E-ISSN: 2829-6257
P-ISSN: 2962-5629

The Influence of Inlet and Outlet Ratios on the Performance of Natural Ventilation in Mosque, Indonesia

Imron Ahmadi[1], Agus Hariyadi[2]
[1,2] Department of Architecture and Planning, Universitas Gadjah Mada, Yogyakarta, Indonesia

Article Info

Article history:

Received April 22nd, 2025
Revised May 30th, 2025
Accepted June 2nd, 2025

Keywords:

Natural Ventilation,
Inlet and Outlet Ratio,
Mosque,
Thermal Comfort,
CFD Simulation,

ABSTRACT

Natural Ventilation is a sustainable passive strategy for enhancing thermal comfort and reducing energy consumption in buildings with fluctuating occupancy levels, such as mosques. This study investigates the effects of varying inlet-to-outlet opening ratios using different schemes, simulated through Computational Fluid Dynamics (CFD) with the RANS model via OpenFOAM (Butterfly plugin in Grasshopper). Based on urban wind profiles, this research evaluates wind-driven Ventilation under isothermal conditions. The simulation examines five inlet-outlet opening ratios (ranging from 1:1 to 1:5), two inlet configurations (single vs. double), five building lengths (5×10, 10×10, 15×10, 20×10, and 25×10 m), and three building heights (3 m, 4 m, and 5 m). Results indicate that an opening ratio of 1:3 provides the optimal balance between airflow efficiency and thermal stability, particularly concerning solar radiation effects. The single-inlet configuration yields higher air velocity at standing height (1.1 m), whereas the double-inlet setup promotes a more uniform vertical airflow distribution. Increasing building height enhances airflow due to reduced ground-level resistance, while elongated floor plans (over 10 m in length) experience flow stagnation in the central zone. These findings underscore that optimal natural ventilation in mosques depends on opening ratios, spatial proportions, and inlet configurations. Such insights contribute to climate-responsive mosque design, especially in dense urban environments, supporting energy-efficient and comfortable indoor conditions during congregational prayers.

Corresponding Author:

Imron Ahmadi,
Department of Architecture and Planning, Universitas Gadjah Mada,
Jl. Grafika No.2, Sendowo, Sinduadi, Mlati, Sleman, Daerah Istimewa Yogyakarta 55284, Indonesia
Email: imronahmadi@mail.ugm.ac.id

1. INTRODUCTION

Mosques differ significantly from all other types of buildings due to their distinctive functional and operational requirements. Mosques are buildings characterized by external loads, often dominated by irregular and fluctuating occupancy schedules. The occupants significantly contribute to these external loads [1]. For such buildings, internal thermal loads are primarily determined by design features influenced by external climate factors, such as orientation, the surrounding environment, and the building envelope, among others [2]. The indoor air quality in mosques is also influenced by outdoor weather and climate conditions [3]. Therefore, optimizing the building's thermal performance is crucial for maintaining thermal comfort for occupants and ensuring energy efficiency [4]. Thermal comfort for worshippers ensures a calm and conducive atmosphere during prayer [5]. Since the quality of worship is the central purpose, it is enhanced by tranquillity during its performance [6].

The HVAC (Heating, Ventilation, and Air Conditioning) system is critical in energy use within mosques [7]. Nowadays, most mosques rely heavily on HVAC systems to provide thermal comfort for worshippers [8]. Many mosques also prioritise aesthetics and symbolism in their design, often neglecting geo-climatic considerations. It significantly impacts thermal performance, leading to inefficient energy usage [9], and makes such buildings prone to excessive energy consumption [10]. Moreover, HVAC systems often fail to maintain comfort in large-capacity spaces [8]. Solar radiation also contributes to indoor discomfort, especially during peak sunlight [11]. Additionally, body heat generated by worshippers standing close together during prayer contributes to internal heat gain [12][1]. Making it a primary factor influencing thermal comfort [13]. From a thermal comfort perspective, indoor space is generally achieved when the Predicted Mean Vote (PMV) falls within the range of -0.5 to +0.5, indicating that most occupants feel thermally satisfied. In warmer seasons, assuming light clothing insulation (around 0.5 clo) and moderate activity levels (1.0–1.2 met), the optimal operative temperature for comfort typically ranges between 23°C and 27°C. The standard also recommends maintaining relative humidity between 30% and 60% to support thermal comfort and indoor air quality [14].

A passive design approach aims to create climate-responsive buildings through the selection of appropriate materials, orientation, and greenery, while facilitating and enhancing natural ventilation strategies. Since daily prayers involve minimal and short-term occupancy, a passive approach can be a sustainable alternative for improving thermal comfort during low-occupancy periods and reducing energy consumption [9]. For buildings like mosques, which are often dominated by external loads, the most crucial design feature for thermal comfort and energy efficiency is the thermal envelope [15][16]. Optimising the thermal performance of a building through the building envelope is the most effective passive method to reduce external thermal loads. Optimising design parameters and operational schedules can only achieve acceptable thermal comfort levels while simultaneously decreasing energy usage [15]. Hence, it is vital to determine the key design components for mosques within specific geo-climatic contexts and to explore energy-efficient design alternatives.

Natural ventilation stands out as a compelling alternative in building envelope design. Its functionality maximises indoor air circulation by allowing air to enter and exit the space, thus lowering indoor temperatures [16] [15]. It is found that ventilation is often problematic in buildings, primarily due to insufficient airflow, which can be mitigated through effective openings [17]. Similarly, [15] found that optimal air temperature is achieved when high wind speeds coincide with minimal solar radiation. Therefore, wind speed and solar radiation are critical factors affecting indoor temperature [18]. Natural ventilation is a wise choice when implemented with strategies that can significantly impact indoor comfort, particularly when wind speed reaches the comfort threshold, thereby helping to minimise discomfort, especially within the limitations present during prayer activities [19] illustrates that varying the ratio between inlet and outlet openings using a cross-ventilation model is a promising and sustainable strategy in mosque design [15]. Additionally, it is reported that the smaller the opening ratio, the greater the pressure drop, which in turn increases indoor wind speed [15]. It further confirms that the number of openings follows the same principle. Another influential factor besides the ratio is the orientation of the incoming wind [20]. Reports indicate that several initiatives have begun to explore these strategies to optimize natural ventilation performance, laying a foundation for examining other related parameters. In this study, the approach of varying the inlet and outlet opening ratios and wind orientation will be further explored for application in mosque buildings. Naturally, these must be adapted to the activities occurring within the mosque, particularly congregational prayers, as the primary function. This research examines the performance of these strategies when applied to mosque models with rectangular layouts, extending laterally and longitudinally, and featuring varying roof heights.

This study examines the impact of varying inlet and outlet ratios at different scales and wind orientations as passive strategies that influence wind patterns, wind speed, and air exchange within mosque interiors. It also integrates multiple mosque models, varying in size, height, and activity patterns, particularly in terms of obligatory prayers. These models are explored using simulations on the Computational Fluid Dynamics (CFD) platform.

