Numerical simulation of air distribution and CO₂ concentration in different air supply locations in winter classroom

Fan Yingying¹, Niu Runping*, Yin kuichao², Geng Lizhi¹

¹ Beijing University of Civil Engineering and Architecture, Beijing, 100044, China
² China Architecture Design & Research Group, Beijing, Beijing, 100044, China
*Corresponding author’s e-mail: niurunping@bucea.edu.cn

Abstract. In winter, temperature difference between indoor and outdoor is large, and the concentration of indoor CO₂ exceeds standard in the closed classroom. In order to achieve better ventilation effect, different outdoor air supply modes are explored to make up for the shortcoming of existing equipment. In this paper, by using FLUENT software, numerical simulation method is used to simulate the temperature field, velocity field changes and differences of classrooms under different air outlets. Also the concentration of CO₂ distribution under this condition is simulated, and the best outlet position is obtained through simulation. With the in-depth study of airflow organization, the fresh air system will provide a more comfortable living condition for human beings and serve production and life.

1. Introduction
In recent years, with the improvement of living standards and the enhancement of people's health consciousness, more and more attention has been paid to indoor air quality [1]. Air flow to some extent determines the level of indoor air quality, not only affects the spatial temperature distribution, but also plays a key role in eliminating indoor pollution [2]. Under air supply conditions, its air distribution is affected by many factors which include: air supply temperature, speed, location and size of the outlet, air volume, indoor heat source and layout.

An American scholar's long-term environmental tracking survey showed that the content of harmful substances in the body blood of people living in different pollution areas is not very different, because people spend about 80%~ 90% of their time in indoor places every day in average. This research had attracted the attention of the people about indoor air quality. Carbon dioxide concentration will seriously affect human health. Usually, it will be used as an indicator of air pollution, so carbon dioxide concentration is often regarded as an indicator of indoor air quality [3]. A survey of air quality of a primary and high school in Shanghai illustrated that the volume fraction of CO₂ in the classroom rose from 0.081% before class to 0.197% at the forth class in the cold season. When the air in the school reached a certain degree. Epidemic diseases such as colds are easy to spread. Primary and secondary school students are in a period of vigorous development. Breathing volume is much higher than that of adults. Under low solubility oxygen and high concentration of carbon dioxide, there will be chest tightness, shortness of breath, dizziness, headache and so on. Therefore, it is very important to study carbon dioxide in the classroom [4].

This paper mainly studies the influence of the coupling of fresh air system and ground heating on indoor air quality in winter environment. Through ANSYS and GAMBIT, the indoor temperature
distribution and airflow organization of different air supply height are simulated and analyzed, and the human body and leakage model are added to the classroom under the condition of CO$_2$. The concentration was simulated, and the best outlet position was analyzed with the simulated cloud chart.

2. Influence of different air inlet height on airflow and temperature distribution in classroom

2.1. Model introduction

Physical model: as is shown in figure 1, the physical dimension of the room is: 9000×6000×4200 mm$^3$; the air outlet of the classroom is on the top of the roof, the center of the symmetry is close to the top, the size of the air outlet is 1000 × 300 mm$^2$; two mechanical air inlets were arranged on the wall side by side, and the distance between the air inlet’s boundary and the wall is 1500mm. The height of bottom margin of air supply port from ground is 100, 300, 600, 1000, 1400, and 2000mm respectively. The body heat source with the size of 5000×4000 mm$^2$ is in the center of the room.

![Figure 1. Room model.](image)

Table 1. Parameter setting.

| Boundary condition      | Parameter setting |
|-------------------------|-------------------|
| Solver                  | Simple solver     |
| Entry boundary type     | Speed entrance    |
| Inlet air temperature   | 294 k             |
| Entry speed             | 0.5 m/s           |
| Export boundary conditions | Pressure outlet   |
| Body heat source        | 4000 W            |

Mathematical model: indoor air flow is a complex three-dimensional irregular flow. In this paper, two equation model were adopted, and the following assumptions are made: a) Indoor air is incompressible and accord with Boussinesq assumption, and the influence of buoyancy force on the airflow was considered. b) Ignoring the thermal radiation between the solid walls, the indoor air is a transparent medium. c) Assuming that the flow field has a high turbulence Reynolds number (Re), and the viscous viscosity of the fluid is isotropic. d) The airflow is a low-speed and incompressible flow, and the heat dissipation caused by the viscous force of the fluid was neglected. The air flow follows the incompressible viscous fluid control equation, and the continuity equation, momentum equation and energy equation are as below. The boundary conditions and parameters were set as table 1.
\[ \frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_i U_j}{\partial x_j} = 0 \]  \hspace{1cm} (2-1)

\[ \frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_i U_j}{\partial x_j} = - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + \rho \beta g \left( T_{ref} - T \right) \]  \hspace{1cm} (2-2)

\[ \frac{\partial \rho H_i}{\partial t} + \frac{\partial \rho H_i U_j}{\partial x_j} = \alpha \frac{\partial}{\partial x_j} \left( \frac{\lambda}{c_p} \frac{\partial H_i}{\partial x_j} \right) + S_H \]  \hspace{1cm} (2-3)

2.2. Analysis of simulation results

To determine the optimum height range of the outlet and explore the effects of different air outlets’ height on velocity and temperature, a typical working area of the horizontal section of the 1.3 m height was taken under the certain conditions of the outlet position. The distributions of wind speed and temperature are shown in the figure 2- figure 7.

