



APPLICATION OF COMPUTATIONAL FLUID DYNAMICS AND OPEN-SOURCE TECHNOLOGIES FOR INDOOR THERMAL COMFORT MODELING: A SYSTEMATIC REVIEW AND EVIDENCE SYNTHESIS OF THE LAST DECADE

Keny Montalvo Morales¹, Henry Javier-Ninahuman², Grimaldo Quispe³,
Abdallah Elshawadfy Elwakeel⁴, John Espita⁵, Jader Rodriguez⁶, Edwin Villagran*⁷

^{1,3}Universidad Nacional Autónoma Altoandina de Tarma, Junín, Perú.

²Universidad Tecnológica del Perú.

⁴Agricultural Engineering Department, Faculty of Agriculture and Natural Resources, Aswan University, Aswan 81528, Egypt.

^{5,6,7}Corporación Colombiana de Investigación Agropecuaria—Agrosavia, Centro de Investigación Tibaitata, Km 14, vía Mosquera-Bogotá, Mosquera 250040, Colombia.

¹<https://orcid.org/0000-0003-4403-4360>, ²<https://orcid.org/0000-0002-3864-4181>, ³<https://orcid.org/0000-0001-6168-8935>,

⁴<https://orcid.org/0000-0002-9360-051X>, ⁵<https://orcid.org/0000-0003-2684-655X>

⁶<https://orcid.org/0000-0002-5346-092X>, ⁷<https://orcid.org/0000-0003-1860-5932>

Email: kmontalvo@unaat.edu.pe, c21259@utp.edu.pe, gquispe@unaat.edu.pe, abdallah_elshawadfy@agr.aswu.edu.eg,
jjespitia@agrosavia.co, jrodriguez@agrosavia.co, *evillagran@agrosavia.co

ARTICLE INFO

Article History

Received: February 7, 2026

Reviewed: June 5, 2026

Accepted: July 5, 2026

Published: July 31, 2026

Keywords:

Thermal comfort,
Heat transfer,
Indoor airflow,
Numerical simulation,
Ventilation,
Energy efficiency.

ABSTRACT

Over the past decade, Computational Fluid Dynamics (CFD) has become a fundamental tool for the assessment, modeling, and optimization of thermal comfort in controlled architectural environments, enabling detailed simulation of airflow behavior, heat transfer processes, and the spatial distribution of thermo-environmental variables within indoor spaces. The integration of human thermophysiological models has enhanced the predictive capability of these simulations, allowing for more realistic estimations of occupants' thermal responses and perceptual comfort. In parallel, the growth and maturation of open-source platforms have democratized access to these technologies, facilitating their implementation across multiple scales and building typologies, ranging from offices and residential buildings to vehicle cabins and urban environments. Recent studies have further addressed the evaluation of passive environmental conditioning strategies such as solar chimneys and natural ventilation indoor air quality assessment, and the adaptation of models to local climatic conditions, reflecting a transition toward more comprehensive, resilient, and sustainable design approaches. Moreover, the incorporation of artificial intelligence techniques and predictive control systems has opened new opportunities for operational automation and improved building energy efficiency. Within this context, the present systematic review synthesizes methodological advances, emerging applications, and current trends in the use of CFD and open-source platforms for indoor thermal comfort assessment in architecture, identifying their main contributions, technical limitations, and future research directions.



Copyright ©2026 by authors and Galileo Institute of Technology and Education of the Amazon (ITEGAM). This work is licensed under the Creative Commons Attribution International License (CC BY 4.0).

I. INTRODUCTION

Over the past decade, Computational Fluid Dynamics (CFD) has assumed a prominent role in the assessment of thermal comfort within controlled architectural environments due to its capacity to model, with a high level of detail, airflow patterns, heat transfer processes, and the spatial distribution of thermo-environmental variables [1].

These capabilities have enabled researchers and architectural design professionals to optimize indoor environmental conditions, contributing both to improved occupant quality of life and enhanced building energy efficiency [2]. A significant advancement in this field has been the incorporation of human thermophysiological models into CFD simulations, which has enriched the interpretation of thermal comfort by integrating the dynamic interaction between the human body and its immediate thermal surroundings. This approach has enabled more realistic estimations of users' physiological and perceptual responses to microclimatic variations [3]. In parallel, the expansion and consolidation of open-source platforms have democratized access to these technologies, extending their implementation beyond specialized laboratories and facilitating their adoption in smaller academic and professional contexts, as well as in studies addressing urban microclimates and outdoor thermal comfort [4]. Numerous studies have leveraged the potential of CFD to analyze comfort conditions across multiple building typologies and spatial scales. In office buildings, for instance, the evaluation of temperature distribution and CO₂ concentrations has provided key evidence for the design of more efficient ventilation systems. Similarly, passive strategies such as solar chimneys have been investigated to enhance natural ventilation and reduce energy consumption [5-7]. In the transportation sector, the integration of CFD with thermophysiological models has enabled the simulation of passenger thermal comfort by accounting for variables such as airflow, temperature, and vehicle occupancy [8]. Complementarily, recent systematic reviews have documented the evolution of thermal comfort models, highlighting the need to incorporate contextual factors including climatic conditions, environmental characteristics, and user profiles to achieve more accurate and adaptive predictions [9]. Urban growth has further stimulated interest in city-scale microclimate studies, where CFD has proven to be a robust tool for analyzing urban thermal behavior and supporting the development of sustainable planning strategies [10]. More recently, the integration of artificial intelligence techniques into thermal management systems has opened new opportunities for predictive control, operational automation, and the simultaneous optimization of comfort and energy consumption through the use of real-time data [11], [12]. Within this context, the present study conducts a systematic review aimed at synthesizing the most significant advances in the application of CFD and open-source tools for thermal comfort assessment in controlled architectural environments over the past decade, identifying methodological trends, technical contributions, current limitations, and future research directions.

II. METHODOLOGY

II.1 STUDY DESIGN

The present study corresponds to a systematic literature review aimed at identifying, analyzing, and synthesizing scientific evidence related to the application of Computational Fluid Dynamics (CFD) and open-source platforms for thermal comfort assessment in controlled architectural environments. The review was conducted in August 2025 and developed in strict accordance with the methodological guidelines established by the PRISMA statement [13], with the objective of ensuring transparency, reproducibility, and traceability throughout the processes of searching, screening, evaluation, and synthesis of the included studies [14]. The flow diagram describing the identification and screening stages of the articles is presented in Figure 1.

