NUMERICAL SIMULATION ON AIR DISTRIBUTION OF A TENNIS HALL IN WINTER AND EVALUATION ON INDOOR THERMAL ENVIRONMENT

X. Sui [1*] – G. Han[2] – F. Chen[3]

[1]School of Environmental Science and Engineering, Chang'an University, Xi'an 710054, Shanxi, China
[2]Xi'an Municipal Commission of Urban-Rural Development, Xi'an 710061, Shanxi, China
[3]China United Northwest Institute for Design and Research, Xi'an 710082, Shanxi, China

ARTICLE INFO

***Article history*:**
Received: 15.12.2013.
Received in revised form: 17.2.2014.
Accepted: 18.2.2014.

Keywords:
Tennis hall
Air distribution
Numerical simulation
Thermal environment
Thermal comfort

Abstract:

Supplying air with ball spout air diffusers is a common air-conditioning system for air distribution in large space stadiums. When supplying hot air with ball spout diffusers in winter, the phenomenon of hot jet upturning may appear, so the design should consider adjusting the spout angle so as to control the rising airflow. The purpose of the paper is to predict and optimize the air distribution of a tennis hall in winter for the purpose of guiding the design and regulation of air-conditioning system. Based on the optimal scheme of summer conditions, using computational fluid dynamics (CFD) technique, the air distribution and indoor thermal environment of a tennis hall in winter were numerically simulated. Two conditions were considered discharging air with spouts downwards with a 30 degree slope and discharging air horizontally. Indoor thermal environment was evaluated from two case studies including the protection of the movement of the ball and thermal comfort of the human body, and consequently, the optimal design was then proposed. The results can provide some guidance for air distribution design and spout regulation in winter conditions of air-conditioning systems in similar tennis halls.

1 Introduction

Stadiums are large space buildings which have large space, large heat transfer of envelopes, large staff, intensive lights, and large air-conditioning load. Therefore, designing rational air distribution to meet the requirements of games and audiences on the premise of low energy consumption of air-conditioning system has great significance. For the air distribution design of large space stadium, the difficulty is how to predict indoor air distribution quickly and accurately to develop a reasonable air distribution scheme. The stadium is a typical type of a large space building, and its internal airflow is more complicated than the one of office buildings. The method of traditional jet analysis cannot be applied to a variety of complex situations such as various distributions of internal heat sources, and

* Corresponding author. Tel.: +86 29 8541 8148; fax: + 86 29 8541 8148

various layouts of air inlets and outlets [1]. CFD simulation has some advantages such as being quick, economical, intuitive, and easy to conduct [2]. It has incomparable superiority in predicting the air distribution of the large space buildings as well as human thermal comfort [3-5].

Supplying air with ball spout air diffusers is the common air distribution form in a large space Stadium [6,7]. Due to a large nozzle section, large air supply velocity and long air jet, the supplying air strongly mixes with indoor air to form a larger recirculation zone. Therefore, setting several diffusers can meet the requirements of the air distribution in such a large space. Also, it has some advantages such as a simple layout, convenient installation and low investment costs. Supplying air with ball spout diffuser is similar to the air supply from the side wall, and the airflow pattern belongs to limited jet. The air distribution of limited jet depends on building geometry, mounting height of the air inlet and other related factors. If the mounting height of the air inlet is too low, the jet will directly come into the personnel activity area; if the mounting height of the air inlet is too high, the thickness of recirculation zone will increase and the air velocity of return air will decrease. Both of the inappropriate mounting modes mentioned above can affect thermal comfort of a human body. When supplying hot air with ball spout diffusers in winter condition, the phenomenon of hot jet upturning may appear, so the design should consider adjusting the spout angle so as to control the rising airflow.

The indoor air distribution and thermal environment in a tennis hall of Shanghai city in winter was numerically simulated by using Airpak2.1 software based on the optimal air supply scheme and ball spout air diffusers in summer condition. Two conditions were considered including discharging air with spouts sloping 30 degrees downward and discharging air horizontally. Indoor thermal environment was evaluated from the aspects of protecting the movement of the ball as well as the thermal comfort of a human body, and the optimal design was then proposed. The results can provide some guidance for air distribution design and spouts regulation in winter conditions of air-conditioning systems in similar tennis halls.

