wind pressure differences across the facade. A zoomed-in view of the velocity distribution inside the rooms, shown on the right, further illustrates these airflow patterns, highlighting the uneven distribution of air within the spaces.

The cross-sectional plot of the pressure distribution in Fig. 10(b) highlights how the static pressure inside the building varies across the

floors, with higher pressure observed on the upper floors. This pressure gradient aligns with the observed airflow patterns, where the second floor exhibited higher airflow velocities compared to the lower floors. In single-sided ventilation, the system relies primarily on wind pressure differences across the building facade to drive airflow. The variations in static pressure contribute to the differences in airflow observed on each floor, as further illustrated in the zoomed-in view of the velocity distribution inside the rooms, which shows the uneven distribution of air within the spaces.

Given the significant increase in average airflow velocity at a wind direction of 70°, which was notably higher than the values recorded at 60° and 80°, further testing was conducted to investigate this phenomenon. Specifically, wind directions at angles of 65° and 75° were examined to pinpoint the onset of this substantial airflow change. The goal was to determine the precise angles at which airflow velocity begins to increase significantly. The velocity vector and contour plots in Fig. 11

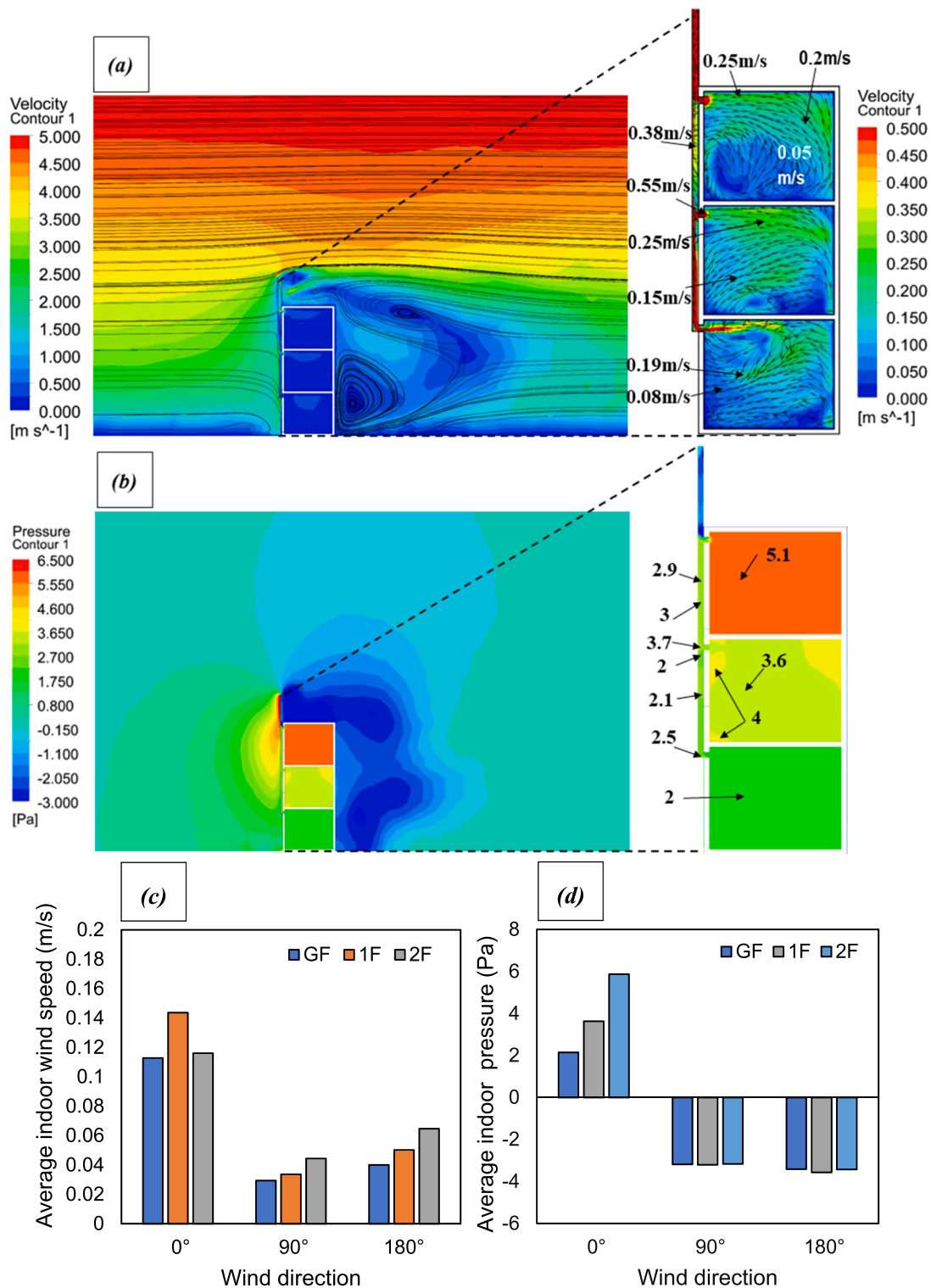


Fig. 12. (a) Cross-sectional plot of the velocity contour in the multi-story building with the proposed system (WWC), including a zoomed-in view of the velocity distribution within the rooms. (b) Cross-sectional plot of the pressure distribution contour in multi-story buildings with the proposed system (WWC). (c) Average indoor airflow speed of the WWC proposed system for various wind directions. (d) Average indoor air pressure with WWC system across various wind directions.

(a) and (b) illustrate the airflow patterns around an inlet at three different impact angles: 65°, 70°, and 75°, showing how these angles significantly influence airflow direction and behavior. At 65°, the effect on airflow is minimal, resulting in a smooth, streamlined flow that remains parallel to the inlet, leading to a lower airflow rate into the interior. As the angle increases to 70°, the airflow pattern shifts, curving and deflecting more sharply towards the inlet, with the airflow colliding

with the inlet tip at around 2 m/s. This change increases the airflow speed into the interior, explaining the higher airflow rate observed on the second floor. At 75°, the airflow continues to curve and deflect, but it shifts slightly farther from the inlet compared to 70°, resulting in a lower airflow rate than at 70°, though still higher than at 65° (Fig. 11(c)).

The analysis of SSV in a multi-story building reveals significant limitations, with overall low airflow values across all floors. The sharp