2. INTRODUCTION

2.1 Selection of CFD Analysis Model

Based on previous studies, the Reynolds-Averaged Navier–Stokes (RANS) model is preferred due to its advantage of significantly lower computational cost compared to Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS). This is because of the Reynolds decomposition assumption, which separates time-dependent turbulent velocity fluctuations from the mean flow velocity [21]. Moreover, the RANS model is frequently applied in studies focused on evaluating turbulence model performance, making it suitable for calculating indoor airflow with high precision [22]. Therefore, this research adopts the RANS model as the foundation for CFD simulation. This study focuses solely on wind-driven ventilation. This research does not consider the effects of ventilation caused by temperature differences (or buoyancy forces). Therefore, all simulations were carried out under constant temperature (isothermal) conditions. The simulation in this study uses Rhinoceros 3D software integrated with Grasshopper and the Butterfly plugin, which serves as an interface for OpenFOAM. Butterfly enables a flexible and parametric setup of CFD simulations, including defining boundary conditions, mesh types, and physical parameters related to airflow. This approach allows researchers to visually and numerically analyse airflow patterns and wind speed distributions across various building opening design scenarios.

2.2 Simulation Process

There are three key stages in the simulation process of this study: 1) Preprocessing: This involves creating a three-dimensional model, configuring input data, setting boundary conditions, and validating the model. 2) Solving: In this stage, the CFD solutions are computed based on the parameters configured during preprocessing. 3) Post-processing: The final step is visualising and analysing the results, typically in graphs or images.

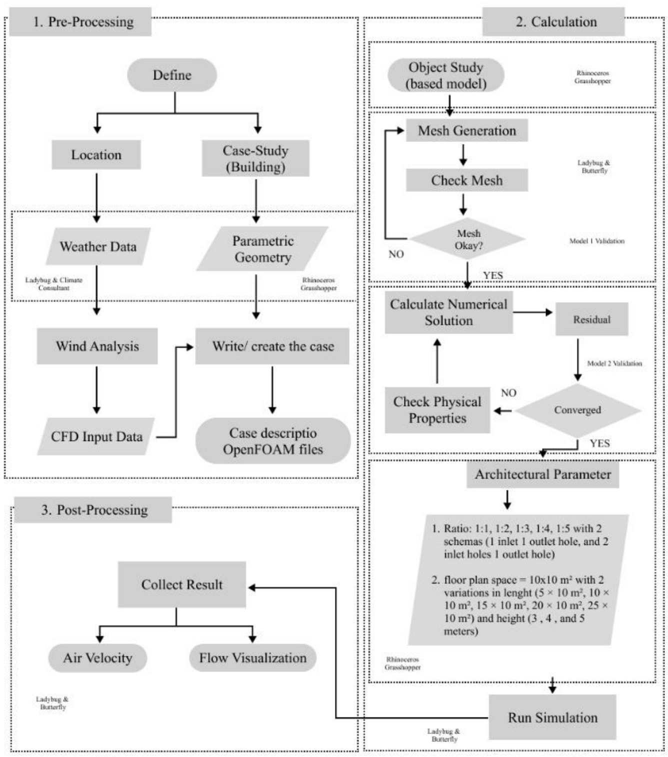


Figure 1. The flow of the CFD simulation process using a parametric 3D model.

2.3 Simulation Parameters

This study adopts a hypothetical case, with the testing conducted in a designated area serving as the simulation space. Jakarta is used as a choice of wind speed data for analysis in the simulation, considering that the city represents one of the most energy-consuming urban areas in Indonesia [22], aligning with the research focus on reducing energy consumption within buildings. Regarding location selection, wind direction is crucial as a primary parameter in the simulation, given its significant impact on ventilation performance [23]. In line with this, wind speed data is required to determine the boundary conditions in the simulation process. Such data can be obtained from the available meteorological agency. EnergyPlus Weather (. EPW) files are the primary data component used as the basis for this research. These data files are integrated with the Climate Consultant software, which processes them to display both visual and numerical data over several years. The software offers options to view the data on a daily, monthly, or yearly basis. The time frequency list can be adjusted based on the specific research needs, and in this study, the average monthly wind speed data is used.

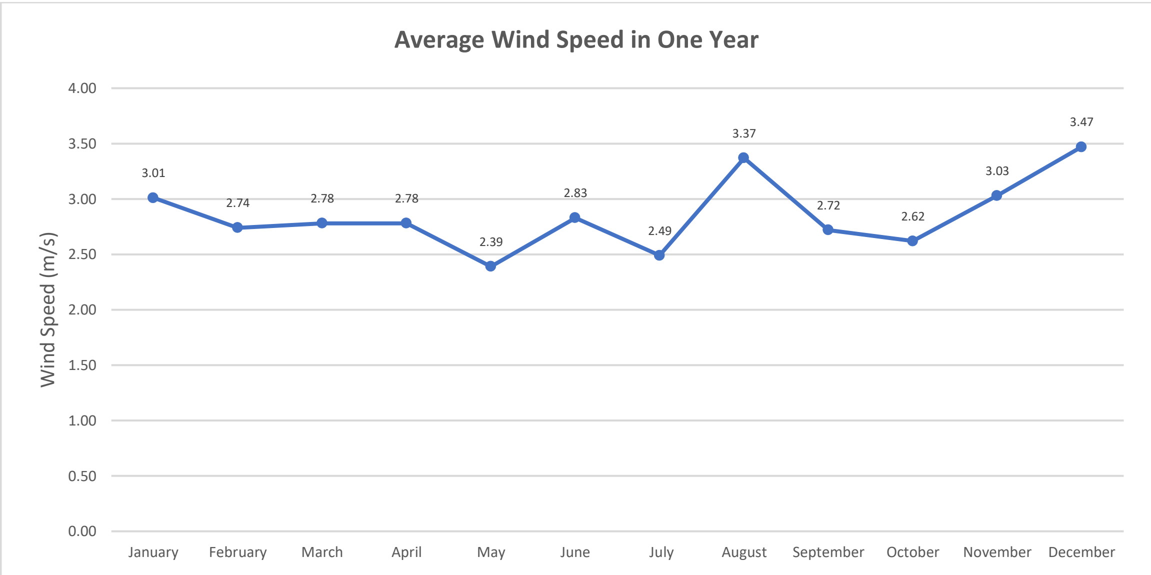


Figure 2. Average Wind Speed in One Year.

Table 1. Simulation Parameters.

PARAMETERS	VALUE
Wind Velocity	2,85 m/s
Wind Tunnel Dimensions (Windward x lateral sides & height x Leeward)	3H x 5H x 15H
Grid Spacing	0,6 meters
Glob Refine Level	Middle 2,3
Landscape Area Size	7 (City Center)
Wind orentation	90° (1,0,0)
Ratios between Inlet and Outlet	1:1, 1:2, 1:3, 1:4, 1:5

Based on Table 2. To realistically model airflow using Computational Fluid Dynamics (CFD), it is essential to accurately represent the actual environmental conditions of the study area. In this research, simulations were conducted using the Butterfly plugin in Grasshopper, where one of the key parameters defined was terrain roughness. This factor plays a crucial role in shaping the wind speed profile at the simulation boundary, particularly at the inlet. It helps ensure that the generated airflow patterns closely resemble real-world conditions. The simulation used wind speed data specific to Jakarta, a city known for its densely built environment. Jakarta's urban landscape is characterized by a mix of high-rise and low-rise buildings, uneven surfaces, and diverse infrastructure elements. Due to this complexity, the area was classified as Category 7 regarding terrain roughness, corresponding to a roughness length (z_0) of 2.0 meters or more. This category is typically applied to environments with chaotic layouts, such as city centres with many obstacles that affect wind movement, like buildings of different heights, vegetation, and narrow roads.

