International Journal of Natural Resources and Environmental Studies
ISSN: 3006-2012 (Print), ISSN: 3006-0834 (Online) | Volume 6, Number 3, Year 2025
DOI: https://doi.org/10.62051/ijnres.v6n3.03
Journal homepage: https://wepub.org/index.php/IJNRES/index

Numerical Simulation of Indoor Comfort Based on FLUENT

Yunqiao Luo

School of Civil Engineering, Central South University of Forestry and Technology, Changsha 410004, China

ABSTRACT

With the continuous development of the national economy, people's demand for using air conditioning systems to regulate thermal comfort in indoor buildings has been increasing, and related research is gradually deepening. However, there is still a lack of specific research on the thermal comfort of specific spaces such as bedrooms. According to research, humans spend about 80% of their lives indoors, so having good indoor conditions has become a key factor affecting the positive development of health, work, and learning efficiency [1]. In recent years, the use of numerical simulation software for numerical simulation analysis of indoor air conditioning to regulate thermal comfort and its application in solving practical engineering problems has gradually become a research hotspot. This method can quickly, efficiently, economically, and reliably simulate and predict indoor airflow distribution, providing safe, reliable, and comprehensive technical analysis results for relevant design departments. Due to the continuous advancement of computer technology and the deepening of research on indoor thermal comfort, the role of Computational Fluid Dynamics (CFD) simulation technology in enhancing thermal comfort and optimizing environmental design in building air conditioning system design will become increasingly significant. The main key factors affecting indoor thermal comfort include air temperature and humidity, wind speed, and thermal radiation. This paper will focus on regulating the temperature and airflow distribution of bedroom air conditioning systems and explore their impact on indoor comfort.

KEYWORDS

CFD Simulation; Air Conditioning Airflow; Temperature; Thermal Comfort.

1. INTRODUCTION

Since the origin of humanity, the exploration to improve the living environment has never ceased. With the progress of civilization, people's demand for buildings has expanded to multiple dimensions such as safety, health, comfort, aesthetics, and efficient work [2]. In the early 20th century, air conditioning equipment that operated throughout the year first appeared in American factories. Since then, air conditioning technology has gradually been applied to various types of artificial environment regulation, achieving precise creation of the environment required for human life and work. Indoor environmental quality is closely related to airflow and temperature. Airflow organization is related to the flow speed of indoor air, temperature distribution, human comfort, relative humidity, and cleanliness. If the airflow organization is not reasonably designed, it can lead to problems such as uneven temperature distribution indoors, direct cold air blowing in the work area causing significant cold sensation to the human body, large temperature difference between head and feet, and accumulation of dirty air in dead airflow zones that is difficult to exhaust [3]. A reasonable air conditioning system can effectively exhaust indoor pollutants, reduce their concentration, improve air quality, and thus enhance thermal comfort, providing a guarantee for a healthy and comfortable indoor environment. At the same time, it is of great significance for improving work efficiency and saving building energy consumption [4]. It can also provide practical references for thermal comfort technicians and designers, promoting research and development in related fields [5].

Currently, as the living standards of Chinese residents continue to improve, their demands for indoor thermal comfort and environmental quality are also on the rise. Research on indoor airflow, temperature, and thermal comfort aligns with the trend of the times. A good thermal comfort environment can maintain a pleasant state and enhance work efficiency. Relevant research shows that a suitable thermal environment can increase production efficiency by 18% [6]. Therefore, the air conditioning industry not only has a consumer attribute but also indirectly creates economic value [7]. Its research results can avoid energy waste caused by unreasonable air conditioning design, create a comfortable environment for people who spend long hours indoors, and contribute to physical and mental health as well as efficiency improvement.

2. PHYSICAL MODEL

2.1. Overview of Research Subjects

In this study, a typical apartment bedroom was constructed as the simulation object. The bedroom is of a rectangular structure, where activities such as sleeping, simple office work, and daily life are carried out. Due to the presence of furniture such as beds, desks, and wardrobes inside the actual bedroom, its complex structure significantly affects indoor airflow. Therefore, during the modeling process, reasonable simplifications were made to balance computational efficiency and accuracy.

