Tech Science Press

Simulation of the Thermal Environment and Velocity Distribution in a Lecture Hall

Guolin Li[*]

Laboratory of Ministry of Petroleum and Natural Gas Education, Southwest Petroleum University, Chengdu, 610500, China
*Corresponding Author: Guolin Li. Email: 2926037954@qq.com

Abstract: The rational design of heating ventilation and air conditioning systems is an important means to achieve energy conservation and sustainable development. The simulation of air-conditioning systems with finite element methods has gradually become an important auxiliary means of complex airspace design. In this paper, a k-ε turbulence model is used to conduct 3D simulations and optimize the summer air conditioning system of a lecture hall. Various conditions are considered in terms of fresh air temperature and flow rate towards the end to improve comfort. The approach used in this paper could also be used in the future as an auxiliary model to promote the design of other lecture halls and ladder classrooms.

Keywords: CFD; lecture theatre; air conditioning system; building energy efficiency

1 Introduction

Numerical simulation is widely used in scientific research, and its applications are mainly concentrated in two parts: on the one hand, it is used to solve problems that cannot be solved by analytical method in theoretical research; on the other hand, it is used to solve engineering problems. In the theoretical problem, numerical analysis of heat and mass transfer in a liquid-filled cavity with a low Prandtl number was conducted to study the natural convection, heat and mass transfer in a rectangular shell [1]. Analyze laminar mixed convection in a square cavity with vertical sidewall movement, and study the effect of Richardson number on unsteady mixed convection in a partially heated square cavity below [2]. The application of numerical simulation to assist in design and optimization has become an important method for scholars to solve engineering problems. Jun et al. taking the medium temperature gravity heat exchanger as the research object, the fluid flow, temperature field and working condition are simulated, and the optimal design is completed [3]. In this paper, numerical simulation is applied to solve the design optimization problem of central air conditioning system in building from the perspective of energy saving.

With the continuous reduction of energy, people are more and more aware of the importance of efficient use of energy. In architecture, the energy consumption of HVAC (Heating, Ventilation and Air Conditioning) is an important part of the whole building energy consumption. The reasonable design of HVAC system will directly affect the building energy consumption. Computational fluid dynamics (CFD) can effectively assist designers to complete the design [4], and gradually become an important auxiliary means of air conditioning

design. Therefore, CFD is widely used in the field of architectural environment design. According to statistics, in the past 20 years, there are nearly 100,000 articles about CFD application in architecture, including almost 1,500 articles about CFD simulation of indoor environment [5]. The research direction of indoor environment simulation mainly focuses on: indoor ventilation, air conditioning design and air quality. The main subjects of the study are atrium, residence, office and greenhouse. Among them, Yu et al. completed the CFD simulation of the air conditioning in the atrium of the hotel [6]. Yu used CFD to simulate the air conditioning in the large atrium of the office building [7]. CFD technology has been successfully applied to residential interior design and energy-saving control [8]. Peng optimized the natural ventilation design of high-rise residential buildings in Jinan area [9]. Dong et al. completed the fluid simulation research on the office thermal environment [10]. CFD technology is also widely used in agricultural research. Fan Aohua completed the simulation of temperature field in winter greenhouse and designed the average temperature control system. However, the simulation of the air conditioning system of the academic lecture hall is less. This paper analyzes the temperature field and velocity field by three-dimensional modeling of a university academic lecture hall, and uses CFD technology to carry out numerical simulation, and gives the optimal design idea of air conditioning system from the perspective of energy saving and comfort.

The rest of the article is organized as follows. In Section 2, the methodology will be introduced in detail. The environmental conditions of the lecture hall are introduced, in Section 3. In the fourth part, The temperature field and velocity field of the lecture hall are simulated by CFD. At the end of the paper, the experimental results will be summarized and analyzed, and the concluding remarks are going to be presented.

2 Methodology

2.1 Geometric Model

This academic lecture hall is located in a university in Chengdu. The external wall (assuming only the rear wall is the external wall) is the thermal insulation wall. Simple setting of air-conditioning tuyere, omitting the simulation of air distribution in the tuyere, only simulating the temperature field and velocity field in the lecture hall. Based on the research purpose, the drawing of air duct is omitted in the modeling process, and only the drawing of air outlet is done. The different runners of air outlet and return air outlet are distinguished. The model is simplified as follows:

- The lecture hall is simplified into a fan-shaped staircase classroom, and the actual staircase steps are replaced by slopes. The number of seats in the lecture hall is 232.
- Assuming that the inclination angle of the stepped classroom is 5.3°, the horizontal distance of the side wall is 17 m, the classroom floor height is 4 m, the rear floor height of the classroom is 2.5 m, the front wall length of the classroom is 15 m, and the back wall length is 20 m.