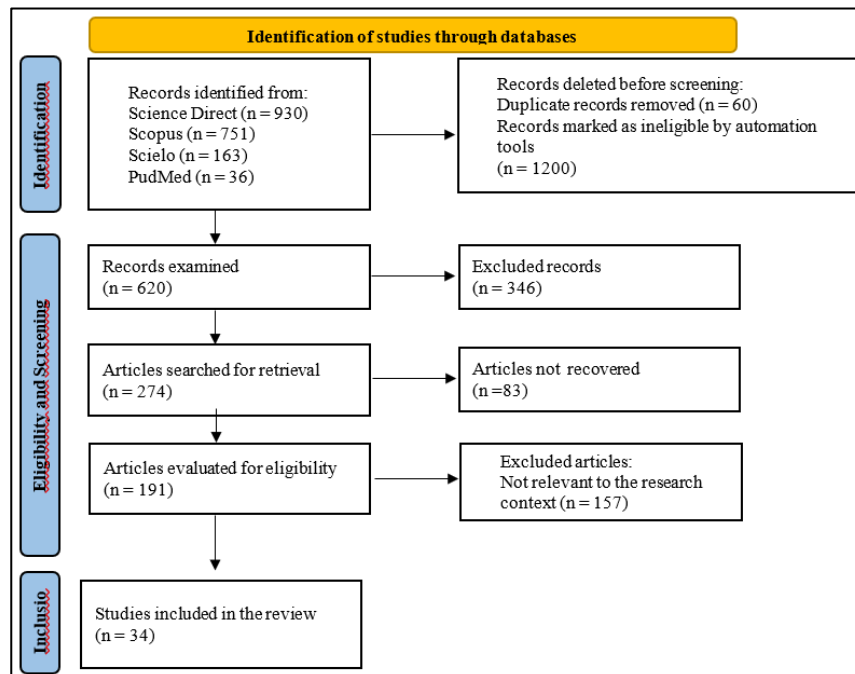


Figure 1: Development approach of the bibliometric review.

Source: Authors, (2026).

II.2 RESEARCH DESIGN

The bibliographic search strategy was carried out by consulting high-impact, multidisciplinary scientific databases, including ScienceDirect, Scopus, PubMed, and SciELO, in order to ensure the comprehensiveness and quality of the retrieved sources. To define the search terms and structure the research question, the PIO framework (Population–Intervention–Outcomes) was adopted, enabling a systematic delimitation of the study scope and facilitating the identification of relevant literature.

Population (P): indoor architectural spaces or controlled built environments in which thermal comfort and occupant well-being are assessed.

Intervention (I): application of Computational Fluid Dynamics (CFD) techniques and the use of open-source platforms or tools for the simulation, modeling, and analysis of thermal conditions and airflow behavior.

Outcomes (O): indicators of thermal comfort, energy performance, indoor environmental quality, and simulated thermo-environmental variables.

This methodological framework enabled a consistent process of data collection, critical selection, and comparative analysis of the studies, supporting the identification of trends, applications, and research gaps within the field (Table 1) [15].

Table 1: Application of the PIO framework for the definition of search equations.

PIO Category	Keywords
Population	"controlled environments", "indoor spaces", "thermal comfort zones", "buildings", "architectural environments"
Intervention	"computational fluid dynamics", "CFD", "thermal simulation", "open source tools", "energy modeling"
Outputs	"thermal comfort evaluation", "temperature analysis", "indoor climate modeling", "comfort improvement", "energy efficiency"

Source: Authors, (2026).

Prior to the selection process, inclusion criteria were established to define the thematic scope and ensure the relevance and quality of the collected evidence. Scientific articles published in English, Spanish, or Portuguese were considered, thereby broadening the geographical and linguistic coverage of the analysis. Studies focused on the application of Computational Fluid Dynamics (CFD) or open-source simulation tools for thermal comfort assessment in architectural spaces or built environments were also included. To ensure the timeliness and relevance of the findings, only studies published within the last ten years were considered. Complementarily, exclusion criteria were defined to discard works not directly related to the use of CFD or thermal comfort assessment, as well as studies addressing psychological, political, or other areas unrelated to architecture and thermo-environmental simulation. Duplicate publications, conference abstracts, opinion papers, systematic reviews, meta-analyses, and other documents that did not provide original evidence were also excluded to preserve methodological rigor and minimize bias in the synthesis of results.

II.3 STUDY POPULATION

The bibliographic search strategy was structured through the systematic combination of keywords and Boolean operators applied to the title, abstract, and keyword fields (TITLE-ABS-KEY), integrating concepts related to *thermal comfort*, *computational fluid dynamics* (CFD), and *thermal simulation*. The search string used was: ((TITLE-ABS-KEY ("thermal comfort") AND TITLE-ABS-KEY ("computational fluid dynamics" OR "CFD") AND TITLE-ABS-KEY ("thermal simulation"))). This approach enabled the precise identification of studies focused on thermal comfort assessment through thermo-aerodynamic simulations in controlled architectural environments. Using this strategy, a total of 34 scientific articles were identified across high-impact specialized databases, including ScienceDirect, Scopus, PubMed, and SciELO. These records constituted the initial corpus of literature for the screening, eligibility assessment, and subsequent qualitative synthesis, providing the theoretical and methodological foundation for the present systematic review (Figure 1).

III. RESULTS AND DISCUSSION

III.1 ANALYSIS OF THE SCIENTIFIC STRUCTURE OF THE RESEARCH FIELD

The search strategy enabled the identification of a total of 1,880 potentially relevant documents. After applying the thematic preselection criteria, the bibliographic records were exported in CSV format and processed within the RStudio environment, using the Bibliometrix package to perform bibliometric analysis and construct keyword co-occurrence networks [16]. This approach facilitated a quantitative exploration of the conceptual structure and thematic relationships present in the literature. Figure 2 presents the keyword co-occurrence map, where node size reflects the frequency of term occurrence and the links represent the strength of semantic relationships among the analyzed studies [17], [18]. The central core of the network is dominated by *computational fluid dynamics*, which acts as the main articulating axis of the field and is closely connected to terms such as *thermal comfort*, *ventilation*, *energy*, and *indoor environment*. This high density of connections highlights the consolidation of CFD as a cross-cutting methodological tool for simulating thermo-aerodynamic behavior and assessing indoor environmental performance. Cluster analysis reveals at least two major groupings: a larger and denser blue cluster associated with direct building applications including thermal comfort, ventilation, indoor air quality, and energy efficiency which represents the predominant research line; and a more compact red cluster linked to methodological developments and the computational foundations of CFD, such as numerical modeling and advanced simulation approaches. The interaction between both clusters suggests a convergence between theoretical advancements and practical applications, indicating a trend toward integrated frameworks that combine computational accuracy with sustainable architectural design solutions. Additionally, the topological structure of the network shows a high degree of interconnectivity among clusters, with multiple cross-links connecting methodological terms to architectural applications, thereby indicating a mature research field undergoing disciplinary integration. The centrality of key nodes and the proximity among subtopics demonstrate that advances in numerical modeling are directly translated into improvements in thermal comfort and energy efficiency outcomes, while the absence of isolated groups suggests low thematic fragmentation. Overall, this configuration reflects an evolution from specialized studies toward interdisciplinary approaches that integrate computational simulation, environmental performance, and sustainable design.