2.2. Principles and Methods of Model Simplification

The modeling follows the principle of "focusing on the primary and ignoring the secondary": retaining objects that significantly affect airflow organization (such as beds, desks, and air conditioners) while ignoring decorative items (such as hanging pictures and small ornaments); simplifying the complex structure of furniture (such as simplifying the carved design on the head of the bed to a flat surface and merging the desk drawers into one). The basis for simplification is as follows:

As the core furniture in the bedroom, the bed frame, with a height of 0.5m, is directly related to the human lying posture and significantly obstructs the airflow near the ground, necessitating precise modeling. The desk, positioned on the air conditioning supply path, with its desktop (height 0.66m) altering the airflow direction, requires the retention of its main structure. As the source of airflow, the air conditioning unit's supply and return air outlet dimensions and positions play a decisive role in the entire flow field, necessitating strict modeling based on actual dimensions.

2.3. Model Parameters and Construction Tools

In this study, the SpaceClaim software in ANSYS 2022R2 was used to construct a three-dimensional model. This tool supports parametric modeling and allows for rapid adjustment of dimensions. The specific parameters of the model are as follows:

Room: Length 6.8m × Width 3.6m × Height 2.7m (conforming to the common dimensions of residential bedrooms in China); Bed: 2.0m × 1.8m × 0.5m (standard size for a double bed), placed on one side of the room, 0.6m away from the wall; Desk: 1.2m × 0.86m × 0.66m, located on the other side of the room, 2.5m horizontally away from the air conditioning outlet; Human body: Simplified as an upright cylindrical model with a height of 1.71m and a weight of 70kg (equivalent to the human body heat dissipation area), located in the central activity area of the room;

Air conditioner: vertical cabinet unit, dimensions 0.7m×0.5m×1.5m, with the air supply outlet (0.55m×0.2m) located in the middle of the front and the air return outlet (0.55m×0.54m) located at the lower part of the front, 0.3m above the ground.

The overall layout of the model is shown in Figure 1. Through simplification, the model retains more than 90% of the factors affecting airflow, while reducing the number of grids by about 30%, significantly improving computational efficiency.

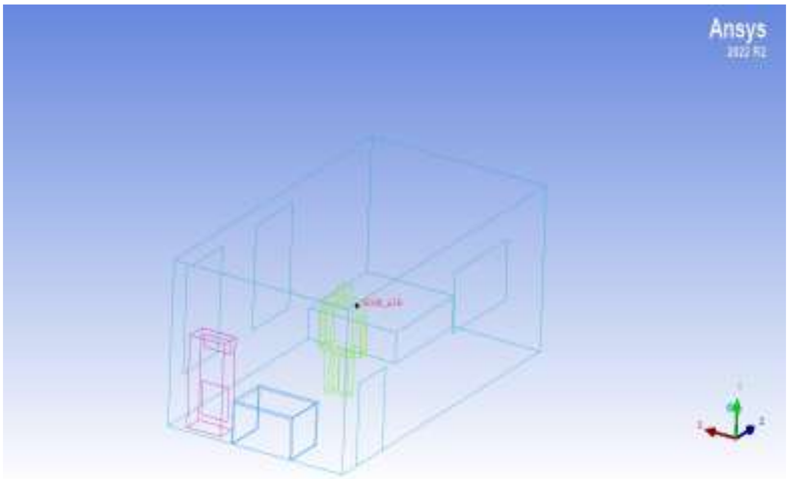


Figure 1: 3D diagram of bedroom

3. GRID DIVISION

After constructing the 3D model using Space Claim and saving it in scdox format, it was imported into ICEM for grid generation. Currently, the grids used in numerical calculations can be divided into two major categories: unstructured grids and structured grids. Given the geometric shape of the bedroom and the research purpose, an unstructured grid was adopted for grid generation, with local densification of the grid in areas with lower quality. The number of grids not only affects the calculation accuracy but also the calculation time. To ensure a reasonable grid generation, the grid was gradually densified locally, resulting in a total of 1,010,276 grids, as shown in Figure 2.