Category 7 was selected to better capture the turbulent and irregular wind flow conditions in Jakarta. It enables the model to account for reduced wind speeds and increased turbulence resulting from physical obstructions. It is especially relevant for evaluating how natural ventilation might perform in dense urban settings, where space and airflow can be restricted. By applying this terrain roughness category to the wind inlet settings in Butterfly, the simulation produces airflow results that more closely reflect the real urban conditions of Jakarta. It helps improve the study's accuracy and supports a deeper understanding of how building design interacts with local microclimates in crowded city areas.

Furthermore, the wind orientation was limited to 0 degrees, representing airflow directly approaching the natural ventilation inlet. This simplification was adopted to reduce the complexity of the simulation variables and represent the most stable and direct wind condition relative to the inlet opening. By focusing on this optimal scenario, the analysis can more accurately assess the natural ventilation performance, eliminating the influence of varying wind directions that may introduce additional complexities [24].

2.4 Computation Domain

The dimensions of the computational domain were defined based on established best practices for CFD simulations in urban atmospheric flow studies, as outlined by [25] and other relevant guidelines [26]. In this setup, the domain boundaries were located at distances of 3H upstream (windward), 5H laterally (from both sides of the building), and 15H downstream (leeward), where H denotes the height of the scaled mosque model. These proportions are widely adopted to ensure the accurate development of the boundary layer and minimize the influence of domain size on simulation results.

The upstream distance of 3H was selected to allow sufficient flow development while maintaining computational efficiency. Although longer windward extents are recommended for high-accuracy turbulence modelling, several studies have demonstrated that a 3H upstream distance is adequate to prevent artificial acceleration or distortion of the inlet flow profile, particularly in cases involving isolated low-rise buildings [27] [28] [29]. The lateral extent of 5H provides adequate space to reduce blockage effects and lateral flow recirculation. At the same time, the downstream length of 15H ensures that wake development and flow separation can be fully captured without reflection effects from the outlet boundary.

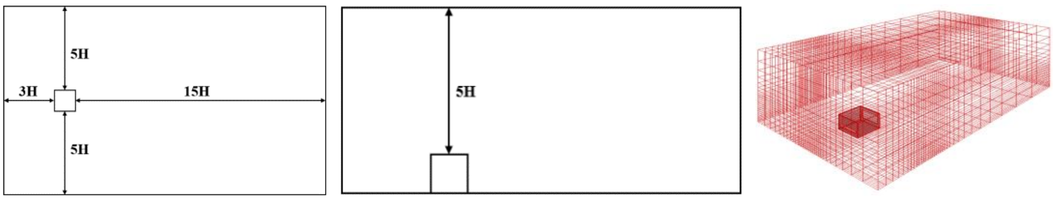


Figure 3. Windtunnel Size as Simulation Domain.

2.5 Solver Setting

The airflow was modelled as turbulent; therefore, the Reynolds-Averaged Navier–Stokes (RANS) equations were combined with the standard k-ε turbulence model [30]. This two-equation model is widely used in natural ventilation studies because it strikes a balance between computational efficiency and accuracy. Previous research has shown that the k-ε model reliably predicts indoor airflow behaviour under various conditions [31] [32]. A steady-state solver for incompressible and turbulent flow was applied to solve the velocity-pressure coupling, using the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm [33]. Under-relaxation factors were set at 0.3 for pressure and 0.7 for velocity, turbulent kinetic energy (k), and dissipation rate (ε), to enhance numerical stability by controlling the update rate of each variable during iteration [34].

Second-order discretisation schemes were adopted for convective and diffusive terms to improve solution accuracy. Specifically, the bounded Gauss linear Upwind [35] scheme was applied for convection terms. In contrast, the Gauss linear limited corrected scheme, with a non-orthogonal correction factor of 0.333, was used for diffusion terms. Convergence was evaluated based on residual thresholds (set at a maximum of 10^5) and monitoring key output parameters such as velocity and pressure at selected probe points, particularly near window openings. The solution converged when residuals stabilised and the solution imbalance remained below 0.5%.

2.6 Mesh Processing and Boundary Conditions

Accurate and stable results in Computational Fluid Dynamics (CFD) simulations depend heavily on the quality of mesh generation. In this research, CFD analyses were conducted using the Butterfly plugin within the Grasshopper environment, which serves as a visual interface for the OpenFOAM solver. Mesh generation was performed using the SnappyHexMesh algorithm, a widely used tool for producing structured and unstructured hexahedral meshes that adapt effectively to complex geometries [36]. To improve the precision of simulations, two mesh refinement levels were implemented: level 2 and level 3. The second level of refinement provides a moderate mesh density, suitable for initial testing and comparative studies.

In contrast, the third level provides finer resolution in key areas such as wall surfaces and openings, which is crucial for capturing detailed flow features and turbulence patterns [37]. Boundary conditions were established based on the physical setup of the model, including specified velocity inlets, pressure outlets, and no-slip wall boundaries, in accordance with standard CFD protocols [38]. The careful setup of mesh parameters and boundary conditions plays a crucial role in ensuring the reliability of CFD outputs, particularly when evaluating natural ventilation performance and airflow patterns in architectural spaces.

Table 2. Grid sensitivity with CFD.

Mesh Type	Refine Levels	Residuals				
		Ux	Uy	Uz	p	Cumulative Continuity
Coarse	2,3	2,0 x 10^{-5}	3,1 x 10^{-6}	9,1 x 10^{-6}	1,1 x 10^{-6}	1,306 x 10^{-6}
Medium	3,4	2,1 x 10^{-5}	3,2 x 10^{-6}	9,3 x 10^{-6}	4,6 x 10^{-6}	1,309 x 10^{-6}
Fine	4,5	3,6 x 10^{-5}	6,4 x 10^{-6}	15,8 x 10^{-6}	4,4 x 10^{-6}	1,305 x 10^{-6}

2.7 Research Object

The ratio between inlet and outlet openings is the primary parameter in this study, with the building model developed using a cross-ventilation approach. The mosque model simulated in this research has a floor plan dimension of 10 × 10 meters. This area is estimated to accommodate 100 worshippers during congregational prayer. The estimation is based on the standard space allocation per individual, approximately 0.72 m² (60 × 120 cm²). Therefore, the total capacity is projected to be around 138 individuals under optimal conditions. However, this figure does not account for the space occupied by structural columns or the designated area for the prayer leader (imam).

Table 3. Simulation Test Model Indicators.

Opening Ratio (Inlet to Outlet Ratio)	1:1 ; 1:2 ; 1:3 ; 1:4 ; 1:5
Number of inlet and outlet difference schemes	1 inlet 1 outlet hole and 2 inlet holes 1 outlet hole
Area	5x10 ; 10x10 ; 15x10 ; 20x10 ; 25x10 meters
Height	3, 4, 5 meters

Beyond the inlet-outlet ratio, this study also examines the influence of two additional parameters: variations in floor area and building height. The floor area variations are limited to five configurations, namely: 5 × 10 m², 10 × 10 m², 15 × 10 m², 20 × 10 m², and 25 × 10 m². These configurations are designed to assess the impact of the inlet-outlet ratio on ventilation performance in buildings with increasing longitudinal dimensions between the inlet and outlet openings.

Building height is assessed through three variations: 3 meters, 4 meters, and 5 meters. It is intended to determine its effect on indoor airflow velocity, using the inlet-outlet ratio as the primary control parameter. These variations in area and height are not combined with other factors, as the study is deliberately limited to a manageable scope due to time constraints and its exploratory nature.

Two additional design schemes are tested, differentiated by the number of inlet openings. This variable is examined to assess its effect on indoor wind velocity, considering that wind speed generally increases with height [39]. However, regarding total opening area, both schemes maintain equivalent inlet surface area to ensure comparability.

