Integrating Natural Ventilation and HVAC Systems for Sustainable Building Design.

Abstract

The building industry is working to overcome energy efficiency and environmental sustainability challenges. One solution gaining traction is combining natural ventilation with HVAC (heating, ventilation, and air conditioning) systems. This approach aims to create sustainable buildings that balance energy efficiency, indoor air quality, and occupant comfort. This paper investigates the advantages and obstacles of merging natural ventilation and HVAC systems, including better air quality, lower energy use, and increased occupant satisfaction by analyzing existing research and real-world examples. To enhance the performance of integrated natural ventilation and HVAC systems, various design approaches and technologies were utilized, including strategic building orientation, façade design, and intelligent control systems. This study also presents case studies, explores natural ventilation techniques, and discusses the benefits and challenges of this approach in achieving energy efficiency and sustainable architecture. The results show that combining natural ventilation and HVAC systems can substantially lower energy consumption and environmental impact while improving occupant health and well-being. This research offers practical guidance for architects, engineers, and building professionals to design sustainable, high-performance buildings that balance human needs with environmental responsibility.

Keywords: Natural ventilation; HVAC systems; sustainable building design; energy efficiency; indoor air quality; occupant comfort.

Introduction

Global energy demand has risen sharply by more than 50% over the past few decades, mainly due to the dual factors of population expansion and economic progress [1]. The substantial rise in non-renewable energy consumption has had far-reaching consequences, imposing significant economic costs and causing widespread environmental harm, including the release of greenhouse gases, acceleration of climate change, and intensification of global warming [2][3]. Globally, buildings represent a significant source of energy consumption, using between 20% to 40% of the world's total energy output, making them one of the largest energy-using sectors [4]. The majority of energy consumed by buildings is attributed to heating, ventilation, and air conditioning (HVAC) systems for temperature control and air quality, and this percentage is projected to increase to 64% by the year 2100 [5][6]. To address the substantial energy consumption and resulting greenhouse gas emissions, numerous countries have incorporated energy efficiency into their building codes. In areas where heating or cooling demands are high, adopting effective passive heating and cooling techniques offers considerable opportunities to decrease HVAC energy consumption. Furthermore, the trend of urban densification, fueled by population growth and economic expansion, has resulted in a surge in the construction of multistory buildings, presenting new challenges and opportunities for energy-efficient design [7][8].

Despite the potential for energy efficiency, many multistory buildings have been designed without adequately incorporating passive climate-responsive strategies, leading to unnecessarily high energy consumption [5]. By embracing efficient passive design strategies for heating and cooling, multistory buildings can substantially lower their energy consumption, reducing their reliance on air conditioning systems. One effective approach is natural ventilation, which harnesses the natural movement of air between indoors and outdoors to enhance thermal comfort, refresh stale air, and create a healthier indoor environment without the need for mechanical devices [9]. Natural ventilation offers a cost-effective alternative to mechanical systems for building ventilation, as it is free from energy costs and requires only minimal upkeep. This advantage is especially significant in regions where cooling demands are high, as air conditioning systems are typically the largest energy consumers in buildings, making natural ventilation a valuable strategy for reducing energy expenses [10]. Although natural ventilation offers numerous benefits, predicting its effectiveness in building design can be difficult due to the intricate physics involved. To maximize its potential, it's crucial to conduct accurate predictions early in the design phase, utilizing suitable methods to evaluate a building's ventilation performance and inform design decisions that optimize natural ventilation's impact.

The effectiveness of natural ventilation is primarily assessed by examining various fluid dynamic factors, including airflow patterns, average velocities, airflow rates, pressure distributions, Mean Age of Air (MAA), and volumetric flow rates. These parameters, along with derived properties, serve as [9]. By examining fluid dynamic features, researchers can not only assess natural ventilation performance but also gain insights into a building's overall indoor environment, including Indoor Air Quality (IAQ) and thermal comfort. A range of techniques exists for predicting and evaluating natural ventilation effectiveness, each with its strengths and weaknesses, providing flexibility in choosing the most appropriate method for a given project. [11]. Therefore, it's crucial to choose the most suitable method or combination of methods, considering the project's specific requirements, available resources, and design stage, to ensure the effective evaluation and optimization of natural ventilation [10].

Research has explored the ventilation performance of dome-shaped roofs and similar structures, with a particular focus on the effectiveness of wind-induced ventilation in these designs and the resulting airflow pressure patterns above the domed roofs [12]. An outline of earlier study, comprising the purpose, methods, and conclusions of experimental, analytical, and numerical CFD modelling studies, will be provided in this part. In an experimental study, [13] conducted wind tunnel tests on a 1/10 scale domed roof model to determine its wind pressure coefficient (CP) at various points. The study focused on the apex and dome collar, examining three scenarios with different opening configurations. The results showed that the CP peaked at the collar, reaching a maximum value of 1 in the first two scenarios, while the minimum value of -2.3 occurred at the apex. [14] conducted an analytical study to investigate the potential of domed roofs to meet thermal requirements during warm seasons. Their analysis considered various factors, including airflow patterns, solar radiation, radiation heat transfer with the sky and ground, and opening configurations. The study evaluated the thermal performance of domed roofs covered with both conventional materials and glazed tiles. The findings revealed that domed roofs covered with glazed tiles provided better thermal conditions during summer compared to flat roofs, highlighting their potential for improved thermal performance. The study also found that the openings in the domed structure created a passive airflow, contributing to a comfortable indoor temperature. The research investigated the optical and thermal properties of the hemispherical dome, revealing that the domed skylight admitted less heat and solar radiation than traditional skylights when the sun was overhead (near-normal zenith angles). However, this advantage diminished when the sun was lower in the sky (high zenith angles). Furthermore, the study showed that increasing the site's latitude resulted in higher total daily solar heat gains for the domed skylight compared to the flat skylight, with increases of 3-9% in summer and up to 232% in winter. [14].

[5] conducted a study to determine the applicability of existing empirical models for calculating ventilation rates in multi-story buildings, aiming to assess their suitability for this purpose. By contrasting the outcomes of current empirical approaches with on-site data and CFD simulation results, the study revealed that single-zone building models are unsuitable for multi-story buildings, as they cannot account for the differences in ventilation rates across various building zones. This highlights the need for continued research and development of empirical and analytical methodologies that cater specifically to the complexities of multi-story buildings. A parametric study by [15] examined the relationship between façade design and natural ventilation in high-rise residential buildings in Singapore. Using an integrated approach that linked BES and CFD, the researchers assessed the ability of natural ventilation to achieve thermal comfort in these buildings, identifying key design factors that influence ventilation performance. Using a combined BES and CFD approach, [15] investigated the effects of various façade designs and ventilation systems on the indoor thermal comfort of naturally ventilated residential units in Singapore, providing insights into optimizing design elements for improved thermal comfort. To overcome the limitation of built-in network airflow models in building energy simulations, which cannot provide detailed airflow velocity data for interior spaces, a linked method was employed. This approach involved developing a data exchange interface to facilitate the transfer of data between Computational Fluid Dynamics (CFD) software and building energy simulation software (ESP-r), enhancing the accuracy and detail of the thermal comfort study's results.