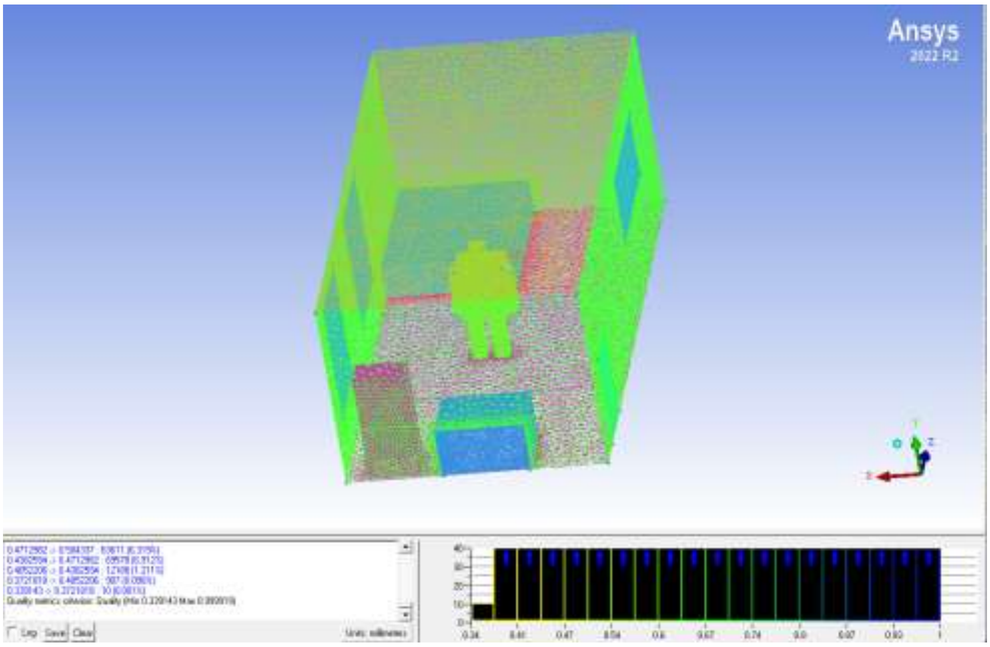


Figure 2: Grid division and quality statistical chart

4. NUMERICAL MODEL

If you follow the "checklist" your paper will conform to the requirements of the publisher and facilitate a problem-free publication process.

4.1. Control Equations for Fluid Flow

4.1.1. (1) The continuity equation of fluid

any form of fluid problem must satisfy the law of conservation of mass [8]. This law can be expressed as: the increase in mass of a fluid element in unit time is equal to the net mass of the inflowing element in the same time interval. The equation of mass conservation is as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

Where ρ represents the density of the fluid, and u represents the velocity vector of the fluid.

4.1.2. (2) Fluid momentum conservation equation

The law of conservation of momentum is also a fundamental law that fluid flow must satisfy. It states that the rate of change of the momentum of the fluid in a micro-element is equal to the sum of all external forces acting on the micro-element.

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right)\right] + \rho g_i + F_i \tag{2}$$

Where p represents the static pressure; Fi and gi represent the external body force and gravitational body force in the direction of i, respectively.

4.2. Numerical Simulation Method

This study employs FLUENT software to conduct simulation analysis on a residential bedroom model. Given that the gas flow field within the bedroom model is turbulent, numerous turbulence models are applicable to FLUENT. Choosing an appropriate turbulence model will impact the accuracy and convergence of the computational results. Due to the standard k−ε model's [9] wide application and adequate computational accuracy, this study utilizes this model to simulate the movement of airflow organization in the bedroom under unsteady conditions. This model necessitates the calculation and solution of the dissipation rate equation and the turbulent kinetic energy equation. The transport equation of the standard k−ε model is as follows:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{3}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \tag{4}$$

Where Gb represents the turbulent energy caused by buoyancy; Gk represents the turbulent kinetic energy generated by the average velocity gradient; YM represents the contribution of pulsating expansion in compressible turbulence; C1ε, C2ε, C3ε represent empirical constants; μ represents the turbulence viscosity; σk and σε represent the Prandtl numbers corresponding to k and ε, respectively; Sk and Sε represent user-defined source terms.