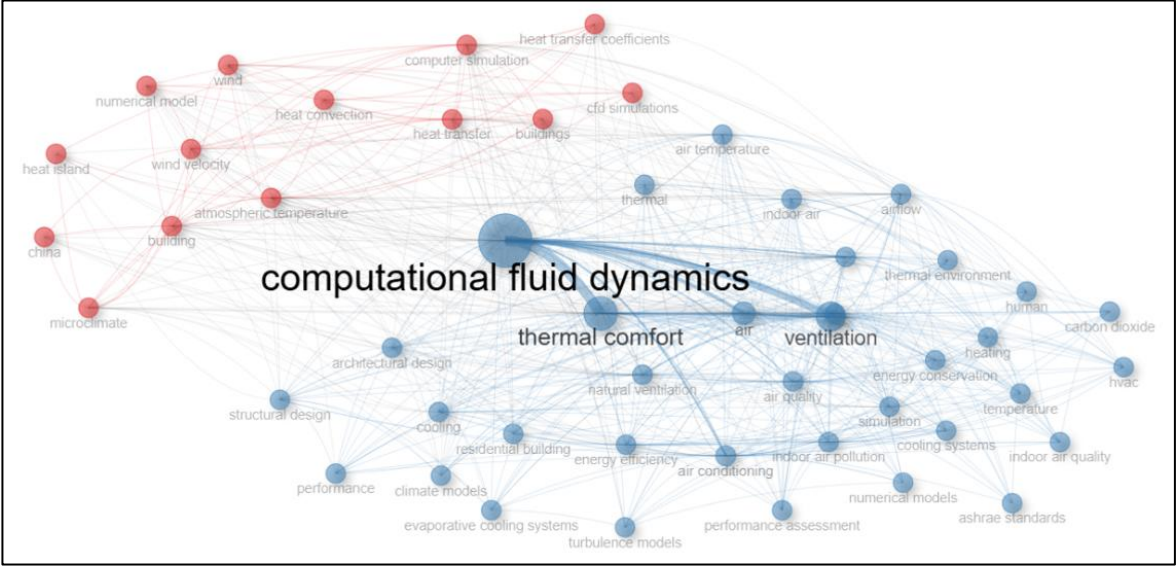


Figure 2: Keyword co-occurrence network.
Source: Authors, (2026).

III.2 COMPARATIVE SYNTHESIS OF THE INCLUDED STUDIES AND APPLICATIONS OF CFD IN THERMAL COMFORT

Table 2 summarizes the studies included in this review, organizing for each publication its specific application of Computational Fluid Dynamics (CFD), the main benefit or contribution, the evaluated parameters, and the reported conclusions. Collectively, the reviewed works demonstrate that CFD functions as a thermo-aerodynamic analysis platform capable of addressing both indoor environmental performance issues such as temperature distribution, air velocity, relative humidity, CO₂/contaminant concentrations, and comfort indices including PMV/PPD and urban microclimate and outdoor ventilation challenges, where specific indicators such as “air delay” have been proposed to characterize ventilation performance in complex urban geometries [19]. This evidence confirms a shift toward multivariable assessments, in which thermal comfort is interpreted alongside air quality and energy consumption indicators, reinforcing the relevance of CFD as a decision-support tool for architectural design. From a thematic perspective, the studies can be grouped into three predominant research lines. The first focuses on natural ventilation and passive building strategies such as solar chimneys, windcatchers, cross-ventilation, and evaporative cooling with the aim of enhancing indoor comfort while reducing reliance on mechanical systems [20-27]. The second integrates CFD with building energy simulation (BES/BPS) or envelopes and material analyses to quantify impacts on cooling/heating demand and overall efficiency [28-33]. The third research line addresses urban microclimate and outdoor thermal comfort, including urban geometry, ventilation within street canyons, and morphological solutions such as elevated buildings, emphasizing the importance of considering non-stationary conditions and realistic configurations [19], [34], [35]. From a methodological standpoint, a consistent pattern emerges: (i) widespread use of Reynolds-Averaged Navier–Stokes (RANS) models and, in specific cases, Large Eddy Simulation (LES) or unsteady simulations to capture transient phenomena; (ii) experimental validation through wind tunnel testing or in situ measurements in ventilation and pollutant dispersion studies, thereby strengthening the reliability of results [20], [36], [37] ; and (iii) increasing adoption of CFD–human thermoregulation model coupling or machine learning–based surrogate models aimed at reducing computational costs and enabling faster evaluations in design scenarios [38], [39]. Overall, the table illustrates the evolution of the field toward integrated approaches that combine numerical accuracy, empirical validation, and practical applicability to improve thermal comfort, indoor environmental quality, and energy efficiency.

Table 2: Main methodologies and findings of the compiled studies.

Study Title	CFD Implementation	Key Benefit	Variables Analyzed	Main Findings
CFD and wind-tunnel analysis of outdoor ventilation in a real compact heterogeneous urban area: Evaluation using "air delay [19].	It applies steady three-dimensional Reynolds-Averaged Navier–Stokes (RANS) models and Large Eddy Simulation (LES) to simulate outdoor urban ventilation conditions.	The study introduces a novel ventilation indicator, “air delay,” to improve the assessment of outdoor ventilation in complex urban environments. It further provides a systematic evaluation of steady RANS and LES approaches for urban ventilation analysis.	Distribution of air age and the “air delay” indicator.	The study offers a comprehensive assessment of RANS and LES approaches for evaluating ventilation conditions in a real urban setting and introduces a novel metric to support this evaluation.

CFD simulation of the wind flow under lift-up buildings using a porous approach [35].	CFD was employed to model the urban microclimate, with particular emphasis on simulating wind flow beneath elevated buildings.	The proposed method enables efficient wind-flow modeling in urban areas with elevated buildings while maintaining a balance between computational cost and accuracy, which is critical for urban thermal comfort assessment and heatwave mitigation.	Wind flow patterns, the impact of ground-floor geometry, and urban thermal comfort conditions.	The porous modeling approach in CFD has proven effective for simulating wind flow beneath elevated buildings, providing an optimal trade-off between predictive accuracy and computational efficiency.
Experimental study and CFD modelling of four-bed hospital ward with all-air wall induction unit for air-conditioning [36].	CFD modeling methods are proposed and experimentally validated to assess the ventilation and thermal performance of an all-air wall induction unit installed in a hospital ward.	The study examines a prototype induction unit designed for hospital wards, featuring a water-free system with low air supply velocity, and integrates experimental measurements with CFD simulations to enable a comprehensive performance evaluation.	Temperature distribution and CO ₂ concentration (used as a proxy indicator for airborne contaminants).	The wall induction system maintains normalized contaminant concentrations near hospital beds by promoting displacement ventilation, with low-flow air supply from the lower section of the wall corner demonstrating effective performance.
Optimization of classroom window design using CFD and the Taguchi method [40].	CFD is employed to evaluate the influence of window position, width, height, and elevation from the floor on CO ₂ concentration levels and draft risk within a classroom environment.	The study optimizes classroom window design to enhance indoor air quality and thermal comfort, employing the Taguchi method in combination with CFD to efficiently evaluate and optimize design parameters.	CO ₂ concentration levels and the risk of draft discomfort.	The study proposes optimized window design parameters aimed at reducing CO ₂ concentration levels and mitigating draft risk in classroom environments.
Assessing the impact of urban microclimate on building energy demand by coupling CFD and building performance simulation [28].	A validated CFD model is integrated with Building Performance Simulation (BPS) to quantify the impact of urban surface composition on urban microclimatic conditions and building energy demand.	The study quantifies the effects of urban design and vegetation on building energy consumption, with particular emphasis on cooling loads during heatwave events.	Building energy demand (cooling and heating loads) and urban microclimatic conditions (temperature).	Urban greening strategies significantly reduce cooling energy demand during heatwave conditions, while the composition of urban surfaces exerts a substantial influence on overall building energy requirements.
Cross-ventilation in a generic isolated building equipped with louvers: Wind-tunnel experiments and CFD simulations [20].	Steady three-dimensional Reynolds-Averaged Navier-Stokes (RANS) simulations are employed to model cross-ventilation in a building equipped with louvers, with CFD results validated against wind tunnel experiments.	The study examines the influence of louvered openings on natural ventilation performance, a critical factor for enhancing indoor air quality and achieving thermal comfort.	Airflow velocity distributions and air exchange efficiency.	The location of the openings significantly influences ventilation performance, and CFD simulations, when experimentally validated, are capable of accurately predicting cross-ventilation behavior.