The number of seats in the lecture hall is 232. Considering the occupancy rate, the source term is calculated according to 200 people. The design of central air conditioning is based on the indoor parameter temperature of 26°C and relative humidity of 60%. The cooling load of air conditioning in summer is 30,912 w. Through the calculation of air distribution, twelve air outlets are set up: six small air outlets of 200 × 400 mm at the top and six small air outlets of 100 × 200 mm at the back and upper of the classroom. There is also a general air outlet and return air outlet located on the side wall. The simplified three-dimensional model of the lecture hall is shown in Fig. 1.

2.2 Mathematical Model

The report hall is facing north and south. The rear wall is directly connected with the outside world, while the other walls are all interior walls. The following assumptions are made:

- Indoor air is viscous incompressible gas.

- The indoor air flow is steady-state turbulent except at the entrance of the air supply and return.
- The wall temperature is uniform and constant.
- Ignoring the radiation heat transfer of the sun through the window.

In addition, the boundary conditions of the model are described as follows:

- The first kind of boundary condition: the wall temperature is uniform and constant.
- The second kind of boundary condition: regard human body and indoor electrical equipment as heat source.
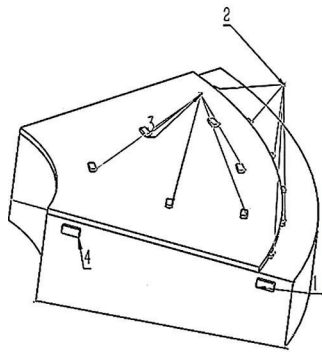

Figure 1: On the left is the three-dimensional model of the classroom and on the right is the real scene of the classroom

According to the conservation of mass, momentum and energy, the continuity equation, momentum equation and energy equation can be established respectively. The equations established by differential method are as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \tag{1}$$

$$\begin{cases} \frac{\partial(\rho u)}{\partial t} + div(\rho uU) = div(ugradu) - \frac{\partial(\rho u)}{\partial x} + S_u \\ \frac{\partial(\rho v)}{\partial t} + div(\rho vU) = div(vgradv) - \frac{\partial(\rho u)}{\partial y} + S_v \\ \frac{\partial(\rho w)}{\partial t} + div(\rho uU) = div(wgradw) - \frac{\partial(\rho u)}{\partial z} + S_w \end{cases} \tag{2}$$

$$\frac{\partial(\rho T)}{\partial t} + div(\rho UT) = div(\frac{k}{c_p} gradT) + S_T \tag{3}$$

Formulas (1)–(3) represent continuity, momentum and energy equation, ρ represents fluid density, u, v, w represents direction velocity, S_u, S_v, S_w is generalized source term, In this model, $S_u = S_v = 0, S_w = -\rho g$, c_p is constant pressure specific heat capacity, K is fluid heat transfer coefficient, S_T is viscous dissipation term.

3 Numerical Calculation

3.1 Initial Condition

The initial condition of the model is the air velocity at the entrance and exit. According to the HVAC design manual, the recommended air velocity at the fresh air entrance of public buildings is found to be 4 m/s [11]. Classroom belongs to public buildings. The velocity of air supply at the intake is 4 m/s, that is to say, the velocity of air supply at the outlets 2 and 3 is 4 m/s. The total air supply speed at the inlets 1 is 5.3 m/s. The fresh air temperature is 18°C. Because the wall shear stress of non-circular pipe is not uniformly distributed, the average value of non-circular pipe can only be calculated. When judging the flow pattern of fluid, it is necessary to calculate Reynolds number. For a circular pipe, the characteristic length can be taken as its diameter, while for a non-circular section, its hydraulic diameter is taken as its characteristic length. The hydraulic diameter and flow pattern of each tuyere are determined as shown in Tab. 1.