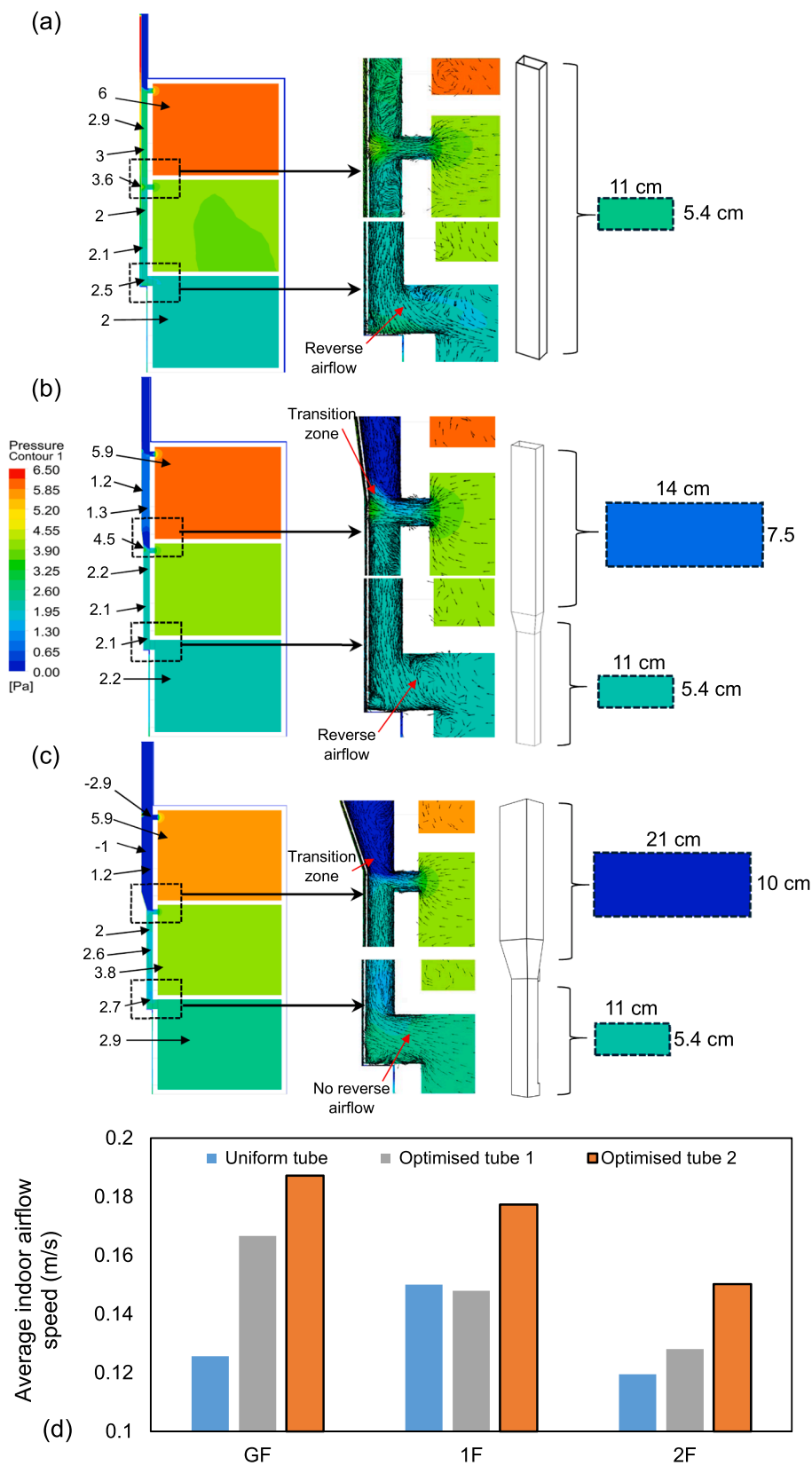


Fig. 13. Comparison of the pressure distribution and velocity vectors in the exhaust tube channel, exhaust outlet first floor, and exhaust outlet ground floor for the (a) WWC system one size exhaust tube (Uniform tube), (b) WWC system optimised tube 1, and (c) WWC system optimised tube 2. (d) Comparison of average indoor airflow speed across all floors for three different tube sizes.

declines observed at 90° and 180° further underscore these limitations, suggesting that relying solely on SSV, especially when wind directions are not optimal, may result in inadequate indoor air movement. These findings highlight the need for more robust ventilation strategies that can maintain consistent airflow across all floors, regardless of wind direction, to ensure effective natural ventilation in multi-story buildings.

4.4. Multi-story building with proposed WWC ventilation system

The same boundary conditions used in the single-sided ventilation (SSV) simulation were applied to the analysis of the proposed WWC system. Fig. 12(a) shows the cross-sectional plot of the velocity contour in the multi-story building with the WWC system, including a zoomed-in view of the velocity distribution within the rooms. Fig. 12(b) displays the pressure distribution contour for the same building. The results indicate a clear improvement in airflow velocity and distribution across all floors compared to the SSV model, demonstrating the effectiveness of the WWC system. According to Fig. 12(c), with a wind direction of 0°, the average airflow was 0.11 m/s on the ground floor, 0.14 m/s on the first floor, and 0.12 m/s on the second floor, representing increases of 118 %, 219 %, and 160 %, respectively, compared to the SSV model. At a wind direction of 90°, the airflow increased by 426 % on the ground floor, 18 % on the first floor and 26 % on the second floor. With a wind direction of 180°, airflow increases were observed at 262 % on the ground floor, 70 % on the first floor, and 149 % on the second floor.

However, the ground floor exhaust outlet acted as an inlet, causing airflow to reverse from the exhaust tube channel back into the room, as shown in Fig. 13(a). This occurred because the pressure in the exhaust tube channel between the first and ground floors (duct zone 1) was lower than the pressure between the first and second floors (duct zone 2), leading to a downward airflow toward the lower pressure area. In contrast, on the first and second floors, the airflow was directed outward. The average airflow in the exhaust channel was 0.8 m/s downward from the first floor to the ground floor, 0.4 m/s upward from the first to the second floor, and 2 m/s upward from the second-floor outlet to the top of the exhaust tube channel.

The proposed WWC system demonstrates improvements over the SSV model by enhancing airflow across all floors, particularly under wind conditions where the SSV struggled. The WWC system's design, including its dedicated exhaust tubes, ensures more efficient air expulsion, which helps to maintain a continuous flow of fresh air and reduces the likelihood of recirculation, addressing a major limitation of the SSV approach. However, the system is not without its challenges, as evidenced by the reverse airflow observed on the ground floor, where the exhaust outlet acted as an inlet. This issue highlights the need for further refinement of the WWC design to ensure consistent and effective ventilation throughout the building.

4.5. Enhancement of the WWC system

This section focuses on enhancing the proposed WWC ventilation system. As noted in Section 4.4, airflow on the first and second floors improved significantly compared to the single-sided model. However, testing the proposed WWC system, as shown in Fig. 13, revealed an issue with ground floor airflow. Specifically, the exhaust outlet functioned as an inlet due to low pressure in the return tube, causing air to flow back down to the ground floor through the exhaust outlet, as shown in (Fig 13 (a)). This reverse flow compromised ventilation effectiveness on the ground floor, as stale air from the first floor could descend into the space, hindering proper air circulation.

To resolve this issue, several modifications were implemented to optimize the pressure gradients within the exhaust tube channel and promote upward airflow. Initially, the exhaust tube channel had the same dimensions as those used in the experiment, extending across three floors with a width of 5.4 cm and a length of 11 cm, resulting in a total cross-sectional area of 59.4 cm². However, the pressure in the exhaust tube channel in duct zone 1, between the ground floor outlet and the first-floor outlet, was lower than in duct zone 2, between the first-floor outlet and the second-floor outlet. This pressure imbalance led to reverse airflow, causing air to flow back into the ground floor.

To address this issue, different duct branch sizes were tested. In the first modification, duct branch zone 1 retained its original dimensions of 5.4 cm width and 11 cm length, resulting in a total cross-sectional area of 59.4 cm². Duct branch zone 2, which extended from the first-floor outlet to the top of the building, was expanded to a width of 7.5 cm and a length of 14 cm. As shown in Fig. 13(d), this modification increased the average indoor airflow velocity notably on the ground floor from 0.126 m/s to 0.167 m/s and slightly improved the second-floor airflow velocity from 0.120 m/s to 0.128 m/s. More importantly, this adjustment did not completely resolve the reverse airflow issue, with air continuing to return to the ground floor through the exhaust outlet, as illustrated in Fig. 13(b).

A second attempt was made using a larger duct branch size, as depicted in (Fig 13(c)). The new design featured a width of 10 cm and a length of 21 cm, increasing the total cross-sectional area to 210 cm². By enlarging duct branch zone 2, a greater pressure differential was created, effectively drawing air upward from the ground floor to the top of the model. This modification successfully resolved the reverse airflow issue, ensuring efficient ventilation on the ground floor without negatively impacting airflow on the first and second floors.

Tube optimization results show that optimized tube 2 enhances airflow across all floors (Fig 13(d)). Although the uniform tube and optimized tube 1 failed to provide effective ventilation on the ground floor due to reversed airflow from the first floor (acting as stale air), optimized tube 2 outperformed both. It increased airflow by 49 % on the ground floor, 18 % on the first floor, and 25.7 % on the second floor compared to the uniform tube. Moreover, optimized tube 2 outperformed the single-sided ventilation model by 261 % on the ground floor, 293 % on the first floor, and 236 % on the second floor.

The results (Fig. 15) demonstrate that the enhanced WWC system generally outperforms the SSV system in enhancing airflow, with a few exceptions. Specifically, at a wind angle of 60° on the ground floor, 70° on the second floor, and 80° on the first floor, the SSV system exhibited higher airflow by 70 %, 28 %, and 18 %, respectively. Overall, the WWC system shows higher performance across most wind directions, with the most significant improvement occurring at 90° reaching 734 %, indicating that even in less favourable wind conditions, the WWC system still holds an advantage, and the smallest improvement is observed at a 50° angle with 6 % increase.

These findings align with the CFD contours in Fig. 14(a) and (b), which depict the velocity and pressure distribution in and around the multi-story building with an enhanced WWC system at a wind angle of 0°. A zoomed-in view of the velocity distribution inside the rooms, shown on the right, further illustrates these airflow patterns, highlighting a more even distribution of airflow within the spaces as compared to SSV.