In summary, the growing prevalence of multi-story buildings, the need for energy efficiency, the potential of natural ventilation to reduce energy consumption in cooling-dominated regions, and the challenges of effectively integrating natural ventilation into building design highlight the importance of utilizing appropriate tools to evaluate and predict natural ventilation during the design phase of multi-story buildings in these regions. The existing literature reveals that various methods have been employed to assess the performance of natural ventilation, emphasizing the need for a comprehensive approach to optimize its benefits. Although existing research has made progress in understanding natural ventilation, it mostly focuses on specialized topics, and its applicability throughout the building design process remains unclear. By understanding the advantages and limitations of each assessment method and considering the design's resources and needs, a cohesive connection can be made between the approaches and design stages. This integrated strategy will facilitate the effective implementation of natural ventilation, reducing energy consumption while preserving occupant thermal comfort.

This paper introduces a model for seamlessly integrating natural ventilation analysis tools throughout the building design process. Initially, the paper explores the unique obstacles encountered when designing natural ventilation systems for multi-story buildings. Following this, it presents a comprehensive literature review that identifies prevalent methods for analyzing natural ventilation, with a specific focus on studies examining its efficacy in cooling-dominated climates and multi-story structures, ensuring a targeted and manageable dataset. The paper then provides an assessment of the advantages and limitations of existing techniques used to evaluate natural ventilation performance. This evaluation is a crucial step in establishing a meaningful connection between the techniques and the design process, and it will aid in selecting the most suitable tools and identifying potential areas for improvement. Ultimately, this study proposes a model for the natural ventilation design process, informed by the benefits and drawbacks of the analysis techniques, to guide the effective integration of natural ventilation into building design.

2.0 Methodology

Wind and buoyancy are the key factors driving natural ventilation in both low-rise and high-rise buildings. However, designing natural ventilation systems for structures poses a significant challenge due to the amplified pressure differentials caused by wind and buoyancy at greater heights. As building height increases, so do wind pressure and speed, resulting in a wider range of pressure fluctuations on the building's facade, which complicates the design process [16]. Figure 1 depicts the atmospheric boundary layer, demonstrating how wind speed increases with height. This illustration reveals that wind pressure loading on a building varies dramatically with elevation, with upper floors experiencing significantly higher wind pressure loads than lower levels. Consequently, natural ventilation design in high-rise buildings is more complex due to the increased wind pressure on higher floors, requiring careful consideration of opening size and shape. Additionally, semi-open areas like balconies on upper floors may become unusable during strong winds due to excessive exposure. To address this challenge, designers have suggested alternative strategies, such as double skin façades, to reduce the impact of wind pressure [17].

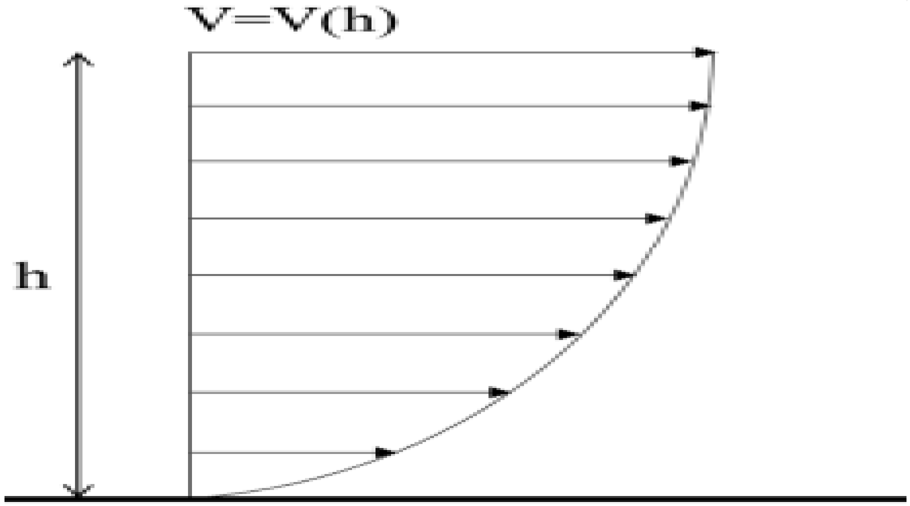

Figure 1: A diagrammatic representation of the atmospheric boundary layer

Buoyancy forces are generated when there is a difference in temperature and height between the air entering and leaving a space. In high-ceilinged areas and structures with vertical channels, like chimneys, the height of the space plays a crucial role in determining the pressure differences driven by buoyancy. According to Etheridge, natural ventilation systems in tall buildings can be categorized into three main groups [14]. The first group consists of buildings where every floor is fully enclosed, and floor openings are not linked to vertical spaces. In these cases, buoyancy forces create a manageable pressure difference, similar to that found in low-rise buildings. However, in high-rise buildings with large central voids and internal openings, the pressure difference becomes a concern. These buildings tend to rely on wind for natural ventilation, and their height plays a significant role in determining the overall pressure difference generated by buoyancy forces, causing the building to behave like a single, unified system. The pressures drop-in lower units can become so pronounced that it requires an unacceptably high force to open windows. To address this issue, [18] suggests dividing the building into separate segments, which helps to reduce the excessive pressure difference caused by buoyancy forces in buildings with central voids. This segmentation approach is similar to how low-rise buildings function, where each segment operates independently, isolated from the others.

2.1 Computational analysis

The literature suggests several simulation techniques, including standalone CFD and CFD combined with multi-zone models or BES programs. Most studies analyzed in this paper employed a combined approach, using CFD alongside experimental measurements. This coupled method has three primary applications:

1. Visualizing and exploring experimental data parameters
2. Providing insights into airflow physics that are hard to obtain experimentally
3. Validating CFD models for future studies with similar goals.