The primary output of this research is the wind speed within the prayer space, which directly influences the thermal comfort and overall experience of mosque users, particularly during obligatory prayers that typically have the most extended duration. Wind velocity vectors are evaluated at a height of 1.1 meters, corresponding to the average standing head height of occupants, based on data from [40]. The inlet and outlet openings are also aligned to this reference height, as illustrated in Figure 4. The complete visualisation of the simulation schemes employed in this study is presented in Figure 5.

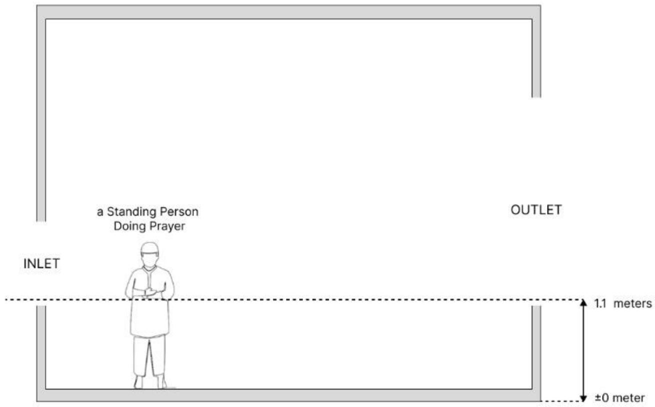


Figure 4. The Schema Integrating Openings and Activity (Praying) in Mosque.

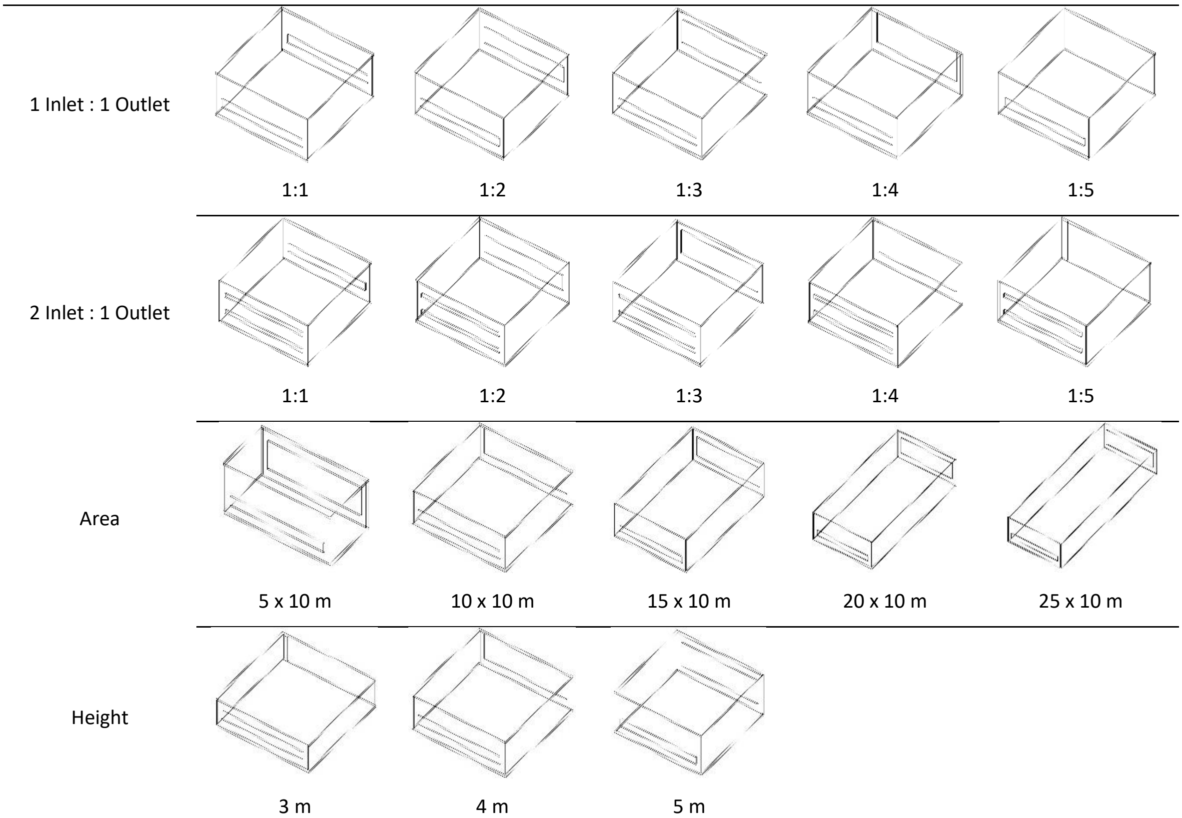
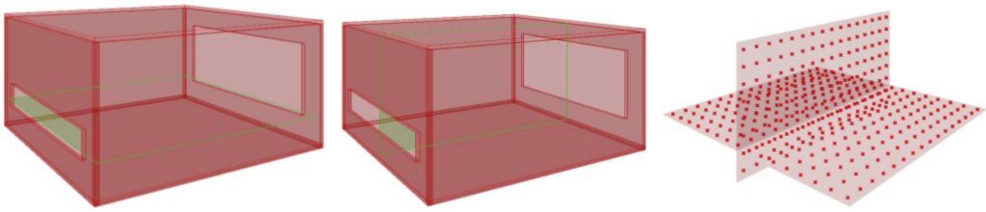


Figure 5. Simulation Scheme

2.8 Post-Processing

Figure 6 represents that the final output of this study evaluates two indicators: 1) the wind speed on the horizontal surface inside the mosque at a height of 1.1 meters, and 2) the airflow pattern occurring within the room, both horizontally and vertically. Each point of the measured wind velocity vectors was planned with a spacing of 0.6 meters.



(a) (b) (c)

Figure 6. Final Output Visualization of This Study
(a) Horizontal Vector Plan, (b) Vertical Vector Plan, (c) Vector Spacing.

2.9 Wind Speed Standard

The categorization of wind speed values is used to help analyze simulation results and reflect possible real-life conditions in the field based on those wind speed values.

Table 4. Indoor Wind Speed Standard.

Wind Speed (m/s)	Effect on Comfort	Cooling Effect (at 30°C)
< 0.25	Not noticeable	0°C
0.25–0.5	Most comfortable	0.5–0.7°C
0.5–1	Still comfortable, but air movement is noticeable	1.0–1.2°C
1–1.5	Maximum comfort level	1.7–2.2°C
1.5–2	Less comfortable, breezy	2.0–3.3°C
> 2	Occupant health affected by high wind speed	2.3–4.2°C

Source: [41]

3. RESULTS AND DISCUSSION

The discussion in this chapter is divided into three sections: 1) the comparison results between buildings with one and two inlets, respectively, 2) differences in height, and 3) what happens when the building becomes increasingly elongated from the inlet side to the outlet.

3.1 Difference Between 1 Hole and 2 Holes Inlet

The two presented graphs show that at a height of 1.1 meters, the configuration with a single inlet generates higher air velocity than the two-inlet configuration. Although the airspeed generally decreases from the starting point to near the outlet, the velocity remains higher in the single-inlet scheme. Moreover, based on Figure 8, the vertical distribution of air velocity within the space appears more even in the single-inlet setup, aligning with the needs of worship activities as shown in Figure 4. The dominant position during prayer is standing, though sitting is also involved, making the presence of airflow at different heights crucial for the thermal comfort of the worshippers.

