Research on Indoor Thermal Environment of Building Corridor Based on Numerical Simulation

Zhongping Yu¹, Jin Yu¹, Gaolei Wu², Dongsheng Xi³ and Mengdi Chen²*

¹ State Grid Xinjiang Economic Research Institute, Urumqi, Xinjiang, 83000, China
² State Grid Xinjiang Electric Power Co., Ltd, Urumqi, Xinjiang, 83000, China
³ Beijing Smart China Energy Internet Research Institute, Beijing, 100176, China
*Corresponding author’s e-mail: chenmengdi@tshny.cn

Abstract. As an important connecting structure between modern buildings, because of its closed structure and high frequency of people's passing, the local "greenhouse effect" formed by solar radiation causes strong discomfort to human body during the summer in northern China. In this paper, a building corridor in Tianjin is modelled by CFD, and the internal environment of the building corridor is simulated by turbulence model, heat transfer model and radiation model. It is found that K - ε model and DO radiation model in the numerical method can be used to simulate the internal thermal environment and solar radiation conditions of the building corridor. According to the distribution of air temperature and velocity in the corridor, a new type of building should be proposed in further study.

1. Introduction
The building corridor was originally a form of ancient Chinese architecture. It is a kind of structure which is connected between buildings. Generally, it has only a roof and no enclosure. From the perspective of modern architecture, corridor is a form of complex high-rise building structure system, which generally refers to a kind of traffic connection between two or several high-rise buildings connected by overhead connection body, in order to meet the requirements of building shape and use function.

At present, there are few special design manuals and researches on the building corridor, and even fewer researches on the ventilation effect of the building corridor. The contents similar to the ventilation of the building corridor mainly focus on the civil air defense engineering design and the energy-saving building solar house [1]. In civil air defense engineering, the guarantee of indoor air quality mainly depends on mechanical ventilation system and air conditioning system. And the design of solar house is mainly to make better use of solar energy[2], in order to reduce the heating heat load of buildings in winter, so there is a big difference among the civil air defense engineering design, the energy-saving building solar house and the building corridor. In addition, the function space similar to the local "greenhouse effect" in the interior of the building corridor is the car under the sun exposure [3]. There are large transparent glass materials around the car. Under sun exposure, the interior environment of the car is similar to the interior environment of the building corridor.

The research methods for the internal environment of the building corridor and the phenomenon of local greenhouse effect are generally experimental methods, and some scholars apply the numerical simulation technology to the related fields. For example, Francesca et al[4]. have carried out a comprehensive study on the internal thermal comfort of different structural buildings in Europe, and
put forward relevant optimization and improvement measures. Chourasia[5] used CFD simulation technology to study the storage environment of vegetable warehouse, and put forward the methods of optimizing the air distribution and improving the environmental conditions, so as to improve the storage efficiency and vegetable quality.

Therefore, in order to ensure the better thermal comfort of the interior of the building corridor, and on the basis of no additional energy consumption, this paper uses the experimental verification and numerical simulation method to study the natural ventilation effect of the interior of the building corridor, and puts forward the corresponding structural optimization measures.

2. Materials and models

2.1. Physical models

According to the actual structure of the building corridor, the simplified model is established as shown in Fig. 1. The corridor is located between the two buildings, and both sides are inlets. For the convenience of calculation, the middle part of corridor is selected as the calculation physical model. In the model, the length in X direction is 10 m, the length in Y direction is 4 m, and the height in Z direction is 3 m, in which the height of vertical baffles on both sides is 1 m. A simplified human body model is set in the centre of the model, and the height of human body is 1.75 m.

![Figure 1. The physical model of building corridor](image)

2.2. Mathematical models

Computational fluid dynamics is a branch of fluid mechanics. It first discretizes the control equations in time and space, uses the iterative function of computer to calculate the discrete algebraic equations, and then displays the numerical solution of the flow field. This method has made a lot of research achievements in the fields of industrial design, architecture, atmospheric environment, hydraulics, energy and power, aerospace and metallurgy. Among them, the advantages of CFD technology in industrial equipment optimization design and energy and power industry application research mainly include: the process of physical and mathematical model building is flexible and low cost; it can carry out multi-directional and multi angle research on working conditions and conditions that are difficult to be realized under experimental conditions; it can also obtain more detailed and accurate basic research data through post-processing, which is helpful to manage on the combination of research and numerical research to guide the experimental research[6].

When CFD numerical simulation technology is used to analyze the flow and heat transfer characteristics inside the heat exchanger, the governing equations on mass, momentum, energy, and species conservation can be written as the following equation[7-9].

\[
\frac{\partial (\rho \phi)}{\partial t} + \text{div} (\rho \nu \phi) = \text{div} (\Gamma \text{grad} \phi) + S_f
\]  

(1)
In this equation, \( \phi \) is a variable that has a value of 1 for mass conservation, \( v \) for momentum conservation, \( T \) for energy conservation, and \( m \) denotes the component solution. \( \rho \) is fluid density, \( \mu \) is the diffusion coefficient, and \( S_i \) is the source term. Parameters of the governing equations can be found in Table 1. Among them, because of the condensation phenomenon in the flow process of the flue gas, there is a change of water vapor pressure variation, so the source term \( \frac{\partial \rho}{\partial x_i} + S_i \) is added in the momentum equation. Similarly, the related source \( c_r \) and \( S_c \) are added to the energy equation and the component equation.