Evaluation of the impact of different wind directions is presented in Fig. 15. On the ground floor, the highest indoor airflow speed for the enhanced WWC system is recorded at a wind angle of 0°, reaching 0.19 m/s, a 261 % increase over the SSV airflow at the same angle (0.05 m/s). Even at the lowest recorded speed of 0.041 m/s at 180° (leeward), the

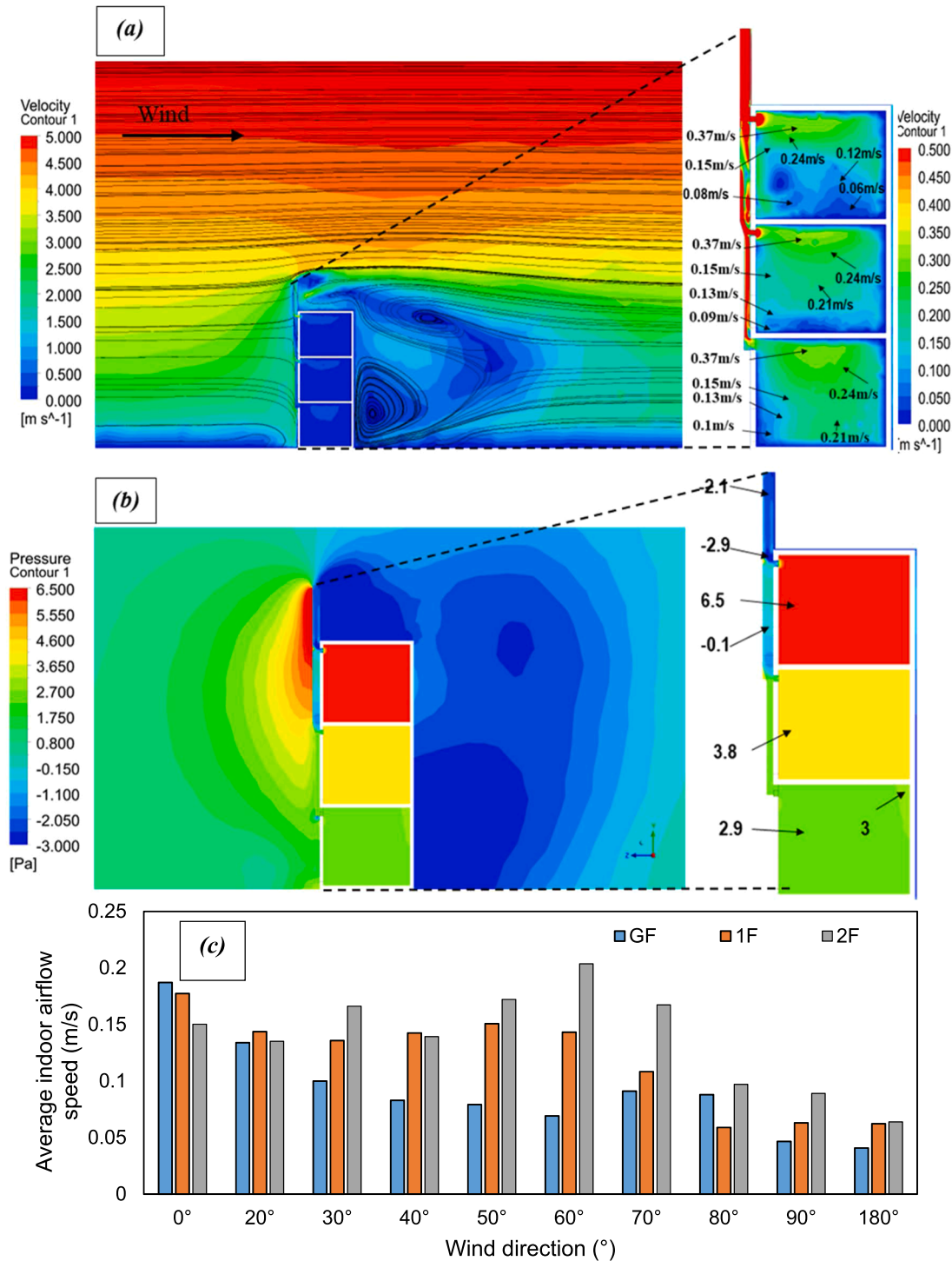


Fig. 14. (a) Cross-sectional plot of the velocity contours in the multi-story building with the optimized proposed ventilation system, featuring a zoomed-in view of the velocity distribution in the rooms. (b) Cross-sectional plot of the pressure distribution contours in the multi-story building with the optimized proposed ventilation system. (c) Average indoor airflow of the enhanced proposed ventilation system for various wind directions.

WWC still performs significantly better, providing a 268 % increase compared to the SSV system's 0.01 m/s. However, at a 60° wind angle, the SSV system marginally outperforms the WWC system, with a value of 0.08 m/s, showing an 18 % reduction in WWC airflow.

On the first floor, the highest airflow for the enhanced WWC system occurs at 0°, with a value of 0.17 m/s, marking a substantial 293 % increase over the SSV at the same angle (0.045 m/s). Even at the lowest speed recorded at 180° (leeward), the enhanced WWC still demonstrates a 110 % improvement over the SSV, registering 0.06 m/s. Across most

wind directions, the WWC system consistently outperforms the SSV on the first floor, except at 80°, where the SSV airflow is 18 % higher than the WWC.

On the second floor, the enhanced WWC system achieves its peak airflow at a wind angle of 60°, reaching 0.20 m/s a 229 % increase compared to the SSV's 0.062 m/s at the same angle. However, at 70°, the SSV system surpasses the WWC, achieving 0.22 m/s, while the WWC records 0.17 m/s, marking a 25 % decrease for the WWC at this angle. Despite this anomaly, the WWC system consistently outperforms the SSV

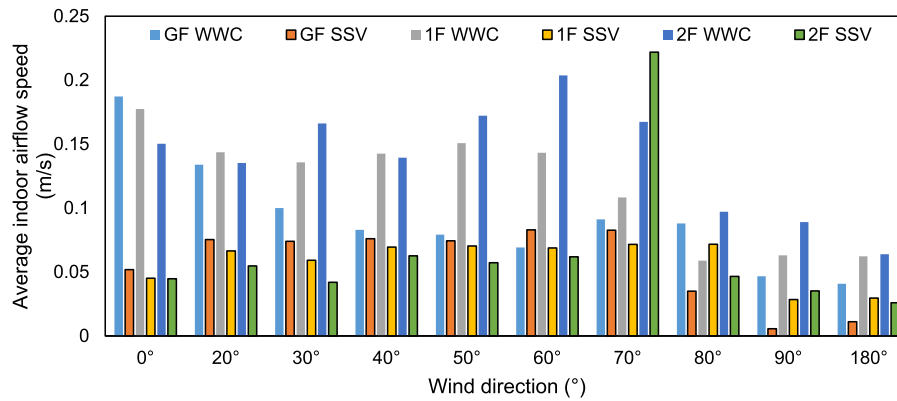


Fig. 15. Comparison of indoor average airflow velocity between single-sided ventilation and the enhanced WWC system under wind directions of 0°, 90°, and 180°.

across most wind directions on the second floor, proving to be a more effective ventilation method overall, except at 70°, where the SSV shows superior performance. Horizontal and vertical cross-sectional velocity contour plots (Fig. A.6–A.8) further compare the airflow results of both systems across various wind directions, including 0° to 90° and 180°.

The enhanced WWC system demonstrates a consistent improvement in average indoor airflow speed compared to the SSV system across different wind angles and floors, with notable exceptions at 60° on the ground floor, 70° on the second floor, and 80° on the first floor. The highest performance of the WWC is seen at 0°, with improvements ranging from 261 % to 293 % across all floors. Even in its lowest performance scenarios, the WWC system still outperforms the SSV system, proving to be a more effective ventilation method overall.

To further highlight the ventilation advantages of the WWC system, a comparison of average airflow velocities at breathing height between the optimized WWC and SSV systems was conducted (Fig. A.9). According to [56], airflow velocities below 0.05 m/s are considered "stagnant" and not appropriate for sedentary occupation. The optimized WWC consistently maintained airflow velocities above this threshold, whereas the SSV frequently fell below or hovered near it. Although this does not provide a comprehensive air quality assessment, these findings clearly demonstrate the benefits of the WWC system and underline the importance of future research that includes detailed analyses of CO₂ concentrations and other pollutants.