2 Air distribution design requirements in stadiums

The main tasks of air distribution design in large space stadiums are to control air velocity in the playing area, meet thermal comfort requirements and provide the acceptable indoor air quality. The main design points are as follows [8]:

1) Indoor airflow should meet the requirements of sports competitions. For various ball sport venues, controlling the air velocity in a playing area is the key point. For tennis halls, the air velocity in the range of tennis activities should not be greater than 0.5 m/s. The height of tennis activity range generally is less than 4 m.
2) The airflow should form a uniform temperature field and velocity field in the audience area, and cold draft should be avoided. Cold asymmetry should be also avoided, and the temperature difference between human head and foot should not exceed 3 ° C. For the design of the air velocity in a working area, the current design code GB50736-2012 “Design code for heating ventilation and air conditioning of civil buildings” prescribes that the indoor air velocity should not be greater than 0.2 m/s in winter and should not be greater than 0.3 m/s in summer [9].
3) Based on meeting the above requirements, the flexible adjustment of a system should be achieved for the purpose of energy-saving.

3 Numerical simulation

3.1 Physical model

The physical model of this tennis hall is shown in Fig. 1. The tennis hall is a large space building with a pitched roof. It has a length of 50.7 m and a width of 38 m. The height of exterior wall is 7.9 m and the total height is 13 m. There is a fabric wall with a 3 m height at the distance of 1.5 m from the surrounding walls. The total area of this tennis hall is 1932 m^2, and the total volume is 17996 m^3. There is no audience. The total number of sportsmen, referees and caddies is 27. There are six all-air systems in the tennis hall, and the air volume of each unit is 6480 CMH (m^3/h). Ball spout air diffusers are employed and arranged on both sides. The air jets are discharging to each other. The geometric dimensions of return air outlets are 800 mm × 1200 mm. The return air outlets are located at the same side with the air inlet system at the bottom of the room. Its hemline is 0.5 m away from the ground. The outdoor design air temperature is -0.1 ° C. The indoor design air temperature is 20 ° C and indoor design relative humidity is 50%.

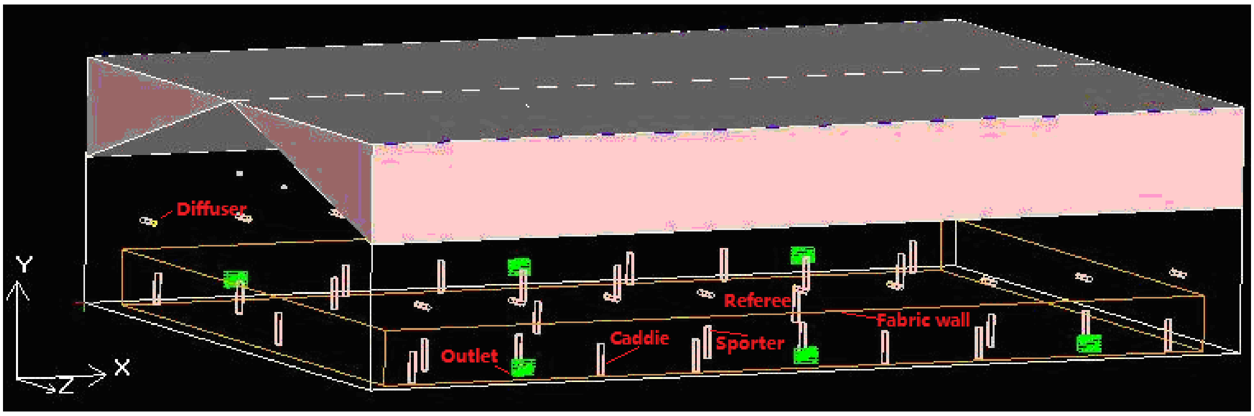


Figure 1. Physical model.

3.2 Numerical simulation methods

In this article, Airpak 2.1 software was employed for numerical simulation. The flow field studied in this article belongs to turbulent flow. As far as numerical simulation methods of turbulent flow, the Reynolds-averaging equations methods are the basic methods for engineering turbulent flow calculation. The Reynolds-averaging equations methods are divided into two categories such as the Reynolds stress equation method and the turbulent viscosity coefficient method. Turbulent viscosity coefficient method is the most widely used method in numerical calculation of engineering turbulent flow, so it was also used in the simulation of this study. For the turbulent viscosity coefficient method and the turbulent flow model, there are some relationship formulae that link turbulent viscosity coefficient (η_t) to time-averaging parameters of the turbulent flow. Depending on the numbers of differential equations determining turbulent viscosity coefficient, the turbulence flow models can be divided into the zero-equation model, one-equation model and two-equation model. Based on a lot of trial calculation simulation result analyses, the $k-\varepsilon$ two equation model was used as the mathematical model combined with the standard wall function.