Meanwhile, the two-inlet scheme shows a unique characteristic: a sharp drop in air velocity up to point 4, followed by stable flow until near the outlet (point 15). Another advantage of the two-inlet scheme, especially at a 1:1 ratio, is that the wind speed remains above 0.4 m/s—unlike in the single-inlet scheme, which drops below that threshold between positions 11 and 13. This finding aligns with the research in [42], which states that a multi-inlet system in large rooms can produce a more symmetrical airflow pattern and be thermally more efficient for space users.

Based on the wind speed trends, the variation in inlet-to-outlet ratios from 1:3 to 1:5 shows a tendency toward stagnation in flow performance improvement. It supports findings from the study [43], which state that there is an optimal point in determining the ratio strategy between the inlet and the outlet. Compared with the wind speed standards referenced in Table 4, the minimum wind speed recorded across all ratio variations still falls within the comfort category for room occupants.

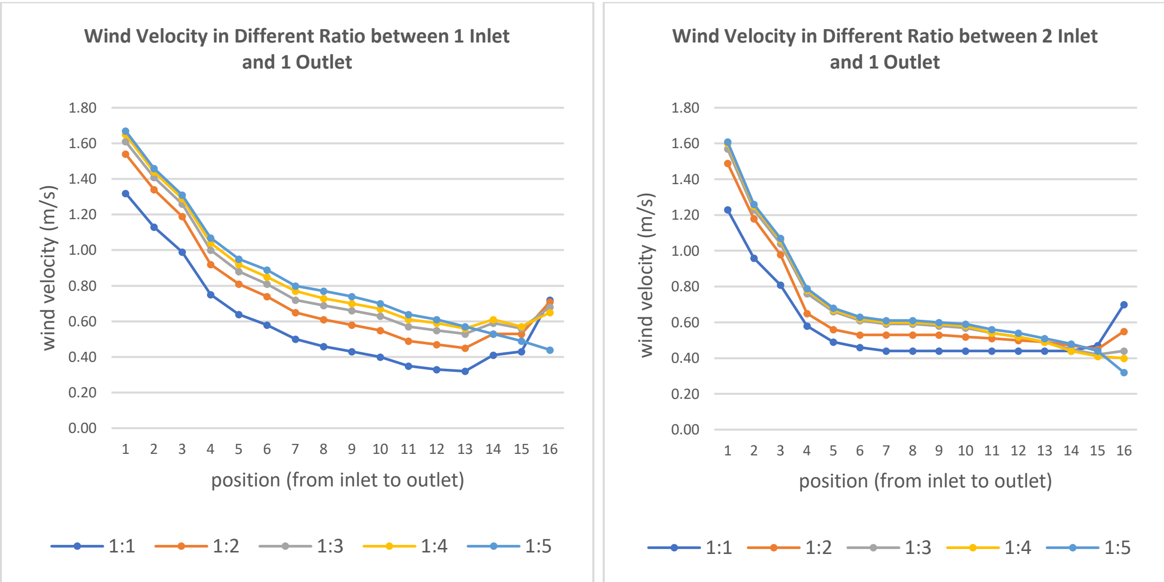
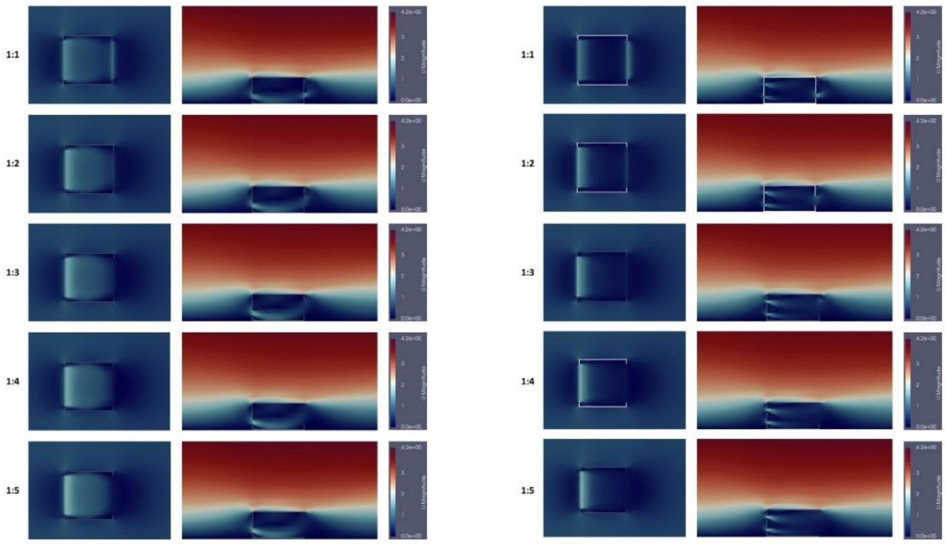


Figure 7. Comparison of Simulation Results between Using 1 and 2 Openings

However, these results must be further examined in subsequent studies, as this simulation has not yet involved human objects directly performing worship activities indoors. Additionally, the strategy of varying inlet and outlet ratios should also consider the influence of solar radiation. While the 1:5 ratio does produce the highest initial wind speed, it also exhibits greater exposure to thermal radiation. The effectiveness of ventilation against radiation influence is greatly affected by the orientation of the inlet toward the heat source and the capacity of incoming air to replace hot air inside the space. Therefore, based on the simulation results, the 1:3 and 1:4 ratios can be more ideal for facing high solar radiation conditions while maintaining airflow stability within the space.



a) b)

Figure 8. Horizontal and Vertical Airflow Patterns in Different Schemes
(a) 1 inlet, (b) 2 Inlets

3.2 Floor Area

The simulation in this study was conducted by varying the floor area of the mosque building, using a fixed inlet-to-outlet opening ratio of 1:3. This ratio was chosen based on initial simulation results that indicated optimal performance in terms of incoming airflow and control over solar radiation, both of which affect thermal comfort within the space.