These findings showed that the enhanced WWC system outperformed the SSV system in both airflow speed and distribution. Small adjustments to the duct size, transition ducts, and outlet size have proven effective in improving performance. This same approach can be applied when adapting the system for taller buildings. However, equal airflow speed distribution still cannot be maintained across all floors at certain wind angles, indicating that further optimization is needed to achieve uniform ventilation under all conditions. Even with this limitation, the WWC system demonstrates improved performance and remains a highly effective alternative to single-sided ventilation.

It should be noted that the initial choice of positioning the inlet openings at the corners was strategically adopted in this study to maximize the spatial separation between supply inlets and exhaust outlets, thereby reducing the potential for air short-circuiting and ensuring more effective mixing of airflow within each space. However, an additional comparative analysis was conducted to evaluate the influence of inlet positioning, corner versus center, on the airflow characteristics and distribution within the rooms, as illustrated in the velocity contours, vectors, and bar charts presented in Fig. A.10.

The findings from this supplementary analysis demonstrated differences in airflow patterns between the two inlet configurations. Specifically, the corner inlet position generated stronger, more localized airflow circulation, promoting enhanced air mixing near the inlet areas. In contrast, the center inlet arrangement provided more uniform airflow distribution throughout the interior space but exhibited reduced velocities near room edges, potentially creating stagnation zones. These highlight the significance of inlet positioning in influencing airflow behavior and distribution uniformity.

Given these findings, future research should involve more detailed optimization and parametric analyses to systematically explore the effects of various inlet placements, configurations, and geometrical parameters on ventilation performance. Such investigations would enable more informed decisions regarding inlet design and positioning to achieve optimal natural ventilation efficiency tailored to specific building designs and occupancy requirements.

5. Conclusion and future works

This study introduced and evaluated a novel wall windcatcher (WWC) system for enhancing natural ventilation in multi-story buildings. Through a combination of field experiments, CFD modeling, and comparisons to a single-sided ventilation (SSV) benchmark, the effectiveness of the WWC system was assessed. The findings demonstrate that the WWC system offers improvements over traditional SSV in terms of airflow enhancement and distribution across multiple floors.

Across various wind directions, the WWC system consistently achieved higher airflow speeds than the SSV model. Even at wind angles where single-sided ventilation performed poorly, such as 90° and 180°, the WWC system maintained higher airflow rates, demonstrating its ability to facilitate air movement under diverse conditions. The WWC also provided a more uniform distribution of fresh air, reducing localized stagnation zones and potentially improving indoor air quality and thermal comfort.

The study validated the CFD simulations against experimental field tests, which confirmed the model's reliability for predicting ventilation performance. While minor discrepancies were observed, particularly in the exhaust tube airflow speeds, the overall trend showed a strong correlation between simulated and real-world results. This validation strengthens confidence in using the CFD model for further parametric studies and design optimizations.

One of the key challenges identified was reverse airflow on the ground floor, where the exhaust outlet sometimes functioned as an inlet

due to undesired pressure differentials in the exhaust duct. To address this issue, modifications were made to the duct size, transition zones, and outlet dimensions. These refinements effectively resolved the reverse airflow problem, ensuring proper air extraction from the ground floor without negatively impacting the performance of the first and second floors. However, despite these improvements, airflow variations still occurred at specific wind angles, such as 60° on the ground floor, 70° on the second floor, and 80° on the first floor, indicating that further optimization is needed to ensure consistent airflow across all levels.

The practicality of the WWC system makes it a potential solution for both new constructions and retrofitting of existing multi-story buildings. Its external mounting and separate supply and exhaust channels eliminate the need for extensive internal ductwork, reducing installation complexity and potential disruption to occupants. The study also demonstrated that small yet targeted adjustments to duct dimensions and airflow pathways can significantly enhance performance, a principle that can be extended to taller buildings and more complex structures.

Future research should focus on refining the optimal design parameters to achieve more uniform airflow distribution under all wind conditions. Additionally, integrating heat recovery, pre-cooling, or other passive ventilation techniques could further enhance the system's energy efficiency and year-round performance. Incorporating detailed simulations that couple airflow analysis with thermal and humidity variations will provide a holistic assessment of the WWC system's effectiveness across diverse climatic conditions.

Furthermore, preliminary analysis indicates that internal heat gains influence airflow patterns due to buoyancy effects (Fig. A.11), highlighting the importance of considering internal thermal conditions in future evaluations. Employing more advanced turbulence modeling techniques, such as Large Eddy Simulation (LES) or the Reynolds Stress Model (RSM), could provide more accurate predictions of complex airflow patterns and turbulence characteristics. A broader range of

experimental tests, including wind tunnel experiments and full-scale real-world applications, would provide additional insights into the long-term viability and adaptability of the WWC system.

The WWC system presents a substantial improvement over conventional single-sided ventilation in multi-story buildings. While some design challenges remain, particularly in maintaining equal airflow across all floors, the WWC system demonstrates strong potential as an innovative and effective natural ventilation strategy. With further refinements, it could play a pivotal role in reducing reliance on mechanical systems, promoting energy efficiency, and enhancing indoor environmental quality in future building designs.

CRediT authorship contribution statement

Salah Almazmumi: Writing – review & editing, Writing – original draft, Visualization, Validation, Software, Resources, Project administration, Methodology, Investigation, Funding acquisition, Formal analysis, Data curation, Conceptualization. **Hao Sun:** Validation. **Miaomiao Liu:** Investigation. **John Calautit:** Writing – review & editing, Supervision. **Carlos Jimenez-bescos:** Supervision.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgement

This research was supported by the Saudi Arabian Cultural Bureau (SACB) in London. The author would like to thank Dr. Christopher Wood for providing access to the test pod.

Appendix

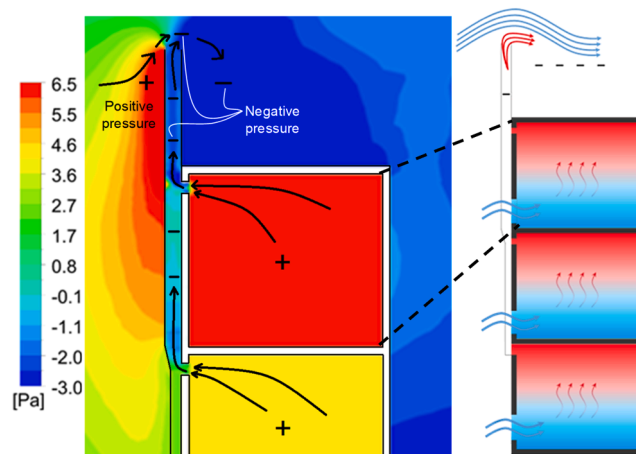


Fig. A.1. Working principle of the wall windcatcher (WWC) system showing pressure distribution and airflow patterns.

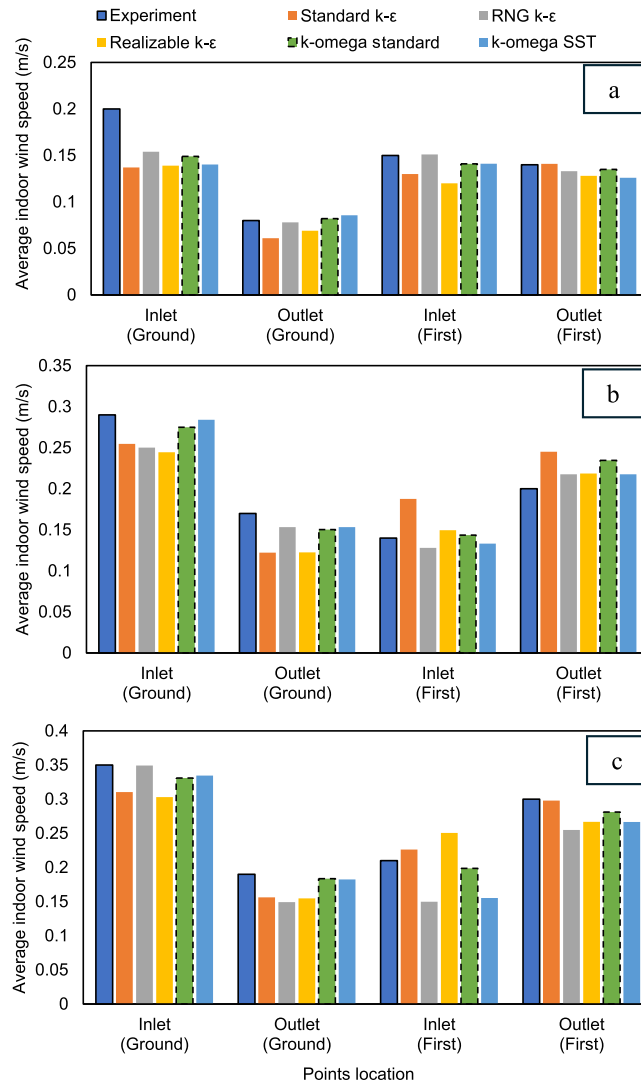


Fig. A.2. Comparison of five turbulence models, Standard k-ε, RNG k-ε, Realizable k-ε, Standard k-ω, and k-ω SST, under three different wind speeds (a) 1.9 m/s, (b) 3.31 m/s, and (c) 4.04.