The two-equation model needs to solve two turbulence parameters in order to make the equations of turbulence model be closed. In the $k-\varepsilon$ two-equation model, k is the kinetic energy per unit mass of the pulsating turbulent flow, which is defined as follows [10]:

$$k = \frac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right) \tag{1}$$

ε is the dissipation rate of pulsating kinetic energy per unit mass of fluid, and it is defined as follows:

$$\varepsilon = \nu\overline{\left(\frac{\partial u'_i}{\partial x_k}\right)\left(\frac{\partial u'_i}{\partial x_k}\right)} \tag{2}$$

The turbulence equations are as follows. The continuity equation is:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{3}$$

The momentum equation is:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\eta_{eff}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \rho\beta(T - T_0)g_i \tag{4}$$

The energy conservation equation is:

$$\frac{\partial \rho T}{\partial t}+\frac{\partial \rho u_{\mathrm{j}} T}{\partial x_{\mathrm{j}}}=\frac{\partial}{\partial x_{\mathrm{j}}}\left[\left(\frac{\eta}{P_{r}}+\frac{\eta_{\mathrm{t}}}{\sigma_{\mathrm{T}}}\right) \frac{\partial T}{\partial x_{\mathrm{j}}}\right]+S \quad (5)$$

The k -equation is:

$$\begin{aligned} & \rho \frac{\partial k}{\partial t}+\rho u_{\mathrm{j}} \frac{\partial k}{\partial x_{\mathrm{j}}}=\frac{\partial}{\partial x_{\mathrm{j}}}\left[\left(\eta+\frac{\eta_{\mathrm{t}}}{\sigma_{\mathrm{k}}}\right) \frac{\partial k}{\partial x_{\mathrm{j}}}\right]+ \\ & \eta_{\mathrm{t}} \frac{\partial u_{\mathrm{i}}}{\partial x_{\mathrm{j}}}\left(\frac{\partial u_{\mathrm{i}}}{\partial x_{\mathrm{j}}}+\frac{\partial u_{\mathrm{j}}}{\partial x_{\mathrm{i}}}\right)-\rho \varepsilon+\beta g_{\mathrm{j}} \frac{\eta_{\mathrm{t}}}{\sigma_{\mathrm{T}}}\left(\frac{\partial T}{\partial x_{\mathrm{j}}}\right) \end{aligned} \quad (6)$$

The ε -equation is:

$$\begin{aligned} & \rho \frac{\partial \varepsilon}{\partial t}+\rho u_{\mathrm{j}} \frac{\partial \varepsilon}{\partial x_{\mathrm{j}}}=\frac{\partial}{\partial x_{\mathrm{j}}}\left[\left(\eta+\frac{\eta_{\mathrm{t}}}{\sigma_{\varepsilon}}\right) \frac{\partial \varepsilon}{\partial x_{\mathrm{k}}}\right]+ \\ & \frac{c_{1} \varepsilon}{k} \eta_{\mathrm{t}} \frac{\partial u_{\mathrm{i}}}{\partial x_{\mathrm{j}}}\left(\frac{\partial u_{\mathrm{i}}}{\partial x_{\mathrm{j}}}+\frac{\partial u_{\mathrm{j}}}{\partial x_{\mathrm{i}}}\right)-c_{2} \rho \frac{\varepsilon^{2}}{k}+ \\ & c_{1} c_{3} \frac{\varepsilon}{k} \beta g_{\mathrm{j}} \frac{\eta_{\mathrm{t}}}{\sigma_{\mathrm{T}}}\left(\frac{\partial T}{\partial x_{\mathrm{j}}}\right) \end{aligned} \quad (7)$$

The turbulent viscosity coefficient is calculated by Eq. (8):

$$\eta_{\mathrm{t}}=\frac{c_{\mu} \rho k^{2}}{\varepsilon} \quad (8)$$

Where ρ is the air density; u is the velocity in the x direction; v is the velocity in the y direction; w is the velocity in the z direction; x_i and x_j are the directions (i=1,2,3; j=1,2,3); u_i is the velocity in the x_i direction; u_j is the velocity in the x_j direction; p is the air pressure; η_{eff} is the effective viscosity coefficient; η_t is the turbulent viscosity coefficient; η is the laminar viscosity coefficient; β is the thermal expansion coefficient; T_0 is the reference temperature (K); T is the air temperature (K); g_i is the gravity acceleration in the i-direction; ν is the molecular viscosity of fluid.

