Enhancing Indoor Environmental Simulations: A Comprehensive Review of CFD Methods

Shahrzad Marashian[a,*], Amir Vadiee [a], Omid Abouali [b], Sasan Sadrizadeh [a,b,*]

[a] *School of Business, Society, and Engineering, Mälardalens University, Västerås, Sweden,*

[b] *Department of Civil and Architectural Engineering, KTH Royal Institute of Technology, Stockholm, Sweden*
shahrzad.marashian@mdu.se | sasan.sadrizadeh@mdu.se

Abstract

Computational Fluid Dynamics (CFD) simulations are extensively used to model indoor environments, including airflow patterns, temperature distribution, and contaminant dispersion. These simulations provide valuable insights for improving indoor air quality, enhancing thermal comfort, optimizing energy efficiency, and informing design decisions. The recent global pandemic has emphasized the importance of understanding airflow patterns and particle dispersion in indoor spaces, highlighting the potential of CFD simulations to guide strategies for improving indoor air quality and public health. Consequently, there has been a significant increase in research focused on studying the transport and dispersion of pollutants in indoor environments using CFD techniques. These simulations are vital in advancing engineers' understanding of indoor environments; however, achieving accurate results requires careful method selection and proper implementation of each step. This paper aims to review the state-of-the-art CFD simulations of indoor environments, specifically focusing on strategies employed for three main simulation components: geometry and grid generation, ventilation strategies, and turbulence model selection. Researchers can select suitable techniques for their specific applications by comparing different indoor airflow simulation strategies.

1. Introduction

Understanding and assessing indoor environments is crucial, given that people spend most of their time indoors. The quality of indoor air, thermal comfort, and energy consumption are key factors that directly impact individuals' well-being and productivity. Poor air quality can lead to various health issues, such as respiratory problems and allergies. After the recent pandemic, there has been a heightened awareness and understanding of the importance of indoor air quality. A growing emphasis is on implementing strategies and technologies that enhance ventilation, filtration, and air purification to create safer and healthier indoor environments. The COVID-19 pandemic has significantly increased research on indoor airflow since the onset of the COVID-19 pandemic in 2019 (Kohanski et al., 2020).

Investigating enclosed environments commonly involves conducting experiments and utilizing computational simulations. The increasing number of numerical research studies in this area can also be attributed to a notable advancement in computer capabilities, as well as the development and refinement of computational fluid dynamics (CFD) methods and software.

Indoor environments can be broadly categorized into private residential spaces and public settings. Public utility buildings include diverse spaces, such as churches, museums, libraries, and hospitals. Each of these environments serves a unique purpose and has specific requirements for indoor air quality. The quality of indoor air in public utility buildings is influenced by various environmental factors, including human activity, characteristics of the indoor area, and the presence of chemical compounds in the surrounding air (Śmiełowska et al., 2017).

Ventilation is an important part of indoor air simulations, which also plays a crucial role in limiting the spread of viruses. Optimizing ventilation rates, eliminating air recirculation, using portable air cleaners with proper maintenance, and avoiding overcrowding in public spaces are some of the recommendations in this area. Implementing these engineering controls alongside other preventive measures will lower airborne pathogen concentrations and reduce infection rates for airborne diseases. It emphasizes the need to prioritize airborne transmission reduction in hospitals and public buildings to protect healthcare workers and the public.(Morawska et al., 2020)

The effectiveness of designing and operating indoor environments relies on accurate and reliable numerical simulations. This paper comprehensively reviews the latest advancements in Computational Fluid Dynamics (CFD) simulations for various indoor settings. Specifically, it focuses on three crucial and challenging aspects: geometry and grid generation, ventilation strategy, and selection of appropriate turbulence models. By examining these areas, this review aims to enhance the understanding of simulation strategies and ultimately improve the overall accuracy and reliability of numerical simulations in the investigation of indoor environments.