5. BOUNDARY CONDITIONS AND SOLUTION SETTINGS

5.1. Setting of Boundary Conditions

Based on the actual environment of the bedroom, the following boundary conditions are set: Air supply outlet: "Velocity Inlet" is adopted, with a wind speed of 3m/s (refer to the conventional air supply speed of a vertical air conditioner), a turbulence intensity of 5%, a turbulence viscosity ratio of 10, and an air supply temperature of 20℃ (typical setting in summer); Air return outlet: "Outflow" is adopted to simulate natural air return under atmospheric pressure; Human body surface: set to "Wall", with a heat flux of 80W/m² (corresponding to a medium metabolic rate, such as sitting still), and no slip boundary (wind speed = 0); Wall and furniture surfaces: both set to "adiabatic wall" (no slip except for heat transfer), with a wall heat transfer coefficient of 100W/(m²·K) (refer to the insulation performance of concrete walls); Floor and ceiling: set to "wall", with a temperature of 25℃ (environmental reference temperature).

5.2. Solver Settings

The simulation was solved using FLUENT 2022R2, with the following specific parameters: solver type: three-dimensional, transient; pressure-velocity coupling algorithm: SIMPLE algorithm (applicable to indoor low-speed flow); spatial discretization scheme: pressure term adopts the Standard scheme, while both k−ε and momentum equations adopt the second-order upwind scheme (to improve computational accuracy); time step: 0.1s, with a total of 5000 iteration steps (to ensure the flow field reaches steady state); convergence criterion: residuals (Residual) less than 1e-6 (continuity equation) and 1e-3 (energy equation), as shown in Figure 3.

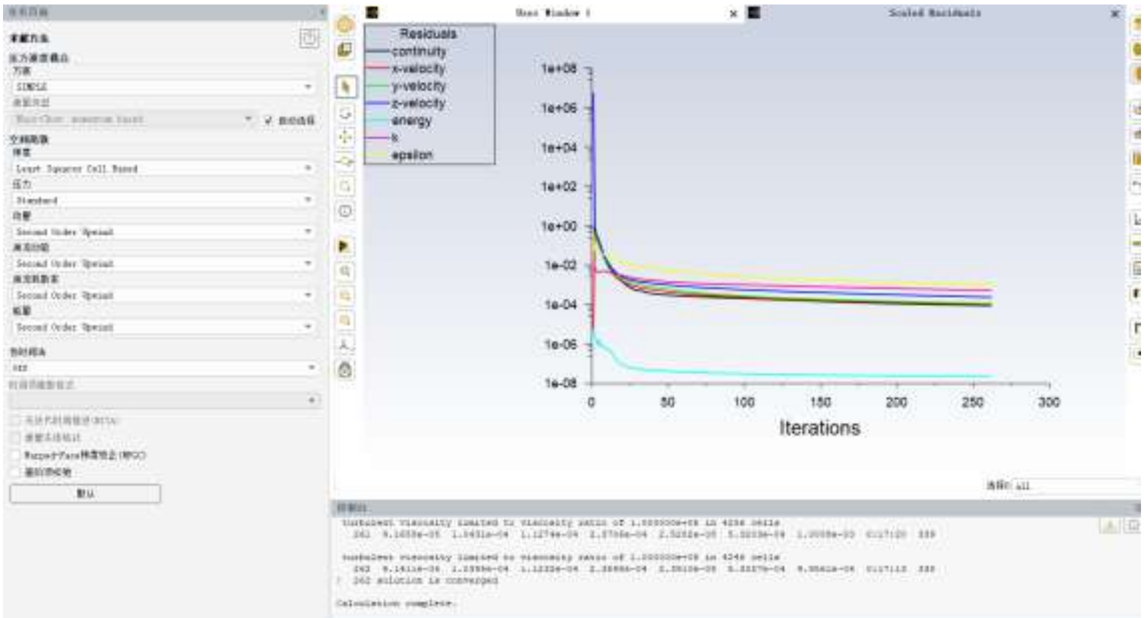
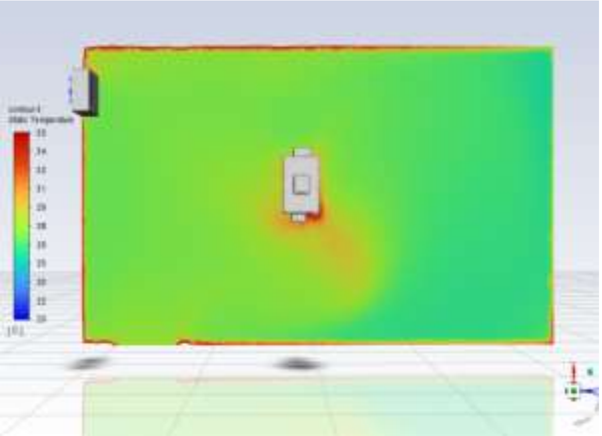


