Research on the Influence of Window Structure on Indoor Ventilation Based on CFD

Daiwei Gan, Junbing Pan*, Jiazhong Xu, Rongli Li, Yuhua Luo, Xin'an Zhou

Engineering Research Center of Aerospace High-end Manufacturing-Universities of Shaanxi Province, Shaanxi Institute of Technology, Xi'an, China

*e-mail: 1210211014@stu.xaut.edu.cn

Abstract. Indoor air circulation is significant in maintaining indoor air quality and comfort. As one of the main channels for indoor and outdoor air exchange, the design of window structure is essential for indoor air circulation. Based on the simulations of three window structures using ANSYS Fluent software using computational fluid dynamics (CFD) method, this study aims to investigate the effects of different window structures on indoor air circulation. The results show that left-right sliding window and push-pull rotating composite window performed well in improving indoor air circulation. These findings can provide a basis for the selection of architectural windows and provide a reference for the optimization of window structures.

Keywords: CFD simulation; Window structure; Indoor air circulation; Indoor air quality; Optimal design

1 Introduction

Recently, the study on indoor air ventilation has been widely concerned as people pay more and more attention to indoor comfort and health [1]. Indoor ventilation system plays as a key part in maintaining fresh indoor air, purifying indoor air, and regulating indoor humidity and temperature [2]. In the indoor ventilation system, window structure serves as a medium between indoors and outdoors, which is crucial to improving indoor ventilation [3].

Computational fluid dynamics (CFD) is an engineering technology based on mathematical methods and computer simulations that can be used to simulate and analyze fluid flow, heat transfer, and mass transfer [2~4]. In the study of window structures, CFD is widely used to simulate and analyze the effects of different window structures on indoor ventilation [5].

The design and selection of window structures are important for indoor ventilation. An excellent window structure can achieve a good ventilation effect, ensuring the inflow of fresh air and the discharge of vitiated air. Through CFD simulation, researchers can predict the airflow distribution, temperature distribution, humidity distribution, and other key parameters of different window structures to optimize the design of window structure and improve indoor ventilation effect.

G. Zhao et al. (eds.), *Proceedings of the 2024 6th International Conference on Civil Architecture and Urban Engineering (ICCAUE 2024)*, Advances in Engineering Research 262,
https://doi.org/10.2991/978-94-6463-688-8_40

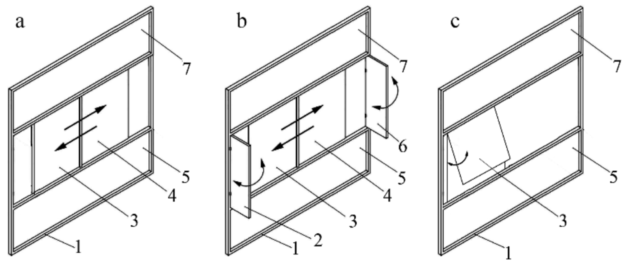
Therefore, the study of the effects of CFD-based window structures on indoor ventilation can provide guidance for the design and optimization of window structures and help to improve indoor air quality, provide a comfortable indoor environment, and reduce the risk of disease transmission and indoor air pollution.

In summary, window structure has a significant impact on indoor ventilation. By studying the diversity of window structures, the flexibility of their opening modes, and the thermal conductivity of the window materials, researchers can better understand the mechanism of the windows influencing indoor ventilation, so as to provide scientific and effective design suggestions to optimize indoor ventilation environment, improve indoor air quality, and create a more comfortable and healthier living environment for residents.

2 Experimental Methods and Calculation Models

2.1 Experimental Methods

In this study, three common window structures were simulated to investigate the impacts of different window structures on indoor ventilation. The three structures were left-right sliding window (L-RSW), push-pull rotating composite window(P-PRCW), and up-down rotating window (U-PRW) (Fig. 1). An indoor CFD flow channel model was established for each window to simulate the indoor airflow under different window structures. During the simulation, the opening state of the window and the outdoor wind speed were considered, and a standard turbulence model and boundary conditions were used for the simulation [3~5].