Table 1: Hydraulic diameter and flow pattern of tuyere

Features	1	2	3	4
Dimension of tuyere (mm)	300 × 600	100 × 200	200 × 400	300 × 600
Hydraulic diameter (mm)	0.432	0.13333	0.26667	0.432
Velocity of flow (m/s)	5.3	4	4	5.3
Reynolds number (Re)	103287	24059	48119	103287
Flow state	Turbulence	Turbulence	Turbulence	Turbulence

3.2 Boundary Conditions

The boundary conditions of this model are mixed heat transfer boundary conditions. According to the HVAC design manual, the heat transfer coefficients of different walls are obtained. In this model, the outer wall is assumed to be only the back wall, and the front, left and right walls are considered as the inner wall. The walls of this report hall are all taken as thermal insulation wall, that is, the wall is inlaid with thermal insulation materials with thermal insulation performance, so as to improve the thermal insulation performance of the building, which is conducive to building energy conservation. The specific thermal parameters of each wall are shown in Tab. 2.

Table 2: Wall thermal parameters

Thermal parameters	Heat transfer coefficient (w/m^2·K)	Temperature (K)
Rear wall	0.7	307
Front wall	1.66	302
Left wall	1.66	302
Right wall	1.66	302
Roof	1.17	299
Bottom	1.17	299

3.3 Numerical Model and Method

The numerical calculation involved in this paper is the simulation of the temperature and velocity fields in the lecture hall. According to the initial and boundary conditions of the lecture hall mentioned above, k – ε model is used to complete the numerical calculation. The model is a semi empirical formula based on

turbulent flow and diffusion rate, which is widely used in flow field simulation and heat exchange calculation. In this model, the intermolecular viscosity is ignored, and the calculation equations are described by Eqs. (4) and (5).

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{4}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial\varepsilon}{\partial x_j}] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \tag{5}$$

In the equation, G_k represents the turbulent kinetic energy term generated by laminar velocity gradient, and G_b is the turbulent kinetic energy term generated by buoyancy. Y_M is the contribution of turbulent pulsation expansion in compressible flow to the dissipation rate in the global flow. C_1, C_2 and C_3 are constants, σ_ε and σ_k are turbulent Prandtl numbers of ε and K equations, respectively, and S_k and S_ε are turbulent energy terms and turbulent dissipation source terms.

Because of the small pressure change in the hall, this paper uses velocity-vorticity method to complete the simulation calculation. The tensor control equations of velocity-vorticity laminar incompressible fluid kinematics and vorticity transport are given. Let $\vec{v}$ be the velocity field and $\vec{w} = \nabla \times \vec{v}$ the vorticity field [12].

$$\nabla^2\vec{v} + \nabla \times \vec{w} = 0 \tag{6}$$

Eq. (6) represents a connection between the velocity and vorticity vector field. It is an elliptic partial differential equation. The fluid movement is governed by the vorticity transport equation, written in a nondimensional form as:

$$\frac{\partial\vec{w}}{\partial t} + (\vec{v} \cdot \vec{\nabla})\vec{w} = (\vec{w} \cdot \vec{\nabla})\vec{v} + \frac{1}{\text{Re}}\nabla^2\vec{w}$$

where Re is the Reynolds number. The velocity boundary conditions are known, while the vorticity boundary conditions are calculated from Eq. (6).

3.4 Numerical Discretization and Grid Independence Test

In this paper, the finite element method is used to simulate the temperature and velocity fields of the lecture hall. The hexahedral structured grid is used to discrete the geometry model of the lecture hall. The whole geometric model is divided into solid and fluid parts, which generate different meshes for different parts, and the mesh at the boundary layer is also properly encrypted. The mesh generation results are shown in Fig. 2.

Grid independence tests were carried out. In this study, the wind velocity and temperature of the air outlet are more concerned, and the gradient is obvious. Therefore, the velocity and temperature of the area below the No. 3 air outlet is selected as the test basis, and this area is located at a working area 1 m from the ground. Three different grids are generated with different sizes, as shown in Tab. 3. The base size is the largest size of the mesh.