### Table 1. Parameters of the governing equations

| Continuity | Momentum | Energy | Component |
|------------|----------|--------|-----------|
| \( \phi \) | \( \Gamma \) | \( \rho \) | \( m \) |
| 1          | 0        | 0      | 0         |
| \( v_i \)  | \( \mu \) | \( \frac{\partial \rho}{\partial x_i} + S_i \) | \( D_i \) |
| \( T \)    | \( c_p \) | \( S_c \) | \( S_i \) |

### 3. Numerical simulation verification and analysis

To ensure the accuracy and rationality of the mathematical model and boundary conditions[10], a building corridor in Beichen District of Tianjin was tested by the method of experimental verification. The test equipment is Omega HH67 temperature tester. The thermometer is specially set in the vertical direction of the horizontal center position of the building corridor through the suspension device. The measuring range of the temperature measuring equipment is \(-50 ~ 150^\circ C\), and the measuring accuracy is 1.5 \(^\circ C\). The comparison results of experiment and numerical simulation are shown in Fig.2.

![Figure 2. The comparison of the experiment and numerical simulation results](image)

A comparison between experimental and simulation results on flue gas temperatures for the winter condition are shown in Fig.2. It is found that the relative error between them is 7.4\%, which is mainly caused by the inaccuracy of experimental temperature test caused by personnel walking during the test process of temperature tester. But the error is less than 10\%. In the process of numerical simulation, the selected mathematical model and boundary conditions can meet the engineering requirements.

### 4. Results and discussion

After the comparison of numerical simulation and experimental verification, it can be seen that the internal environment of the building corridor can be simulated by the numerical simulation method.
First, Tianjin area is also selected as the simulation site, and summer solstice day (14:00 on June 21) is selected as the calculation day and calculation time. Other boundary conditions are the same as the setting method in Section 3 experiment and numerical simulation verification. The specific simulation results are shown in Fig. 3.

Fig. 3 shows the temperature distribution inside the building corridor. The vertical section is the center section of the corridor, and the horizontal section is the corridor ground. It can be seen that the average temperature inside the corridor is 72.5 ℃, which is higher than the human body temperature, and which has an uncomfortable impact on the human body. From the ground temperature distribution, it can be seen that both sides and the top of the corridor are glass fences, and the sunlight can be directed into the interior of the corridor, resulting in the temperature on both sides of the corridor ground higher than the middle of the ground. On the ground, there is a low temperature area near the human body. Through analysis, it can be seen that this area is the shadow area formed by the human body under the direct sunlight. The temperature in this area is significantly lower than that in other surrounding areas, which shows that the shelter in the corridor can effectively reduce the internal temperature of the corridor.

Fig. 3 shows the velocity distribution inside the corridor, in which the two sections are the central sections in the X and Y directions of the corridor. It can be seen that the air velocity around the human body in the corridor changes more obviously than that in other areas, mainly because the temperature inside the corridor is higher, while the temperature around the human body is lower, when the air with different temperatures come together, there will be obvious air flow. It can also be seen from the y-axis that the air temperature around the human body is higher than the surface temperature of the human body, which makes the surrounding air form a upward flow trend, especially in the position of the human head, which can form a more obvious velocity gradient stratification. This also has a significant impact on the thermal comfort of the environment in which the human body is located.

5. Conclusion
Through the modelling of a building corridor in Tianjin area, and the simulation analysis of the internal environment of the building corridor, the numerical simulation method is used to carry out comparative simulation and research. The K-ε model and DO radiation model can be used to simulate the internal thermal environment and solar radiation conditions of the building corridor. Based on the simulation study of the actual corridor structure, according to the distribution rule of air temperature and velocity in the corridor, a new structure of building corridor structure should be proposed.
References

[1] C.C. Michelsen, R. Madlener, Switching from fossil fuel to renewables in residential heating systems: An empirical study of homeowners' decisions in Germany, Energy Policy 89 (2016) 95-105.

[2] S.A. Kalogirou, Solar thermal collectors and applications, Prog. Energy Combust. Sci. 30(3) (2004) 231-295.

[3] S. Derakhshan, A. Tavaziani, N. Kasaeian, Numerical Shape Optimization of a Wind Turbine Blades Using Artificial Bee Colony Algorithm, J Energy Resour Technol Trans ASME 137(5) (2015).

[4] G.M. Stavrakakis, P.L. Zervas, H. Sarimveis, N.C. Markatos, Optimization of window-openings design for thermal comfort in naturally ventilated buildings, Appl. Math. Model. 36(1) (2012) 193-211.

[5] M.K. Chourasia, T.K. Goswami, Steady state CFD modeling of airflow, heat transfer and moisture loss in a commercial potato cold store, International Journal of Refrigeration 30(4) (2007) 672-689.

[6] M.M. Aslam Bhutta, N. Hayat, M.H. Bashir, A.R. Khan, K.N. Ahmad, S. Khan, CFD applications in various heat exchangers design: A review, Applied Thermal Engineering 32(1) (2012) 1-12.

[7] K. Elsayed, C. Lacor, CFD modeling and multi-objective optimization of cyclone geometry using desirability function, Artificial neural networks and genetic algorithms, Appl. Math. Model. 37(8) (2013) 5680-5704.

[8] W. Cao, Y. Xue, Numerical and experimental study of sensitivity factors on heat island of residence community, Environmental Fluid Mechanics 17(6) (2017) 1189-1205.

[9] W.-x. Cao, X.-y. You, The inverse optimization of exhaust hood by using intelligent algorithms and CFD simulation, Powder Technol. 315(2017) (2017) 282-289.

[10] W. Cao, F. Liu, X.Y. You, Performance of casting aluminum-silicon alloy condensing heating exchanger for gas-fired boiler, Heat Mass Transfer 54(7) (2018) 1951-1960.