Based on these supplementary tests, we concluded that while longer domains are typically advisable, the 5H configuration used in this particular scenario did not introduce significant inaccuracies for the building geometry and flow conditions studied. Nevertheless, we acknowledge that the recommended 10H–15H domain might be more appropriate for more complex or sensitive configurations.

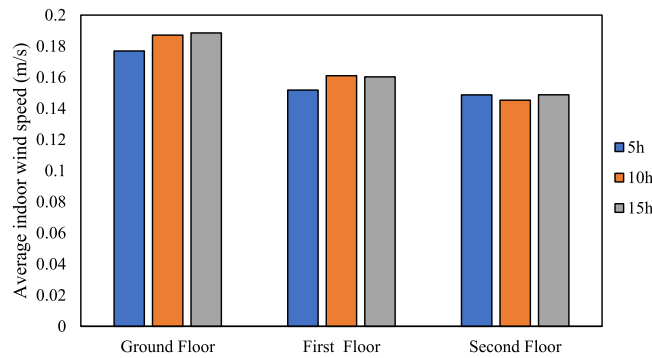


Fig. A.3. Comparison of average indoor airflow across three domain sizes: 5 h, 10 h, and 15 h.

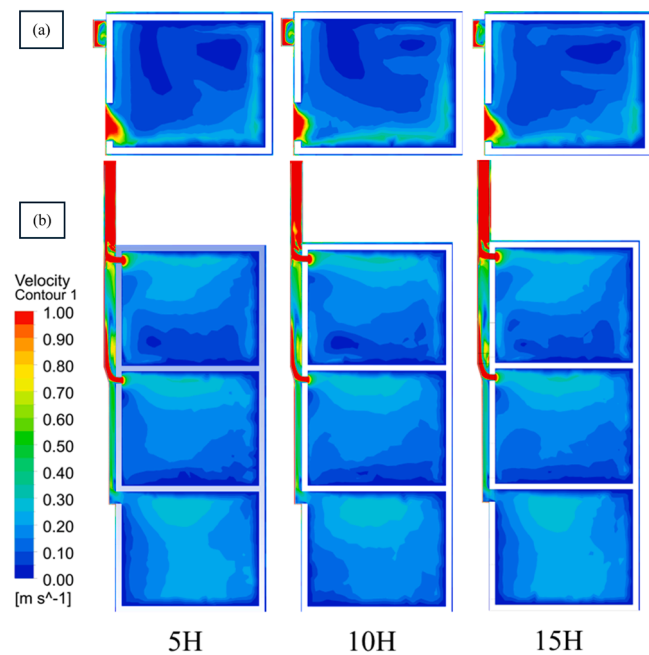


Fig. A.4. Velocity contour plots for three different domain sizes at downstream locations 5H, 10H, and 15H: (a) Horizontal cross-sectional views of the velocity contours. (b) Vertical cross-sectional views of the velocity contours.

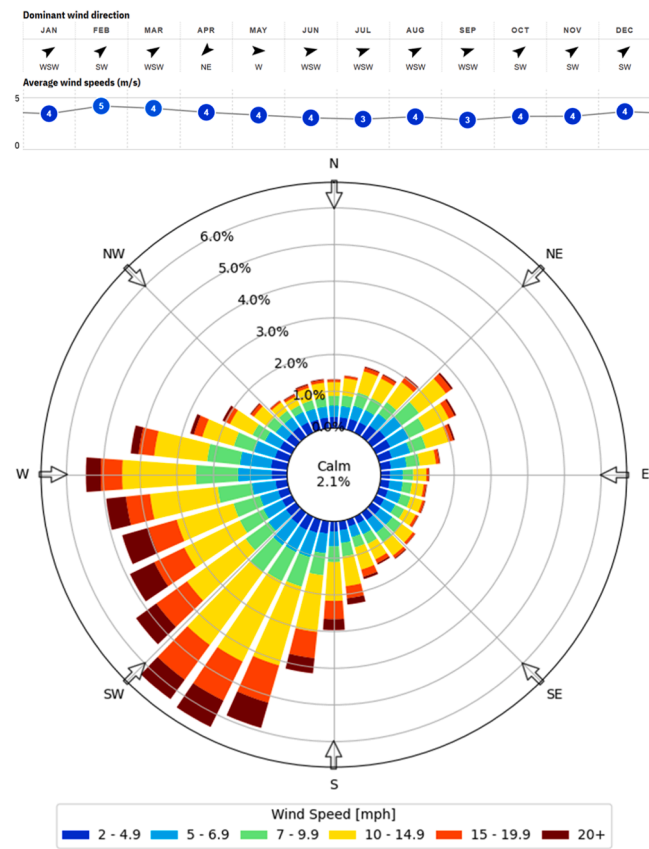


Fig. A.5. Nottingham, UK (a) monthly wind direction and speed (m/s), and (b) wind rose [54,55].

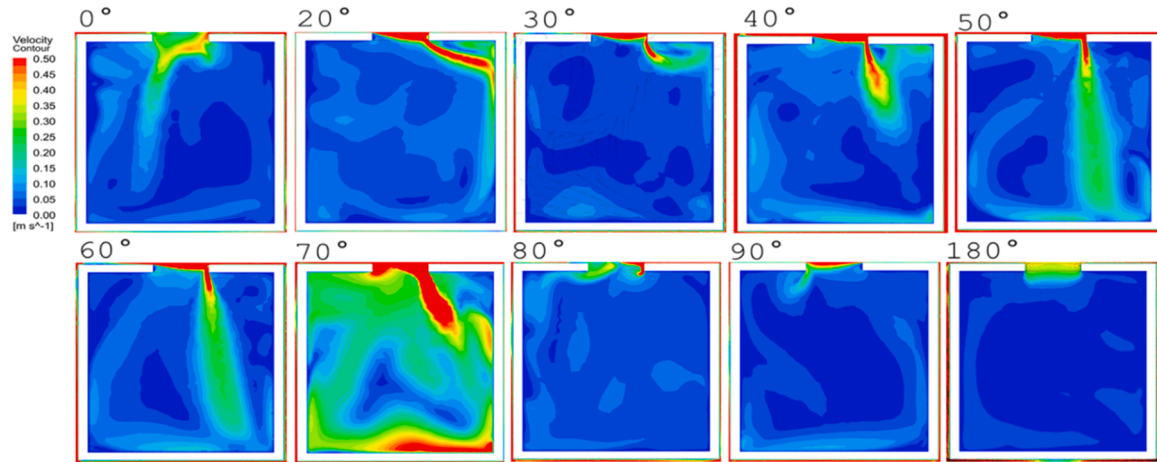


Fig. A.6. Horizontal cross-sectional plots of the velocity contours for single-sided ventilation on the second floor, illustrating results for various wind directions: 0° to 90° and 180°.