The accuracy of the numerical simulation is improved with the increase of grids [13]. It can be seen from Tab. 3 that the relative error between Mesh 1 and the other two sets of grids is larger, and the relative error between Mesh 2 and Mesh 3 is smaller. Therefore, the accuracy of Mesh 2 satisfies the calculation requirements. As a compromise between accuracy and CPU time, it was decided to use Mesh 2 for subsequent calculations.
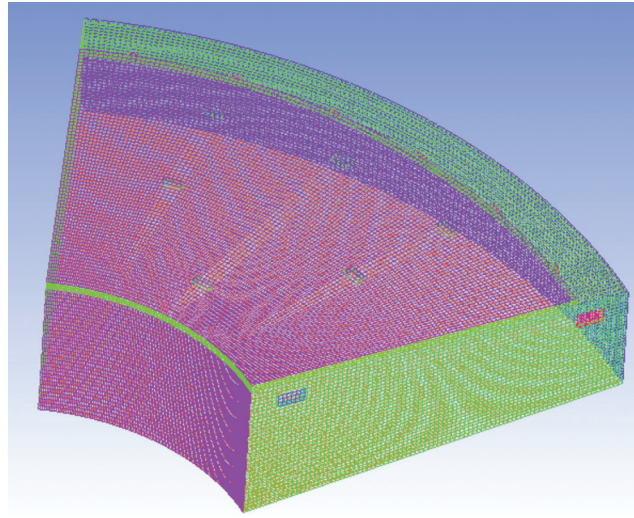
Figure 2: Geometry discrete mesh of the hall

Table 3: Effect of the grid arrangements on numerical solution

Conditions	Basic size (m^2)	Number of grids	Temperature (k)	Velocity (m/s)
Mesh 1	2e−2	293276	292.36	1.53
Mesh 2	1.5e−2	442488	293.42	1.42
Mesh 3	1e−2	672264	293.39	1.41

4 Results and Discussion

4.1 Primitive Experiment

The geometric model of the lecture hall is imported into the pre-processing software of CFD to complete the mesh generation, and then it is imported into the CFD calculation software module to complete the numerical calculation. In this paper, the whole implicit multi grid coupling iterative solution technology is used. When the residual value of each parameter is less than the error limit, the iteration converges. The error limit of parameter residual set in this paper is 10e−6.

After calculation, the maximum temperature is 300.6621 K, and the maximum temperature appears in the rear wall, the minimum temperature is 291 K, and the minimum temperature is located in the air outlet. The overall temperature cloud is shown in Fig. 3. From Fig. 3, we can get the temperature gradient change of each wall: the fresh air temperature of the supply air is the lowest, and with the distance from the fresh air outlet increasing, its temperature will rise correspondingly, and there will be obvious temperature gradient. The evaluation criterion of air conditioning performance is whether it can meet the requirements of temperature and humidity of the staff in the working area. Therefore, different Z-value planes are drawn (Figs. 4 and 5), and the temperature and velocity distributions are further analyzed and studied.

By analyzing the temperature and velocity fields of z = 1 and z = 1.8 m, we can find that the average plane temperature of z = 1 m is about 296 K, which can meet the temperature needs of the students in the classroom. However, at the lower part of the inlet, the air velocity is more than 1 m/s, which does not meet the comfort requirements. Moreover, the average temperature at z = 1.8 m is about 296 K. However, this area is no longer a student's work area, so it will become a waste of energy. Therefore, the initial test conditions are modified and further simulation analysis is carried out.
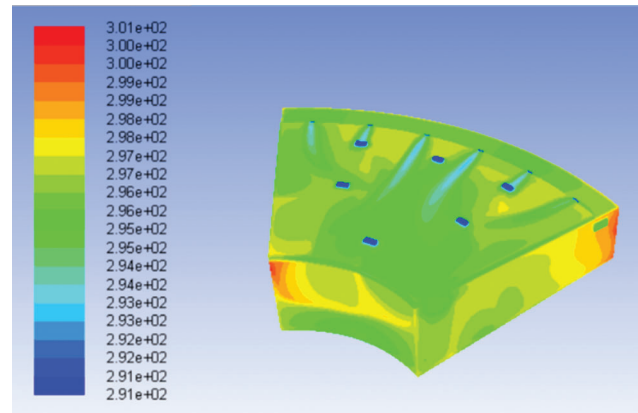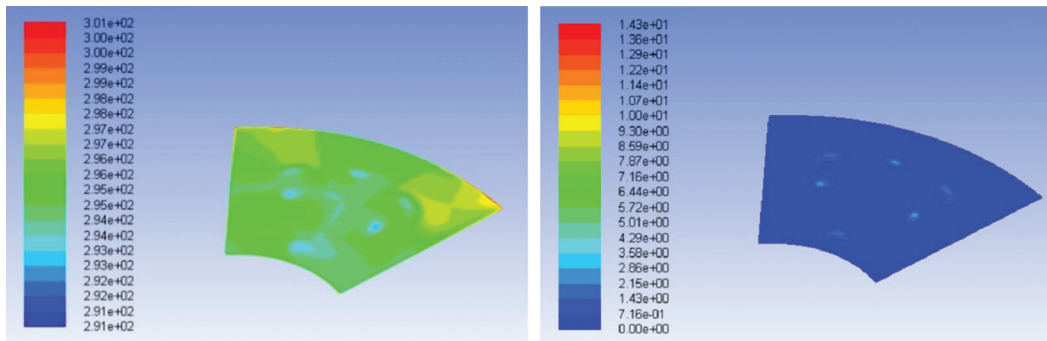
Figure 3: Overall temperature cloud