As mentioned above, the continuity equation, momentum equation, energy equation, k -equation, ε -equation, and the Eq. (8), constitute the basic fluid flow and heat transfer equations of indoor air. The two-equation model coefficients are shown in Table 1.

The Boussinesq assumption was adopted to reflect the influence of buoyancy force [10]. The coupling between pressure and velocity was calculated by SIMPLE algorithm. The constant heat flux boundary condition was adopted for the solid wall according to load calculation results. The velocity inlet boundary condition was adopted for the air supply inlet, and the pressure outlet boundary condition was adopted for the return air inlet.

3.3 Simulated conditions

The tennis hall is located in Shanghai city; its air-conditioning load in summer is larger than in winter, the design and selection of the air conditioning system depends on the summer condition. Concerning the summer condition, the author has conducted optimization design on air distribution [11]. According to the simulation results, in the double side directions, the design should respectively arrange three ball spout diffusers with the diameter of 400 mm, the distance from its center to ground is 5.95 m and the air supply velocity is 4.78 m/s. The supplying volume of summer condition is equal to that of winter condition. According to the load calculation results, the supply air temperature should be 22.5 ° C. When the ball spout diffuser is used to supply hot air for heating, the design usually adopts the downward blowing air supply mode to solve the hot air rising problem due to thermal jet floating.

Table 1. Coefficients of the $k-\varepsilon$ two-equation model

Coefficient	c_μ	c_1	c_2	σ_k	σ_ε	σ_T
Value	0.09	1.44	1.92	1.0	1.3	0.9~1.0

The author conducted numerical simulation of two schemes including discharging air with spouts sloping 30 degrees downward and discharging air horizontally. The reasonable airflow distribution was determined after analyzing, evaluating and comparing airflow distribution differences of these two schemes.

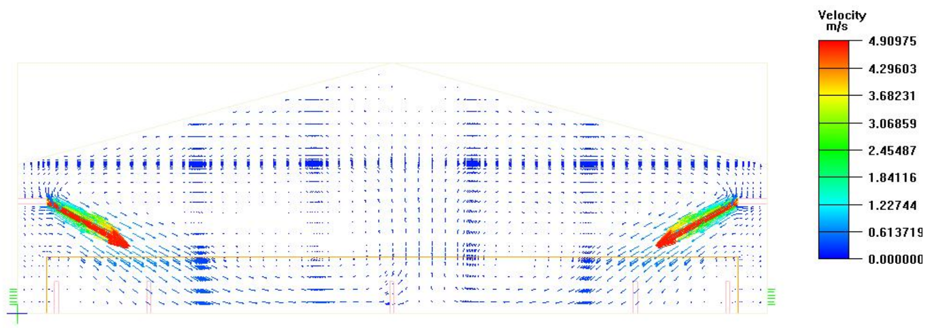
4 Numerical simulation results

In the analysis of numerical simulation results, four representative vertical sections were considered including a vertical section through the air inlet (x=3.4m), a vertical section through the return air outlet (x=8.3m), two sections through the human body (z=6.6m, y=1.4m) and three sections (y=4.3m, y=4.5m, y=5.95m) through spouts.

4.1 Supplying air with spouts sloping 30 degree downward (Condition A)

The velocity and speed fields of x=3.4m are shown in Fig. 2. The velocity and speed fields of x=8.3m are shown in Fig. 3. As the figures have shown, the air is supplied downward, but the upturned phenomenon of thermal jet is not obvious, so the thermal jet directly comes into the staff activity area. Due to the short range of thermal jet and less air entrainment in workspace, the work area cannot form a circumfluence zone. The airflow forms updrafts in the middle of the room after finishing the attenuation of its speed, then crosses the cloth obscuration and discharges from the return air outlet. This air distribution form makes the upper non-air-conditioned area form a recirculation zone, while the lower personnel activity area does not form a recirculation zone. The air distribution in personnel activity zone is uneven, and the speed of partial zone has exceeded 0.5 m/s, which cannot meet the comfort requirements and the requirement of design velocity in the movement zone of the ball. The main reason that the thermal airflow does not rise is that the supply air temperature difference is relatively small and the influence of the buoyancy effect caused by the temperature difference is far less than the influence of the air supply momentum.

