

CFD Applications in Environmental Engineering

Filiberto Hueyotl-Zahuantla ^{1,2}  and Mario A. Aguirre-López ^{2,*} 

¹ Secretariat of Science, Humanities, Technology and Innovation, Mexico City 03940, Mexico; fhueyotl@secihti.mx

² Faculty of Sciences in Physics and Mathematics, Autonomous University of Chiapas, Tuxtla Gutiérrez 29050, Mexico

* Correspondence: mario.aguirre@unach.mx

Environmental engineering is increasingly relying on advanced numerical tools to address the complex challenges associated with energy efficiency, air quality, pollutant transport, and sustainable infrastructure design. The pursuit of improvements in these regards is a global priority and, as such, considerable efforts have been made on various fronts, particularly through the use of computational fluid dynamics (CFD) simulations. The versatility of CFD has enabled the investigation of applications ranging from the analysis and engineering of wind energy devices to the study of indoor and outdoor environmental processes at various scales, such as those presented in this Special Issue.

The six contributions included in this edition are as follows:

- “Hybridization of a Micro-Scale Savonius Rotor Using a Helical Darrieus Rotor” [1];
- “Mitigating Airborne Infection Transmission in the Common Area of Inpatient Wards—A Case Study” [2];
- “Experimentally Validated Discrete Phase Model for PM2.5 and PM10 with Numerical Transport Mapping” [3];
- “Optimizing Air Change Rates: A CFD Study on Mitigating Pathogen Transmission in Aircraft Cabins” [4];
- “Turbulent Flow Analysis of a Representative Low-Height Urban Landscape in Mexico” [5];
- “Vertical Dense Jets in Cross-Flows: A Preliminary Study with Lattice Boltzmann Methods” [6].

They highlight recent developments in CFD-based research, encompassing improvements in vertical-axis wind turbines; air and particle dispersion in enclosed environments, including hospital waiting rooms, bedrooms, and cabins; wind flow and pollutant transport in urban areas; and vertical dense jets in cross-flows in a wastewater dispersion scenario.

Together, these studies illustrate the growing role of CFD not only as a predictive tool for understanding fluid-flow phenomena, but also as a decision-support framework for environmental engineering. In all cases, CFD is employed to evaluate alternative designs, identify optimal operating conditions, assess environmental risks, or support planning strategies before implementation, demonstrating how CFD is increasingly integrated into the early stages of engineering design and environmental management. This evolution reflects a broader transition from CFD as a purely analytical methodology toward a practical tool for supporting sustainable and evidence-based engineering decisions.

The contributions can be viewed within three complementary themes: (i) CFD-supported design and optimization of energy systems, which is represented by the hybrid wind turbine study [1]; (ii) CFD-guided assessment of indoor environmental quality, including ventilation [2], particle dispersion [3], and pathogen transmission [4]; and (iii) CFD-informed analysis of outdoor environmental processes, such as urban airflow [5] and wastewater discharge [6] dynamics.



Received: 26 June 2026

Revised: 10 July 2026

Accepted: 12 July 2026

Published: 16 July 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and

conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

The collection opened with the contribution by Moreno et al. [1] that addresses the challenge of improving energy efficiency and operational range in micro-scale vertical axis wind turbines (VAWTs) by proposing a hybrid turbine consisting of a Savonius rotor contained within a Darrieus rotor. From CFD simulations based on the finite volume method through a commercial software (ANSYS® Fluent 2022) and controlled experimental tests with a 3D-printed prototype, the authors demonstrated that, at the optimal tip-speed ratio, the hybrid turbine achieves a 180% improvement compared to the performance of a turbine equipped solely with the Savonius rotor. They found that the hybrid design effectively combines the high torque achieved at low wind speeds with the Savonius rotor and the aerodynamic efficiency of a Darrieus turbine. The gap in the state of the art is covered in the sense that previous studies frequently rely solely on either computational simulations or experimental tests, which can limit the robustness and practical applicability of their findings. Furthermore, previous hybrid Savonius–Darrieus rotor designs often favor one rotor type over the other. A comprehensive review on the strengths and weaknesses of both types of rotors can be found in [7]. Meanwhile, ref. [8] reviews the various strategies that have been used to improve the performance of Savonius wind turbines, including hybrid designs. Thus, the trend is toward the development of hybrid designs, as those presented in [9] integrating deflection plates in a hybrid wind turbine design; and the study by [10] focused on enhancing the efficiency of hybrid vertical turbines through design modifications of internal and external blades and optimization of the internal rotor configurations, both using CFD as a key tool. Despite the efforts made so far, there are still outstanding issues to be solved, such as how turbulence and unsteady inflow in real-world conditions affect the performance and efficiency of hybrid systems compared to numerical simulations and controlled wind tunnel environments. And even beyond, what will the environmental impacts and lifecycle assessments of the manufacture, deployment, and disposal of these hybrid turbines be?

