Numerical study of double-diffusive mixed convection of heavy gas in ventilation room

Ying Zhang\(^{1*}\), Tao You\(^1\), Xiaoqi Wen\(^1\), Chenbo Zhao\(^1\), and Angui Li\(^{1,2}\)

\(^1\)School of Building Services Science and Engineering, Xi’an University of Architecture and Technology, Xi’an 710055, PR China
\(^2\)Department of Applied Physics and Electronics, Umeå University, SE-90187 Umeå, Sweden

Abstract. More than 90% of the toxic gas in the industrial production process is heavy gas. However, under non-isothermal conditions, due to the uncertainty of the direction of buoyancy, the traditional ventilation method cannot be used to discharge heat and pollutants. In this study, we adopted the method of numerical simulation. The model used in this study is a two-dimensional double-diffusion model. The typical heavy-specific gravity gas sulphur hexafluoride (SF\(_6\)) is used for the pollutants. When the buoyancy ratio is -1, the diffusion law of heavy gas pollutants under non-isothermal conditions is discussed. The results show that the temperature distribution and the pollutant distribution are mainly affected by the size of the vortex on the surface of the pollution source and the heat source. The temperature ventilation efficiency and the average Nu trend are similar, and put forward the optimal ventilation intensity relational formula in the square cavity.

1 Introduction

The transmission of high specific gravity polluting gas is the result of the combined action of the combined buoyancy force of the concentration difference and temperature difference and the inertial force of forced convection of the air conditioning. The transmission mechanism of heavy gaseous pollutants is not clear, so the ventilation method is not clear.

The distribution of indoor pollutants is closely related to indoor convection. Indoor convection is the combined effect of natural convection caused by thermal buoyancy and gravity of pollutants and forced convection caused by external mechanical ventilation. The two kinds of convection together lead to double-diffusion mixed convection. Many works have studied natural convection in the presence of only a heat source. Most of these studies focus on the influence of the location of the inlet and outlet on the heat transfer and convection in the cavity. Koufi studied the effect of different inlet and outlet positions on the convective heat transfer in the ventilation cavity. It has been found that the upper and lower rows on the same side have higher heat transfer efficiency [1]. Mamun[2] added a cylinder in the centre of the ventilated square cavity, and the airflow entered from the lower left corner and flowed out from the upper right corner. It was found that the larger the diameter of the cylinder, the larger the Nu number. Ampofo[3] did a natural convection experiment and obtained high-precision experimental benchmark data, which provided a powerful verification for natural convection computational fluid dynamics. The indoor environment is not only affected by the heat source, but also by the gravity of the pollution source. Younsi[4] studied the effect of increasing carbon dioxide concentration on the temperature field in the cavity, and concluded that the indoor carbon dioxide concentration should be controlled below 3000ppm, on this basis, Serrano-Arellano also analyzed four different outlet positions and found that the configuration when the exhaust outlet was close to the heat source had higher thermal comfort and air quality [5]. Deng [6] proposed the concepts of parallel flow line, mass transport line, and heat transport line, which provided a powerful tool for describing the visualization of heat and mass transfer.

There are few studies on the double diffusion of heavy air with mechanical ventilation in the current research. In this paper, the typical heavy gas sulphur hexafluoride is used as the research gas, and the best ventilation method is found by analysing the results of temperature field, concentration field, heat transfer Nu number, mass transfer Sh number.

2 Method

Compared with the 3D model, the 2D model is easier to represent the data and laws. In this study, a typical 2D open square cavity is used as the research model. This study is based on the CFD numerical simulation method, which is currently widely used in double diffusion studies.

* Corresponding author: zhangying@xauat.edu.cn
2.1 Physical model

The physical model is a two-dimensional square cavity, as shown in Figure 1, the contaminant source is in the centre of the left wall, the heat source is located in the centre of the right wall, the side length of the square cavity is L, H is the size of the contaminant source and the heat source, where H/L=0.5, and h is the size of the air inlet and the air outlet, where h/L=0.1. In this study, the surface temperature of the heat source is \(t_h\), the concentration of SF6 is \(c_l\), the temperature of the contaminant source surface is \(t_l\), and the concentration of SF6 is \(c_h\), \(t_h>t_l\), \(c_h>c_l\). The velocity of the inlet air is \(u_i\). In this study, the Prandtl number (Pr) is 0.74 and the Lewis number (Le) is 2.18. This model corresponds to common practical scenarios, such as a transformer workshop with a heat source on one side and SF6 leakage on the other side, or other situations where there is heating on one side of a building and rooms with contaminant emissions on the other side.