2. Methodology

The method used in this study involves gathering information from recent articles by searching for keywords such as indoor airflow, ventilation, CFD simulation, and indoor air quality in academic databases such as Web of Science, SAGE journals, and Science Direct. The focus is on studies conducted since 2012. The collected information is then compared and analyzed, specifically emphasizing geometry and grid generation, ventilation strategies, and the selection of turbulence models. By examining and summarizing these aspects, this study aims to understand the latest advancements and strategies used in simulating indoor environments.

3. Overview of CFD Methods for Indoor Environment Simulations

3.1. Yearly publication distribution

For this review, an analysis was performed on 25 previous research studies in the field of CFD simulation with a specific emphasis on indoor airflow. *Figure 1* illustrates the yearly distribution of reviewed papers. It highlights the observed trend in the selected research, showing a focus on recent articles from 2020 until the present.

3.2. Simulation tools

The software tool ANSYS Fluent is widely recognized and extensively used for conducting CFD simulations in most research papers reviewed here. Fluent has gained popularity among researchers and engineers for its exceptional capabilities and user-friendly interface, making it a preferred choice for modeling and analyzing fluid flow, heat transfer, and other related simulations. This software is also commonly used for simulating indoor airflows and conducting CFD analyses in indoor air quality and ventilation. It gives researchers and engineers the tools to model and analyze various aspects of indoor airflows, including air distribution, temperature profiles, pollutant dispersion, and ventilation effectiveness. The choice of CFD software depends on the specific research objectives and the researchers' expertise. A few studies also employed other CFD tools for their simulations. For instance, some researchers utilized ANSYS CFX (Kalliomaki et al., n.d.), OpenFOAM (Arpino et al., 2023), or STAR-CCM+ (Chang et al., 2023).

3.3. Geometry and grid generation

Figure 2 categorizes the selected papers based on the different types of indoor spaces studied using CFD. These categories include various indoor environments, including Hospital Environments, Transportation Spaces, Educational Spaces, Offices, Restaurants, Residential Spaces, and Museums. Among these categories, Figure 2 shows that hospital ward research represents the most significant proportion, accounting for 36% of all the selected research studies. *Table 1* provides detailed information on the subsections within each category and lists the corresponding relevant research studies.

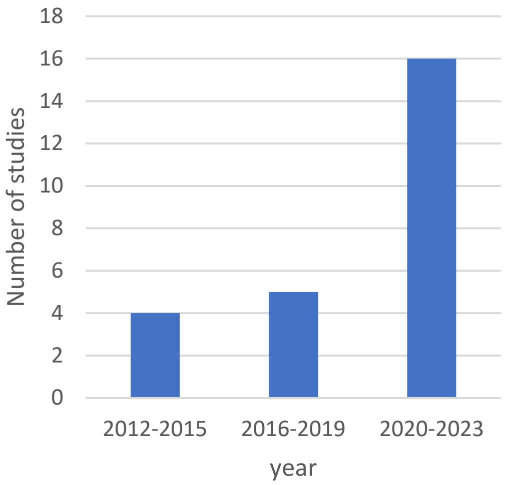

Figure 1: Yearly distribution of reviewed studies

Grid generation is a vital aspect of conducting indoor airflow CFD simulations. It involves creating a grid or mesh that discretizes the indoor environment into smaller elements, allowing for the numerical solution of governing equations. (Liu et al., 2022a) conducted a comparison of three different meshing strategies, namely hexahedral, tetrahedral, and polyhedral meshes, for simulating indoor airflow in geometries with varying levels of complexity to assess the performance and suitability of each mesh type in capturing the airflow behavior within indoor environments

All the articles studied here utilized three-dimensional geometry, and most employed unstructured grids due to their complex geometry. Various types of cells are used in grid generation in studied articles, including Tetrahedral, Hexahedral, poly-hexcore.