The wide scope of this Special Issue allows exploration of the applications of CFD in diverse environments to mitigate indoor infections and pathogen transmission, as well as particle dispersion. The following contributions illustrate recent advances in this field of research. Li et al. [2] cover an important gap rarely investigated: airflow and aerosol transport in a full hospital ward and emphasize that shared spaces are important because healthcare workers spend substantial time there and these spaces can facilitate cross-room transmission. By implementing Reynolds-averaged Navier–Stokes (RANS) simulations with the SST $k - \omega$ turbulence model in Ansys CFX 2024R1, the authors simulated aerosol transport in the common area of an actual hospital ward, comprising various types of patient rooms and equipped with a mixing ventilation system. This study extends the use of CFD beyond single rooms and provides a spatially resolved assessment of infection risk. They tested an improved ventilation layout over a current real ventilation condition, provided insight into how ventilation designs influence aerosol dispersion in hospital settings, and suggested that strategic vent placements can mitigate airborne transmission. Several challenges and unresolved issues continue to motivate ongoing research; to mention a few: the impact of the movement of people and the operation of doors on the transport of aerosols between rooms and common areas; the real-time airflow monitoring and adaptive ventilation control requirements for dynamic reduction of transmission risks; and practical challenges and costs to retrofit existing hospital wards with optimized ventilation systems. The importance of developing research in this stimulating area of investigation became clearly evident during the COVID-19 pandemic. For example, ref. [11] discusses strategies to minimize the airborne transmission of COVID-19 infection in enclosed spaces, emphasizing the importance of engineering controls in public buildings. In turn, the review [12] analyzes how ventilation strategies

have evolved in response to COVID-19, highlighting key findings and gaps in the research categories of detection methods, observational analyses, CFD simulations, and policy recommendations. Finally, in [13] specific scenarios focusing on real-world conditions and infection risk in Japanese hospital rooms have been presented, evaluating ventilation configurations and the probability of infection.

Continuing with CFD studies in controlled environments, the contribution of Estaquio et al. [3] addresses the challenge of accurately mapping the distribution of indoor particle pollution: PM2.5 and PM10 particulate matter (2.5μ and 10μ diameter, respectively) to suggest sensor placement strategies. The key contribution of this study is the development and experimental validation of a transient CFD framework coupling RANS airflow modeling (using $k - \epsilon$ model) and Lagrangian particle tracking in a full-scale bedroom equipped with a fixed fan and a window. Through Ansys Meshing 2024 R2, the study introduces an exposure rating that combines maximum and average concentrations, along with persistence metrics, to identify optimal sensor locations, highlighting that no single fixed point can effectively represent both particle sizes. The challenge of sensor placement and the importance of accurate modeling in poorly ventilated spaces is also highlighted in [14]; this recent study integrates CFD with experimental measurement to analyze the distribution of CO₂ in environments with a low air exchange rate. The analysis of different ventilation scenarios becomes evident in the removal of harmful substances and particles from the air. In this regard, ref. [15] employs a Eulerian–Lagrangian method to numerically investigate the behavior of PM10 under different ventilation scenarios; its findings on particle deposition and removal efficiencies contribute to understanding indoor air quality management. The work [16] critically assesses the gaps in the literature of exposure to PM2.5, particularly the reliance on outdoor data, bridging perspectives from indoor and outdoor predictive models. In [17], human factors in airflow dynamics have been identified to understand the transport of pollutants in built environments through manikin-involved CFD modeling of indoor air quality. These recent studies highlight the importance of analyzing pollution dispersion and mitigation strategies.

In turn, Benn & Tian [4] contributed to our Special Issue also using ANSYS Fluent 2024 by investigating how varying air change rates influence on the dispersion and residence time of airborne infectious agents in aircraft cabins. Their study considers a high-fidelity CAD model representing a section of a Boeing 737 cabin and simulates four scenarios of air change per hour rates, with 4μ particles injected into the cabin depicting pathogens of a single infectious agent. They provide new insights into the optimization of aircraft cabin ventilation systems, specifically focusing on optimizing ventilation efficiency, reducing energy consumption, and maintaining passenger comfort, which have not been comprehensively addressed in previous studies. Following the context of closed environment investigations, ref. [18] evaluates the influence of ventilation and ambient temperature on the airborne transmission of viruses in clinic waiting rooms using CFD, providing practical strategies to improve safety. In turn, ref. [19] analyzes the effectiveness of different ventilation systems in aircraft cabins by comparing various design variables, which contributes to understanding how to optimize airflow to reduce airborne transmission. Finally, ref. [20] explores the relationship between airborne transmission and thermal comfort in train cabins, using a CFD approach similar to that of [4]; the study highlights the importance of optimizing ventilation strategies to balance infection control and passenger comfort, making it a pertinent follow-up for those interested in mitigating pathogen transmission in confined spaces.