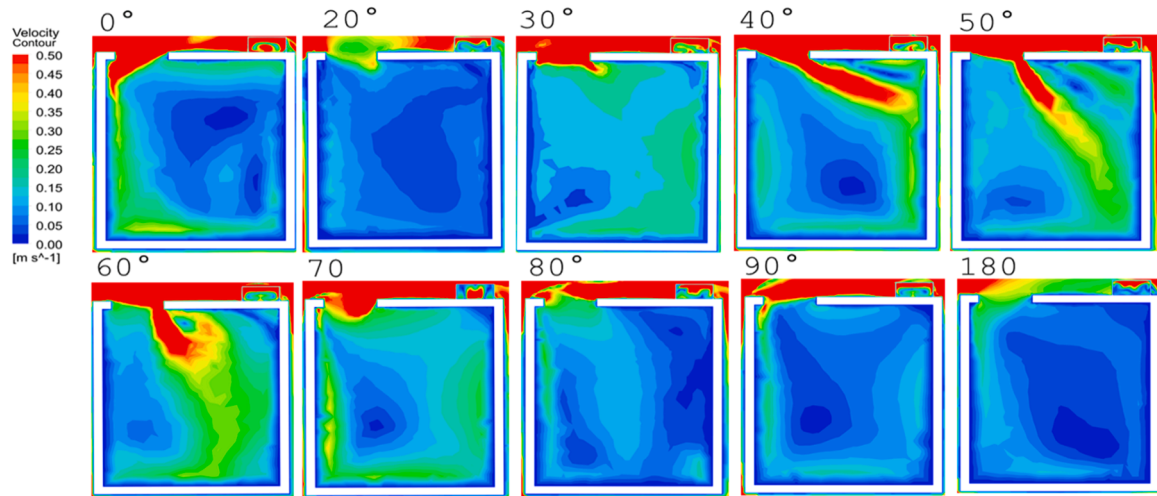


Fig. A.7. Horizontal cross-sectional plots of the velocity contours for enhanced proposed system ventilation (WWC) on the second floor, illustrating results for various wind directions: 0° to 90° and 180°.

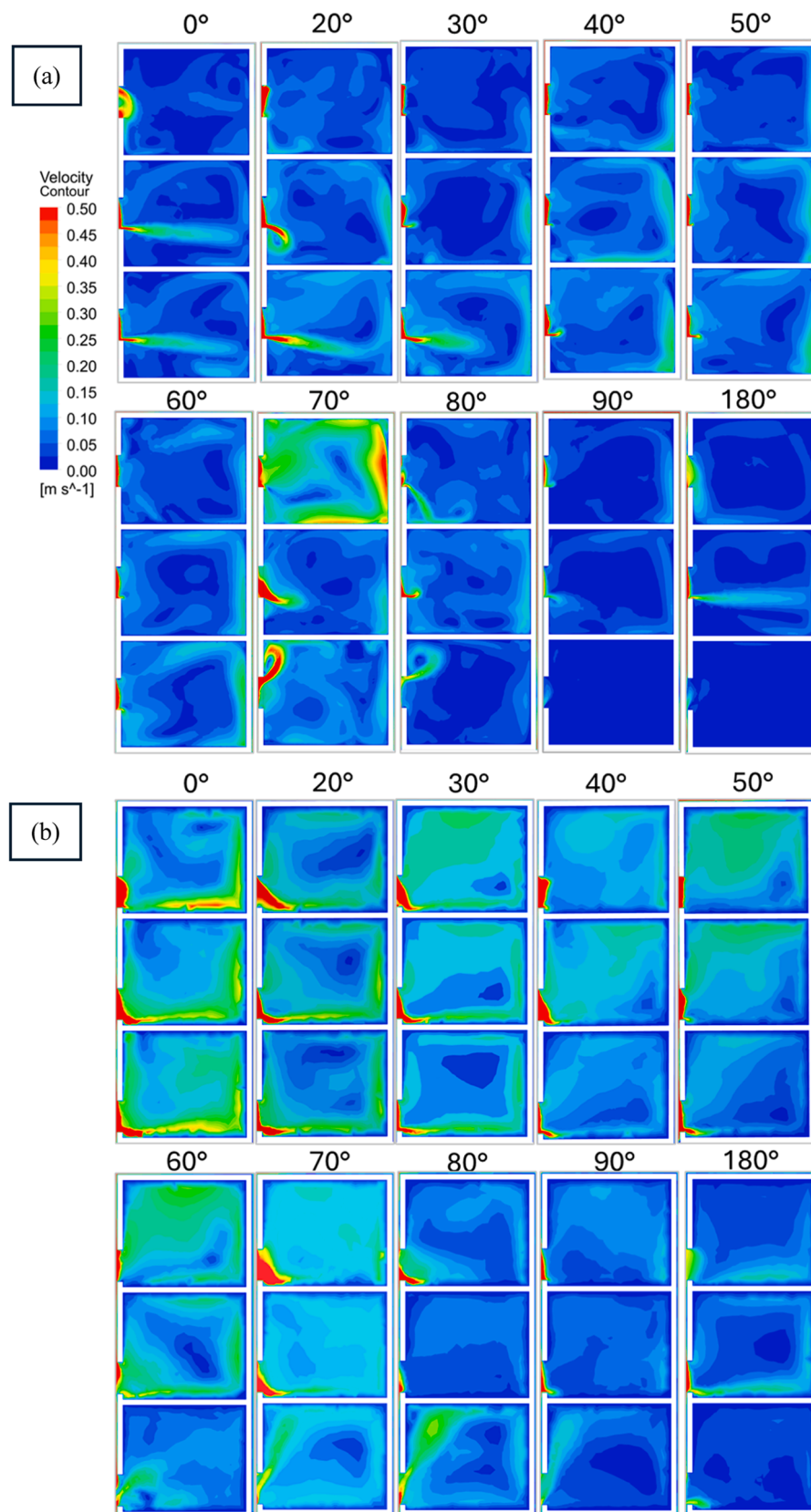


Fig. A.8. Vertical cross-sectional plots of the velocity contours for (a) single sided ventilation (SSV) and (b) enhanced proposed system ventilation (WWC).

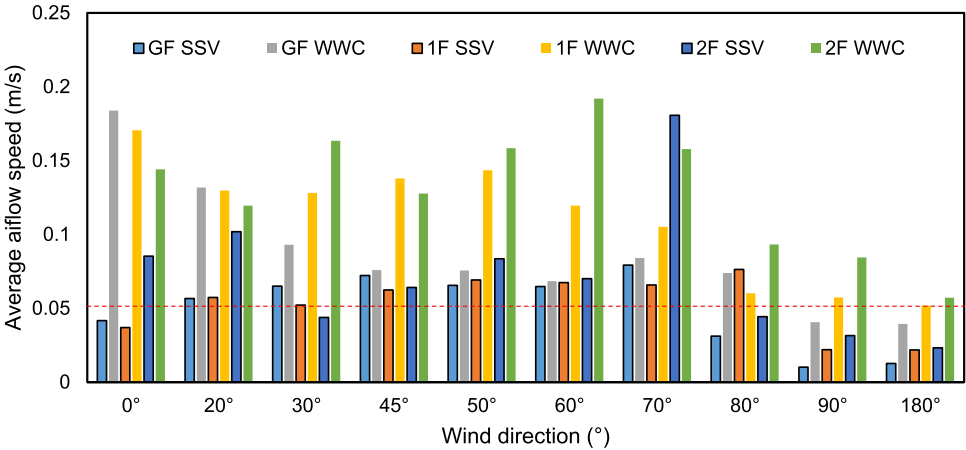


Fig. A.9. Average airflow speed (m/s) at breathing height level comparing SSV and enhanced WWC. Red dashed line indicates “stagnant” [37].