Figure 4: z = 1 m temperature (left) and velocity cloud (right)
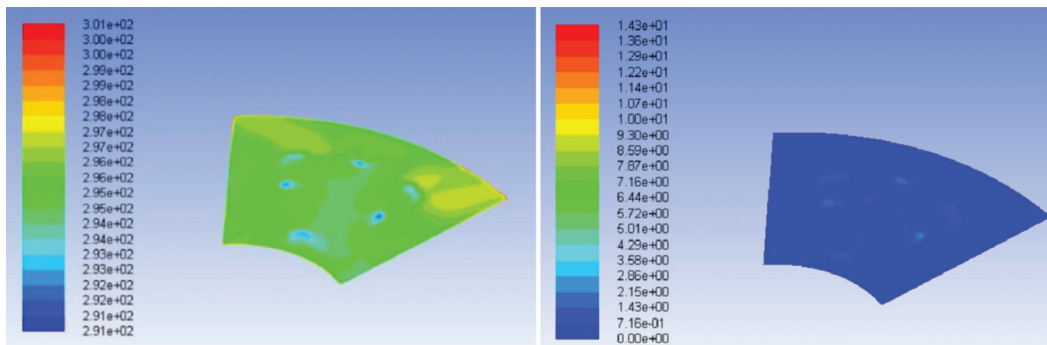

Figure 5: z = 1.8 m temperature (left) and velocity cloud (right)

4.2 Improvement Experiments

According to the results of the first simulation, it is found that in the range of z = 1 m to 1.8 m, that is, the main area of classroom students' activities, the average temperature is 296 K or 23°C. Although this temperature can meet the temperature requirements of classroom activists, it is not conducive to energy saving and green building requirements. In summer, the indoor temperature of air conditioning is controlled at 26°C, which is an ideal condition. On the other hand, in the right place of the outlet, there will be a more obvious sense of cold air, affecting comfort. Therefore, the parameters before are improved, the fresh air velocity of the air supply outlet is set to 3 m/s, and the calculation is carried out again.

After re-calculation, the maximum temperature is 302.2085 K, and the maximum temperature appears in the rear wall, the minimum temperature is 291 K, and the minimum temperature is located in the air supply outlet. The overall temperature cloud is shown in Fig. 6.

The temperature and velocity clouds of Z = 1 m and z = 1.8 m planes are drawn (as shown in Figs. 7 and 8), and the results are further analyzed to determine whether the improved scheme meets the design requirements of energy saving in green buildings.
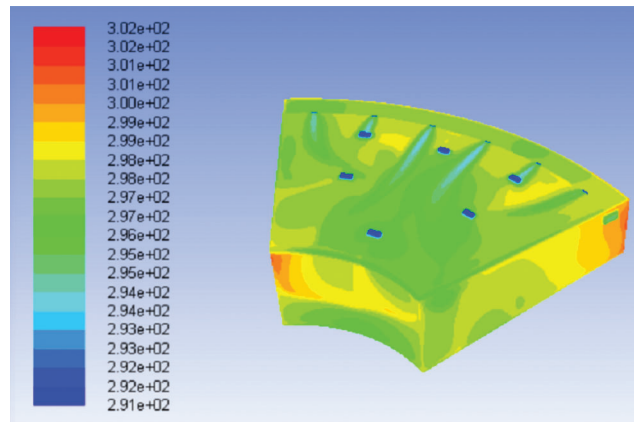