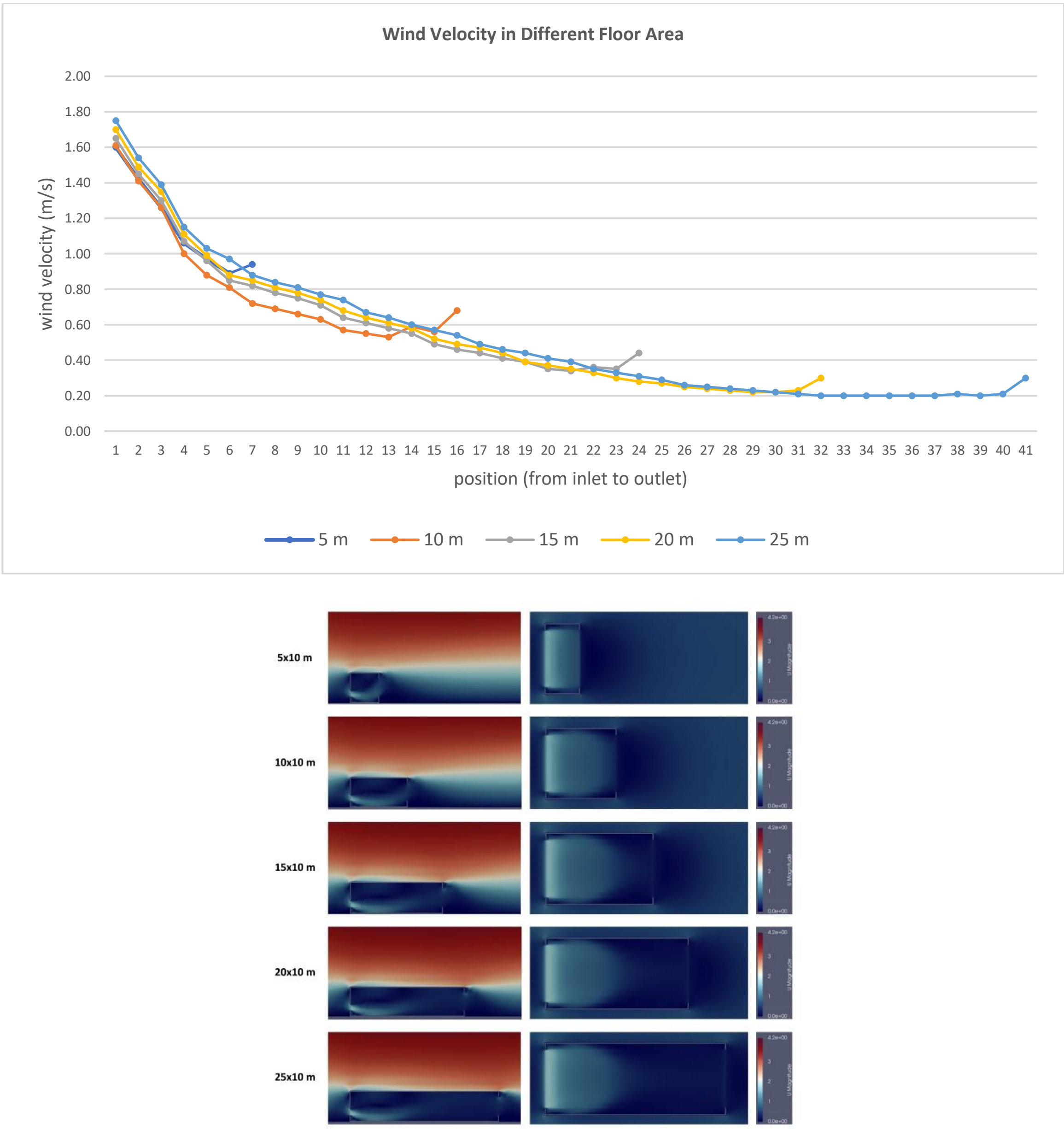


Figure 9. Wind Velocity Average and Visualization in Different Floor Area

The simulation results show that across all room length variations, wind speed tends to be high near the inlet (approximately 1.6–1.8 m/s), but drops sharply as the distance from the inlet increases. Flow stagnation becomes more significant in rooms with longer dimensions, such as 20×10 m and 25×10 m, as visualised by the dark blue areas in the horizontal diagram (Figure 9), indicating wind speeds approaching zero in much of the central area up to near the outlet.

Interestingly, from dimensions of 10×10 m to 25×10 m, there is no significant improvement in wind speed distribution despite the increase in room length. It suggests that increasing room length enlarges the stagnation zone and reduces the effectiveness of natural ventilation. Therefore, in the context of mosque design that relies on passive cross ventilation, it is important to consider limiting room length or implementing additional architectural strategies to maintain overall thermal comfort. This issue is caused by reduced thermal pressure differences and a weakened stack effect in longer spaces [44].

3.3 Height

In the height variation scheme, the inlet-to-outlet opening ratio was also limited to 1:3, similar to the floor area variation scheme. This approach serves as a preliminary exploration for this study. Figure 10 illustrates that, in general, an increase in building height correlates with an increase in the average indoor wind speed. At the initial positions, the different heights (5, 4, and 3 meters) show significant differences in wind speed. However, this speed progressively decreases along the path toward the outlet (position 16), with a tendency toward stagnation occurring after position 10.

Wind speed generally increases with building height. This is because, near ground level, wind is obstructed by elements such as vegetation, buildings, and surface terrain. As height increases, these obstructions diminish, allowing wind to move more freely. This phenomenon aligns with the landscape selection used in the Butterfly simulation, specifically landscape no. 7 [45] elaborates that this is known as the "speed or height effect," where wind blows stronger at higher elevations due to reduced surface drag and the absence of aerial obstructions.

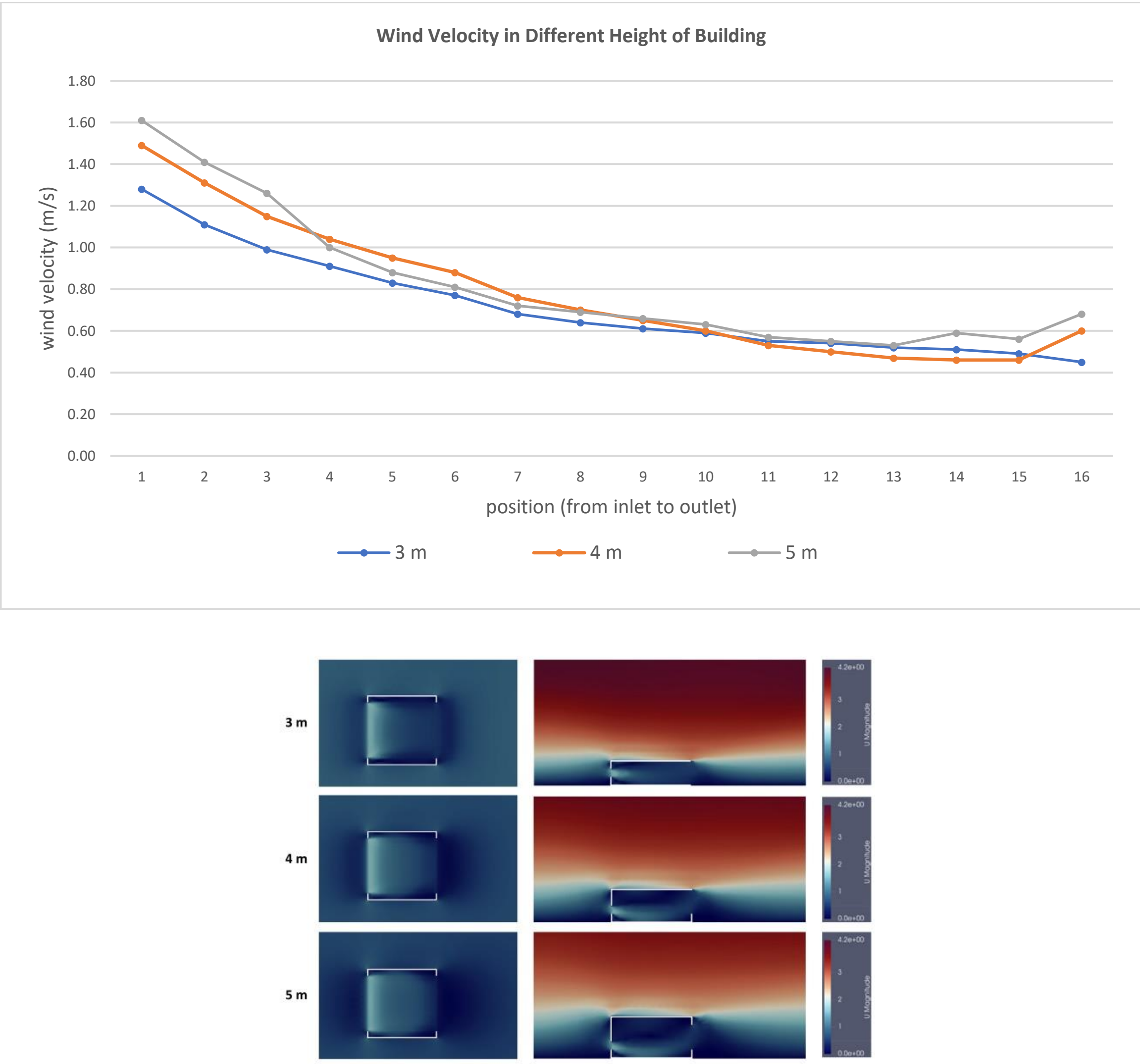


Figure 10. Wind Velocity and Visualization in Different Height of Building

Therefore, within this study, a height of 5 meters tends to be the most effective option, even though between positions 4 and 8, the 4-meter height exhibits slightly higher wind speeds. Furthermore, based on Table 4, all three height scenarios maintain wind speeds above 0.4 m/s, indicating that none fall below the threshold for comfortable indoor wind speed.