Figure 3 Simulation result diagram

6. RESULT ANALYSIS

This article employs the standard k-ε turbulence model for calculations [10]. The calculation results reveal that the cold air emitted from the air conditioning supply outlet first passes through the position of the bed, encounters the wall, and then diffuses to the side, gradually circling around the other side of the room, passing by the desk, and returning to the air conditioning unit through the return air inlet. After the calculation results converge, the room is in dynamic equilibrium. The cold air flows through the entire room, gradually reducing the high summer room temperature to a comfortable temperature for humans, as shown in Figures 4 and 5.



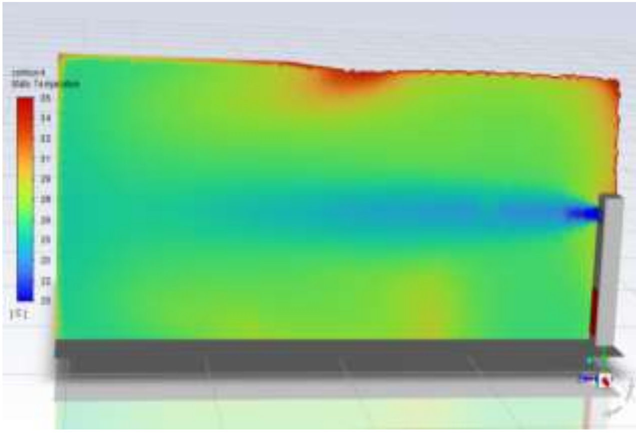


Figure 4 Temperature diagram of horizontal section **Figure 5** Temperature diagram of vertical section

In this way, the temperature meets the requirements for thermal comfort. From the position of the desk, the ambient air temperature is approximately 27°C, which is slightly high but meets the comfort requirements for human work, enhancing work efficiency; from the position of the bed, the temperature is approximately 26°C, which is a very comfortable temperature for sleep. In the middle activity area, the temperature of the layer of air closest to the human body reaches 31°C, as shown in Figures 8(a) and (b). This is due to the high heat flux set for simulating individuals with high metabolic rates. To meet the thermal comfort needs of such individuals, factors such as adjusting indoor relative humidity, human activity metabolic rate, and clothing thermal resistance can be implemented. Through the above analysis, it can be concluded that in this bedroom, whether one is working or sleeping, the environment is relatively comfortable.

In addition, the simulation results indicate that the velocity field of the bed rest area is mostly at 0.4/s, and the head area is at 0.5/s. Therefore, the room airflow velocity obtained under this design condition is relatively high, and the distribution of the velocity field does not meet the design requirements, as shown in Figure 6.