![Fig. 1. Geometric model.](image1)

2.2 Numerical validation and mesh sensitivity

The validation in this paper is carried out in two parts. First, results were compared with those of Béghem[7]. Figure 2 shows the calculation results of this simulation program and those of Béghem. The figure shows that the Nu and Sh are consistent with the results of Béghem's calculation. As shown in Figure 2, the numerical simulation method of this time is in good agreement with the results of other authors, so it is believed that the simulation program of this study can accurately simulate the calculation results.

![Fig. 2(a). Local Nusselt number](image2)

![Fig. 2(b). Local Sherwood number](image3)

2.3 Boundary condition

Using L, \(u_i\), \(\Delta T (\Delta T = t_h - t_l)\), \(\Delta C (\Delta C = c_h - c_l)\) as eigenvalues to dimensionless for length, velocity, temperature and concentration values. Dimensionless excess temperature: \(T = (t - t_0)/\Delta T\); concentration: \(C = (c - c_0)/\Delta C\), velocity \((U,V) = (u_i, v)/u_i\). Other dimensionless numbers are: Re=\(u_iL/\nu\), Ra=\(g\beta_f\Delta T H^3/\nu\alpha\), and N=\(\beta_f\Delta C/\beta_l\Delta T\).

Boundary conditions:
- inlet: \(U = 1, V = 0, T = 0, C = 0\);
- outlet: \(P = 0\);
- wall: \(T = 1\), \(C = 0\) at the heat source, \(T = 0\), \(C = 1\) at the pollution source, and other walls are nonslip walls.

3 Results and discussion

Under the condition that the buoyancy ratio is -1 (the intensity of the heat source and that of the pollution source are equal), the Ra number and Re number is discussed. The Ra number represents the intensity of the heat source, and the Re number represents the intensity of the forced convection.
3.1 Flow pattern

Figure 4 is the streamline diagram, temperature field cloud diagram and concentration field cloud diagram under different Re numbers when Ra=5×10^5. When Re=1315, the strength of the mechanical ventilation is weak, and most of the square cavity is dominated by a counterclockwise vortex generated by natural convection. As the intensity of forced convection increases, the Re increases to 2236, a new vortex generated by the force of gravity on SF6 appears in the lower left corner of the square cavity, and the vortex generated by forced convection continues to increase and moves to the middle of the cavity. As the Re increases to 4605, the vortex generated by the force of gravity on the SF6 in the left side of the cavity decreases and moves to the upper part of the cavity, and forced convection is dominant. It can be found that the concentration field and the temperature field are closely related to the streamline diagram.

![Streamline Concentration Temperature](image)

**Fig. 4.** The streamline diagram, temperature field and concentration field distribution under different Re numbers of Ra=5×10^5

3.2 Heat and Mass Transfer Efficiency

The Nu and Sh represent the average heat transfer and mass transfer rates on the surface of the heat source and the contaminant source, respectively. Figures 5 and 6 show the Nu and Sh, respectively.

In addition to the above working conditions of Ra=5×10^5, two working conditions of Ra=1×10^5 and Ra=1×10^6 are added, and it is found that the laws are similar. Figure 5 shows that the growth rate of the Nu corresponding to various Re ranges is different. As the Ra increases, stronger mechanical ventilation is required to change the flow state of the heat source surface, so the Re corresponding to “turning point” gradually increases.

![Average Nusselt Number](image)

**Fig. 5.** Average Nusselt number

![Average Sherwood Number](image)

**Fig. 6.** Average Sherwood number

Figure 6 shows the Sh corresponding to different values of the Re. For Ra=1×10^6, when Re<921, the Sh increases with increasing value of the Re. As forced convection continues to increase, the flow caused by the forced convection is mainly an updraft at the contaminant source, which cancels the downdraft generated by the force of gravity on the pollution source. Therefore, the Sh gradually decreases. When Re=4605, the Sh reaches the minim value for the second time.