4. CONCLUSION

This study examines the impact of variations in the inlet-to-outlet opening ratio, as well as differences in building height and floor dimensions, on the performance of natural ventilation in mosque buildings, particularly in densely populated urban areas. The results indicate that an opening ratio 1:3 provides a more optimal balance between airflow and thermal stability. A single-inlet configuration generates higher air velocity at standing height (1.1 meters), a critical parameter for comfort during prayer, while a dual-inlet configuration offers more stable airflow distribution advantages.

Furthermore, taller buildings, especially those with a height of 5 meters or more, exhibit improved airflow due to reduced upper-level obstruction. However, elongated buildings (15 meters or more) tend to experience poor airflow in the central zone, compromising ventilation efficiency. It highlights the significant role of spatial form and proportion in the design of passive ventilation systems.

Based on these findings, several potential directions for future research are proposed:

1. Incorporating simulations of occupant presence to represent actual usage conditions during prayer.
2. Integrating dynamic occupancy modelling to reflect realistic heat loads during worship activities.
3. Including the influence of solar radiation and thermal mass to enhance the accuracy of ventilation performance under realistic climatic scenarios.
4. Exploring the application of hybrid ventilation systems that combine passive and mechanical strategies to overcome performance limitations in large or irregularly shaped prayer halls.
5. Conducting experimental validation through field measurements to support and calibrate simulation results.

This study provides valuable insights for architects in designing mosques that prioritise thermal comfort and energy efficiency, particularly in urban environments where indoor climate control presents unique challenges

ACKNOWLEDGEMENTS

We would like to thank ourselves for now being able to complete this study together. We also thank the reviewers and editors for their essential comments, which enhanced the manuscript.

REFERENCES

[1] N. A. Azmi, M. Arıcı, and A. Baharun, "A review on the factors influencing energy efficiency of mosque buildings," *J. Clean. Prod.*, vol. 292, p. 126010, Apr. 2021, doi: 10.1016/J.JCLEPRO.2021.126010.

[2] M. S. Al-Homoud, "Envelope thermal design optimization of buildings with intermittent occupancy," *J. Build. Phys.*, vol. 33, no. 1, pp. 65–82, 2009, doi: 10.1177/1744259109102799.

[3] A. B. Atmaca and G. Z. Gedik, "Determination of thermal comfort of religious buildings by measurement and survey methods: Examples of mosques in a temperate-humid climate," *J. Build. Eng.*, vol. 30, p. 101246, Jul. 2020, doi: 10.1016/J.JOBE.2020.101246.

[4] Q. Li, L. Zhang, L. Zhang, and X. Wu, "Optimizing energy efficiency and thermal comfort in building green retrofit," *Energy*, vol. 237, p. 121509, Dec. 2021, doi: 10.1016/J.ENERGY.2021.121509.

[5] A. Yüksel, M. Arıcı, M. Krajčík, M. Civan, and H. Karabay, "Cooling operation strategies in a typical neighborhood mosque: A Case study on electricity consumption, CO2 emissions, and thermal comfort," *Energy Build.*, vol. 310, p. 114073, May 2024, doi: 10.1016/J.ENBUILD.2024.114073.

[6] A. Rahman, M. Qorib, and Z. Zuliana, "Pelatihan Tahsin Bacaan Sholat Dalam Meningkatkan Kualitas Bacaan Sholat Melalui Metode Muqotha'Ah Pada Ibu-Ibu Aisyiyah Ranting Puji Mulio," *JMM (Jurnal Masy. Mandiri)*, vol. 7, no. 6, p. 6158, 2023, doi: 10.31764/jmm.v7i6.17846.

[7] A. Yüksel, M. Arıcı, M. Krajčík, M. Civan, and H. Karabay, "Developing a strategy to reduce energy consumption and CO2 emissions from underfloor heating systems in mosques: A case study of a typical neighbourhood mosque," *Energy Build.*, vol. 287, p. 112984, May 2023, doi: 10.1016/J.ENBUILD.2023.112984.

[8] N. A. Azmi, A. Baharun, M. Arıcı, and S. H. Ibrahim, "C," *Front. Archit. Res.*, vol. 12, no. 2, pp. 361–385, Apr. 2023, doi: 10.1016/J.FOAR.2022.07.001.

[9] N. A. Azmi and S. H. Ibrahim, "A comprehensive review on thermal performance and envelope thermal design of mosque buildings," *Build. Environ.*, vol. 185, p. 107305, Nov. 2020, doi: 10.1016/J.BUILDENV.2020.107305.

[10] A. Sayed Hassan Abdallah, "Improved energy consumption and smart eco system for mosques in hot arid climates," *Ain Shams Eng. J.*, vol. 14, no. 7, p. 101997, Jul. 2023, doi: 10.1016/J.ASEJ.2022.101997.

[11] Q. Dong, S. Li, and C. Han, "Numerical and experimental study of the effect of solar radiation on thermal comfort in a radiant heating system," *J. Build. Eng.*, vol. 32, p. 101497, Nov. 2020, doi: 10.1016/J.JOBE.2020.101497.

[12] N. A. Azmi, A. Baharun, M. Arıcı, and S. H. Ibrahim, "Improving thermal comfort in mosques of hot-humid climates through passive and low-energy design strategies," *Front. Archit. Res.*, vol. 12, no. 2, pp. 361–385, Apr. 2023, doi: 10.1016/J.FOAR.2022.07.001.

[13] Y. Zhang, X. Zhou, Z. Zheng, M. O. Oladokun, and Z. Fang, "Experimental investigation into the effects of different metabolic rates of body movement on thermal comfort," *Build. Environ.*, vol. 168, p. 106489, Jan. 2020, doi: 10.1016/J.BUILDENV.2019.106489.

[14] A. ASHRAE, "ASHRAE Standard 55: Thermal environmental conditions for human occupancy," *Am. Soc. Heating, Refrig. Air-Conditioning Eng. Atlanta, GA, USA*, 2017.

[15] M. Marzouk, M. El-Maraghy, A. El-Shihy, and M. Metawie, "Evaluating energy retrofit strategies in enhancing operational performance of mosques: A case study of Al-Imam Al-Hussein Mosque," *Clean. Energy Syst.*, vol. 9, p. 100144, Dec. 2024, doi: 10.1016/J.CLES.2024.100144.