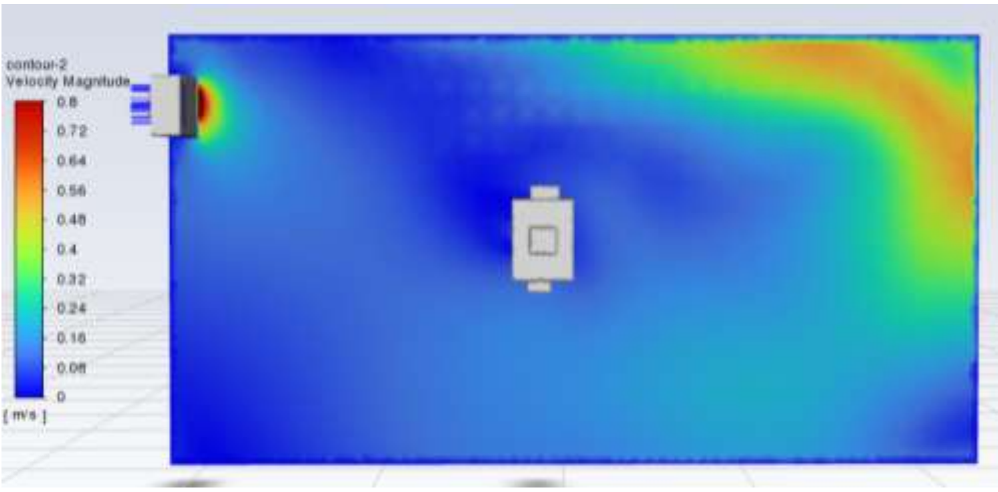


Figure 6 Velocity diagram of horizontal section

In the living area of the human body, most of the temperature is controlled at 27~28°C: the velocity field of the resting parts of the bed is mostly between 0.08-0.24m/s, and the head is at 0.25 m/s. The simulation results are relatively close to the design requirements, with ideal airflow velocity, but the temperature is slightly higher, as shown in Figure 7.

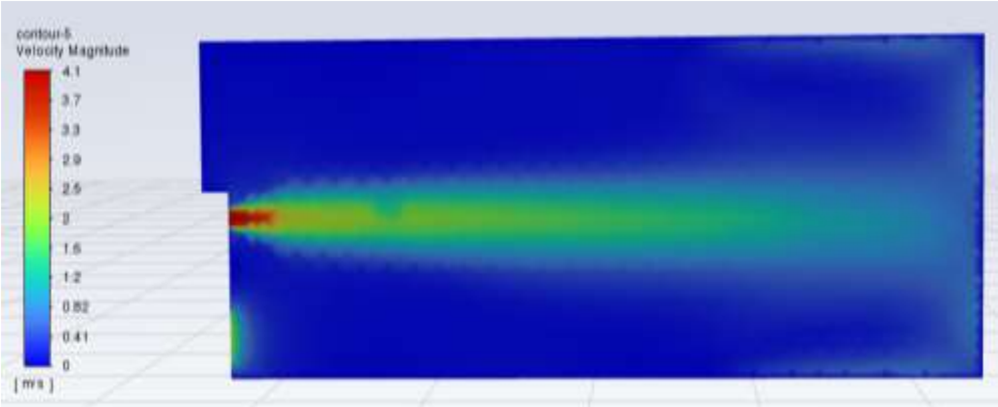


Figure 7 Velocity diagram of vertical interface

Based on the above operating conditions, the air supply and delivery energy are increased, and the size of the air supply and return outlets is enlarged to 1000mm×200mm, while other parameters remain unchanged. Simulation data analysis: The temperature in the human activity zone is controlled at 26.5℃, and the temperature at the bedside is controlled at 27℃. The velocity field in most parts of the bed rest area is at 0.25 m/s, and the velocity in the head area is close to 0.2 m/s. The simulation results are relatively close to the design requirements, with ideal air conditioning temperature and airflow velocity that basically meets the requirements. This operating condition basically meets the design requirements.