(a)



(b)

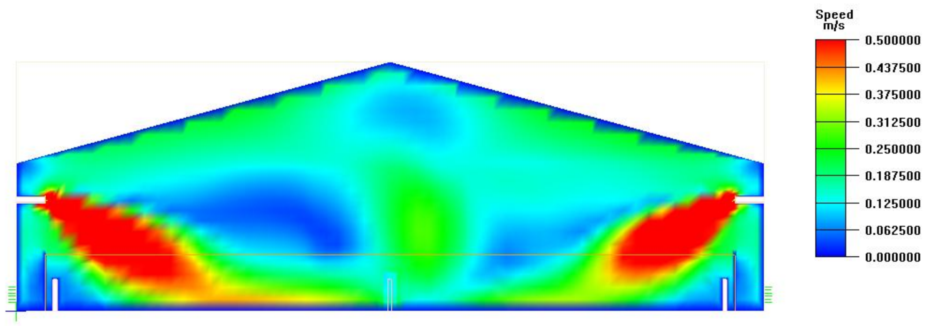


Figure 2. Velocity and speed fields of x=3.4m section in condition A. (a) Velocity field; (b) Speed field.

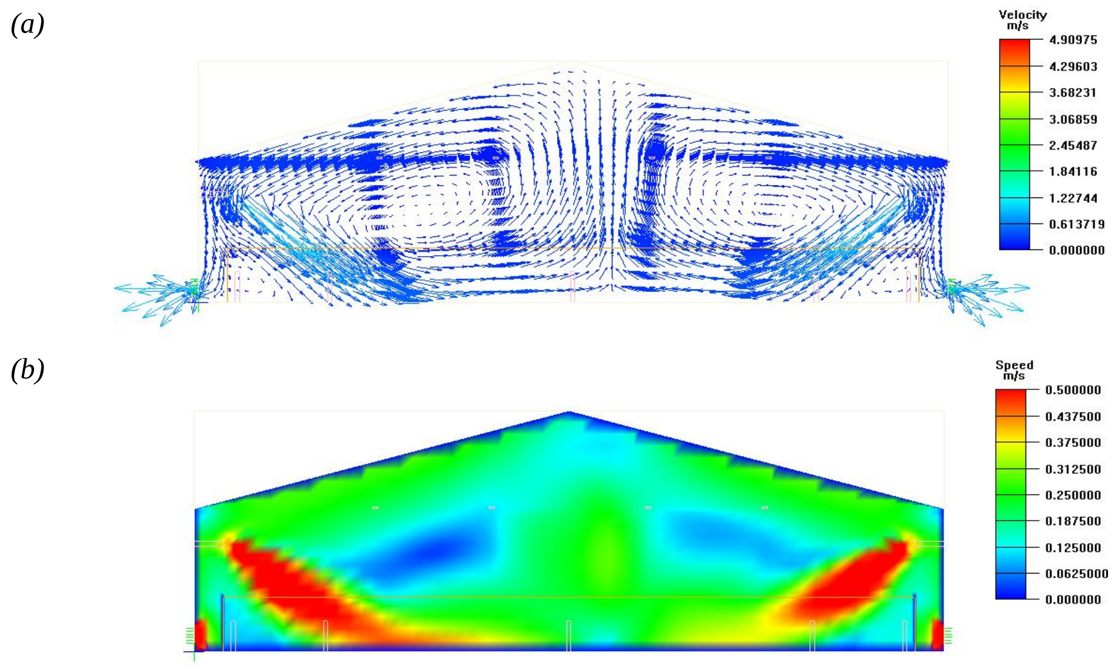


Figure 3. Velocity and speed fields of x=8.3m section in condition A. (a) Velocity field; (b) Speed field.