This section focuses on the combined simulation methods, deviating from the experimental methods section. Computational Fluid Dynamics (CFD) is used to directly calculate the fluid dynamic parameters that govern airflow movement by solving the fundamental Navier-Stokes equations, which describe the behavior of fluids in motion [19][20]. This simulation method offers a comprehensive analysis of airflow patterns in and around buildings, but it comes at a high computational cost. CFD provides detailed information on temperature, pressure, air velocity, and particle concentration distributions in the studied area, whether inside or outside. Nevertheless, the accuracy of CFD results relies on the quality of the grid, proper boundary conditions, and the reasonableness of assumptions made in the model, all of which impact the reliability of the results. Extensive research has verified the accuracy of CFD as a predictive tool, confirming its dependability across various contexts [21]. Furthermore, a detailed examination of CFD's role in simulating wind-induced natural ventilation in buildings is presented, highlighting the method's effectiveness. CFD has been extensively used by numerous researchers as the sole method for investigating natural ventilation, and its popularity continues to grow in airflow-related studies. A case in point is [22] who utilized CFD to examine natural ventilation in a multi-story residential building's courtyard, showcasing its effectiveness in this area of research. The study's outcomes highlighted the benefits of courtyard design in enhancing natural ventilation and air circulation in multi-story buildings, especially in warm and humid environments. A follow-up study employed the same approach to examine natural ventilation techniques in subtropical areas, further expanding the understanding of this concept [22]. The study suggested that while general guidelines can be useful, more sophisticated methods should be employed as a design evolves. Furthermore, [18] investigated a innovative double-skin façade design for high-rise office buildings in hot and humid climates using CFD. CFD was utilized to explore the feasibility of natural ventilation in a multi-story office building and evaluate the impact of airflow patterns. The study found that the double-skin façade design proposed can achieve suitable interior thermal conditions. Additionally, CFD has been employed to investigate the impact of balconies on natural ventilation, examining their role as a façade design element and how they influence airflow [23]. The application of CFD in the reviewed papers showcases its versatility in simulating various topics, design alternatives, and operational scenarios, including diverse temperature and wind speed conditions. CFD simulations can provide a comprehensive range of flow characteristics and quantities, which is precisely what makes CFD an invaluable tool for designers, enabling them to conduct in-depth analysis and informed design decisions.

Several studies on natural ventilation, particularly in multi-story buildings, explored the combination of CFD with multi-zone and Building Energy Simulation (BES) methods. Researchers used CFD in conjunction with building thermal analysis to investigate daytime and nighttime cooling ventilation. CFD modeling predicted airflow patterns, which were then used to establish boundary conditions for the building thermal analysis. Finally, Fanger's comfort model was applied to evaluate the building's thermal comfort, based on the results of both thermal analysis and CFD simulation.

Figure 2 shows the approach that Carrilho da Graça et al., [24]

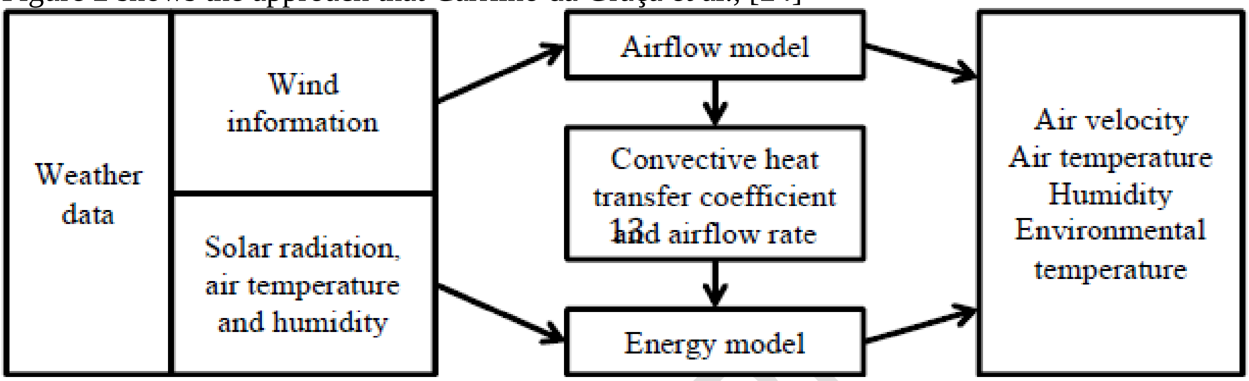

Figure 2: Diagram of the coupled strategy [24]

2.3 Governing equation

The governing equations are discretized using an explicit Finite Volume Method. The Fractional Step Method is employed to resolve the velocity-pressure coupling. By utilizing the enthalpy method, a fixed grid can be used instead of tracking the interface, simplifying the process. Additionally, the momentum source term, which accounts for the presence of solids, only needs to be included in control volumes containing both liquid and solid, rather than in volumes with only solids. The coefficient (S) of the source term assumes a particular form based on the approximation used to describe the flow behavior in the mixed-phase "mushy zone". While the solid-liquid interface should theoretically have zero width at constant phase change temperature, our simulations impose a minimum width of one control volume. Therefore, the physical formulation of the source term is relatively unimportant, as long as it successfully reduces velocity to zero in predominantly solid control volumes and becomes negligible in purely liquid volumes.

2.3.1 Continuity Equation

Governs the conservation of mass and ensures that the rate of mass flow into a control volume equals the rate of mass flow out.

$\nabla \cdot \mathbf{u} = 0$ (1)

Here, $\mathbf{u} = (u, v, w)$ is the velocity vector

2.3.2 Mass Equation

$\frac{\partial p}{\partial t} + \nabla.(\rho u) = Sm$ (2)

Where,

ρ is the fluid density

t is the time

$\boldsymbol{u}$ is the flow velocity vector field

$\boldsymbol{S}$-m is a Constant

2.3.3 Momentum Equations:

Describes the conservation of momentum for each direction (x, y, z).

In the x-direction:

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}\,v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+v\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+gx \quad (3)$$

In the y-direction

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+v\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+gy \quad (4)$$

In the z-direction

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}v\frac{\partial u}{\partial z}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+v\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+gz \quad (5)$$

$$\frac{\partial u}{\partial x}+(u.\nabla)\,u=-\nabla p+\mu\nabla 2u+\rho g \quad (6)$$

These equations account for the effects of pressure, viscosity (μ), and external forces like gravity(g)

Where,

P is the pressure

ρ is the air density

V is the kinematic vicsosity

gx,gy,gz, are the components of the gravitational acceleration

2.3.4 Energy equation

$$\frac{\partial(\rho H)}{\partial t}+\frac{\partial}{\partial xj}(\rho * uj * cp * \mathbf{T})=\frac{\partial \rho}{\partial t}\left(\lambda.\frac{\partial T}{\partial xj}\right)+\mathbf{S_E} \quad (8)$$

Where,

ρ is the mass density

$\mathbf{S_E}$ is the source term

T is the temperature

Cp is the specific heat capacity at constant pressure

2.3.5 Navier stokes equation

$$\left(\rho\frac{\partial ui}{\partial t}+uj\frac{\partial ui}{\partial xj}\right)=-\frac{\partial p}{\partial xi}+\mu\frac{\partial^2 ui}{\partial xj\,\partial xj}+fi \quad (9)$$

In this equation, f denotes additional body forces (per unit volume) like gravity or centrifugal force. The shear stress term ∇.T is simplified to μ∇^2v, where μ is the dynamic viscosity and ∇^2 represents the vector Laplacian operator, which describes the rate of change of the velocity field.

The systems are configured to allow air to flow through the tubes, while phase change materials are positioned in the shell section of the module. A conjugate steady-state CFD heat transfer analysis [6] has been conducted to study the flow and temperature distribution of the heat transmission fluid within the system. This analysis enables the optimization and evaluation of geometrical and flow parameters, as well as the solidification properties of the phase change materials, under specific boundary conditions.