Figure 2. The air inlet is 100 mm in height.

Under the height of 100 mm air inlet, as is shown in figure 2, the wind speed range is 0~0.271 m/s, the airflow in the whole area is uniform, and there is no obvious disturbance. The temperature ranges from 295.468 K to 317.227 K, and the temperature at the outlet side of the body heat source is higher, and the temperature around it is homogeneous.

Under the height of 300 mm air inlet, as is shown in figure 3, the wind speed range is 0~0.251 m/s, the airflow in the whole area is uniform, and there is no obvious disturbance. The temperature range is 295.423~317.019 K, and the temperature at the outlet side of the body heat source is higher. Compared with figure 2, it shows that the air turbulence is stronger.

Figure 4. The air inlet is 600 mm in height.

Under the height of 600 mm air inlet, as is shown in figure 4, the wind speed range is 0~0.231 m/s, the airflow in the whole area is uniform, and there is no obvious disturbance. The temperature ranges from 295.187 K to 317.425 K, where the temperature near the outlet area is relatively high, the temperature near the air inlet is more uniform, and the temperature gradient appears in the middle area.

Under the height of 1000 mm air inlet, as is shown in figure 5, the range of wind speed is 0~0.50 m/s, the highest wind speed appears near the air inlet, and there are vortices generated. The other regions have uniform airflow and enhanced disturbance. The temperature range is 294~315.801 K, where the temperature near the outlet and the boundary is higher, the overall temperature distribution is more uniform, and there is obvious regional temperature gradient.

Figure 6. The air inlet is 1400 mm in height.

When the height of air outlet is 1400 mm, as is shown in figure 6, the range of wind speed is
0~0.320 m/s. Due to the obstruction of the heat source, the air distribution in the whole area is uniform. There are obvious vortices near the side walls and return air outlets of the air inlet. The temperature range is 297.592~304.847 K, the temperature near the side wall and the outlet of the outlet is higher, the overall temperature distribution is uniform, and there is obvious regional temperature gradient.

At the height of 2000 mm air inlet, as is shown in figure 7, the wind speed range from 0 to 0.210 m/s, there is obvious airflow vortex; the temperature range is 299.473~325.033 K, the temperature in the middle area of the air inlet is higher, the overall temperature distribution is uniform, and there is obvious regional temperature gradient.

From the above analyses, we can discover that there is a clear vortex at the height of 1400 mm and 2000 mm of the air outlet, which will lead to dust emission. At the same time, the overall temperature difference is bigger. In winter, when the doors and windows are tightly closed, teachers and students will feel tired because of the uncirculated air. From the value of 1000 mm, there are vortex formation and the air flows over the body heat source with the greatest amount, carries the most heat away, which has an obvious regional temperature gradient. In the height range of 100 mm~600 mm, the whole area has uniform airflow and no obvious disturbance. The temperature keep in a stable and comfortable range.

3. Establishment of the model - CO₂ concentration analysis

3.1. Physical model
In order to reduce the difficulties of simulation, the simplified processing was as follows: a) The human body model is simplified to 400×300×1200 mm³; b) The simplification of human nostrils, and human respiration is the main source of CO₂ release. The nostril is simplified to a square small wind hole of 30×30 mm². The model was established in GAMBIT to mesh, and the hexahedron grid as shown in figure 8 was divided into 3.47 million.

3.2. Mathematical model
The air flow in the classroom is a turbulent motion, opening the energy equation, equation and component transfer equation. It is assumed that the air movement of the CO₂ distributed in the room has a good following property [6], and the ground heating in the classroom is the only heat source. A set of closed equations are established. [7].

\[
\frac{\partial (\rho \varphi)}{\partial t} + \nabla (\rho U \varphi) = \nabla (\Gamma_{\varphi} \nabla \varphi) + S_{\varphi}
\]  

(3-1)

3.3. Parameter setting and solution
Three dimensional incompressible unsteady Navier-Stokes equation was chosen as the governing equation [8]. The numerical simulation parameters were listed as table 2.

| Boundary condition | Parameter setting |
|--------------------|-------------------|
| Solver             | SIMPLE            |
| Turbulence model   | κ−ε Two equation model |
| Energy             | Open              |
| Time               | Unsteady state    |
| Component transport model | CO₂-air        |
4. Simulation results and analysis

In the Run Calculation of Fluent, Time Step Size was set to 1, Number of Time Steps was 10, and Max Steps was set to 20 calculated. The typical horizontal section of the respiratory plane $Z=0.9$ m and the vertical section of $Y=0.8$ m were respectively taken as the study surfaces. The indoor CO$_2$ concentration distribution was compared and analyzed under the condition that the fresh air outlet height was 100 mm, 300 mm and 600 mm.