A compelling observation emerging from the previous studies in closed environments is that there is a clear and growing need for context-specific studies. This leads us to the following general questions: How could customized extraction systems be designed to pre-

vent cross-contamination between passengers/users? Adaptive or customized ventilation systems could be developed to optimize air quality for each passenger/user or for specific areas? In any case, real-time monitoring of airflow and aerosol/particle concentrations is required to obtain information that allows dynamic adjustments to ventilation rates.

This Special Issue is complemented and enriched by the contribution presented by Ibarra-Hernández et al. [5] using the Renormalization Group $k - \epsilon$ model in an external environment through ANSYS FLUENT V.18.1. The article examines turbulent airflow patterns in a specific urban area with low-rise buildings in Monterrey, Mexico. The authors aimed to understand how wind interacts with irregularly arranged buildings to identify zones of high and low wind velocity, which influence the dispersion and retention of pollutants. They modeled a representative urban landscape based on satellite imagery and real local weather data, simulating three wind scenarios with varying velocities. This study covers the gap of a detailed analysis of turbulent flow patterns in realistic, irregular, low-rise urban environments typical of Latin American cities, particularly in Mexico. The methodology enables the identification of regions with a high probability of pollution retention and regions with a high pollutant transport capacity, providing information for customized green strategies and urban planning. Related studies have been introduced by [21], which evaluates various CFD approaches to simulate wind and pollutant dispersion in urban canyons. In turn, ref. [22] explores the effect of the architectural form at block scale using a hybrid methodology that integrates CFD with geographic information system data and machine learning algorithms, providing insights into the impact of architectural morphology on airflow. Moreover, ref. [23] introduces an AI-enhanced approach to urban wind simulations, which offers innovative methodologies applicable to urban wind flows. All of these investigations contribute to the understanding of established flow patterns in specific urban areas and, again, highlight the need for customized studies that consider the complexity of specific areas. This leads to the following questions: What are the implications of these flow patterns for public health policies aimed at reducing exposure to air pollutants? What are the long-term effects of urban expansion and changing building configurations on local airflow and pollution dynamics? What are the potential benefits and limitations of using CFD simulations for environmental policy development, in particular in Latin American cities?

Finally, this Special Issue is complemented by the contribution of Giordano et al. [6], which explores the use of the Lattice Boltzmann Method (LBM) combined with Large Eddy Simulation (LES) to simulate dense vertical jets in cross-flows. Although this is a preliminary study, it has great potential to understand wastewater dispersion scenarios, such as brine from desalination plants. Through sophisticated visualizations, they demonstrate that the simulations qualitatively reproduce the expected flow characteristics, including jet trajectories and vortex structures, such as counter-rotating vortex pairs, whereas quantitative discrepancies were observed, such as the underestimation of rise heights and impact distances. The study provides preliminary insights into the potential of LBM for modeling environmental flows, highlighting its scalability for high-performance computing. The versatility of LBM is evidenced in [24]; this review systematically analyzes the advances of LBM in urban wind simulations. On the other hand, ref. [25] proposes a novel LBM framework for multiphase flows, capable of handling large density ratios and complex interactions. The method has also been used in [26] to study free-surface flows over partially submerged structures. Despite the widespread use of LBM in various fluid dynamics applications, the authors in [6] noted that no specific attempt appears to have been made to numerically investigate the three-dimensional near-field dispersion of debris in ambient water using such a method. A couple of questions arise at this stage: Could adaptive mesh refinement (AMR) techniques be integrated into LBM frameworks

to better capture near-field dynamics without prohibitive computational cost? Can the methodology be extended to simulate multi-component or multiphase flows relevant to real wastewater discharges?

Altogether, the contributions from this Special Issue demonstrate a clear evolution in the role of CFD within environmental engineering. They are examples of that CFD is increasingly employed as a decision-support methodology, beyond its traditional use as a predictive and analytical tool. In detail, they show how numerical simulations can guide the optimization of renewable energy devices, improve indoor air quality and infection-control measures, support urban planning decisions, and contribute to the assessment of wastewater discharge scenarios. This is an indicative of that CFD is becoming an integral component of the early design and planning stages of environmental engineering projects, including risk-mitigation strategies.