Evaluation of supervised machine learning regression models for CFD-based surrogate modelling in indoor airflow field reconstruction [38].	CFD simulations are employed to generate training datasets for machine learning models aimed at predicting indoor airflow distribution, with machine learning acting as a surrogate modeling approach to reduce computational costs and accelerate predictions.	The study investigates the application of machine learning techniques to efficiently predict indoor airflow patterns, thereby reducing the computational burden associated with CFD simulations.	Airflow velocity fields and spatial distribution data.	Decision tree-based approaches and support vector regression models represent viable alternatives to neural networks for the prediction of indoor airflow patterns.
Pollutant cross-transmission in courtyard buildings: Wind tunnel experiments and computational fluid dynamics (CFD) evaluation [37].	CFD simulations are employed to assess the cross-transmission of indoor and outdoor contaminants in courtyard-type buildings, with results validated against wind tunnel experiments.	The study examines how courtyard design configurations may promote contaminant cross-transmission, highlighting a critical indoor air quality issue in residential building complexes.	Pollutant concentration levels and airflow distribution patterns.	Courtyard design configurations can significantly influence the transmission of contaminants between indoor and outdoor environments.
Sustainable ventilation strategies in buildings: CFD research [21].	CFD is employed to investigate natural ventilation strategies in buildings, particularly wind-driven ventilation and stack-effect mechanisms.	The study evaluates the potential of natural ventilation as a sustainable alternative to mechanical air conditioning, enhancing human thermal comfort and reducing overall energy consumption.	Natural ventilation, wind-induced ventilation, stack-effect ventilation, and human thermal comfort.	The article highlights the critical role of natural ventilation strategies in architectural design for enhancing human thermal comfort and promoting environmental sustainability.
Experimental and CFD Investigation on the Application for Aerogel Insulation in Buildings [33].	CFD simulations are employed to evaluate the thermal performance of aerogel insulation materials in building applications.	The study investigates the potential of aerogels as a high-performance insulation material to enhance energy efficiency in buildings.	Thermal insulation performance, aerogel materials, and heat transfer mechanisms.	The study provides valuable insights into the application of aerogel insulation systems to improve overall building energy performance.
Unsteady-state CFD simulations on the impacts of urban geometry on outdoor thermal comfort within idealized building arrays [34].	Unsteady-state CFD simulations are employed to assess the impact of urban geometry on outdoor thermal comfort conditions.	The study assesses the dynamic effects of building layout on airflow behavior and outdoor thermal comfort in urban environments, highlighting their critical importance for urban planning and sustainable design	Outdoor thermal comfort conditions, urban morphology, building configuration, and airflow distribution patterns.	The findings underscore the importance of accounting for unsteady wind conditions when evaluating urban thermal comfort.
CFD modeling of the building integrated with a novel design of a one-sided wind-catcher with water spray: Focus on thermal comfort [22].	CFD simulations are employed to model and evaluate a building integrated with an innovative single-sided wind catcher incorporating a water spray system.	The study evaluates the potential of this innovative design to enhance natural ventilation performance and improve indoor thermal comfort in buildings	Natural ventilation strategies, evaporative cooling, wind catcher design configurations, and thermal comfort performance.	The study evaluates the effectiveness of the proposed wind catcher design in enhancing indoor thermal conditions and overall thermal comfort.

Indoor Thermal Environment in Different Generations of Naturally Ventilated Public Residential Buildings in Singapore [41].	CFD is used to evaluate and compare indoor air velocities and the thermal environment in different generations of naturally ventilated public residential buildings.	The study provides information on the evolution of natural ventilation strategies in residential buildings and their impact on indoor thermal comfort.	Indoor air velocity, thermal environment, natural ventilation, residential buildings.	The research analyzes the effectiveness of natural ventilation in providing comfortable indoor environments under Singapore's climatic conditions.
CFD Evaluation of Thermal Conditioning in a House of Social Interest with a Solar Chimney Arrangement in Guanajuato, Mexico [23].	CFD is used to evaluate the thermal conditioning performance of a social housing unit that incorporates a solar chimney system.	The study evaluates the potential of solar chimneys to provide passive cooling and improve indoor thermal comfort in dwellings.	Thermal conditioning, solar chimney, natural ventilation.	The research provides information on the design and effectiveness of solar chimneys for thermal regulation in residential buildings.
Natural Ventilation in Low-Cost Housing: An Evaluation by CFD [24].	CFD is used to evaluate the performance of natural ventilation in low-cost housing.	The study investigates how natural ventilation can be optimized to improve indoor comfort in affordable housing solutions.	Natural ventilation, low-cost housing, indoor comfort.	The research provides information on the design of effective natural ventilation systems for residential buildings, particularly in the context of low-cost housing.
Energy simulation and CFD coupled analysis for the optimal operation of combined convection and radiant air conditioning considering dehumidification [29].	CFD is used to simulate thermal environments in residences, specifically those with combined radiant panels and air-conditioning systems, including the effects of dehumidification.	The combined use of radiant panels and air conditioning provides an optimal solution for thermal control in residences, balancing energy efficiency and comfort. The developed simulation tool may assist in product development and building material design.	Energy consumption, thermal comfort, relative humidity, and dehumidification effectiveness.	The study proposes a case where energy efficiency and comfort are achieved through the joint use of air conditioners and radiant panels, with a limited number of panels enabling balanced operation.
Influence of cross-ventilation cooling potential on thermal comfort in high-rise buildings in a hot and humid climate [25].	CFD simulation is used to investigate the effectiveness of natural ventilation in a high-rise building in India, together with building energy simulation. It determines wind-induced pressure distribution to evaluate indoor ventilation.	Cross-ventilation improves ventilative cooling efficiency, ensuring acceptable indoor thermal conditions even in adverse climates. It reduces cooling energy demand and electricity consumption.	Indoor thermal comfort, operative temperature, discomfort hours, natural ventilation cooling potential, night structural cooling.	Cross-ventilation significantly reduces discomfort hours and operating temperatures, providing a passive cooling solution for energy savings in high-rise buildings.
Improving Natural Ventilation Multi-Story Buildings within Hot and Dry Climates: A CFD Study of Windcatcher Performance [26].	CFD is used to study the performance of wind catchers in multi-story buildings to improve natural ventilation.	The study explores how wind catchers can enhance natural ventilation, which is crucial for energy and carbon conservation.	Wind catcher designs and their impact on natural ventilation performance in hot and dry climates.	The research contributes to strategies for improving natural ventilation through wind catchers, focusing on energy conservation.
Evaluation of passive cooling through natural ventilation strategies in historic residential buildings using CFD simulations [27].	CFD simulations are used to evaluate the efficiency of natural ventilation strategies in a historic residential structure.	Natural ventilation can save energy by reducing the need for mechanical systems in historic buildings, helping their preservation.	Efficiency of different natural ventilation approaches, thermal comfort, energy-saving potential.	The study demonstrates that natural ventilation strategies, especially cross-ventilation, can provide energy savings and maintain thermal comfort in historic buildings.