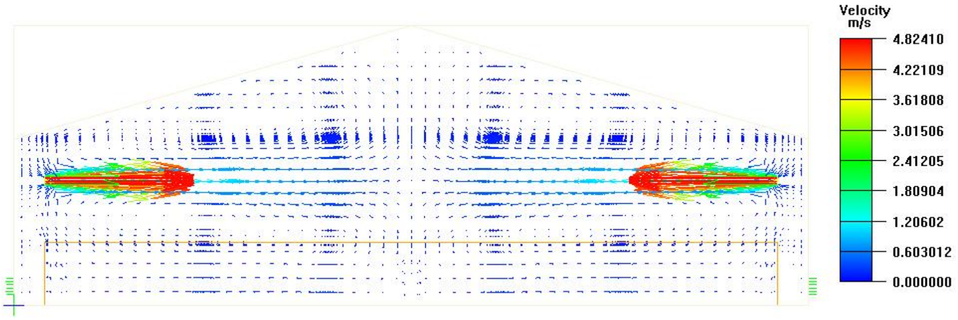
4.2 Supplying air with spouts horizontally (Condition B)

Fig. 4 shows the velocity and speed fields of the x = 3.4m section at the air inlet. Fig. 5 shows the velocity and speed fields of x=8.4m section at the air outlet. Fig. 6 (a) and Fig. 6 (b) show the temperature field through the inlet section and the outlet section, respectively. From Figs. 4 and 5, it can be seen that the two jets overlap in the middle of the Z-direction above the ball movement area, and most of air flows into the lower part of ball movement area and forms reflux, which meets the requirements for fresh air of the work area. Due to higher air supply temperature and rising of hot air, the phenomenon of jet sinking is not obvious compared with condition 4.1. There is a light subsidence phenomenon of warm air in Fig. 5. It is mainly because of the entrainment of warm air due to the recirculation air zone formed near the air outlet. The air velocity of recirculation zone is less than 0.3 m/s, and it can meet the design requirements. As can be seen from Fig. 6, the temperature field in the working area is consistent and its value is about 22.1 ° C.

Fig. 7 shows the velocity field of y=5.95m section. As can be seen from the figure, several parallel jets intersect at the jet boundary, and then superimpose with each other. It makes indoor airflow evenly distribute and makes dead ends be avoided in room.

The PMV and PPD indices are widely used to predict thermal comfort of a human body in steady-state thermal environment [12]. The PMV (predicted mean vote) is an index that predicts the mean value of thermal sensations of the majority of persons in the same environment, based on the heat balance of the human body. The PPD is an index (predicted percentage dissatisfied) that establishes a quantitative prediction of the percentage of people dissatisfied with a thermal environment, people who feel too cool or too warm. The GB50736-2012 Standard recommends the values of PMV/PPD indices to be adopted: -1≤PMV≤1; PPD≤27% [12]. In the calculation of the PMV and PPD indices, the average metabolic rate of people is 2.0 met, and the clothing insulation value is 0.5 clo. Fig. 8 shows the PMV and PPD values of z=6.6m section. Fig. 9 shows the PMV and PPD values of y=1.4m section. (0 to 2 m), and the PPD values are less than 15%. It can be seen from the figures that the PMV values are between +0.7 and- 0.7 in human activity range. The thermal comfort indices meet the specification limit of GB50736-2012 Standard [9].

(a)



(b)

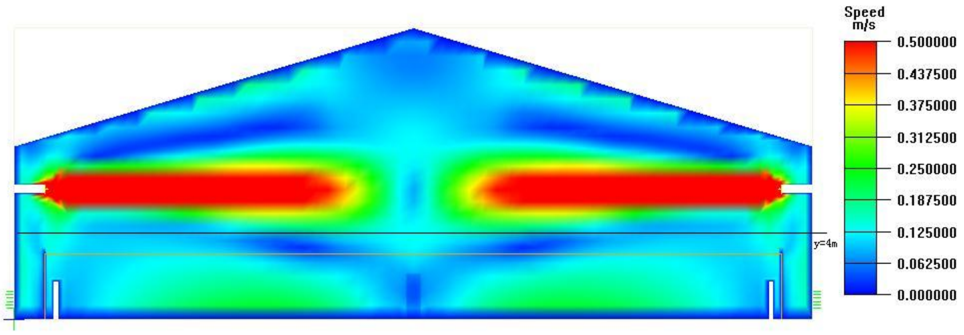
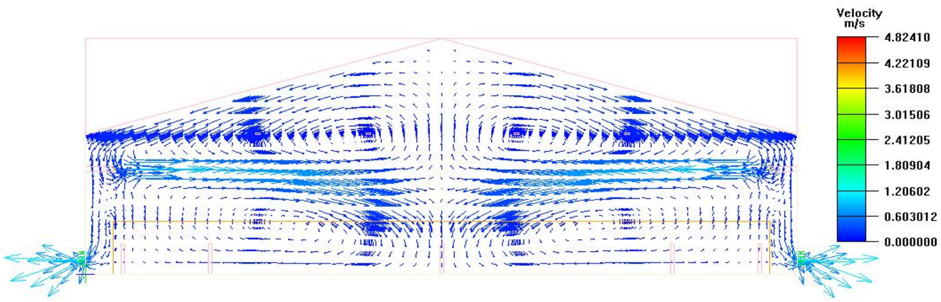