To enhance the performance of building structures, a comparative analysis was performed using CFD research. The study employed ANSYS code to investigate various CFD models with different configurations, including tubes with heat transfer fluid (HTF) and pins or fins, as well as phase change material (PCM) surrounding the tube. The CFD-DEM model enabled the simulation of complex fluid-solid or fluid-particle interactions, allowing for the evaluation of different design configurations, such as fin arrangements, to optimize building structure efficacy. A typical CFD-DEM model utilizes the Discrete Element Method (DEM) to track the motion of discrete particles or solids by applying Newton's laws of motion to each particle. In contrast, the flow of continuous fluids is governed by the locally averaged Navier-Stokes equations, solvable by conventional CFD methods. Newton's third law provides a robust framework for modeling the interactions between the fluid and solid phases, capturing the equal and opposite forces exchanged between them.

This is the area's geometry used in the simulation. It makes up the room and the site plan. Figure 3 displays the cross-ventilated building model that was utilized for this model. The dimensions of the computational model were Area × Loop Length = 1200 mm × 260 mm. The reference length scale H was the building height. The back of both windward walls had two holes built, measuring 120 mm by 40 m (height by breadth). The site's environmental conditions include a temperature range of 28.5 to 40°C, relative humidity between 35% and 90%, and wind speeds varying from 4.0 to 7.0 m/s. For the computational study's worst-case scenario, the minimum wind speed of 4.0 m/s was used as the maximum value. The governing equations are then discretized and solved for each subdomain after meshing. This study utilized ANSYS Fluent for simulations, which employs the finite volume method to discretize the equations.

2.3.6 Boundary conditions.

For a comparative analysis with experimental data, CFD results for two wind direction angles, θ = 0° and θ = 35°, were selected from the TPU database. The boundary conditions for the external domain's planes are adjusted based on their normal direction, denoted as n. Specifically, an inlet boundary condition is applied if the dot product of the inlet velocity (U_in) and n is positive, an outflow boundary (with zero static pressure) if the dot product is negative, and a symmetry condition (with zero normal velocity and gradients) if the dot product is zero. To replicate the velocity and turbulence profiles from wind tunnel studies, an Atmospheric Boundary Layer (ABL) based on the logarithmic law is applied to the inlet patches, which determines the inlet velocity and turbulence variables (k and ∈).

$$|U_{in}| = \frac{U^*}{K}\ln\left((z+z_0)/z_0\right) \quad \text{where } K = \frac{U^{*2}}{\sqrt{C_\mu}}, \in = \frac{U^{*3}}{K(z+z_0)} \qquad (7)$$

The equation includes the friction velocity (U*), von Karman's constant (κ) of 0.41, turbulent viscosity coefficient (C_μ) of 0.09, vertical coordinate (z) with z = 0 representing the ground, and surface roughness height (z_0) of 0.04 m. The friction velocity (U*) is estimated using this equation.

$$U^* = k\frac{U_{ref}}{\ln((z+z_0)/z_0)} \tag{8}$$

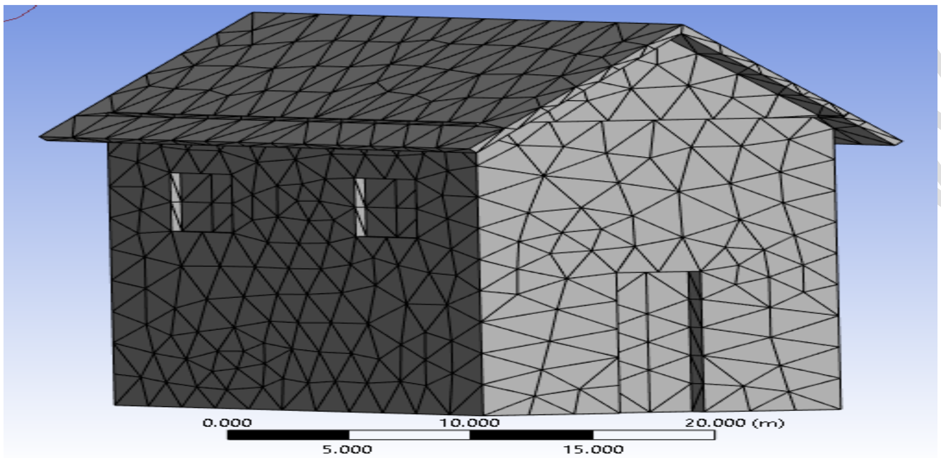

Figure 3: Mesh geometry domain

3. Result and Discussion

3.1 Air quality analysis

A CFD analysis was performed to assess the air characteristics in a lecture room. The initial simulation showed an air change rate of 3.1 per hour at a low wind velocity of 0.05 m/s, which increases with higher wind velocities. In contrast, Ansys Fluent 2023 simulations revealed a higher average air change rate of 4-6 per hour in a lecture room with moderate wind velocities (5-20 m/s), as depicted in Figure 5. The computational analysis of air quality yielded important insights, with simulation results indicating that areas with high population density and traffic congestion had the highest concentrations of particulate matter, as illustrated in Figure 4. This suggests that human activities are major contributors to air pollution in urban areas. Furthermore, the analysis revealed that ventilation rates in buildings significantly impact indoor air quality, with buildings having higher ventilation rates showing lower levels of various pollutants. This underscores the crucial role of thoughtful building design and ventilation strategies in preserving excellent indoor air quality. Additionally, the research revealed that air purifiers and filters can substantially decrease indoor pollutant levels, but their efficacy varies based on factors like filter type, airflow rate, and pollutant type. Finally, the computational analysis demonstrated that combining natural ventilation with HVAC systems can enhance air quality while reducing energy consumption, indicating that a hybrid approach to building design and operation can yield numerous benefits for both occupants and the environment.