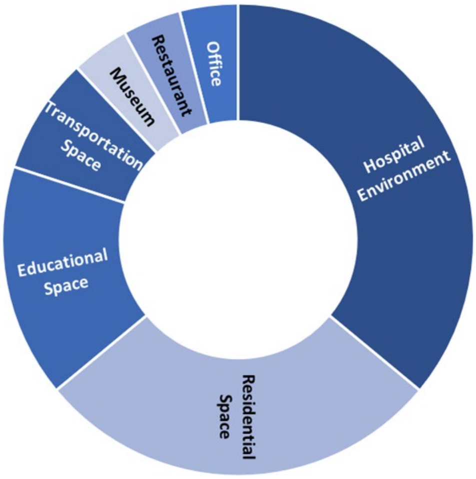

Figure 2: Different Types of Indoor Spaces in Reviewed Studies and Their Proportions

As depicted in *Figure 3*, half of the studies utilized a tetrahedral mesh for their simulations. Around 20% of the studies used hexahedral mesh. Only 10% of recent studies used a poly-hexcore mesh, which is also a new option available in recent versions of Fluent meshing software.

The volume of the simulated geometry varied across the different studies. The smallest volume, 0.128 m³, was associated with a modeled room (Marashian et al., 2022), while the largest volume, 724.56 m³, was observed in the context of an open museum space (Bakry et al., 2022).

The average mesh density is another grid property determined based on the ratio of the geometric model's volume to the total number of cells. (Liu et al., 2022a). This parameter also can represent the complexity of the generated grid. In Table 1, the average mesh density is calculated and reported for the studied papers.

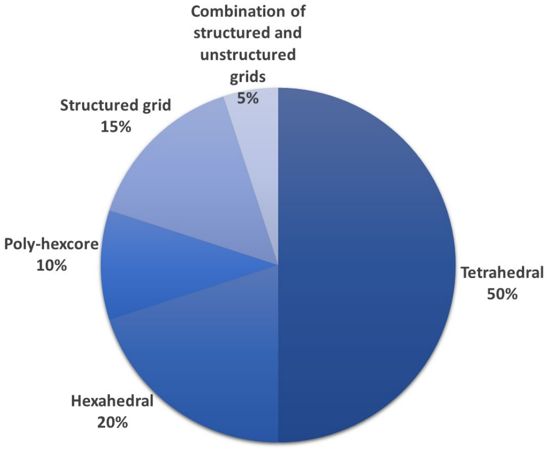

Figure 3: Cell Types Used in Grid Generation in Studied Articles

3.4. Ventilation strategies

Natural and mechanical ventilation are the two main methods to ventilate indoor spaces. Natural ventilation relies on natural forces such as wind or buoyancy to create airflow. On the other hand, mechanical ventilation utilizes mechanical systems like fans, blowers, or air conditioners to control indoor airflow.

The number of inlets, outlets, and openings like windows and their locations are crucial factors that significantly influence indoor air simulations. These parameters are critical in determining air distribution, ventilation effectiveness, and contaminant removal within indoor environments. The placement of inlets and outlets affects the airflow patterns and the distribution of fresh air within the indoor environment. Strategic placement of supply air diffusers and exhausts can ensure efficient air mixing, reduce the residence time of contaminants, and promote thermal comfort.

According to (Xu et al., 2022), Mixing ventilation (MV) and Displacement ventilation (DV) are two common approaches for distributing air within indoor environments. In mixing ventilation, high-speed air is released from upper diffusers, causing the supplied air to mix with the surrounding air. Displacement ventilation, on the other hand, involves supplying cool air from lower diffusers, utilizing convective thermal flow around heat sources, and expelling it from the top of the room. Personalized ventilation (PV) is another ventilation strategy that has recently been used for indoor environments. In personal ventilation, fresh air can be directly supplied for inhalation purposes or exhaled aerosols can be directly exhausted from their source. Downward ventilation (DWV), Protected zone ventilation (PZV), and Stratum ventilation (SV) are among other strategies used in literature.