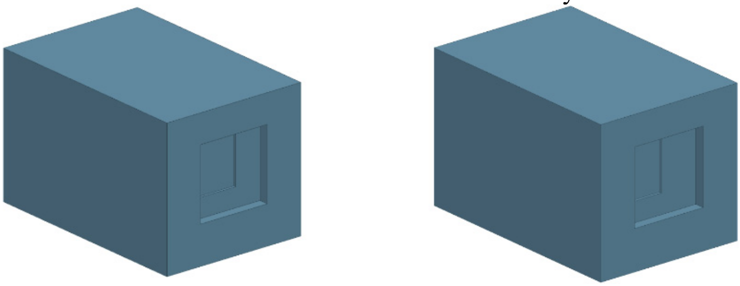


a. Left-right sliding window b. Push-pull rotating composite window c. Up-down rotating window
1. Window frame 2. Left rotating sash 3. Left push-pull sash 4. Right, push-pull sash
5. Lower fixed sash 6. Right rotating sash 7. Upper fixed sash

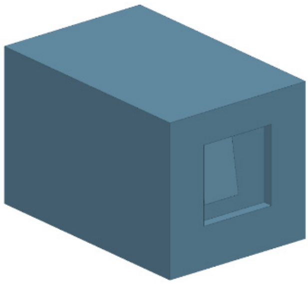
Fig. 1. Window structures

2.2 Mesh Division

Corresponding 3D models of the window structures were established in the 3D modeling software UG, and relevant flow channels were extracted (Fig. 2) and imported into Gambit software for mesh division (Fig. 3). For the mesh type, the tetrahedral unstructured mesh with strong adaptability was used. The minimum mesh was 4 mm in size, the maximum mesh was 15 mm in size, the mesh augmentation ratio was 1.05, and the meshes were encrypted under the inlet and outlet boundary conditions. The total number of divided meshes for the left-right sliding window flow channel was 89,856, the total number of divided meshes for the push-pull rotating composite window flow channel was 91,385, and the total number of meshes for the up-down rotating window flow channel was 98,233. All the divided meshes were saved in *.msh format. In order to meet the calculation requirements, the *.msh meshes were opened with Fluent software, and one end of the window inlet was set as inlet boundary condition, and the other end of the outlet was set as outlet boundary condition.



a. Left-right sliding window flow channel b. Push-pull rotating composite window flow channel



c. Up-down rotating window flow channel

Fig. 2. Window flow channel models

2.3 Mathematical Model

When the air passes the window, the Reynolds number of the airflow in the room was R_e=9.8×10^4, and the flow type was regarded as turbulent. Standard *k-ε* was used in the Fluent calculation, and the airflow was incompressible which satisfies the continuity equation:

$$\frac{\partial V_x}{\partial x}+\frac{\partial V_y}{\partial y}+\frac{\partial V_z}{\partial z}=0 \quad (1)$$

Where, V_x, V_y, and V_z are velocity vectors in X, Y, and Z directions, respectively.

Momentum conservation equation:

$$\left.\begin{aligned}\frac{\partial(\rho u)}{\partial t}+\operatorname{div}(\rho u)&=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+F_x\\ \frac{\partial(\rho v)}{\partial t}+\operatorname{div}(\rho v)&=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+F_y\\ \frac{\partial(\rho w)}{\partial t}+\operatorname{div}(\rho w)&=-\frac{\partial p}{\partial z}+\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+F_z\end{aligned}\right\} \tag{2}$$

Where, p is the pressure, Pa; τ_{xx}, τ_{xy}, and τ_{xz} are the components of viscous stress τ in the three directions, respectively, Pa; F_x, F_y, and F_z are the mass forces. If the mass force only includes gravity, then F_x=0, F_y=0, and F_z=-G。

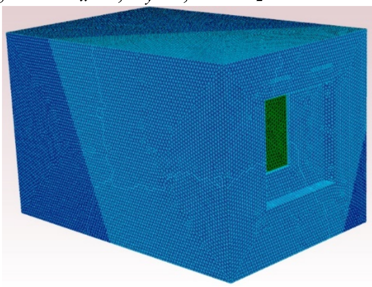
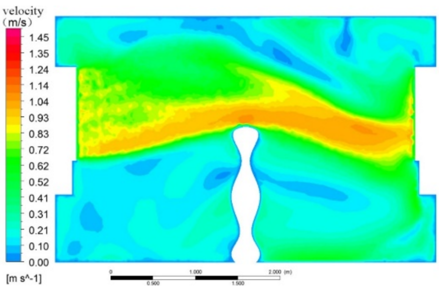