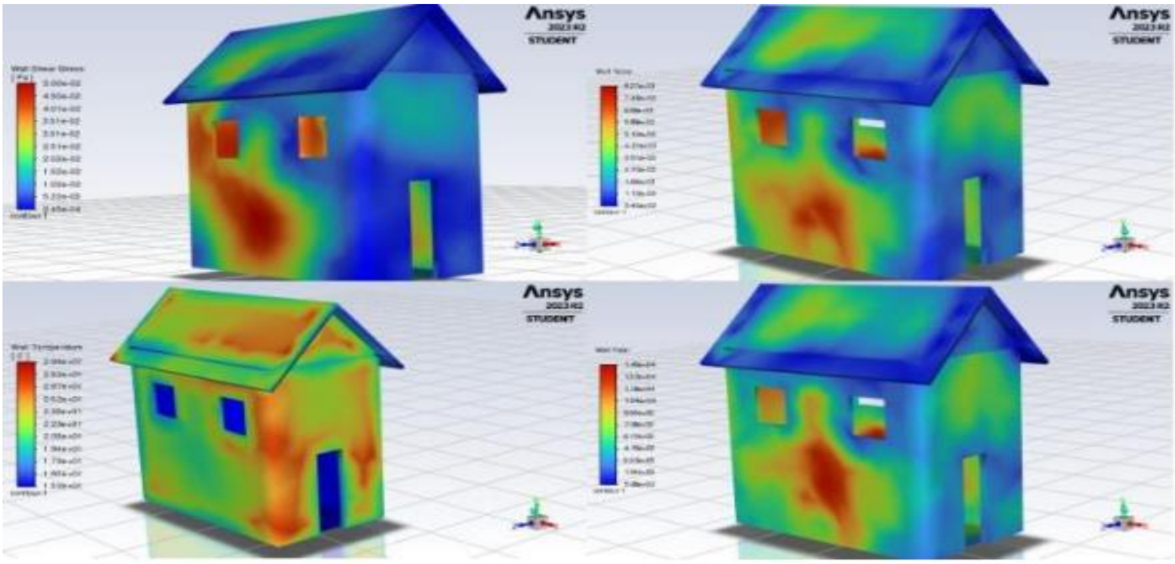

Figure 4 Velocity streamlines of wind in the mechanical lecture room under different inlet velocity conditions

3.1.2 Airflow and temperature distribution in the lecture hall

Figure 5 illustrates that air velocities are more pronounced near the windows, consistent with Bernoulli's principle, which states that airflow slows down and pressure increases as it enters the room through the windows and expands into the space. Notably, the room's core and perimeter regions show negligible air circulation. The initial configuration, where air enters from the side with two windows, may not be ideal for occupants in those areas, leading to stagnant air. However, when the airflow enters from the sides with three windows, a greater portion of the space experiences noticeable air circulation, improving ventilation.

The stratified distribution of air temperature over the entire space is shown in Figure 6. Interestingly, the side with three windows has the lowest temperature, and the areas with two windows have the highest temperature. The higher convective heat transfer close to the three-window side and the increased velocities brought on by the venturi effect are the causes of this phenomenon. As a result, the areas nearest the three windows get more ventilation than the areas nearest the two windows, demonstrating how the number of windows affects the dynamics of airflow. The computational analysis of the lecture hall revealed several key findings related to flow and thermal conditions. the simulation results showed that the air flow pattern in the lecture hall was characterized by a stratified flow regime, with warmer air accumulating near the ceiling and cooler air sinking to the floor. This led to a temperature gradient of up to 2°C between the head and foot levels, which could potentially cause discomfort for occupants. Also, the analysis revealed that the ventilation system was unable to effectively remove heat gains from the occupants and equipment, leading to a rise in temperature during peak usage hours. This highlights the need for a more efficient ventilation strategy, such as the use of personalized ventilation or radiant cooling systems. This study found that the placement of supply and exhaust grilles had a significant impact on the flow and thermal conditions in the lecture hall. Specifically, the location of supply grilles near the ceiling led to a significant amount of cool air being wasted, while the placement of exhaust grilles near the floor resulted in a lack of removal of warm air. The computational analysis showed that the use of natural ventilation strategies, such as opening windows and using solar chimneys, could significantly improve the flow and thermal conditions in the lecture hall. However, this would require careful design and control to avoid overheating and discomfort during peak summer months.

Table 1 illustrates the impact of air mass flow rate on both thermal comfort and air quality, revealing a direct correlation between increased mass flow rate and improved air quality. Initially, an air mass flow rate of 2.8 kg/s achieves an air change rate of 3.10 per hour. Notably, ASHRAE guidelines recommend a ventilation rate of 7.5 cfm per person for public spaces like classrooms. Consequently, a well-ventilated system for a 100-student classroom should provide 750 cfm to maintain optimal air quality.

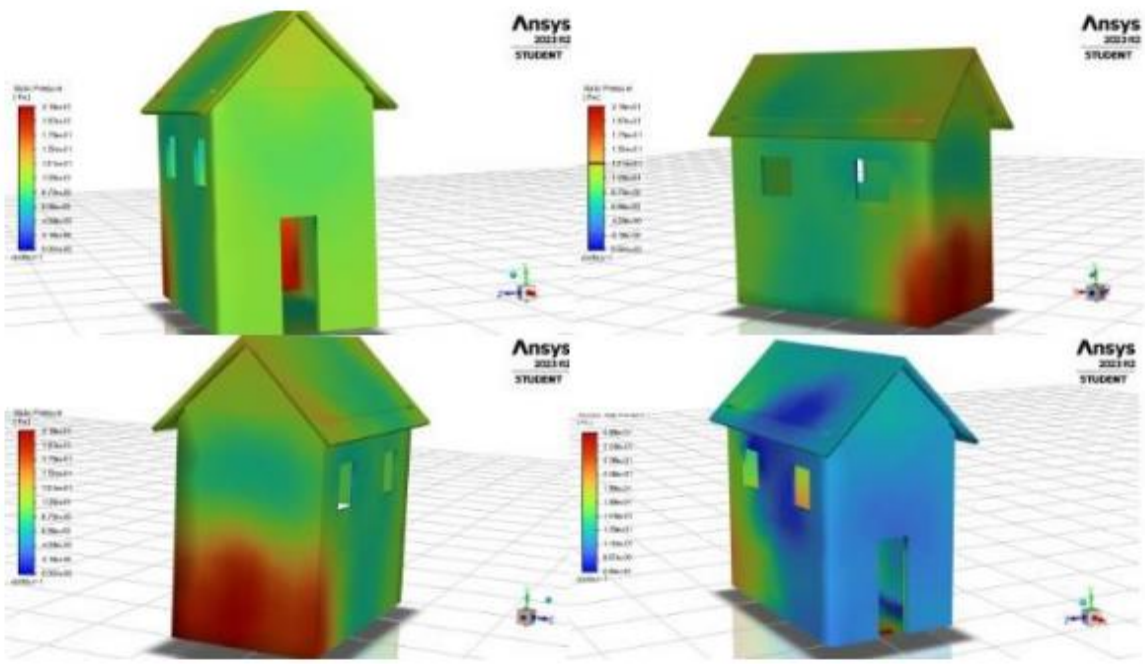

Figure 5. Pressure and temperature vector of air in the mechanical lecture room at different inlet velocities