[16] F. Ghaleb Noman, N. Kamsah, and H. Mohamed Kamar, "Jurnal Teknologi Full Paper IMPROVEMENT OF THERMAL COMFORT INSIDE A MOSQUE BUILDING," vol. 78, pp. 2180–3722, 2016, [Online]. Available: www.jurnalteknologi.utm.my

[17] R. T. Novita, L. H. Sari, and A. Ariatsyah, "an Appraisal of Indoor Thermal Comfort in a Naturally-Ventilated Modified Old Mosque in Banda Aceh, Indonesia," *J. Islam. Archit.*, vol. 7, no. 3, pp. 364–370, 2023, doi: 10.18860/JIA.V7I3.17131.

[18] B. Song, L. Bai, and L. Yang, "Analysis of the long-term effects of solar radiation on the indoor thermal comfort in office buildings," *Energy*, vol. 247, p. 123499, May 2022, doi: 10.1016/J.ENERGY.2022.123499.

[19] W. F. M. Yusoff, "The effects of various opening sizes and configurations to air flow dispersion and velocity in cross-ventilated building," *J. Teknol.*, vol. 82, no. 4, pp. 17–28, 2020, doi: 10.11113/jt.v82.14537.

[20] I. F. Ridzqo, E. Rochimah, and J. Soedarno, "Quantifying Window Effectiveness To Affect Wind Velocity By Experimental Study," *Sinergi*, vol. 25, no. 2, p. 127, 2021, doi: 10.22441/sinergi.2021.2.003.

[21] W. Zhong, Y. Pan, W. Xiao, and T. Zhang, "Identifying bioclimatic techniques for sustainable low-rise high-density residential units: Comparative analysis on the ventilation performance of vernacular dwellings in China," *J. Build. Eng.*, vol. 80, p. 108008, Dec. 2023, doi: 10.1016/J.JOBE.2023.108008.

[22] Zai, "Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part-1," *Ashrae*, vol. 13, no. 6, pp. 1–26, 2012.

[23] X. Zhang, A. U. Weerasuriya, and K. T. Tse, "CFD simulation of natural ventilation of a generic building in various incident wind directions: Comparison of turbulence modelling, evaluation methods, and ventilation mechanisms," *Energy Build.*, vol. 229, p. 110516, Dec. 2020, doi: 10.1016/J.ENBUILD.2020.110516.

[24] N. Sakiyama, J. Frick, T. Bejat, and H. Garrecht, "Using CFD to Evaluate Natural Ventilation through a 3D Parametric Modeling Approach," *Energies*, vol. 14, Apr. 2021, doi: 10.3390/en14082197.

[25] Y. Tominaga *et al.*, "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings," *J. Wind Eng. Ind. Aerodyn.*, vol. 96, no. 10–11, pp. 1749–1761, 2008, doi: 10.1016/j.jweia.2008.02.058.

[26] J. Franke, A. Hellsten, K. H. Schlünzen, and B. Carissimo, "The COST 732 Best Practice Guideline for CFD simulation of flows in the urban environment: A summary," *Int. J. Environ. Pollut.*, vol. 44, no. 1–4, pp. 419–427, 2011, doi: 10.1504/IJEP.2011.038443.

[27] R. Ramponi and B. Blocken, "CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters," *Build. Environ.*, vol. 53, pp. 34–48, 2012.

[28] B. Blocken, T. Stathopoulos, and J. Carmeliet, "CFD simulation of the atmospheric boundary layer: wall function problems," *Atmos. Environ.*, vol. 41, no. 2, pp. 238–252, 2007.

[29] B. Blocken, J. Carmeliet, and T. Stathopoulos, "CFD evaluation of wind speed conditions in passages between parallel buildings—effect of wall-function roughness modifications for the atmospheric boundary layer flow," *J. Wind Eng. Ind. Aerodyn.*, vol. 95, no. 9–11, pp. 941–962, 2007.

[30] T. Cebeci, J. P. Shao, F. Kafyeke, and E. Laurendeau, "Computational fluid dynamics for engineers: from panel to Navier-Stokes methods with computer programs," *(No Title)*, 2005.

[31] Z. Zhang, Z. Wei, Z. Zhiqiang John, and Q. Y. and Chen, "Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 2—Comparison with Experimental Data from Literature," *HVAC&R Res.*, vol. 13, no. 6, pp. 871–886, Nov. 2007, doi: 10.1080/10789669.2007.10391460.

[32] T. G. Farea, D. R. Ossen, S. Alkaff, and H. Kotani, "CFD modeling for natural ventilation in a lightwell connected to outdoor through horizontal voids," *Energy Build.*, vol. 86, pp. 502–513, Jan. 2015, doi:

10.1016/J.ENBUILD.2014.10.030.
[33] S. V. Patankar and D. B. Spalding, "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows," *Int. J. Heat Mass Transf.*, vol. 15, no. 10, pp. 1787–1806, Oct. 1972, doi: 10.1016/0017-9310(72)90054-3.
[34] H. K. Versteeg, *An introduction to computational fluid dynamics the finite volume method, 2/E*. Pearson Education India, 2007.
[35] R. F. Warming and R. M. Beam, "Upwind second-order difference schemes and applications in aerodynamic flows," *AIAA J.*, vol. 14, no. 9, pp. 1241–1249, 1976.
[36] H. Jasak, A. Jemcov, and Z. Tukovic, "OpenFOAM: A C++ library for complex physics simulations," in *International workshop on coupled methods in numerical dynamics*, Dubrovnik, Croatia), 2007, pp. 1–20.
[37] F. M. L. M. M. Darwish, *The finite volume method in computational fluid dynamics*. 2016.
[38] J. H. Ferziger and M. Perić, *Computational methods for fluid dynamics*. Springer, 2002.
[39] G. Fang, M. Wei, L. Zhao, K. Xu, S. Cao, and Y. Ge, "Site- and building height-dependent design extreme wind speed vertical profile of tropical cyclone," *J. Build. Eng.*, vol. 62, p. 105322, Dec. 2022, doi: 10.1016/J.JOBE.2022.105322.
[40] ASHRAE-55, "Thermal Environmental Conditions for Human Occupancy-2017," *ANSI/ASHRAE Stand. 55*, vol. 7, p. 60, 2017.
[41] H. Frick and C. Koesmartadi, *Ilmu Fisika Bangunan*, vol. 8. Kanisius, 2008.
[42] A. G. Safitra, L. Diana, J. Pratilastiarso, and N. Hidayat, "Numerical study the effect of inlet and outlet ventilation configurations for passive cooling air conditioning system," 2024, doi: 10.31284/j.jmesi.2024.v4i2.6659.
[43] D. Maher *et al.*, "Effect of inlet/outlet on thermal performance of naturally ventilated building," *Int. J. Low-Carbon Technol.*, vol. 16, no. 4, pp. 1348–1362, 2021, doi: 10.1093/ijlct/ctab055.
[44] S. Liu and C. Huang, "Optimization of natural ventilation of 'yinzi' dwellings in western hunan based on orthogonal experiment and CFD," *Buildings*, vol. 6, no. 3, 2016, doi: 10.3390/buildings6030025.
[45] A. Kaushik, S. Singh, and M. Kapshe, "Changing vertical wind profile and its importance for wind-induced natural ventilation in high-rise buildings–a case of Pune," *J. Eng. Des. Technol.*, no. ahead-of-print, 2025.

BIOGRAPHIES OF AUTHORS

Imron Ahmadi	Student in the Department of Architecture and Planning, Universitas Gadjah Mada. His research interest is related to structural engineering and building science.
Agus Hariyadi	full-time lecturer in the Department of Architecture and Planning, Universitas Gadjah Mada. His research interests are Environment engineering, Facade design, Thermal and visual comfort in building.