Air Changes per Hour (ACH) is an important term of ventilation design that refers to the number of times the entire volume of air within a space is exchanged with fresh air per hour. ACH is a key parameter for quantifying ventilation rates and determining indoor air quality. Increasing the ventilation rate alone does not always ensure better contamination control. Therefore, it's important to consider the ventilation airflow pattern and the efficiency of air changes in order to achieve effective contamination control (Wang et al., 2018a).

Table 1 provides comprehensive details regarding the type of room, the ventilation strategy employed, and the Air Changes per Hour (ACH) values, wherever available, for the studies examined in this review. ACH values in the reviewed studies varied significantly, ranging from 0.5 for one of the residential building cases to as high as 100 for specific cases, such as operating rooms.

3.5. Turbulence models

Turbulence modeling is crucial in simulating indoor airflow in CFD simulations. Indoor environments are often characterized by complex flow patterns, including turbulence, which can significantly impact factors such as air quality, thermal comfort, and energy efficiency.

Researchers and engineers can better understand and optimize indoor airflow conditions by accurately modeling turbulence. The main turbulent models commonly used in CFD simulations are Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), and Reynolds-Averaged Navier-Stokes (RANS). DNS resolves all the relevant spatial and time scales of the flow and none of the eddies are modeled but it needs a very high computational cost and resources. Hence it is mainly used for basic flows in simple geometries.

RANS models, such as the k-ε, k-ω, and their variations, are widely used in indoor airflow simulations due to their computational efficiency compared to DNS and LES. RANS models solve the time-averaged Navier-Stokes equations and provide insights into the mean flow characteristics.

The renormalization group k-ε turbulence model (RNG), realizable k-ε turbulence model, standard k-ε turbulence model, and Shear stress transport model k-ω(SST) are among the most employed RANS turbulence models in indoor airflow simulations. These models involve the estimation of turbulence kinetic energy (k) and its dissipation rate (ε or ω) to calculate the turbulent viscosity and model the turbulence effects.

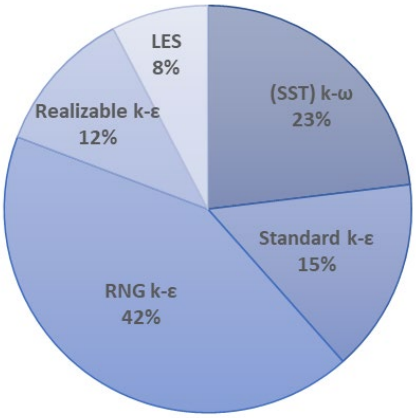

Figure 4: Turbulent models Used in selected Articles.

According to *Figure 4*, (RNG) k-ε was the most prevalent turbulence model among the reviewed studies, accounting for 42% of the cases. LES is a turbulence modeling approach that resolves the larger eddies in the flow while modeling the smaller dissipative scales. Due to its high computational expense, this model was used less frequently compared to other turbulence models and represents 8% of the models used in the reviewed studies.

Table 1: Geometry Category and Relevant Research Studies; Type of Room; Ventilation Strategy, and ACH (Where Available); Average Mesh Density; Turbulent Model

Types of Indoor Spaces		**Study**	**Ventilation Strategy**	**ACH**	**Average Mesh Density cell/m3**	**Turbulent model**
Hospital Environments	Patient ward	(Almhafdy et al., 2023)			29167	SST k-ω
		(Satheesan et al., 2020)	Mechanically ventilated (with a positive pressure towards the corridor), with and without local exhaust grilles	3 6 9 13		RNG k-ε
		(Lu et al., 2020)	Stratum ventilation Mixing ventilation Downward ventilation Displacement ventilation	12	8283	RNG k-ε
		(Aganovic et al., 2019)	Protected occupied zone ventilation	1.57 2.36 3.15 3.94 4.73	108696	SST k-ω