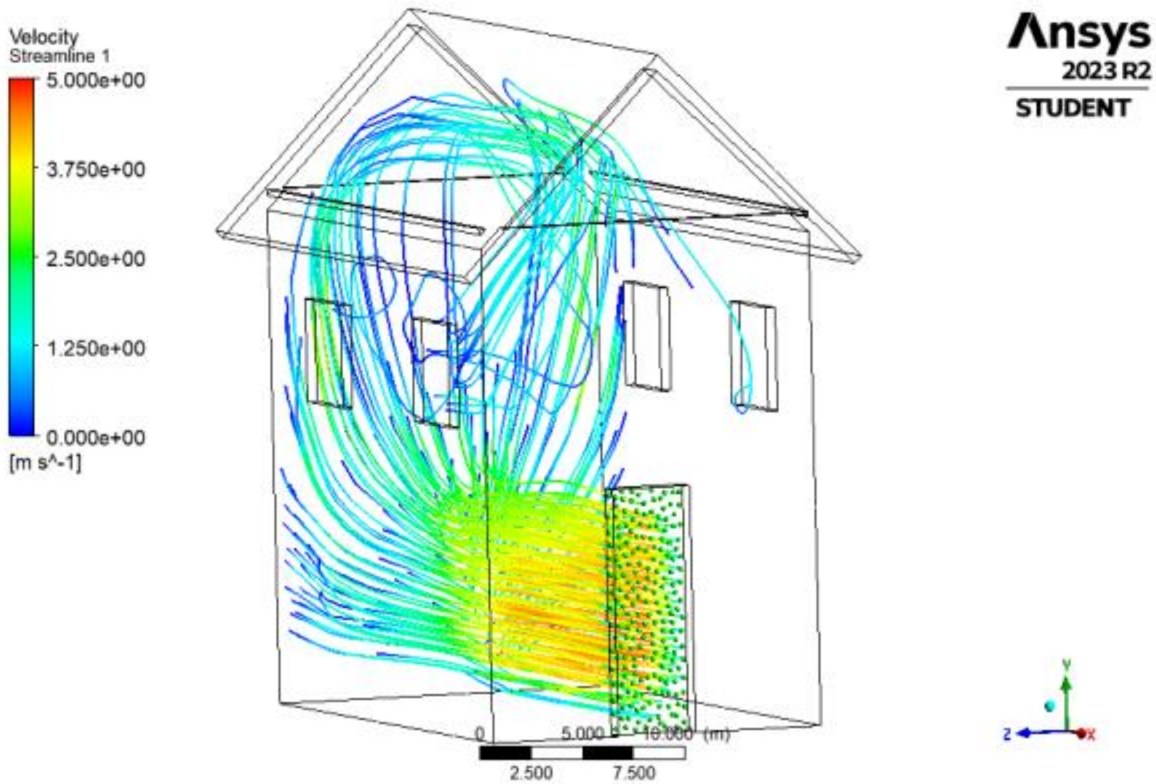

Figure 6. Streamlines of air in the mechanical lecture room.

Table 1. Mass flow rate and corresponding indoor air quality (ACH).

Wind velocity (m/s)	Mass flow (kg/s)	ACH (1/h)
0.05	2.8	3.10
0.5	28.68	31.79
1.0	57.36	63.58
1.5	86.04	95.36
2.0	114.72	127.15

3.3 Room ventilation improvement methods

These models have clearly shown that the lecture hall lacks proper ventilation, especially during low wind speeds. Various approaches are necessary to fix this. The first thing we advise doing is adding more apertures to the current ventilation system. These apertures could be vents or windows placed in strategic locations. By doing this, we can improve the amount of fresh air entering the space and promote healthier circulation. Moreover, take into account how these apertures are arranged in space. Reorienting opening pairs such that they face one another and create crossflows can greatly enhance air exchange. Furthermore, vents near the top of the room facilitate the more effective escape of hot, stale air. This buoyancy effect that occurs naturally helps to keep the interior comfortable. Envision a lecture hall that is enveloped in a harmonious circulation pattern, with warm air exiting on one side and cool air entering through strategically positioned vents. These deliberate design changes improve indoor air quality (IAQ) for speakers and attendees while also optimizing thermal comfort. A lecture room chimney model was created using Equation (7), yielding the graph seen in Figure 7. Notably, Figure 8 shows how the lecture hall's interior air quality can be improved by installing a passive ventilation system, such as a solar chimney that is the right size. It's interesting to note that the graph displays a parabolic tendency, suggesting that certain chimney diameters could not work well in the given area. As chimney size increases, the air change per hour declines beyond the peak value of 100 m³.

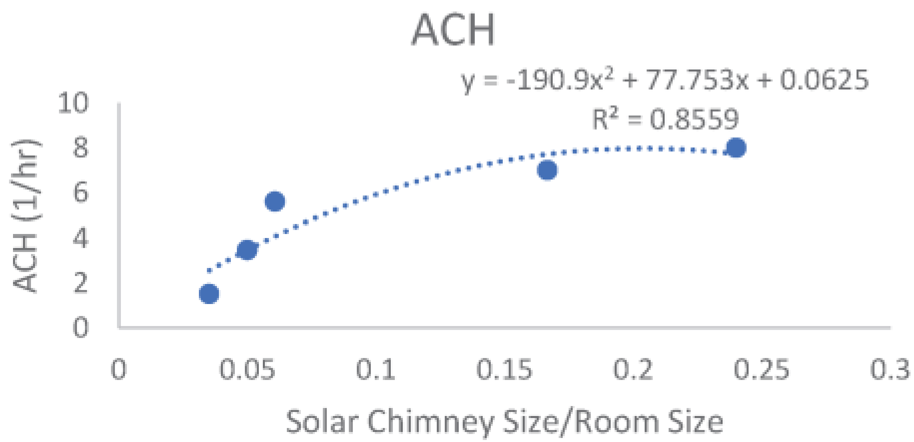

Figure 7: Thread-line graph of solar chimney model

3.4 Mechanical ventilation system's cost analysis

An evaluation of the potential energy savings contrasted mechanical and passive ventilation. Installation Cost: The installation cost includes the cost of the system's purchase as well as the installation of the fans. Table 3 displays the yearly energy cost of the mechanical ventilation system device, and Table 2 displays the energy demand data of a ventilation unit. The average cost of each fan was ₦10,000, and the installation cost was ₦1000. Figure 8 illustrates how the size of the solar chimney affects the amount of air that enters the lecture hall each hour. The relationship between solar chimney and fan expenses over time is depicted in Figure 9. Notably, annualised costs for operating mechanical ventilation (fans) grew by more than 30%, whereas expenses for passive ventilation stayed lower at about 12.5%. This demonstrates how solar chimneys are an economical and energy-efficient option for ventilation. The relative cost rise when employing fans over time is seen in Figure 10. In particular, fans caused a 63.63% increase in cost over the course of seven years in a lecture hall when compared to solar chimneys. These results highlight the advantages of passive ventilated systems over mechanical ventilation in terms of energy and cost savings. Table 4 presents the overall cost implications, while Table 5 presents cumulative expenses together with the corresponding percentage increases.