3.3 Ventilation efficiency

Ventilation efficiency is the ability to remove pollutants and heat from a room. This paper applies the pollutant ventilation efficiency ε_c and temperature ventilation efficiency ε_T of pollutants. As shown in Figure 7 and Figure 8.

![Pollutant Ventilation Efficiency](image)

**Fig. 7.** The ε_c under different values of the Re

![Temperature Ventilation Efficiency](image)

**Fig. 8.** The ε_T under different values of the Re
As shown in Figure 7, $\varepsilon_T$ also undergoes a sudden increase with increasing Re. Figure 8 shows the relationship between the $\varepsilon_C$ and the Re under different Ra value. When the Re number is small, fresh air flows directly through the surface of the pollution source. As the Re increases, the flow rate increases, and the ventilation efficiency increases. However, as the Re number continues to increase, due to the generation of vortices on the surface of the pollution source, the $\varepsilon_C$ drops rapidly.

3.4 Discussion

Compare Figure 5 and Figure 7, $\varepsilon_T$ also undergoes a sudden increase with increasing Re. The value of the Re at the turning point is the same as that in Figure 9. For each Ra number, there will be a Re corresponding turning point, to determine the relationship between the ventilation efficiency turning points Ra and Re, this study adds three groups of working conditions where Ra=2×10^6, 5×10^6, and 1×10^7. The relationship between the Ra number and the Re number is obtained as shown in the figure below:

When the buoyancy ratio is -1, only when the ventilation intensity is greater than Re', can the indoor heat be effectively removed.

4 Conclusions

Based on the two-dimensional double-diffusion model, this study discusses the ventilation control method of heavy gas. The following main conclusions are drawn.

1. The temperature distribution and pollutant distribution in the square cavity are affected mainly by the size of the vortex on the surface of the pollution source and the heat source.
2. The temperature ventilation efficiency and the average Nu trend are similar, when the heat source heat transfer rate is higher, the temperature ventilation efficiency is also larger.
3. For the temperature field, the Ra and the effective ventilation intensity satisfy Ra=0.00167(Re')^{2.5114}, and the intensity of mechanical ventilation must be higher than Re'.

Acknowledgement

This research project was supported by the Natural Science Special Fund of Xi'an University of Architecture and Technology No. ZR19027.

References

1. L. Kou, Z. Younis, Y. Cherif, H. Naji, International Journal of Thermal Sciences Numerical investigation of turbulent mixed convection in an open cavity: Effect of inlet and outlet openings, 116 (2017)
2. M.A.H. Mamun, M.M. Rahman, M.M. Billah, R. Saidur, A numerical study on the effect of a heated hollow cylinder on mixed convection in a ventilated cavity, International Communications in Heat and Mass Transfer. 37 (2010) 1326–1334.
3. F. Ampofo, T.G. Karayiannis, Experimental benchmark data for turbulent natural convection in an air filled square cavity, International Journal of Heat and Mass Transfer. 46 (2003) 3551–3572.
4. L. Koufi, Z. Younsi, H. Naji, Numerical investigation of turbulent double-diffusive mixed convection in a slot ventilated enclosure with supply air flow ports, (2017) 1–8.
5. J. Serrano-Arellano, J. Xamán, G. Álvarez, Optimum ventilation based on the ventilation effectiveness for temperature and CO2 distribution in ventilated cavities, International Journal of Heat and Mass Transfer. 62 (2013) 9–21.
6. Q.H. Deng, J. Zhou, C. Mei, Y.M. Shen, Fluid, heat and contaminant transport structures of laminar double-diffusive mixed convection in a two-dimensional ventilated enclosure, International Journal of Heat and Mass Transfer. 47 (2004) 5257–5269.
7. C. Béghein, F. Haghighat, F. Allard, Numerical study of double-diffusive natural convection in a square cavity, International Journal of Heat and Mass Transfer. 35 (1992) 833–846.