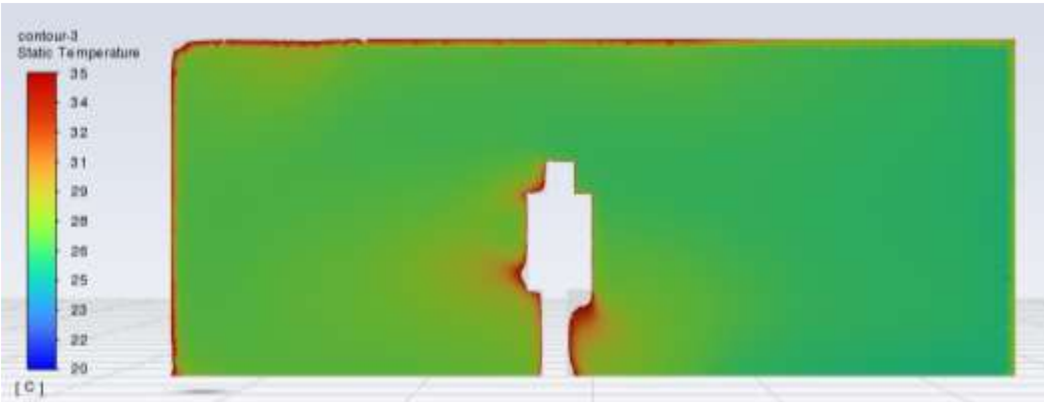


Figure 8(a) Temperature map around the human body

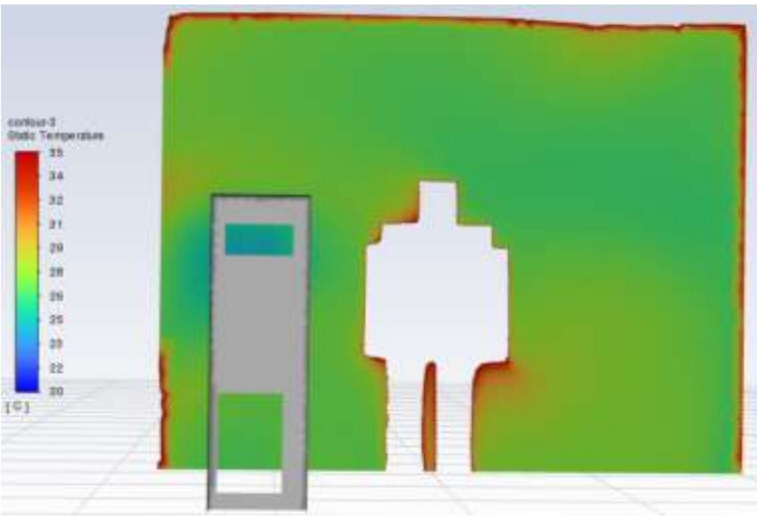


Figure 8(b) Temperature map around the human body

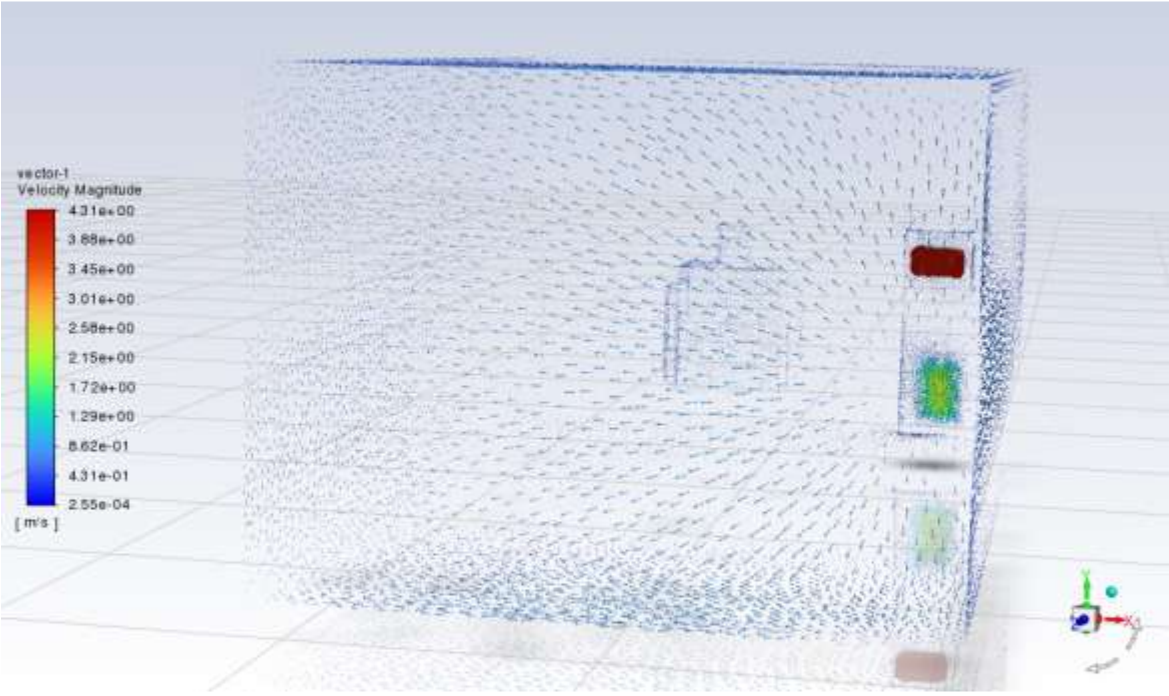


Figure 9 Velocity vector diagram

7. CONCLUSION AND OUTLOOK

This article employs the CFD simulation software ANSYS FLUENT to conduct numerical simulation and analysis of the temperature and flow fields under air conditioning conditions within a bedroom model. The results indicate that FLUENT can effectively simulate indoor temperature distribution and air flow velocity, ensuring that individuals in the bedroom are in a relatively comfortable state during work and rest. The research presented in this article provides verification of the use of CFD software for numerical simulation of indoor environments, offering a basis and experience for the use of numerical simulation in regulating more complex indoor air conditioning and ventilation systems. It also serves as a reference for the design of indoor air conditioning thermal comfort. By utilizing the professional software SpaceClaim to establish models and combining it with Fluent for iterative calculations, data with minimal errors can be obtained, providing a basis for indoor flow and temperature fields. Through modeling and calculating the bedroom, SpaceClaim and Fluent can achieve good numerical simulation. However, the selection of model parameters and calculation parameters can introduce certain deviations to the calculation results, which requires further exploration. In this model, the influence of objects such as wardrobes and curtains is simplified and neglected. In the future, a refined model can be constructed, taking into account humidity factors, actual comfort, and combining temperature and humidity index analysis. Furthermore, it can be extended to discuss different seasons and conduct comparative studies on different types of air conditioning (such as wall-mounted and ducted air conditioning).

REFERENCES