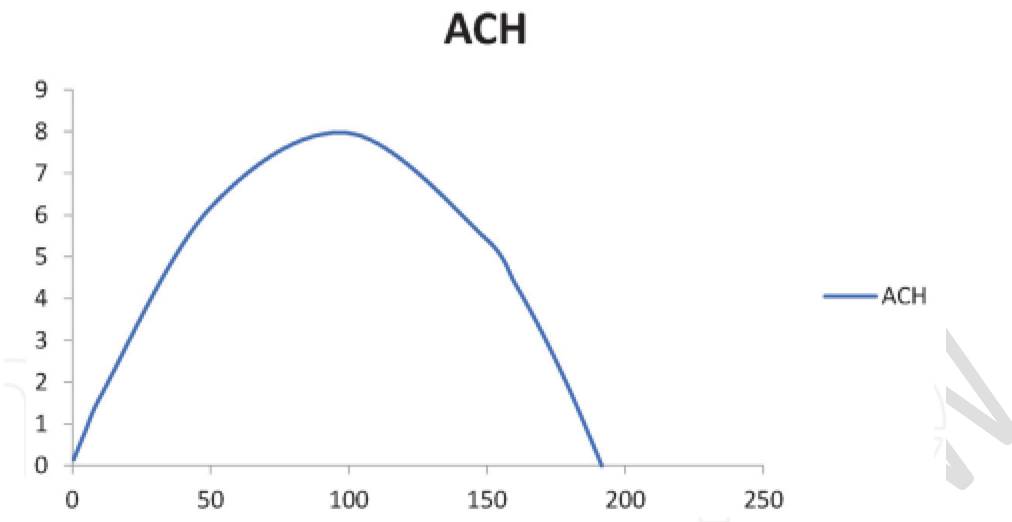

Figure 8. Effect of solar chimney size on the air change per hour of the lecture room.

Table 2. Energy demand information of a unit of the ventilation device.

system	Unit watt(w)	Number of fans	Energy demand	Hours/day	Kwh/day	Kwh/year
Fan	25	10	0.225	8	1.8	657

Table 3. Annual energy cost for the mechanical ventilation system.

System	Cost (₦) per kWh	Cost/day (₦)	Cost/month (₦)	Cost/year (₦)
1 fan	25	5	150	1825
10 fans	250	50	1500	18,250

Table 4. Total cost of ventilation systems.

	DC (₦)	IC (₦)	OC (₦)	MC (₦)	TT (₦)
Mechanical System (Fans)		46,000	16,420	4500	66,920
Passive System (Solar Chimney)	25,000	10,000		5000	40,000

Table 5. Cumulative cost and percentage increase.

Years	**MVS (₦)**	**PVS (₦)**	**Increase (₦)**	**% Increase**
1	66,920	40,000	26,920	40.23
2	87,840	45,000	42,840	48.77
3	108,760	50,000	58,760	54.03
4	129,680	55,000	74,680	57.59
5	150,600	60,000	90,600	60.16
6	171,520	65,000	106,520	62.10
7	192,440	70,000	122,440	63.63

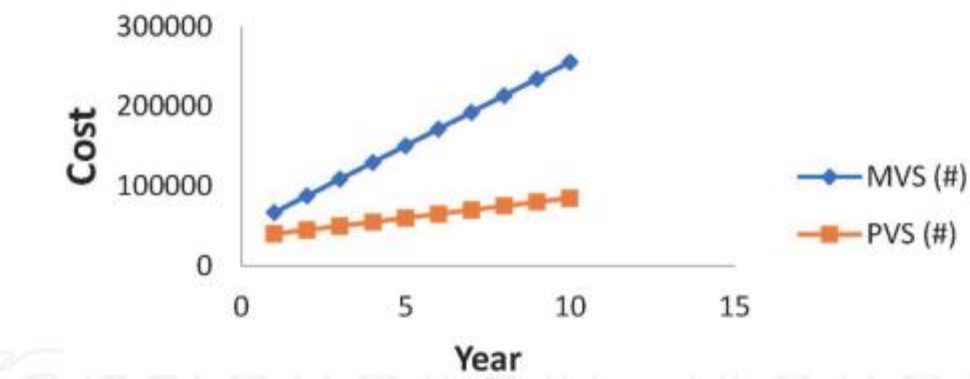

Figure 9. Graph of the cost of solar chimney and fans against number of years

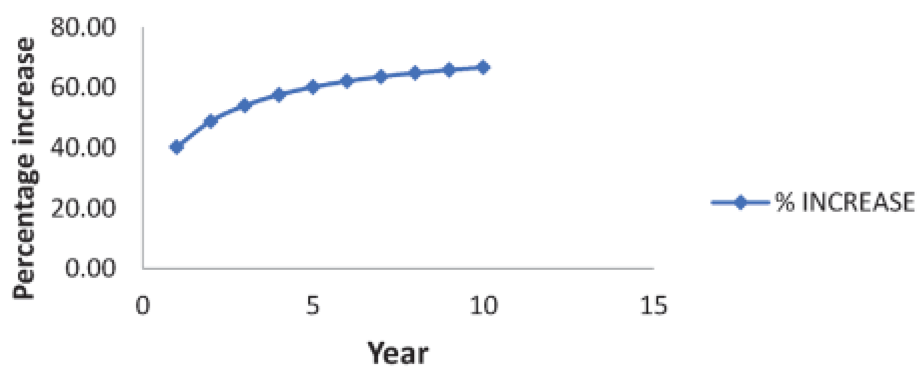

Figure 10. Graph of percentage increase against number of years.

4. Conclusion

To achieve sustainable building design, integrating natural ventilation with HVAC systems is vital. By merging the benefits of natural ventilation with the reliability of HVAC systems, designers and engineers can create buildings that prioritize energy efficiency and occupant well-being. This research demonstrates that integrated systems can significantly reduce energy consumption, improve indoor air quality, and increase occupant satisfaction. Effective implementation requires a holistic design approach that considers factors like building orientation, façade design, and smart controls to create buildings that excel in performance and sustainability. By embracing an integrated approach, the building industry can shift towards designing sustainable, resilient, and healthy buildings that minimize environmental impact while promoting occupant health, productivity, and well-being. As the sector continues to advance, it's crucial to prioritize innovative and sustainable design solutions that tackle pressing issues like climate change, energy efficiency, and human health. By combining natural ventilation and HVAC systems, we can create buildings that reduce their environmental footprint while providing a healthier, more comfortable, and productive environment for occupants. This research seeks to advance sustainable building design practices that balance human needs with environmental stewardship, ultimately contributing to a healthier planet and its inhabitants.

References

[1] M. H. Ghadiri, N. Lukman, N. Ibrahim, and M. F. Mohamed, “Computational analysis of wind-driven natural ventilation in a two sided rectangular wind catcher,” *Int. J. Vent.*, vol. 12, no. 1, pp. 51–61, 2013, doi: 10.1080/14733315.2013.11684002.

[2] O. O. Alabi, O. J. Gbadeyan, A. Bala, G. O. Ogunsiji, and N. Deenadayalu, “Study of Combustion Characteristics of Diesel-Vegetable Oil Blends Utilizing an Industrial Fuel Burner,” *Fuel Commun.*, vol. 18, no. October 2023, p. 100104, 2023, doi: 10.1016/j.jfueco.2023.100104.