Fig. 3. Mesh division of window flow channel models

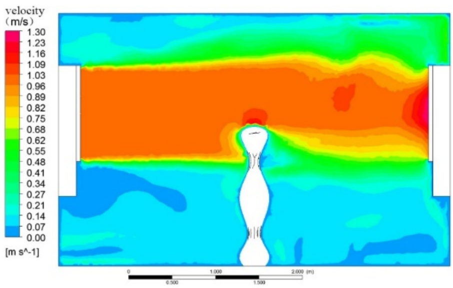
3 Simulation Results and Analyses

3.1 Simulation Results

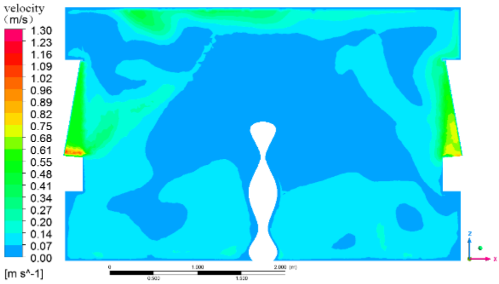
Before simulation calculations, the space needs to be divided into mesh models first to accurately present the window structure and the airflow flow status (Fig.3). When defining the boundary conditions and parameters, the material, size, and opening mode of the window, as well as the airflow rate and temperature should be considered. The accuracy and reliability of the calculation model should be also ensured to achieve reliable simulation results. During the simulation calculations in Fluent software, the airflow rate was set to 1 m/s, and the number of iteration steps was set to 50,000 steps. In each iteration step, the software would calculate according to the preset boundary conditions and parameters and update the distribution of the airflow velocity field. During the calculation process, the change of the residual curve was monitored until the curve converges when it means the calculation results have been stabilized. Finally, the simulation calculations were performed using Fluent software, and three simulation airflow velocity cloud maps were obtained. Fig. 4a was the simulation velocity cloud of left-right sliding window; Fig. 4b was the simulation velocity cloud of push-pull rotating composite window; Fig. 4c was the simulation velocity cloud of up-down rotating window. These simulation results can intuitively show the airflow status of different types of windows, which can provide a reference basis for window design and airflow optimization.



a. Simulation velocity cloud of left-right sliding window



b. Simulation velocity cloud of push-pull rotating composite window



c. Simulation velocity cloud of up-down rotating window

Fig. 4. Simulation results cloud maps of different window structures

3.2 Analyses of Simulation Results

Fig.4 shows the airflow of three different window structures and their effects on indoor air circulation. As shown, the airflow directions of the three window structures were all from left to right. In Figure 4a, the maximum velocity of the left-right sliding window was 0.98 m/s, and the mean value was 0.62 m/s. The airflow was mainly distributed in the direction of the two-end window streamlines, with a large energy loss, and the indoor air circulation was relatively fast; in Figure 4b, the maximum velocity of the push-pull rotating composite window was 1.26 m/s, and the mean value was 0.89 m/s. The airflow was mainly distributed in the direction of the two-end window streamlines with a high flow rate, and the airflow can promote indoor air circulation; in Figure 4c, the maximum velocity of the up-down rotating window was 0.92 m/s, and the mean value was 0.25 m/s. The airflow was disturbed in the room, and finally flew out from the exit window, with less energy loss, the lowest airflow velocity, the longest indoor air exchange time, and low air exchange efficiency. To sum up, push-pull rotating composite window has obvious advantages in air exchange efficiency, which can quickly and effectively accomplish indoor air circulation and improve indoor air quality. The following window type is left-right sliding window which can change air quickly but with larger energy loss. The up-down rotating window has the lowest air exchange efficiency. Although it can reduce energy loss, its air exchange speed is the lowest. Therefore, suitable window types should be selected according to the actual needs and environmental conditions to improve indoor air quality and comfort.

4 Experimental Verification

In this study, the Shima AS836 handheld anemometer (Fig.5) was used to detect the wind speed in the indoor space corresponding to the three windows, and the windows were oriented in the east-west direction. The external environment temperature is 18 °C, westerly wind, wind speed 1m / s. From the east side of the window to the west side of the window, 1.7 m from the ground height measurement (the height of the person standing breathing), every 0.5 m as a measurement interval. The experimental results are shown in Fig.6. By comparing the experimental results with the simulation results, the error is within 10%, which meets the engineering requirements and proves the reliability of the simulation.