Thermal performance of an advanced smart fenestration systems for low-energy buildings [32].	CFD is used in a new model that combines ray-tracing and CFD to investigate the thermal and optical properties of smart fenestration systems.	The study explores a new adaptive fenestration system that can offer dynamic regulation of solar energy, improving building energy efficiency.	Solar transmittance, solar heat gain coefficient (SHGC), and dynamic thermal performance in summer and winter.	The proposed system can reduce solar transmittance and heat gain, resulting in better thermal performance with lower summer gains and lower winter heat losses.
Thermal Environment Analysis and Optimization for Large Space Buildings with Radiant Cooling Floors: A Case Study of Xianyang International Airport [42].	CFD is used to investigate the thermal environment of a large building and to analyze control strategies for radiant cooling floors combined with ventilation systems.	The study aims to find control strategies to achieve energy conservation and environmental improvement in large-space buildings.	Supply air temperature, cooling surface temperature, cooling capacity, and predicted mean vote (PMV).	The study determines optimal control variables to balance cooling capacity and thermal comfort.
Efficiency Assessment on Roof Geometry and Trombe Wall Shape for Improving Buildings' Heating Performance [43].	ANSYS-CFX software is used to simulate the thermal and fluid dynamic behavior of a Trombe wall system.	The study examines the impact of Trombe wall designs and roof types on building energy and operational efficiency.	Trombe wall designs, roof types, indoor climates, thermal comfort standards, energy efficiency.	The research provides information to optimize building design for improved heating performance under specific climatic conditions.
Impact of Building Envelope Materials on Energy Usage and Performance of Evaporative Cooling System in Residential Building [31].	ANSYS is used to simulate and analyze the performance of an evaporative cooling system in a residential building considering envelope materials and building orientation.	The research proposes an energy-efficient cooling system, showing that the proposed system combined with efficient envelope materials can achieve energy savings of about 35–43%.	Envelope components, building orientation, energy use, temperature reduction.	The study concludes that the proposed system reduces indoor temperature and that envelope characteristics significantly affect energy performance, leading to substantial energy savings.
Investigating Efficient Thermal Distribution in a House Room by combining Statistics with Computational Fluid Dynamics [44].	CFD analysis combined with statistical methods is used to determine cost-effective configurations for heating systems by investigating thermal distribution.	The study explores energy efficiency by identifying optimal heating configurations.	Theoretical temperatures at different indoor points.	The study uses numerical simulation to determine efficient thermal distribution.
Impact of Wind Pressure Coefficients on the Natural Ventilation Effectiveness of Buildings through Simulations [45].	Building energy simulation (BES) models are used to evaluate the effectiveness of natural ventilation using wind pressure coefficients as boundary conditions.	The research quantifies when outdoor airflow can be used as a cooling strategy to achieve indoor thermal comfort.	Airflows, cooling loads, heat rates.	Wind pressure coefficients are essential for evaluating airflow and ventilation effectiveness.
Significance of external wind conditions on the convective heat transfer coefficients (CHTC) and energy performance in multi-zone high-rise buildings [30].	A coupled BES-CFD method is employed to improve the accuracy of convective heat transfer coefficient calculations in high-rise buildings.	Improved predictions of CHTC and better understanding of wind influence on building energy performance.	Convective heat transfer coefficients and energy performance.	External wind conditions significantly affect CHTC and energy performance.

Improvement of the hygrothermal efficiency numerically using CFD modeling in a full-scale ventilated room with Moroccan climates [46].	CFD is used to examine moisture distribution and heat transfer in a ventilated room under winter conditions.	Ventilation configurations are analyzed to find the optimal configuration for thermal comfort.	Moisture and temperature distribution.	The study identifies the optimal ventilation configuration through comfort indicators.
The influence of zonal air supply on thermal comfort in a classroom located in a hot and humid environment: a case study from Jeddah—Saudi Arabia [47].	CFD simulations predict air temperature, velocity, and humidity distribution, evaluating zonal air supply designs in classrooms.	Explores zonal air supply strategies to improve comfort in hot and humid regions.	Air velocity, air temperature, relative humidity, globe temperature, PMV, PPD.	Parametric analysis identifies effective configurations.
Experimental Validation of the Potential of Cross-Ventilation Strategy as a Natural Cooling Technique Integrated in a Real Historic Building [48].	Simulations analyze temperature fluctuations under free-running mode in naturally ventilated historic buildings.	Validates cross-ventilation as a natural cooling technique.	Temperature fluctuations.	Provides insights into thermal performance through cross-ventilation.
Thermal Comfort Assessment for Simultaneous Operation of Cooling and an Energy Recovery Ventilator in a Residential Building During Summer [49].	The article evaluates thermal comfort.	The research explores simultaneous operation of cooling and energy recovery ventilators.	Thermal comfort.	The study provides an evaluation under specific conditions.
Assessment of Floor-Level Impact on Natural Ventilation and Indoor Thermal Environment in Hot-Humid Climates: A Case Study of a Mid-Rise Educational Building [50].	The study evaluates natural ventilation and the indoor thermal environment.	Investigates the impact of floor level on ventilation in educational buildings.	Natural ventilation and indoor thermal environment.	Provides an assessment of thermal conditions at different floor levels.
The Effect of the Isolation Hotel Facade Attachment on the Inter-Flat Transmission of Aerosols [51].	The study investigates aerosol transmission.	Examines façade attachments and their effect on aerosol transmission between apartments.	Aerosol transmission.	Presents findings relevant to infection prevention.
Integrating CFD and thermoregulation models: A novel framework for thermal comfort analysis of non-uniform indoor environments [39]	CFD is integrated with thermoregulation models to analyze thermal comfort in non-uniform indoor environments.	Enables a more detailed analysis of occupant-centered radiant cooling strategies.	Operative temperature, heat balance, physiological responses.	Introduces a new numerical framework for evaluating thermoregulation and comfort.
Sensitivity analysis of influence factors on multi-zone indoor airflow CFD simulation [52].	CFD simulation is used for sensitivity analysis of factors influencing multi-zone indoor airflow.	Improves understanding of how different factors affect airflow simulation results.	Factors influencing multi-zone airflow.	Provides a sensitivity analysis of influencing factors in CFD airflow simulations.

Source: Authors (2026).

III.3 TRENDS IN THE USE OF CFD IN ARCHITECTURAL SPACES

Computational Fluid Dynamics (CFD) has become, over the past decade, a key methodological tool for analyzing the environmental and energy performance of the built environment, expanding its scope beyond strictly engineering applications to address challenges specific to architecture and urban planning. Traditionally employed in the field of computational wind engineering, CFD has demonstrated a strong capability to model complex phenomena such as airflow behavior, heat transfer, and contaminant dispersion, which has encouraged its adoption in studies related to pedestrian wind comfort, urban ventilation, air quality, and outdoor thermal conditions [19]. This versatility has enabled more accurate assessments of the interaction between urban morphology, local microclimates, and occupants' thermal well-being, thereby establishing CFD as a relevant technical support tool for sustainable urban planning and design.