[3] O. O. Alabi, G. O. Ogunsiji, and S. A. Dada, “Performances evaluation of blended alternative refrigerant in vapour compression refrigeration system.,” *Fed. Trend Sci. Technolojy Journal.*, vol. 8, no. 2, pp. 37–44, 2023.

[4] J. M. Gimenez, F. Bre, N. M. Nigro, and V. Fachinotti, “Computational modeling of natural ventilation in low-rise non-rectangular floor-plan buildings,” *Build. Simul.*, vol. 11, no. 6, pp. 1255–1271, 2018, doi: 10.1007/s12273-018-0461-9.

[5] C. Alkalah, S. Omrani, H. Garcia, C. Veronica, and R. Bianca, “Natural ventilation in multi-storey buildings: Design process and review of evaluation tools.,” vol. 19, no. 5, pp. 1–23, 2016, doi: 10.1016/j.buildenv.2017.02.012.This.

[6] A. O. Adeaga, O. O. Alabi, and S. A. Akintola, “Experimental investigation of the potential of liquified petroleum gas in vapour compression refrigeration system.,” *LAUTECH J. Eng. Technol.*, vol. 17, no. 1, pp. 1–7, 2003, doi: http://www.laujet.com/index.php/laujet/article/view/544.

[7] Y. Peng, Y. Lei, Z. D. Tekler, N. Antanuri, S. K. Lau, and A. Chong, “Hybrid system controls of natural ventilation and HVAC in mixed-mode buildings: A comprehensive review,” *Energy Build.*, vol. 276, pp. 0–1, 2022, doi: 10.1016/j.enbuild.2022.112509.

[8] W. Guo, X. Liu, and X. Yuan, “Study on Natural Ventilation Design Optimization Based on CFD Simulation for Green Buildings,” *Procedia Eng.*, vol. 121, pp. 573–581, 2015, doi: 10.1016/j.proeng.2015.08.1036.

[9] M. Gil-Baez, Á. Barrios-Padura, M. Molina-Huelva, and R. Chacartegui, “Natural ventilation systems in 21st-century for near zero energy school buildings,” *Energy*, vol. 137, pp. 1186–1200, 2017, doi: 10.1016/j.energy.2017.05.188.

[10] A. Mukhtar, M. Z. Yusoff, and K. C. Ng, “The potential influence of building optimization and passive design strategies on natural ventilation systems in underground buildings: The state of the art,” *Tunn. Undergr. Sp. Technol.*, vol. 92, no. July, 2019, doi: 10.1016/j.tust.2019.103065.

[11] C. H. Lim *et al.*, “Design configurations analysis of wind-induced natural ventilation tower in hot humid climate using computational fluid dynamics,” *Int. J. Low-Carbon Technol.*, vol. 10, no. 4, pp. 332–346, 2015, doi: 10.1093/ijlct/ctt039.

[12] M. Aram and O. Abessi, “Optimal design of green buildings using computational fluid dynamics and climate simulation tools,” *Int. J. Environ. Sci. Technol.*, vol. 17, no. 2, pp. 917–932, 2020, doi: 10.1007/s13762-019-02403-6.

[13] B. Wang and A. Malkawi, “Design-based natural ventilation evaluation in early stage for high performance buildings,” *Sustain. Cities Soc.*, vol. 45, pp. 25–37, 2019, doi: 10.1016/j.scs.2018.11.024.

[14] F. Sher, H. Sadiq, P. Mert Cuce, T. Guclu, and A. B. Besir, “Sustainable ventilation strategies in buildings: CFD research 1 2 Erdem Cuce a,” pp. 1–29, 1800.

[15] B. Chenari, J. Dias Carrilho, and M. Gameiro Da Silva, “Towards sustainable, energy-efficient and healthy ventilation strategies in buildings: A review,” *Renew. Sustain. Energy Rev.*, vol. 59, pp. 1426–1447, 2016, doi: 10.1016/j.rser.2016.01.074.

[16] M. Mora-Pérez, I. Guillen-Guillamón, G. López-Patiño, and P. A. López-Jiménez, “Natural ventilation building design approach in mediterranean regions-a case study at the valencian coastal regional scale (Spain),” *Sustain.*, vol. 8, no. 9, 2016, doi: 10.3390/su8090855.

[17] Z. Soleimani, J. K. Calautit, and B. R. Hughes, “Computational analysis of natural ventilation flows in geodesic dome building in hot climates,” *Computation*, vol. 4, no. 3, 2016, doi: 10.3390/computation4030031.

[18] M. Mora-Pérez, I. Guillén-Guillamón, and P. A. López-Jiménez, “Computational analysis of wind interactions for comparing different buildings sites in terms of natural ventilation,” *Adv. Eng. Softw.*, vol. 88, pp. 73–82, 2015, doi: 10.1016/j.advengsoft.2015.06.003.

[19] F. Xu, S. Xu, U. Passe, and B. Ganapathysubramanian, “Computational study of natural ventilation in a sustainable building with complex geometry,” *Sustain. Energy Technol. Assessments*, vol. 45, no. March, p. 101153, 2021, doi: 10.1016/j.seta.2021.101153.

[20] O. O. Alabi, O. A. Adeaga, and S. A. Akintola, “Numerical Modeling and Investigation of Flow of Incompressible Non-Newtonian Fluids through Uniform Slightly Deformable Channel,” *2023 Int. Conf. Sci. Eng. Bus. Sustain. Dev. Goals*, vol. 1, no. 1984, pp. 1–6, 2020, doi: 10.1109/SEB-SDG57117.2023.10124471.

[21] Z. A. Elhassan, “Energy consumption performance using natural ventilation in dwelling design and CFD simulation in a hot dry climate: A case study in Sudan,” *Front. Built Environ.*, vol. 9, no. March, pp. 1–14, 2023, doi: 10.3389/fbuil.2023.1145747.

[22] J. C. Salcido, A. A. Raheem, and R. R. A. Issa, “From simulation to monitoring: Evaluating the potential of mixed-mode ventilation (MMV) systems for integrating natural ventilation in office buildings through a comprehensive literature review,” *Energy Build.*, vol. 127, pp. 1008–1018, 2016, doi: 10.1016/j.enbuild.2016.06.054.

[23] H. Zhang *et al.*, “A critical review of combined natural ventilation techniques in sustainable buildings,” *Renew. Sustain. Energy Rev.*, vol. 141, no. January, p. 110795, 2021, doi: 10.1016/j.rser.2021.110795.

[24] R. Z. Homod and K. S. M. Sahari, “Energy savings by smart utilization of mechanical and natural ventilation for hybrid residential building model in passive climate,” *Energy Build.*, vol. 60, no. 2013, pp. 310–329, 2013, doi: 10.1016/j.enbuild.2012.10.034.