Fig. 5. Shima AS836 handheld anemometer

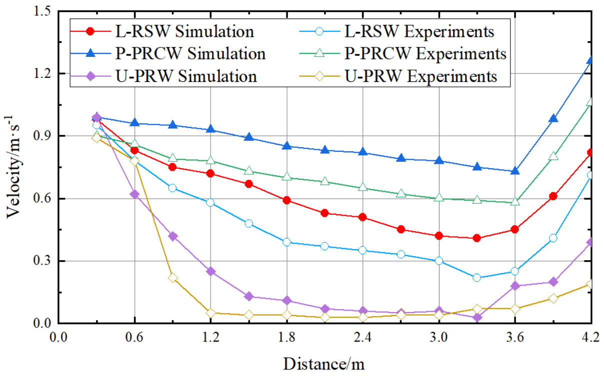


Fig. 6. Experimental and simulation results

5 Results and Discussions

Results: According to the simulation results, different window structures have significant effects on indoor air circulation. When designing a new building, windows should be selected not only for aesthetic and lighting purposes but also for their effects on the indoor air quality. Especially in modern buildings with high airtightness, the choosing of window structures is particularly important. Up-and-down rotating windows can limit indoor air circulation, causing uneven distribution of indoor airflow and decreasing the indoor air quality. In contrast, left-right sliding windows and push-pull rotating composite windows can better refresh indoor air and improve indoor air quality. In particular, push-pull rotating composite windows can maximize the exchange of indoor and outdoor air, and effectively improve indoor air quality.

Discussions: In modern architectural design, the window is not only a tool for lighting and ventilation, its structure and design directly affect the indoor air exchange rate and energy consumption. The left-right sliding window and the push-pull rotating composite window have a large ventilation area, which can speed up the indoor air update speed and effectively improve the indoor air quality. In particular, the push-pull rotating composite window have the largest ventilation area, which can maximize the indoor and outdoor air exchange and improve the indoor air quality. Appropriate window layout can promote air circulation, reduce dependence on mechanical ventilation systems, and thus reduce energy consumption.

The results of this study have shown the significant effects of window structures on indoor air circulation. Architects should choose suitable window structures when designing buildings so as to achieve good indoor air circulation. Based on the CFD simulation method, building designers can better understand the effects of different window structures on indoor air circulation and provide a reference basis for optimal designs. This is of great significance for improving the indoor environmental quality and living comfort of buildings. In the later stage, the study of window structure on indoor air quality will continue.

6 Conclusions

This study evaluates the impact of various window structures on indoor ventilation using CFD simulations in ANSYS Fluent, comparing left-right sliding, push-pull rotating composite, and up-down rotating windows. The results show that push-pull rotating composite and left-right sliding windows significantly improve indoor air circulation, offering insights for window selection in architectural design to enhance air quality and comfort.

Acknowledgment

This work was financially supported by 2024 Scientific Research Project of Shaanxi Institute of Technology of China (Pt24-05，Gfy24-06) and The sub-project of China Vocational Education Professional Resource Library (ZKZYK-2019Z16).

References

1. Vita, G., Woolf, D., Avery-Hickmott, T., et al. (2023) A CFD-based framework to assess airborne infection risk in buildings. Building and Environment., 233:110099-110099. https://doi.org/10.1016/j.buildenv.2023.110099.
2. Gao, N., Wang, R., Wu, Y., et al. (2022) Study on impact factors of tracer gas method in investigations of gaseous pollutant transport and building ventilation. Building Simulation., 16:413-426. https://doi.org/10.1007/s12273-022-0947-3.
3. Larkermani E., Cao G., and Georges L. (2022) Characterization of the density-driven counter-flow through a doorway using Large Eddy Simulation. Building and environment., 221. https://doi.org/10.1016/j.buildenv.2022.109319.
4. Zhou K., Li S., Cao R., et al. (2022) Optimal technology innovation of teaching buildings for primary and middle schools based on compound negative pressure ventilation. Applied Nanoscience., 13:2037-2048. https://doi.org/10.1007/s13204-021-02066-1.
5. Sinha K., Yadav M. S., Jayakrishnan R., et al. (2022) Field Experiments to Identify and Eliminate Recirculation Zones to Improve Indoor Ventilation: Comparison with CFD. Transactions of the Indian National Academy of Engineering: an international journal of engineering and technology., 7:911-926. https://doi.org/10.1007/s41403-022-00335-1.