At the building scale, the synthesized evidence reveals a clear trend toward applications aimed at optimizing indoor thermal comfort through passive and low-energy strategies, such as natural ventilation, cross-ventilation, solar chimneys, wind catchers, and evaporative cooling systems. In these studies, CFD is employed to analyze variables including air temperature, airflow velocities, CO₂ concentration, relative humidity, and comfort indices (PMV/PPD), enabling the evaluation of design configurations and the validation of architectural solutions with greater quantitative rigor. Furthermore, increasing integration with human thermo physiological models and experimental validation techniques has been observed, strengthening the reliability of predictions and aligning simulations more closely with real occupancy conditions [41]. Complementarily, a methodological evolution toward hybrid approaches that combine CFD with building energy performance simulation tools and data-driven predictive models has been identified. This coupling allows for the simultaneous evaluation of local thermo-aerodynamic behavior and its global impact on energy demand, enabling the quantification of how urban microclimate, building envelope characteristics, and conditioning systems influence energy consumption and overall performance [32]. Collectively, these trends indicate a shift from isolated modeling applications toward integrated analytical frameworks, in which CFD serves as a bridge between architectural design, human comfort, and energy sustainability.

III.4 CHALLENGES IN THE APPLICATION OF CFD

Despite its growing adoption in the analysis of thermal comfort and the environmental performance of the built environment, the application of Computational Fluid Dynamics (CFD) faces several technical and methodological limitations that affect the accuracy and feasibility of its results. One of the primary challenges lies in the faithful representation of real urban environments, which are characterized by highly complex geometries, morphological heterogeneity, and multiple spatial scales. Due to these difficulties, many studies rely on simplified or idealized configurations, which may reduce the ability of simulations to capture local phenomena such as ventilation patterns, recirculation zones, or heat accumulation typical of real urban contexts. From a computational perspective, CFD simulations particularly transient or high-fidelity approaches such as Large Eddy Simulation (LES) require considerable computational time and hardware resources, limiting their applicability in parametric studies or iterative design processes. Consequently, many studies adopt steady-state models based on Reynolds-Averaged Navier–Stokes (RANS), which offer a reasonable trade-off between accuracy and computational cost, albeit with reduced capability to resolve complex turbulent structures [40]. In addition, simulation results are highly sensitive to technical decisions such as mesh resolution, turbulence model selection, boundary condition definition, and the quality of climatic and geometric input data, factors that may introduce significant uncertainty if not properly controlled. In indoor applications, further challenges persist regarding the modeling of occupants' dynamic behavior, variable furniture layouts, and real space-use patterns, all of which directly influence airflow distribution, effective ventilation, and thermal perception. Finally, CFD model validation remains a critical requirement to ensure the reliability of predictions; however, acquiring high-resolution experimental data whether through in situ measurements or wind tunnel testing is often costly and complex, particularly in dense urban environments or specific configurations, thereby limiting the empirical verification of numerical models [38]. Overall, these challenges highlight the need to advance toward hybrid methodologies, robust validation frameworks, and more computationally efficient tools that can broaden the applicability of CFD in architectural practice.

III.5 OPPORTUNITIES FOR THE USE OF CFD IN ARCHITECTURAL SPACES

Computational Fluid Dynamics (CFD) offers a wide range of opportunities to strengthen environmental analysis and the sustainable design of the built environment, particularly in the evaluation of urban ventilation strategies aimed at improving air quality and mitigating pollutant accumulation in densely built areas. Its ability to simulate complex flow patterns, air dispersion, and renewal processes enables the identification of morphological configurations and design solutions that enhance pedestrian comfort and reduce exposure to atmospheric contaminants, thereby contributing to resilient and healthy urban planning [20]. At the building scale, CFD enables the optimization of thermal performance through the design and validation of passive strategies such as natural ventilation, cross-ventilation, and convective cooling, reducing reliance on mechanical conditioning systems and lowering the energy and carbon footprint of buildings. Moreover, the detailed analysis of variables such as air temperature, relative humidity, airflow velocities, and spatial heat distribution allows for the estimation of thermal comfort indices with higher spatial resolution, supporting evidence-based design decisions adapted to local climatic conditions [32]. Complementarily, the integration of CFD with building performance simulation (BPS) tools significantly expands its analytical potential by enabling the coupled evaluation of interactions among urban microclimate, architectural envelope characteristics, and overall energy demand. This hybrid approach facilitates the comparison of design scenarios and the development of optimized solutions that balance thermal comfort, operational efficiency, and environmental sustainability [28]. Furthermore, the flexibility of CFD modeling allows the analysis of specific health and safety contexts, such as contaminant or aerosol transmission in highly occupied spaces, providing relevant technical insights for the design of safer and more resilient buildings [24]. Collectively, these applications highlight the potential of CFD as a strategic tool for informed decision-making in architecture and urban planning.

IV. CONCLUSIONS

The evidence gathered in this systematic review confirms that Computational Fluid Dynamics (CFD) has become, over the past decade, a central tool for assessing thermal comfort and environmental performance in architectural and urban spaces. The bibliometric analysis, comparative synthesis of studies, and characterization of applications reveal a sustained expansion in its use, both in passive ventilation and thermal conditioning strategies and in urban microclimate modeling and building energy optimization. Furthermore, the increasing adoption of open-source platforms, the coupling with building energy simulations (BPS/BES), and the integration with thermo physiological models and machine learning techniques reflect a transition toward more accessible, integrated, and data-driven decision-making approaches. Nevertheless, the review also identifies persistent challenges that affect the reliability and scalability of these applications, including the realistic geometric representation of complex environments, the high computational requirements of transient high-fidelity models, the sensitivity of results to numerical parameters, and the need for rigorous experimental validation.

These limitations underscore the importance of advancing toward hybrid methodologies, robust verification frameworks, and surrogate models capable of balancing predictive accuracy with operational efficiency. Overall, the findings indicate that CFD holds significant strategic potential during the early stages of architectural design and urban planning, enabling the evaluation of scenarios, comparison of alternatives, and anticipation of thermal and energy impacts prior to physical implementation. Its consolidation as a decision-support tool for environmentally conscious design not only enhances thermal comfort and indoor air quality but also contributes to reducing energy consumption and the carbon footprint of the built environment. Future research should further integrate empirical data, artificial intelligence, and open-source workflows in order to broaden the applicability of CFD and strengthen its role as a bridge between computational modeling, human well-being, and sustainability.

V. AUTHOR'S CONTRIBUTION

Conceptualization: Keny Montalvo Morales, Henry Javier-Ninahuan, Grimaldo Quispe, Abdallah Elshawadfy Elwakeel, John Espita, Jader Rodriguez and Edwin Villagran.

Methodology: Keny Montalvo Morales, Henry Javier-Ninahuan, Grimaldo Quispe, Abdallah Elshawadfy Elwakeel, John Espita, Jader Rodriguez and Edwin Villagran.

Investigation: Keny Montalvo Morales, Henry Javier-Ninahuan, Grimaldo Quispe, Abdallah Elshawadfy Elwakeel, John Espita, Jader Rodriguez and Edwin Villagran.

Discussion of results: Keny Montalvo Morales, Henry Javier-Ninahuan, Grimaldo Quispe, Abdallah Elshawadfy Elwakeel, John Espita, Jader Rodriguez and Edwin Villagran.

Writing – Original Draft: Keny Montalvo Morales, Henry Javier-Ninahuan, Grimaldo Quispe, Abdallah Elshawadfy Elwakeel, John Espita, Jader Rodriguez and Edwin Villagran.

Writing – Review and Editing: Keny Montalvo Morales, Henry Javier-Ninahuan, Grimaldo Quispe, Abdallah Elshawadfy Elwakeel, John Espita, Jader Rodriguez and Edwin Villagran.

Resources: Keny Montalvo Morales, Henry Javier-Ninahuan, Grimaldo Quispe, Abdallah Elshawadfy Elwakeel, John Espita, Jader Rodriguez and Edwin Villagran.