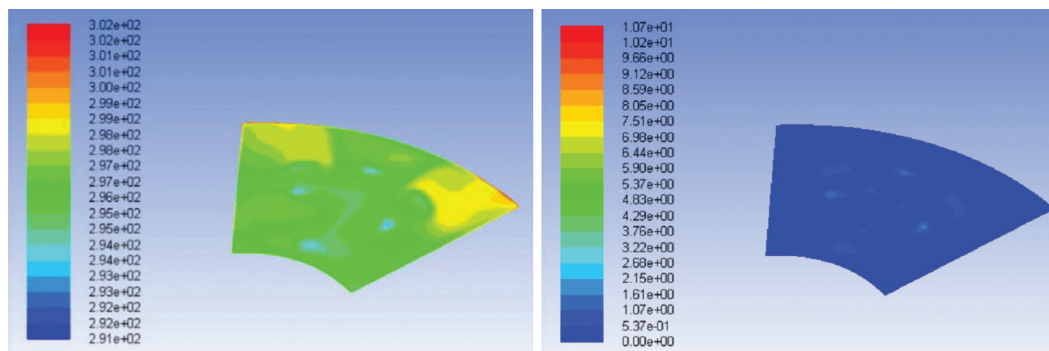**Figure 6:** Improved overall temperature cloud

Figure 7: Temperature (left) and velocity (right) cloud with Z = 1 m improved
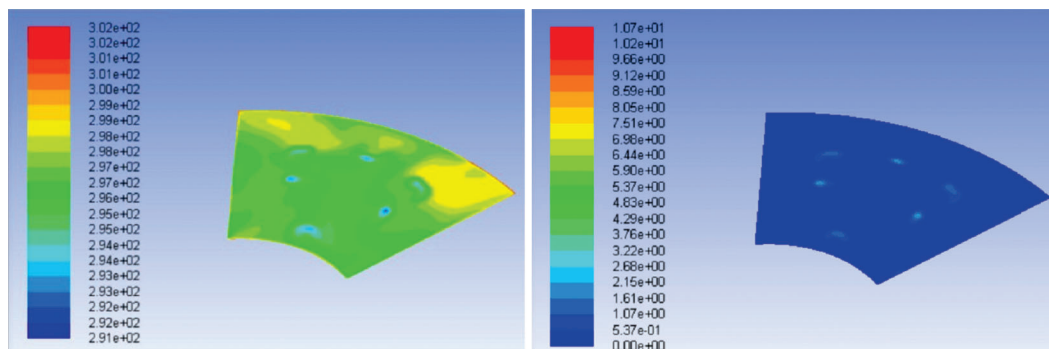

Figure 8: Temperature (left) and velocity (right) cloud with Z = 1.8 m after improvement

By analyzing the above-mentioned temperature and velocity clouds of z = 1 m and z = 1.8 m plane, the following conclusions can be drawn: on the one hand, the average temperature in this area is about 26°C, which meets the design requirements. On the other hand, when the flow rate is revised from 4 m/s to 3 m/s, the wind speed under the inlet decreases significantly, and the sense of cooling air also decreases significantly, which greatly improves the comfort of air conditioning. In addition, the cooling capacity of the refrigeration unit will decrease correspondingly due to the decrease of the air velocity at the inlet, which will play an energy-saving role. Furthermore, the decrease of the air velocity will greatly reduce the noise caused by the flow velocity and improve the comfort.

4.3 Experimental Discussion on the Comparison Results

Based on ASHRAE ventilation standard and meteorological conditions in Gaza City, Faouzi et al. simulated the air conditioning design scheme, and verified the actual measurement data, and achieved convincing simulation results [14]. In this paper, the same idea is used to complete the simulation of the air conditioning system design scheme, and to modify the unreasonable part of the existing design scheme and optimize the design scheme. This is reasonable consistent with the measurement research in other literature [15–16].