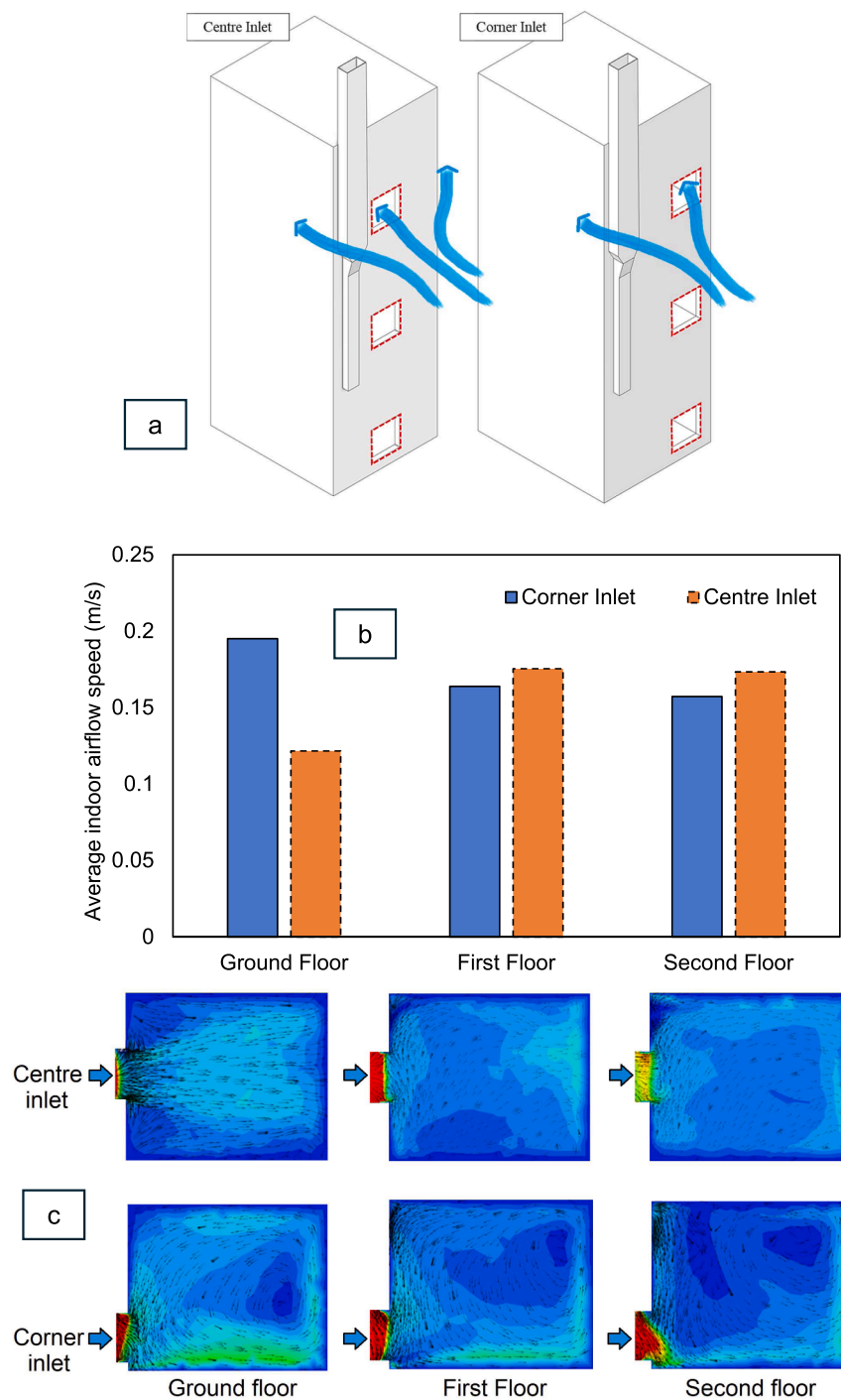


Fig. A.10. (a) 3D models of two supply inlet location: corner inlet and centre inlet (b) Average indoor airflow speed for central and corner inlets across three floors. (c) Horizontal cross-sectional plots of the velocity vectors for the central and corner inlets.

To preliminarily examine the effects of thermal conditions, we performed an additional CFD simulation comparing the airflow characteristics of the WWC system under two scenarios: without internal heat sources (baseline) and with internal heat sources present in the rooms. In this simulation, three cylindrical tubes, each emitting a heat output of 80 W, were introduced into each room to represent typical internal heat gains. The outdoor temperature at the inlet was fixed at 21.5 °C. Fig. A.10 presents the velocity contour comparisons between these two scenarios. The contours indicate differences in airflow patterns and velocities within the rooms due to the presence of internal heat sources, in particular the ground floor. These highlight the impact of internal heat gains on airflow distribution, emphasizing that thermal buoyancy can influence the WWC system's performance beyond purely wind-driven mechanisms. This initial assessment underscores the importance of including internal and external thermal conditions, in evaluating natural ventilation systems comprehensively.

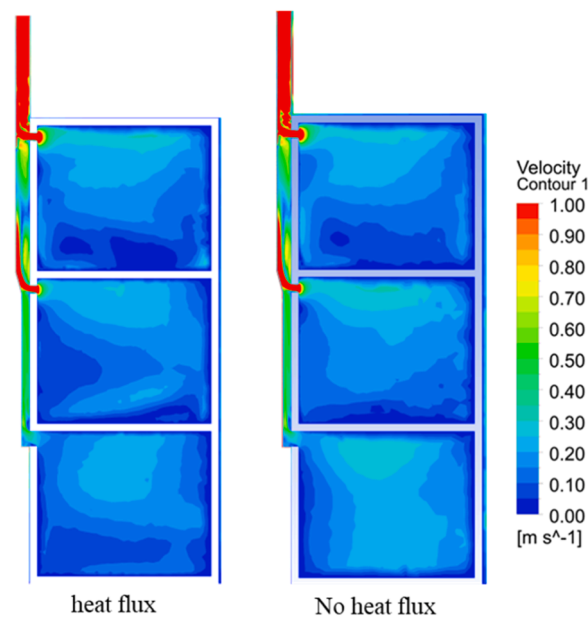


Fig. A.11. Influence of heat flux on the airflow distribution.

Data availability

Data will be made available on request.

References

- [1] World Economic Forum. "Could taller buildings bring about a better quality of life?" <https://www.weforum.org/agenda/2021/06/cities-development-urban-infrastructure/> (accessed).
- [2] F. Muhsin, W.F.M. Yusoff, M.F. Mohamed, A.R. Sopian, CFD modeling of natural ventilation in a void connected to the living units of multi-storey housing for thermal comfort, *Energy Build.* 144 (2017) 1–16.
- [3] J. Yang, B. Shi, Y. Shi, S. Marvin, Y. Zheng, G. Xia, Air pollution dispersal in high density urban areas: research on the triadic relation of wind, air pollution, and urban form, *Sustain. Cities. Soc.* 54 (2020) 101941.
- [4] G.A. Ganesh, S.L. Sinha, T.N. Verma, S.K. Dewangan, Investigation of indoor environment quality and factors affecting human comfort: a critical review, *Build. Environ.* 204 (2021) 108146.
- [5] IEA. "CO2 Emissions in 2023." <https://www.iea.org/reports/co2-emissions-in-2023>.
- [6] N. Khan, Y. Su, S.B. Riffat, A review on wind driven ventilation techniques, *Energy Build.* 40 (8) (2008) 1586–1604.
- [7] X. Shi, Y. Li, T. Long, N. Zhao, S. Liu, T. Li, Buoyancy-driven natural ventilation performance of a multi-story building with vertical shaft designed by using dimensionless design approach, *Sustain. Cities. Soc.* 99 (2023) 104997.
- [8] H.L. Gough, J.F. Barlow, Z. Luo, M.F. King, C.H. Halios, C.S.B. Grimmond, Evaluating single-sided natural ventilation models against full-scale idealised measurements: impact of wind direction and turbulence, *Build. Environ.* 170 (2020) 106556.
- [9] T. van Hooff, B. Blocken, Y. Tominaga, On the accuracy of CFD simulations of cross-ventilation flows for a generic isolated building: comparison of RANS, LES and experiments, *Build. Environ.* 114 (2017) 148–165.
- [10] B.R. Hughes, J.K. Calautit, S.A. Ghani, The development of commercial wind towers for natural ventilation: a review, *Appl. Energy* 92 (2012) 606–627.
- [11] N.N. Ziarani, M.J. Cook, P.D. O'Sullivan, Experimental evaluation of airflow guiding components for wind-driven single-sided natural ventilation: a comparative study in a test chamber, *Energy Build.* 300 (2023) 113627.
- [12] H.Y. Zhong, J. Sun, C. Lin, et al., Single-sided natural ventilation in buildings: a critical literature review, *Build. Environ.* 212 (2022) 108797.
- [13] C.R. Chu, Assessment of year-round wind-driven ventilation by an integrated ventilation model, *Build. Environ.* 243 (2023) 110710.
- [14] W. Pan, S. Liu, Y. Wang, X. Cheng, H. Zhang, Z. Long, Measurement of cross-ventilation rate in urban multi-zone dwellings, *Build. Environ.* 158 (2019) 51–59.
- [15] H.Y. Zhong, J. Sun, C. Lin, et al., CFD simulation of pumping ventilation in a three-story isolated building with internal partitioning: effects of partition widths, heights and locations, *Build. Simul.* 17 (2024) 267–284, <https://doi.org/10.1007/s12273-023-1068-3>.
- [16] C.R. Chu, Y.H. Chiu, Y.W. Wang, An experimental study of wind-driven cross ventilation in partitioned buildings, *Energy Build.* 42 (5) (2010) 667–673.
- [17] X. Zhang, J.W.G. Buddhika, J. Wang, A.U. Weerasuriya, K.T. Tse, Numerical investigation of effects of trees on cross-ventilation of an isolated building, *J. Build. Eng.* 73 (2023) 106808.
- [18] P. Nejat, J.K. Calautit, M.Z.A. Majid, B.R. Hughes, F. Jomehzadeh, Anti-short-circuit device: a new solution for short-circuiting in windcatcher and improvement of natural ventilation performance, *Build. Environ.* 105 (2016) 24–39.
- [19] F. Jomehzadeh, P. Nejat, J.K. Calautit, M.B.M. Yusof, S.A. Zaki, B.R. Hughes, M.N. A.W.M. Yazid, A review on windcatcher for passive cooling and natural ventilation in buildings, part 1: indoor air quality and thermal comfort assessment, *Renew. Sustain. Energy Rev.* 70 (2017) 736–756.
- [20] P. Nejat, M.S. Ferwati, J. Calautit, A. Ghahramani, M. Sheikhshahrokhdehordi, Passive cooling and natural ventilation by the windcatcher (Badgir): an experimental and simulation study of indoor air quality, thermal comfort and passive cooling power, *J. Building Eng.* 41 (2021) 102436.
- [21] X. Gao, A. Li, C. Yang, Study on natural ventilation driven by a restricted turbulent buoyant plume in an enclosure, *Energy Build.* 177 (2018) 173–183.
- [22] J. Shaeri, M. Mahdavejad, M.H. Pourghasemian, A new design to create natural ventilation in buildings: wind chimney, *J. Build. Eng.* 59 (2022) 105041.
- [23] Y. Ji, M.J. Cook, V. Hanby, CFD modelling of natural displacement ventilation in an enclosure connected to an atrium, *Build. Environ.* 42 (3) (2007) 1158–1172.
- [24] G. Gan, S.B. Riffat, A numerical study of solar chimney for natural ventilation of buildings with heat recovery, *Appl. Therm. Eng.* 18 (12) (1998) 1171–1187.
- [25] K. Pelletier, C. Wood, J. Calautit, Y. Wu, The viability of double-skin façade systems in the 21st century: a systematic review and meta-analysis of the nexus of factors affecting ventilation and thermal performance, and building integration, *Build. Environ.* 228 (2023) 109870.
- [26] Z.T. Ai, C.M. Mak, Comparison of single-sided ventilation characteristics between single-storey and multi-storey buildings due to wind effect, *Int. J. Ventilation* 14 (2) (2015) 181–190.
- [27] J.O.P. Cheung, C.H. Liu, CFD simulations of natural ventilation behaviour in high-rise buildings in regular and staggered arrangements at various spacings, *Energy Build.* 43 (5) (2011) 1149–1158.
- [28] L. Pérez-Lombard, J. Ortiz, I.R. Maestre, J.F. Coronel, Constructing HVAC energy efficiency indicators, *Energy Build.* 47 (2012) 619–629.
- [29] Z.T. Ai, C.M. Mak, Determination of single-sided ventilation rates in multistory buildings: evaluation of methods, *Energy Build.* 69 (2014) 292–300.
- [30] N.C. Daish, G. Carrilho da Graça, P.F. Linden, D. Banks, Impact of aperture separation on wind-driven single-sided natural ventilation, *Build. Environ.* 108 (2016) 122–134.
- [31] A. Afkari, N. Mahyuddin, M.R. Baharum, The influence of single-sided ventilation towards the indoor thermal performance of high-rise residential building: a field study, *Energy Build.* 126 (2016) 146–158.
- [32] C. Zhou, Z. Wang, Q. Chen, Y. Jiang, J. Pei, Design optimization and field demonstration of natural ventilation for high-rise residential buildings, *Energy Build.* 82 (2014) 457–465.
- [33] P. Prajongsan, S. Sharples, Enhancing natural ventilation, thermal comfort and energy savings in high-rise residential buildings in Bangkok through the use of ventilation shafts, *Build. Environ.* 50 (2012) 104–113.