[1] Jing, B. K. (2023). A study on the measurement of indoor thermal environment and optimization of air - conditioning system in large - scale building spaces [Dissertation]. Shandong Jianzhu University. DOI: 10.27273/d.cnki.gsajc.2023.000861.

[2] Hu, C. Y., Kang, S. Y., Li, H., & others. (2023). A study on the simulation and optimization of indoor air distribution in a sports venue [J]. Jiangsu Architecture, (1), 128 - 132.

[3] Wang, K. Y., Zheng, Z. Z., Chen, M. J., & others. (2023). Research on natural ventilation schemes for offices based on CFD simulation [J]. Building Energy Efficiency (Chinese & English), 51(12), 102 - 108.

[4] Zhou, W. T., Wu, X. W., Jia, Y. T., & others. (2023). Optimization study of indoor thermal and humidity environment in old communities based on CFD and field measurement [J]. Smart City, 9(5), 4 - 6. DOI: 10.19301/j.cnki.zncs.2023.05.002.

[5] Hao, H., Luo, W., Shi, G. W., & others. (2023). Numerical simulation analysis of indoor air quality in gyms [J]. Journal of Shenyang Jianzhu University (Natural Science Edition), 39(5), 923 - 931.

[6] Sun, X. M. (2023). A study on the impact of the office fresh air system on indoor air quality and thermal comfort [Dissertation]. Hebei University. DOI: 10.27103/d.cnki.ghebu.2023.000475.

[7] An, Y. H., Ma, X. Y., Wang, A. Q., & others. (2024). Simulation analysis of indoor air distribution in existing natatoriums based on natural ventilation. Journal of Shenyang Jianzhu University (Natural Science Edition), 40(1), 148–157.

[8] Concilio C, Benito A P, Ramírez P C, et al. CFD simulation study and experimental analysis of indoor air stratification in an unventilated classroom: A case study in Spain[J]. Heliyon, 2024, 10(12): e32721-.

[9] ChaoLong G, XuYanran Z, Ning C.Optimizing indoor air quality: CFD simulation and novel air cleaning methods for effective aerosol particle inhibition in public spaces.[J]. Environmental science and pollution research international, 2023, 30(57):120528-120539.

[10] Shipeng X, Guangwei Z, Xiaoping L, et al. CFD modelling of infection control in indoor environments: A focus on room-level air recirculation systems[J]. Energy & Buildings, 2023, 288.