Liu Jiying also uses k – ε model to simulate the temperature and velocity field of the airport [17]. The temperature and velocity field results of the plane 1.7 m away from the ground are compared with the improved experimental results. The results show that the simulation results of this paper are close to those of Liu, which also proves the reliability of the simulation results of this paper. However, due to the simplification of the model, the return air is concentrated in one air outlet, and the maximum velocity also appears there, while other places are small, which leads to the velocity gradient is not obvious. In order to judge the effect of the scheme before and after improvement more intuitively, the speed data of the air supply area are sorted out and the following curve chart is drawn.
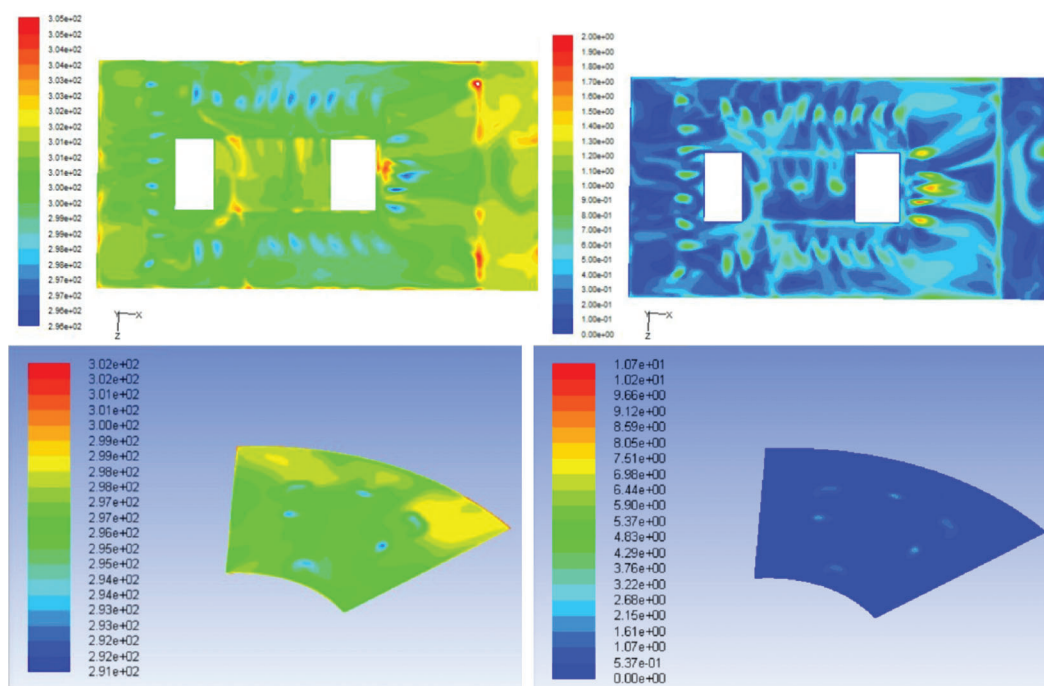

Figure 9: The above two pictures show the temperature and speed map of the airport 1.7 m above the ground, and the following is the temperature and speed map of the lecture hall 1.8 m above the ground

As shown in Fig. 10, V1 and V2 respectively represent the air inlet flow rate before and after improvement. Although the average velocity of the whole area is only slightly different, but for the area below the tuyere, the velocity has been significantly reduced. In addition, the reduction of the wind speed is more conducive to energy conservation.
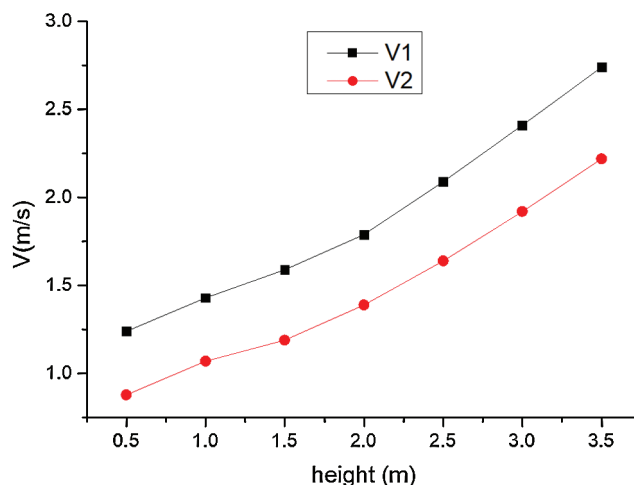

Figure 10: Velocity in the area below the inlet

5 Conclusion

Traditional HVAC design depends on the experience of experts. For the interior design of complex structure, the effect is often poor. Unreasonable design will not only cause economic loss, but also cause energy waste. Based on CFD, this paper puts forward suggestions on energy saving and comfort of traditional HVAC design.