	Operating room	(Sadeghian et al., 2022)			40000	RNG k-ε / Realizable k-ε
		(Rahman et al., 2018)			6667	RNG k-ε
		(Wang et al., 2018)	Vertical laminar airflow ventilation Temperature-controlled airflow Mixing ventilation	26 46 100	28925	Realizable k-ε
		(Sadrizadeh et al., 2014)	Mixing ventilation system	47	19048	RNG k-ε
	Isolation-room	(Kalliomaki et al., 2020.)	Overhead mixing ventilation Local downward ventilation with background mixing ventilation Zonal downward ventilation	12	46939	SST k–ω LES
	Intensive care units (ICU)	(Ismail et al., 2023)	Vertical laminar airflow Horizontal laminar airflow Temperature controlled airflow		7206	Standard k-ε
Transportation Spaces	Aircraft cabin	(Rai & Chen, 2012)			85263	RNG k-ε
	Vehicle cabin	(Chang et al., 2023)			77778	SST k-ω/ RNG k-ε
Educational Spaces	Classroom	(Mirzaie et al., 2021)			6458	RNG k-ε
		(Pirouz et al., 2021)			7211	k-ε
	Lecture Room	(Arpino et al., 2023)			8205	SST k-ω
		(Lin et al., 2015)			369	RNG k-ε
Office		(Pirouz et al., 2021)			9059	k-ε
Restaurant		(Li et al., 2021)				RNG k-ε
Residential Spaces		(Bahramian et al., 2023)	Mixing ventilation	15	35000	RNG k-ε
		(Zong et al., 2023)	Displacement ventilation	0.51	27679	SST k-ω
		(Liu et al., 2023)	Make-up air organization from window		63272	Standard k-ε
		(Feng et al., 2020)			66667	LES
		(A. Zhang et al., 2019)			36765	Standard k-ε
		(Plana-Fattori et al., 2014)			15714	Standard k-ε
Museum		(Bakry et al., 2022)			18450	Realizable k–ε

4. Summary and Discussions

In conclusion, this review paper analyzed 25 research papers focusing on geometry and grid generation, ventilation strategies, and turbulence model selection in the context of indoor airflow simulations. This review investigated various geometries with varying dimensions and complexities, reflecting the diverse indoor environments. The tetrahedral mesh was the most frequently employed cell type among the different mesh types. The RNG k-ε model was the most used in studied papers regarding turbulence models. Furthermore, a comprehensive overview of the various ventilation strategies employed in each type of indoor environment is presented. This information provides valuable insights into the diverse approaches to ensure optimal air quality and circulation in various settings.

References

Aganovic, A., Steffensen, M., & Cao, G. (2019). CFD study of the air distribution and occupant draught sensation in a patient ward equipped with protected zone ventilation. *Building and Environment*, *162*. https://doi.org/10.1016/j.buildenv.2019.106279

Almhafdy, A., Korany, H. Z., AlSaleem, S. S., & Cao, S.-J. (2023). Airflow distribution in hospital isolation rooms with different ventilation and exhaust vent configurations. *Indoor and Built Environment*, 1420326X2311774. https://doi.org/10.1177/1420326X231177460

Arpino, F., Cortellessa, G., D'Alicandro, A. C., Grossi, G., Massarotti, N., & Mauro, A. (2023). CFD analysis of the air supply rate influence on the aerosol dispersion in a university lecture room. *Building and Environment*, *235*, 110257. https://doi.org/10.1016/j.buildenv.2023.110257

Bahramian, A., Mohammadi, M., & Ahmadi, G. (2023). Effect of indoor temperature on the velocity fields and airborne transmission of sneeze droplets: An experimental study and transient CFD modeling. *Science of the Total Environment*, *858*. https://doi.org/10.1016/j.scitotenv.2022.159444

Bakry, M. S., Hamdy, M., Mohamed, A., & Elsayed, K. (2022). Energy saving potential in open museum spaces: A comparative hygrothermal microclimates analysis. *Building and Environment*, *225*. https://doi.org/10.1016/j.buildenv.2022.109639