- [34] T. Ahmed, P. Kumar, L. Mottet, Natural ventilation in warm climates: the challenges of thermal comfort, heatwave resilience and indoor air quality, *Renew. Sustain. Energy Rev.* 138 (2021) 110669.
- [35] M. Liu, P. Nejat, P. Cao, C. Jimenez-Bescos, J.K. Calautit, A critical review of windcatcher ventilation: micro-environment, techno-economics, and commercialisation, *Renew. Sustain. Energy Rev.* 191 (2024) 114048.
- [36] P.K. Sangdeh, N. Nasrollahi, Windcatchers and their applications in contemporary architecture, *Energy Built Environ.* 3 (1) (2022) 56–72.
- [37] M. Alsallani, H. Montazeri, A. Rezaeiha, Towards optimal aerodynamic design of wind catchers: impact of geometrical characteristics, *Renew. Energy* 168 (2021) 1344–1363.
- [38] M. Sadeghi, B. Samali, G. Wood, R. de Dear, Comfort cooling by wind towers in the Australian residential context—Experimental wind tunnel study of comfort, *J. Wind Eng. Industr. Aerodyn.* 196 (2020) 104014.
- [39] M. Liu, C. Jimenez-Bescos, J.K. Calautit, Performance evaluation of wind tower natural ventilation with passive solid tube heat recovery based on CO₂ levels, *J. Build. Eng.* 72 (2023) 106457.
- [40] F. Jomehzadeh, H.M. Hussien, J.K. Calautit, P. Nejat, M.S. Ferwati, Natural ventilation by windcatcher (Badgir): a review on the impacts of geometry, microclimate and macroclimate, *Energy Build.* 226 (2020) 110396.
- [41] J.K. Calautit, B.R. Hughes, Wind tunnel and CFD study of the natural ventilation performance of a commercial multi-directional wind tower, *Build. Environ.* 80 (2014) 71–83.
- [42] J.K. Calautit, B.R. Hughes, A passive cooling wind catcher with heat pipe technology: CFD, wind tunnel and field-test analysis, *Appl. Energy* 162 (2016) 460–471.
- [43] J.K. Calautit, D. O'Connor, P.W. Tien, S. Wei, C.A.J. Pantua, B. Hughes, Development of a natural ventilation windcatcher with passive heat recovery wheel for mild-cold climates: CFD and experimental analysis, *Renew. Energy* 160 (2020) 465–482.
- [44] M. Liu, C. Jimenez-Bescos, J.K. Calautit, Passive heat recovery wind tower: assessing the overheating risk in summertime and ventilation heat loss reduction in wintertime, *Sustain. Energy Technol. Assessments* 58 (2023) 103310.
- [45] P. Nejat, J.K. Calautit, M.Z.A. Majid, B.R. Hughes, I. Zeynali, F. Jomehzadeh, Wind tunnel and numerical data on the ventilation performance of windcatcher with wing wall, *Data Brief.* 9 (2016) 448–452.
- [46] Y. Zhang, Y. Zhao, Novel design of a circulation control airfoil with cylinder rotation, *Phys. Fluids* 35 (8) (2023).
- [47] S.Y. Hu, J.H. Cheng, Innovative designs for ducted wind turbines, *Renew. Energy* 33 (7) (2008) 1491–1498.
- [48] Y.K. Yang, M.Y. Kim, Y.W. Song, S.H. Choi, J.C. Park, Windcatcher louvers to improve ventilation efficiency, *Energies (Basel)* 13 (17) (2020) 1–20.
- [49] L.G. Carreto-Hernandez, S.L. Moya, C.A. Varela-Boydo, A. Francisco-Hernandez, Numerical-experimental study of mixed convection in a wind tower-room system, *Build. Environ.* 237 (2023) 110294.
- [50] Ansys Inc. "Ansys fluent theory guide 2021R2 [online].," <https://www.ansys.com/> (Accessed 12 June 2024).
- [51] S. Pashardes, C. Christofides, Statistical analysis of wind speed and direction in Cyprus, *Solar Energy* 55 (5) (1995) 405–414.
- [52] Y. Tominaga, et al., ALJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, *J. Wind Eng. Industr. Aerodyn.* 96 (10–11) (2008) 1749–1761.
- [53] P.J. Roache, "Perspective: a method for uniform reporting of grid refinement studies," 1994.
- [54] Iowa Environmental Mesonet. "Wind roses." https://mesonet.agron.iastate.edu/site/s/windrose.phtml?network=GB_ASOS&station=EGNX (accessed 01/05/2025).
- [55] Windfinder. "Wind & weather statistics." https://www.windfinder.com/windstatistics/nottingham_watnall (accessed 01.05.2025).
- [56] CIBSE Journal T. Dwyer, Room air distribution for effective ventilation, CIBSE J. (2022). Available: (accessed 01.05.2025).