Considering the engineering practice, this paper selects an academic lecture hall as the research object, simulates the temperature and velocity fields of the lecture hall under different conditions without changing the original design scheme, and finds that when the air supply temperature is 291 K and the velocity is 4 m/s, although it meets the design requirements, on the one hand, it will lead to excessive cooling capacity, resulting in increased energy consumption. On the other hand, if the flow velocity is too fast at the right place of the inlet, there will be an obvious sense of cold air. When the air supply speed is changed to 3 m/s, the comfort is obviously improved, which also meets the requirements of building energy saving and reduces the energy consumption of air conditioning unit. Therefore, the use of finite element simulation method for complex airspace air conditioning system design can provide optimization and verification means for traditional design methods, and provide a certain reference for the future design of air conditioning system in the complex airspace of step classroom class.

Conflicts of Interest: The authors declare that they have no conflicts of interest to report regarding the present study.

References

1. Nouri, S., Abderrahmane, G., Said, A., Pierre, S. (2018). A numerical study of the transitions of laminar natural flows in a square cavity. *Fluid Dynamics & Materials Processing, 14(2),* 121–135.
2. Sacia, K., Fatima, Z. B., Nawel, F., Saadoun, B. (2019). Effect of Richardson number on unsteady mixed convection in a square cavity partially heated from below. *Fluid Dynamics & Materials Processing, 15(2),* 89–105.

3. Du, J., Wu, X., Li, R. N., Cheng, R. R. (2019). Numerical simulation and optimization of a mid temperature heat pipe exchanger. *Fluid Dynamics & Materials Processing, 15(1),* 77–87.

4. Stavridou, A. (2015). Breathing architecture. Conceptual architectural design based on the investigation into the natural ventilation of buildings. *Frontiers of Architectural Research, 4,* 127–145.

5. Emanuele, N., Sang Hoon Lee, D., Fabbri, K. (2017). Thermal comfort-CFD maps for architectural interior design. *Procedia Engineering, 180,* 110–117.

6. Yu, T., Hu, X. W., Hu, T. (2016). CFD simulation of stratified air conditioning in Hotel Atrium. *Low Temperature and Superconductivity, 44(3),* 80–83.

7. Yu, X. (2017). *A numerical simulation study of air conditioning in large office buildings using CFD.* Beijing: Beijing University of Architecture.

8. Su, D. H. (2010). *Application of CFD technology in energy-saving control of residential interior design.* Chongqing: Chongqing University.

9. Peng, Y. L. (2015). *Research on natural ventilation optimum design strategy of high-rise residential buildings in Jinan area based on CFD simulation.* Shandong: Shandong Architectural University.

10. Dong, Z. C., Lou, J. (2011). CFD simulation of thermal environment in an office. *Refrigeration and Air Conditioning, 25(1),* 102–106.

11. Lu, Y. Q. (2008). *Practical heating and air conditioning design manual.* Beijing: China Construction Industry Press.

12. Tibaut, J., Kerget, L., Ravnik, J. (2017). Acceleration of a BEM based solution of the velocity–vorticity formulation of the Navier-Stokes equations by the cross approximation method. *Engineering Analysis with Boundary Elements, 82,* 17–26. DOI 10.1016/j.enganabound.2017.05.013.

13. Dai, Z. Y., Li, T., Zhang, W. H., Zhang, J. Y. (2020). Numerical study on aerodynamic performance of high-speed pantograph with double strips. *Fluid Dynamics & Materials Processing, 16(1),* 31–40.

14. Faouzi, N., Faris, A., Rached, N., Chaouki, A. (2018). Design and simulation of a novel solar air-conditioning system coupled with solar chimney. *Sustainable Cities and Society, 40,* 667–676.

15. Meng, X., Hu, W. T., Cao, Y. C., Huang, Y. S., Du, J. F. et al. (2018). Numerical optimization on thermal performance characteristics of interior walls based on air-conditioning intermittent running. *Case Studies in Thermal Engineering, 12,* 608–619.

16. Liu, X. B., Du, J. R., Wan, J. H., Zheng, Z. Y., Xu, W. M. (2012). Simulation investigation of mixed absorption refrigeration air-conditioning system. *Proceedings of the 2nd International Conference on Civil Engineering, Architecture and Building Materials*, Yantai, 2517–2520.

17. Liu, J. Y. (2015). A numerical study of the indoor thermal environment in an air-conditioned large space building. *Proceedings of the 8th International Conference on Intelligent Computation Technology and Automation,* Nanchang, 69–72.