Chang, T. B., Lin, Y. S., & Hsu, Y. T. (2023). CFD simulations of effects of recirculation mode and fresh air mode on vehicle cabin indoor air quality. *Atmospheric Environment*, *293*. https://doi.org/10.1016/j.atmosenv.2022.119473

Feng, G., Bi, Y., Zhang, Y., Cai, Y., & Huang, K. (2020). Study on the motion law of aerosols produced by human respiration under the action of thermal plume of different intensities. *Sustainable Cities and Society*, *54*. https://doi.org/10.1016/j.scs.2019.101935

Ismail, Y. A., Eldosoky, M. A. A., Rashed, M. R., & Soliman, A. M. (2023). Numerical investigation of indoor air quality in health care facilities: A case study of an intensive care unit. *Journal of Building Engineering*, *68*. https://doi.org/10.1016/j.jobe.2023.106143

Kalliomaki, P., Koskela, H., Waris, M., & Wei-Tze Tang, J. (2020). *Assessing the risk to healthcare workers of hospital-acquired infection from patients infected with aerosol-transmissible pathogens Transmission of respiratory viruses View project Rhinovirus View project*. www.iosh.com/reducing-hospital-infections

Kohanski, M. A., Lo, L. J., & Waring, M. S. (2020). Review of indoor aerosol generation, transport, and control in the context of COVID-19. *International Forum of Allergy and Rhinology*, *10*(10), 1173–1179. https://doi.org/10.1002/alr.22661

Li, Y., Qian, H., Hang, J., Chen, X., Cheng, P., Ling, H., Wang, S., Liang, P., Li, J., Xiao, S., Wei, J., Liu, L., Cowling, B. J., & Kang, M. (2021). Probable airborne transmission of SARS-CoV-2 in a poorly ventilated restaurant. *Building and Environment*, *196*. https://doi.org/10.1016/j.buildenv.2021.107788

Lin, S., Tee, B. T., & Tan, C. F. (2015). Indoor airflow simulation inside lecture room: A CFD approach. *IOP Conference Series: Materials Science and Engineering*, *88*(1). https://doi.org/10.1088/1757-899X/88/1/012008

Liu, Y., Li, C., Ma, H., & Dong, J. (2023). Investigation on the indoor environment during a whole cooking process under constant make-up air organization in a Chinese-style residential kitchen. *Indoor and Built Environment*. https://doi.org/10.1177/1420326X231152554

Liu, Y., Long, Z., & Liu, W. (2022). A semi-empirical mesh strategy for CFD simulation of indoor airflow. *Indoor and Built Environment*, *31*(9), 2240–2256. https://doi.org/10.1177/1420326X221089825

Lu, Y., Oladokun, M., & Lin, Z. (2020). Reducing the exposure risk in hospital wards by applying stratum ventilation system. *Building and Environment*, *183*. https://doi.org/10.1016/j.buildenv.2020.107204

Marashian, S., Sadrizadeh, S., & Abouali, O. (2022). Modeling particle distribution in a ventilated room with modified discrete random walk methods. *International Journal of Ventilation*. https://doi.org/10.1080/14733315.2022.2143062

Mirzaie, M., Lakzian, E., Khan, A., Warkiani, M. E., Mahian, O., & Ahmadi, G. (2021). COVID-19 spread in a classroom equipped with partition – A CFD approach. *Journal of Hazardous Materials*, *420*. https://doi.org/10.1016/j.jhazmat.2021.126587

Morawska, L., Tang, J. W., Bahnfleth, W., Bluyssen, P. M., Boerstra, A., Buonanno, G., Cao, J., Dancer, S., Floto, A., Franchimon, F., Haworth, C., Hogeling, J., Isaxon, C., Jimenez, J. L., Kurnitski, J., Li, Y., Loomans, M., Marks, G., Marr, L. C., … Yao, M. (2020). How can airborne transmission of COVID-19 indoors be minimised? In *Environment International* (Vol. 142). Elsevier Ltd. https://doi.org/10.1016/j.envint.2020.105832