Supervision: Keny Montalvo Morales, Henry Javier-Ninahuan, Grimaldo Quispe, Abdallah Elshawadfy Elwakeel, John Espita, Jader Rodriguez and Edwin Villagran.

Approval of the final text: Keny Montalvo Morales, Henry Javier-Ninahuan, Grimaldo Quispe, Abdallah Elshawadfy Elwakeel, John Espita, Jader Rodriguez and Edwin Villagran.

VI. REFERENCES

- [1] G. A. O. Rocha, M. A. Pichimata, and E. Villagran, "Research on the Microclimate of Protected Agriculture Structures Using Numerical Simulation Tools: A Technical and Bibliometric Analysis as a Contribution to the Sustainability of Under-Cover Cropping in Tropical and Subtropical Countries," *Sustain.* 2021, Vol. 13, Page 10433, vol. 13, no. 18, p. 10433, Sep. 2021, doi: 10.3390/SU131810433.
- [2] Y. Toparlar, B. Blocken, B. Maiheu, and G. J. F. van Heijst, "A review on the CFD analysis of urban microclimate," *Renew. Sustain. Energy Rev.*, vol. 80, pp. 1613–1640, 2017.
- [3] A. Jamin, W. Bosschaerts, I. Nastase, and B. Janssens, "Review of the existing state of the art regarding the use of CFD and human thermophysiological models for the vehicular comfort assessment," in *IOP Conference Series: Earth and Environmental Science*, 2021, vol. 664, no. 1, p. 12012.
- [4] K. Setaih, N. Hamza, M. Mohammed, S. Dudek, and T. Townshend, "CFD modeling as a tool for assessing outdoor thermal comfort conditions in urban settings in hot arid climates," *J. Inf. Technol. Constr.*, 2014.
- [5] H. Kabrein, A. Hariri, A. M. Leman, M. Z. M. Yusof, and A. Afandi, "Experimental and CFD modelling for thermal comfort and CO2 concentration in office building," in *IOP Conference Series: Materials Science and Engineering*, 2017, vol. 243, no. 1, p. 12050.
- [6] M. Dhahri and H. Aouinet, "CFD investigation of temperature distribution, air flow pattern and thermal comfort in natural ventilation of building using solar chimney," *World J. Eng.*, vol. 17, no. 1, pp. 78–86, 2020.
- [7] E. Villagran, "Implementation of ventilation towers in a greenhouse established in low altitude tropical climate conditions: numerical approach to the behavior of the natural ventilation," *Rev. Ceres*, 2021, doi: 10.1590/0034-737x202168010002.
- [8] A. Palmowska and I. Sarna, "CFD modelling of thermal comfort in the passenger coach," *Archit. Civ. Eng. Environ.*, vol. 15, no. 4, pp. 133–146, 2023.
- [9] I. Lourenço Niza and E. E. Broday, "Development of thermal comfort models over the past years: a systematic literature review," *Int. J. Ambient Energy*, vol. 43, no. 1, pp. 8830–8846, 2022.
- [10] H. Huo et al., "Simulation of the urban space thermal environment based on computational fluid dynamics: A comprehensive review," *Sensors*, vol. 21, no. 20, p. 6898, 2021.
- [11] G. H. Merabet et al., "Artificial intelligence-assisted energy and thermal comfort control for sustainable buildings: an extended representation of the systematic review," *arXiv Prepr. arXiv2006.12559*, 2020.
- [12] E. Villagran, J. J. Espitia, F. A. Velázquez, A. Sarmiento, D. A. S. Velandia, and J. Rodriguez, "Artificial Intelligence in Solar-Assisted Greenhouse Systems: A Technical, Systematic and Bibliometric Review of Energy Integration and Efficiency Advances," *Technologies*, vol. 13, no. 12, p. 574, 2025.
- [13] M. J. Page et al., "The PRISMA 2020 statement: an updated guideline for reporting systematic reviews," *bmj*, vol. 372, 2021.
- [14] E. Villagran, J. J. Espitia, F. A. Velázquez, and J. Rodriguez, "Solar Dryers: Technical Insights and Bibliometric Trends in Energy Technologies," *AgriEngineering*, vol. 6, no. 4, pp. 4041–4063, 2024.