In this regard, several challenges remain for the next generation of CFD applications in environmental engineering. First, there is a growing need for real-time and data-driven CFD technologies capable of assimilating sensor measurements and adapting predictions to changing environmental conditions. Second, the increasing complexity of environmental systems requires more realistic representations of multiple factors: human behavior, urban growth, multiphase processes, and transient boundary conditions. Third, computational cost remains a significant limitation, implying higher energy consumption and then, motivating the development of reduced-order models, adaptive meshing techniques, high-performance computing implementations, and hybrid CFD–AI approaches. Finally, greater attention should be devoted to uncertainty quantification and model validation to strengthen confidence in CFD-based recommendations used in engineering and policy-making. Addressing these challenges will further continue consolidating CFD as a key technology for sustainable environmental engineering and informed decision-making.

We thank the authors who contributed to the first edition and look forward to their follow-up research and new proposals.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Moreno, M.; Trejo-Zúñiga, I.; Terrazas, J.; Díaz-Ponce, A.; Pérez-Terrazo, A. Hybridization of a Micro-Scale Savonius Rotor Using a Helical Darrieus Rotor. *Fluids* **2025**, *10*, 63. [\[CrossRef\]](#)
2. Li, X.; Kevin, K.; Lam, W.K.; Ooi, A.; Zachreson, C.; Geard, N.; Tsigaras, L.; Bates, S.; McGain, F.; Morawska, L.; et al. Mitigating Airborne Infection Transmission in the Common Area of Inpatient Wards—A Case Study. *Fluids* **2025**, *10*, 267. [\[CrossRef\]](#)
3. Estaquio, R.P.; Canlas, M.K.; Astrologo, N.; Encarnacion, J.I.; Agar, J.; Fernandez, K.B.; Lusto, J.R.; Reyes, J.G. Experimentally Validated Discrete Phase Model for PM2.5 and PM10 with Numerical Transport Mapping. *Fluids* **2026**, *11*, 90. [\[CrossRef\]](#)
4. Benn, J.; Tian, L. Optimising Air Change Rates: A CFD Study on Mitigating Pathogen Transmission in Aircraft Cabins. *Fluids* **2025**, *10*, 74. [\[CrossRef\]](#)
5. Ibarra-Hernández, C.; Hernández-García, L.; Nájera-Sanchez, R.; Arriaga-Gomez, E.; Martínez-Delgadillo, S.; Medellín-Salazar, D.; García, A.A. Turbulent Flow Analysis of a Representative Low-Height Urban Landscape in Mexico. *Fluids* **2026**, *11*, 23. [\[CrossRef\]](#)
6. Giordano, M.G.; Jacob, J.; Fusco, P.; Tangaro, S.; Malcangio, D. Vertical Dense Jets in Crossflows: A Preliminary Study with Lattice Boltzmann Methods. *Fluids* **2025**, *10*, 159. [\[CrossRef\]](#)
7. Fazylova, A.; Alipbayev, K.; Aden, A.; Oraz, F.; Iliev, T.; Stoyanov, I. A Comparative Review of Vertical Axis Wind Turbine Designs: Savonius Rotor vs. Darrieus Rotor. *Inventions* **2025**, *10*, 95. [\[CrossRef\]](#)
8. Sengupta, A.R.; Rakshit, S.; Majumder, H. A Comprehensive Review on Various Approaches for Performance Improvement of Savonius Wind Turbines. *Discov. Appl. Sci.* **2025**, *8*, 79. [\[CrossRef\]](#)
9. Hajong, G.; Raman, R.; Sarma, S.; Raji, R.; Mohammad, N.; Das, T.P. Computational Investigation of a Hybrid Vertical Axis Wind Turbine using Deflector Plates. *Sādhanā* **2025**, *50*, 271. [\[CrossRef\]](#)
10. Tomar, S.S.; Dewan, A. Enhancing Efficiency of Hybrid Vertical Axis Turbine: Effect of Various Internal and External Blades, Collapsed Rotor and Reduced Internal Rotor Velocity. *Renew. Energy* **2025**, *255*, 123791. [\[CrossRef\]](#)