Pirouz, B., Palermo, S. A., Naghib, S. N., Mazzeo, D., Turco, M., & Piro, P. (2021). The role of hvac design and windows on the indoor airflow pattern and ach. *Sustainability (Switzerland)*, *13*(14). https://doi.org/10.3390/su13147931

Plana-Fattori, A., Trelea, I. C., Le Page, J. F., Souchon, I., Pollien, P., Ali, S., Ramaioli, M., Pionnier-Pineau, E., Hartmann, C., & Flick, D. (2014). A novel approach for studying the indoor dispersion of aroma through computational fluid dynamics. *Flavour and Fragrance Journal*, *29*(3), 143–156. https://doi.org/10.1002/ffj.3190

Rahman, M. N. Y., Razlan, Z. M., Izham, M., Omar, M. I., Zambri, N. A. A., Shahriman, A. B., Zunaidi, I., & Wan, W. K. (2018). A Test of Possibility on Relative Humidity Function in Minor Operation Theatre. *IOP*

Conference Series: Materials Science and Engineering, *429*(1). https://doi.org/10.1088/1757-899X/429/1/012089

Rai, A. C., & Chen, Q. (2012). Simulations of ozone distributions in an aircraft cabin using computational fluid dynamics. *Atmospheric Environment*, *54*, 348–357. https://doi.org/10.1016/j.atmosenv.2012.02.010

Sadeghian, P., Bi, Y., Cao, G., & Sadrizadeh, S. (2022). Reducing the risk of viral contamination during the coronavirus pandemic by using a protective curtain in the operating room. *Patient Safety in Surgery*, *16*(1). https://doi.org/10.1186/s13037-022-00332-x

Sadrizadeh, S., Tammelin, A., Ekolind, P., & Holmberg, S. (2014). Influence of staff number and internal constellation on surgical site infection in an operating room. *Particuology*, *13*(1), 42–51. https://doi.org/10.1016/j.partic.2013.10.006

Satheesan, M. K., Mui, K. W., & Wong, L. T. (2020). A numerical study of ventilation strategies for infection risk mitigation in general inpatient wards. *Building Simulation*, *13*(4), 887–896. https://doi.org/10.1007/s12273-020-0623-4

Śmiełowska, M., Marć, M., & Zabiegała, B. (2017). Indoor air quality in public utility environments—a review. *Environmental Science and Pollution Research*, *24*(12), 11166–11176. https://doi.org/10.1007/s11356-017-8567-7

Wang, C., Holmberg, S., & Sadrizadeh, S. (2018a). Numerical study of temperature-controlled airflow in comparison with turbulent mixing and laminar airflow for operating room ventilation. *Building and Environment*, *144*, 45–56. https://doi.org/10.1016/j.buildenv.2018.08.010

Xu, C., Liu, W., Luo, X., Huang, X., & Nielsen, P. V. (2022). Prediction and control of aerosol transmission of SARS-CoV-2 in ventilated context: from source to receptor. *Sustainable Cities and Society*, *76*. https://doi.org/10.1016/j.scs.2021.103416

Zhang, A., Zhen, Q., Zheng, C., Li, J., Zheng, Y., Du, Y., Huang, Q., & Zhang, Q. (2023). Assessing the impact of architectural and behavioral interventions for controlling indoor COVID-19 infection risk: An agent-based approach. *Journal of Building Engineering*, *74*. https://doi.org/10.1016/j.jobe.2023.106807

Zhang, B., Guo, G., Zhu, C., & Ji, Z. (2019). Transport of aerosol by coughing in an air-conditioned space. *Proceedings of the Thermal and Fluids Engineering Summer Conference*, *2019-April*, 1341–1353. https://doi.org/10.1615/TFEC2019.hbe.028480

Zong, J., Ai, Z., & Ma, G. (2023). Accurate evaluation of inhalation exposure based on CFD predicted concentration in the breathing zone towards personalized and smart control. *Journal of Building Engineering*, *71*. https://doi.org/10.1016/j.jobe.2023.106404