- [15] D. A. Pinzon, G. M. Amado, J. Rodriguez, and E. Villagran, "Global Research Trends and Thematic Evolution of Blueberry (*Vaccinium* spp.) Science: A Bibliometric Analysis," *Horticulturae*, vol. 11, no. 12, p. 1501, 2025.
- [16] G. A. O. Rocha, A. N. C. Medina, L. G. Arias, J. F. A. Caíta, and E. Villagran, "Análisis sobre la actividad científica referente a las estrategias de climatización pasiva usada en invernaderos: Parte 1: Análisis bibliométrico," *Cienc. Lat. Rev. Científica Multidiscip.*, vol. 6, no. 5, pp. 4596–4623, 2022.
- [17] J. J. Espitia et al., "Solar Energy Applications in Protected Agriculture: A Technical and Bibliometric Review of Greenhouse Systems and Solar Technologies," *Agronomy*, vol. 14, no. 12, p. 2791, 2024.
- [18] G. Herrera-Franco, N. Montalván-Burbano, P. Carrión-Mero, B. Apolo-Masache, and M. Jaya-Montalvo, "Research trends in geotourism: A bibliometric analysis using the scopus database," *Geosci.*, 2020, doi: 10.3390/geosciences10100379.
- [19] N. Antoniou, H. Montazeri, H. Wigo, M. K.-A. Neophytou, B. Blocken, and M. Sandberg, "CFD and wind-tunnel analysis of outdoor ventilation in a real compact heterogeneous urban area: Evaluation using 'air delay,'" *Build. Environ.*, vol. 126, pp. 355–372, 2017.
- [20] K. Kosutova, T. Van Hooff, C. Vanderwel, B. Blocken, and J. Hensen, "Cross-ventilation in a generic isolated building equipped with louvers: Wind-tunnel experiments and CFD simulations," *Build. Environ.*, vol. 154, pp. 263–280, 2019.
- [21] E. Cuce, F. Sher, H. Sadiq, P. M. Cuce, T. Guclu, and A. B. Besir, "Sustainable ventilation strategies in buildings: CFD research," *Sustain. Energy Technol. Assessments*, vol. 36, p. 100540, 2019.
- [22] J. Foroozesh et al., "CFD modeling of the building integrated with a novel design of a one-sided wind-catcher with water spray: Focus on thermal comfort," *Sustain. Energy Technol. Assessments*, vol. 53, p. 102736, 2022.
- [23] S. Rodriguez Miranda, G. O. Gamboa, M. A. Zamora-Antuñano, N. Farrera-Vázquez, and R. García-García, "CFD evaluation of thermal conditioning in a house of social interest with a solar chimney arrangement in Guanajuato, Mexico," *Processes*, vol. 11, no. 4, p. 1286, 2023.
- [24] L. A. Rocha et al., "Natural ventilation in low-cost housing: an evaluation by CFD," *Buildings*, vol. 13, no. 6, p. 1408, 2023.
- [25] R. Stasi, F. Ruggiero, and U. Berardi, "Influence of cross-ventilation cooling potential on thermal comfort in high-rise buildings in a hot and humid climate," *Build. Environ.*, vol. 248, p. 111096, 2024.
- [26] J. A. Qadourah, "Improving Natural Ventilation Multi-Story Buildings within Hot and Dry Climates: A CFD Study of Windcatcher Performance," *energy Conserv.*, vol. 28, p. 29, 2024.
- [27] L. Iskandar, E. Bay-Sahin, A. Martinez-Molina, and S. T. Beeson, "Evaluation of passive cooling through natural ventilation strategies in historic residential buildings using CFD simulations," *Energy Build.*, vol. 308, p. 114005, 2024.
- [28] J. Brozovsky, J. Radivojevic, and A. Simonsen, "Assessing the impact of urban microclimate on building energy demand by coupling CFD and building performance simulation," *J. Build. Eng.*, vol. 55, p. 104681, 2022.
- [29] T. Yamamoto, A. Ozaki, K. Aratsu, and R. Fukui, "Energy simulation and CFD coupled analysis for the optimal operation of combined convection and radiant air conditioning considering dehumidification," *Heliyon*, vol. 9, no. 7, 2023.
- [30] K. Ding, J. K. Calautit, and C. Jimenez-Bescos, "Significance of external wind conditions on the convective heat transfer coefficients (CHTC) and energy performance in multi-zone high-rise buildings," *Energy Build.*, vol. 320, p. 114570, 2024.
- [31] S. Thongsuk, P. Songsukthawan, P. Lertwanitrot, S. Ananwattanaporn, S. Yoomak, and C. Pothisarn, "Impact of Building Envelope Materials on Energy Usage and Performance of Evaporative Cooling System in Residential Building," *Energies* (19961073), vol. 17, no. 15, 2024.
- [32] Y. Ming, Y. Sun, X. Liu, X. Liu, and Y. Wu, "Thermal performance of an advanced smart fenestration systems for low-energy buildings," *Appl. Therm. Eng.*, vol. 244, p. 122610, 2024.
- [33] S. Golder, R. Narayanan, M. R. Hossain, and M. R. Islam, "Experimental and CFD investigation on the application for aerogel insulation in buildings," *Energies*, vol. 14, no. 11, p. 3310, 2021.
- [34] G. Chen, L. Rong, and G. Zhang, "Unsteady-state CFD simulations on the impacts of urban geometry on outdoor thermal comfort within idealized building arrays," *Sustain. Cities Soc.*, vol. 74, p. 103187, 2021.
- [35] C. Nevers, A. Kubilay, J. Carmeliet, and D. Derome, "CFD simulation of the wind flow under lift-up buildings using a porous approach," *Build. Environ.*, vol. 263, p. 111867, 2024.
- [36] S. Sheng, T. Yamanaka, T. Kobayashi, N. Choi, and N. Chou, "Experimental study and CFD modelling of four-bed hospital ward with all-air wall induction unit for air-conditioning," *Build. Environ.*, vol. 222, p. 109388, 2022.
- [37] H. Sun et al., "Pollutant cross-transmission in courtyard buildings: Wind tunnel experiments and computational fluid dynamics (CFD) evaluation," *Build. Environ.*, vol. 264, p. 111919, 2024.
- [38] X. Li, W. Sun, C. Qin, Y. Yan, L. Zhang, and J. Tu, "Evaluation of supervised machine learning regression models for CFD-based surrogate modelling in indoor airflow field reconstruction," *Build. Environ.*, vol. 267, p. 112173, 2025.
- [39] J. M. Zambrano and L. Baldini, "Integrating CFD and thermoregulation models: A novel framework for thermal comfort analysis of non-uniform indoor environments," *Energy Build.*, vol. 335, p. 115570, 2025.
- [40] Y. E. Cetin, B. E. Yuce, M. Kriegel, P. Wargocki, and P. V. Nielsen, "Optimization of classroom window design using CFD and the Taguchi method," *J. Build. Eng.*, vol. 106, p. 112580, 2025.
- [41] J.-Y. Deng et al., "Indoor thermal environment in different generations of naturally ventilated public residential buildings in Singapore," *Atmosphere (Basel)*, vol. 13, no. 12, p. 2118, 2022.

- [42] R. Hu, H. Wang, J. Liang, X. Li, W. Zheng, and G. Liu, “Thermal Environment Analysis and Optimization for Large Space Buildings with Radiant Cooling Floors: A Case Study of Xianyang International Airport,” *Buildings*, vol. 14, no. 5, p. 1355, 2024.
- [43] M. Dhahri, A. Yüksel, H. Aouinet, D. Wang, M. Arıcı, and H. Sammouda, “Efficiency assessment on roof geometry and Trombe wall shape for improving buildings’ heating performance,” *Buildings*, vol. 14, no. 5, p. 1297, 2024.
- [44] J. Aparecido Martins, A. Francisco Siqueira, and E. Claro Romao, “Investigating Efficient Thermal Distribution in a House Room by combining Statistics with Computational Fluid Dynamics,” *Eng. Technol. Appl. Sci. Res.*, vol. 14, no. 4, 2024.
- [45] N. R. M. Sakiyama, J. C. Carlo, F. I. H. Sakiyama, N. Abdessemed, J. Frick, and H. Garrecht, “Impact of wind pressure coefficients on the natural ventilation effectiveness of buildings through simulations,” *Buildings*, vol. 14, no. 9, p. 2803, 2024.
- [46] E. H. Sebbar, Y. Khattari, Y. Chaibi, and T. El Rhafiki, “Improvement of the hygrothermal efficiency numerically using CFD modeling in a full-scale ventilated room with Moroccan climates,” *Sci. African*, vol. 26, p. e02408, 2024.
- [47] Z. Jastaneyah, H. M. Kamar, A. Hashmi, F. A. Ghaleb, and H. Al Garalleh, “The influence of zonal air supply on thermal comfort in a classroom located in a hot and humid environment: a case study from Jeddah—Saudi Arabia,” *Discov. Sustain.*, vol. 5, no. 1, p. 526, 2024.
- [48] T. Palomo Amores, F. Ruda Sarria, D. C. Medina, T. C. Valera, J. Sánchez Ramos, and S. Álvarez Domínguez, “Experimental Validation of the Potential of Cross-Ventilation Strategy as a Natural Cooling Technique Integrated in a Real Historic Building,” *Appl. Sci.*, vol. 15, no. 4, p. 2174, 2025.
- [49] K. Kang and D. D. Kim, “Thermal Comfort Assessment for Simultaneous Operation of Cooling and an Energy Recovery Ventilator in a Residential Building During Summer,” *Buildings*, vol. 15, no. 4, p. 582, 2025.
- [50] E. J. Mba et al., “Assessment of Floor-Level Impact on Natural Ventilation and Indoor Thermal Environment in Hot–Humid Climates: A Case Study of a Mid-Rise Educational Building,” *Buildings*, vol. 15, no. 5, p. 686, 2025.
- [51] N. Zhang, Y. Li, Y. Ji, and X. Li, “The Effect of the Isolation Hotel Facade Attachment on the Inter-Flat Transmission of Aerosols,” *Build.*, vol. 15, no. 5, 2025.
- [52] J. C. P. Cheng, H. H. L. Kwok, A. T. Y. Li, J. C. K. Tong, and A. K. H. Lau, “Sensitivity analysis of influence factors on multi-zone indoor airflow CFD simulation,” *Sci. Total Environ.*, vol. 761, p. 143298, 2021.