11. Morawska, L.; Tang, J.W.; Bahnfleth, W.; Bluyssen, P.M.; Boerstra, A.; Buonanno, G.; Cao, J.; Dancer, S.; Floto, A.; Franchimon, F.; et al. How Can Airborne Transmission of COVID-19 Indoors be Minimised? *Environ. Int.* **2020**, *142*, 105832. [[CrossRef](#)] [[PubMed](#)]
12. Nikoopayan Tak, M.S.; Mousavi, E. Ventilation and Infection Control in Healthcare Facilities: A Post-COVID-19 Literature Synthesis. *Air* **2025**, *3*, 30. [[CrossRef](#)]
13. Tanaka, Y.; Kikuta, K.; Nakazawa, T.; Sasai, Y.; Ogawa, Y.; Gu, Y.; Mitsumoto-Kaseida, F.; Yanagihara, K.; Tateda, K.; Hayashi, M. Ventilation Characteristics and Infection Risk in Japanese Hospital Wards: An Assessment Based on Field Measurements and CFD. *Indoor Environ.* **2026**, *3*, 100161. [[CrossRef](#)]
14. Jaeschke, M.; Wallburg, F.; Weickhardt, C.; Satke, P.; Schoenfelder, S. Experimental and Numerical Analysis of CO₂ Distribution in Low Air Change Rate Environments: A Case Study of a Meeting Room. *Build. Environ.* **2026**, *288*, 113893. [[CrossRef](#)]
15. Hu, X.; Zhang, M.; Fan, F. Numerical Investigation of Indoor Particle Behaviors in Different Ventilation Scenarios. *Aerosol Air Qual. Res.* **2025**, *25*, 35. [[CrossRef](#)]
16. Lu, Q.O.; Chang, W.H.; Chen, H.L.; Lee, C.C. Unveiling the Hidden Gaps of PM_{2.5} Exposure: A Critical Review Bridging Perspectives from Indoor and Outdoor Predictive Models. *Results Eng.* **2026**, *29*, 109830. [[CrossRef](#)]
17. Zhou, Y.; Zheng, X. Manikin-Involved CFD Modeling and Applications for Air Flow Analysis: A Comprehensive Review. *Renew. Sustain. Energy Rev.* **2026**, *229*, 116561. [[CrossRef](#)]
18. Zhu, Z.; Gao, G.; Hu, Y.; Zhao, X. Impact of Ventilation and Ambient Temperature on COVID-19 Transmission in Clinic Waiting Rooms: A Computational Fluid Dynamics Approach. *PLoS ONE* **2025**, *20*, e0328154. [[CrossRef](#)] [[PubMed](#)]
19. Rajendran, R.R.; Turcanu, F.E.; Tawfikur, R.M.; Askarpour, H. Computational Fluid Dynamic Analysis of Corona Virus Patients Breathing in an Airplane. *Phys. Fluids* **2023**, *35*, 035129. [[CrossRef](#)]
20. Xu, R.; Wu, F.; Shen, L.; Pan, X.; Huang, Z.; Yu, J.; Huang, Y. Investigation of Airborne Transmission and Thermal Comfort in a Train Cabin: Effects of Air Change Rate and Supply Mode. *Case Stud. Therm. Eng.* **2025**, *76*, 107387. [[CrossRef](#)]
21. Liang, W.; Zheng, X. CFD Simulation of Mixed Convection and Pollutant Dispersion in Street Canyons: A Comparative Assessment of LES, RANS, and SAS. *Sustain. Cities Soc.* **2026**, *138*, 107126. [[CrossRef](#)]
22. Guo, B.; Chen, M.; Zhu, X.; Wang, Z.; Li, L.; Pei, L.; Chen, H.; Chen, P.; Guo, T. Exploring the Effect of the Architecture Morphology on Urban Ventilation at Block Scale Using CFD-GIS and Random Forest Combined Method. *Sustain. Cities Soc.* **2025**, *122*, 106241. [[CrossRef](#)]
23. Tian, G.; Qin, S.; Geng, D.; Zhan, D.; Yang, S.; Liu, P.; Gao, N.; Wang, L.L. AI-Enhanced Large-Eddy Simulation for Urban Wind Flows: Accelerating Spin-Up with Fourier Neural Operator Initialization. *Build. Environ.* **2026**, *289*, 114046. [[CrossRef](#)]
24. Hu, Y.; Han, M. Review of Lattice Boltzmann Method in Urban Wind Simulation: Progress, Challenges, and Outlook. *Build. Environ.* **2026**, *288*, 113915. [[CrossRef](#)]
25. Piredda, M.M.; Asinari, P. Lattice Boltzmann Framework for Multiphase Flows by Eulerian–Eulerian Navier–Stokes Equations. *Computation* **2025**, *13*, 164. [[CrossRef](#)]
26. Guo, B.; Meng, J.; Xie, Z.; Ning, D.; Pan, S. A Lattice Boltzmann Method for Free Surface Flows over Partially Submerged Structures. *Comput. Phys. Commun.* **2025**, *317*, 109852. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.