![Figure 9. The air inlet is 100 mm in height.](image)

![Figure 10. The air inlet is 300 mm in height.](image)

![Figure 11. The air inlet is 600 mm in height.](image)

Under the height of 100mm air inlet, as is shown in figure 9, CO$_2$ concentration has obvious characteristics of density, and CO$_2$ was released from the leakage port. Under the disturbance of indoor plane airflow, CO$_2$ followed the movement, and the concentration far away from the breathing area of the personnel was obviously low. Its value is between 400–600 ppm, and the concentration of breathing area is mostly distributed between 800–1200 ppm.

Under the height of 300 mm air inlet, as is shown in figure 10, the disturbance of indoor plane airflow is stronger than that in figure 9, and CO$_2$ followed the movement more clearly. The concentration value far away from the breathing area is between 400–600 ppm, and the concentration of breathing area is mostly distributed between 600–1200 ppm. The influence of the fresh air inlet on the diffusion of CO$_2$ concentration is greater than that of 100 mm.

At the height of 600 mm air inlet, as is shown in figure 11, CO$_2$ follows the movement of air, and its concentration density decreases. The location concentration value far away from the breathing area is between 200–600 ppm. The concentration of respiratory area is mostly distributed between 600–1000 ppm. The total CO$_2$ concentration in the room decreased, the oxygen content in the gas increased, the CO$_2$ solubility reaches the comfort range, and the inlet of the fresh air has great effect on the concentration. The ventilation effect is obviously better than the above two conditions.

5. Conclusion

In the simulation, the control variable method was used to set the other conditions to study the influence of the air inlet location. In order to simplify the simulation, the contrast results were obtained. According to whether the human body model and the CO$_2$ vent (nostril) were added, the indoor air distribution of the new air system coupled with the ground heating was simulated. Finally, the optimal air supply position at the height of 600 mm was obtained.

When the air inlet is at the height of 1400 mm and 2000 mm, the airflow distribution has obvious...
swirl which will lead to dust emission, and the temperature difference is large, and the air is not circulated in the enclosed environment. When the air inlet is at 1000 mm height, the air supply is larger than the working plane, and the overall temperature gradient is obvious. In the 100 mm–600 mm height range, the whole area has uniform airflow and no obvious disturbance, and the temperature distribution is uniform.

During the simulation period, the CO2 concentration was higher at the height of 100mm and 300mm. When the air inlet is at 600 mm height, the overall CO2 concentration is low, and distributes in a reasonable concentration range, and the distribution is more uniform at this time. Ventilation effect is obviously better than other cases.

Acknowledgments
This research was supported by State Key Laboratory of Air-Conditioning Equipment and System Energy Conservation, Guangdong Key Laboratory of Refrigeration Equipment and Energy Conservation Technology (No. ACSKL2018KT1208).

Reference
[1] Ma, C. (2019) Numerical Simulation of Air Distribution of Up-Supply and Down-Return in an Air-Conditioned Room. Contamination Control & Air-Conditioning Technology, 4: 41-44.
[2] Li, Y., Gou, Y., Liu, Y., Kong, F., Liu, J. (2019) Simulation Effect of Formaldehyde Concentration Bedroom on Different Air. Environmental Science & Technology, 42: 158-161.
[3] Lu, Y., Ma, Z., Zou, P. (2015) Heating, Ventilating and Air Conditioning. China Architecture & Building Press, Beijing.
[4] Zhu, N. (2016) Detection and regulation system of carbon dioxide concentration in classroom based on 51 MCU. Electronic Technology, 45: 38-39,34.
[5] Yue, H., Deng, Z. (2019) Simulation and Analysis of CO2 Distribution in an Office Room. Refrigeration and Air Conditioning, 33: 444-448.
[6] Wang, F. (2004) Computational fluid dynamics analysis -- principles and applications of CFD software. Tsinghua University Press, Beijing.
[7] Chen, J., Wang, Y., Jiang, Z. (2013) Numerical simulation of blasting dust concentration distribution and diffusion regularities in stope. JOURNAL OF CHINA COAL SOCIETY, 38: 147-152.
[8] Zhang, Y. The Experimental Research Into the Influence of Temperature and Carbon Dioxide to Human Comfort in Small Space. [D]. Chongqing University, 2018.