Figure 4. Velocity and speed fields of x=3.4m section in condition B. (a) Velocity field; (b) Speed field.

(a)



(b)

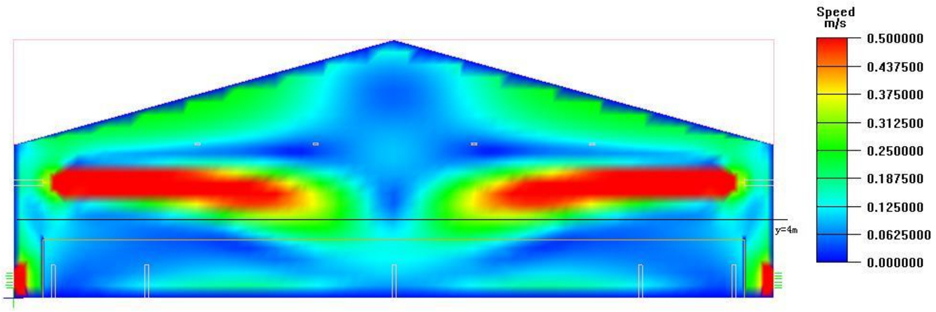


Figure 5. Velocity and speed fields of x=8.3m section in condition B. (a) Velocity field; (b) Speed field.

(a)

(b)

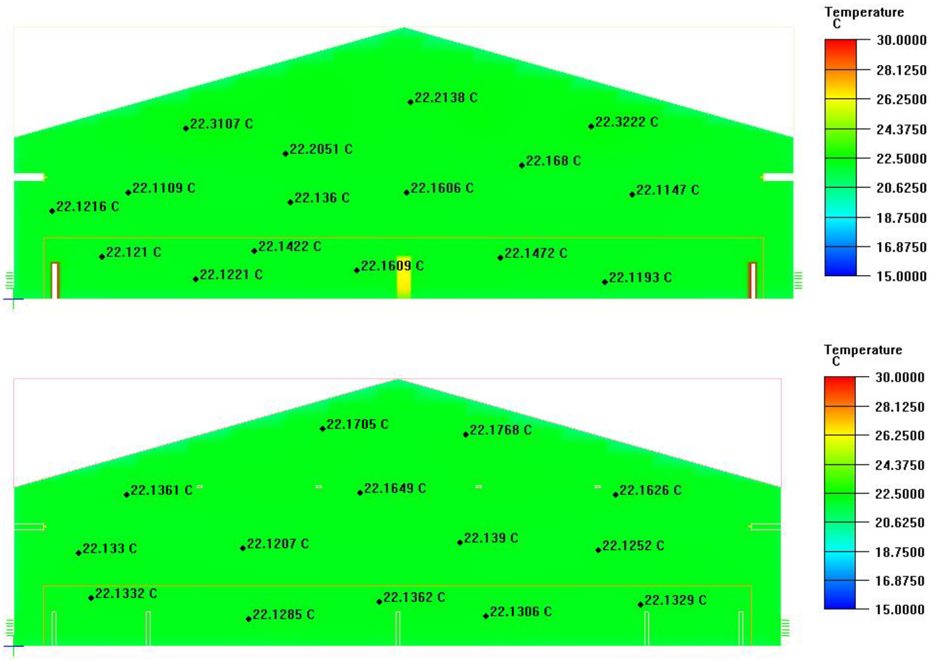


Figure 6. Temperature field of sections through inlet and outlet. (a) x=3.4m; (b) x=8.3m.

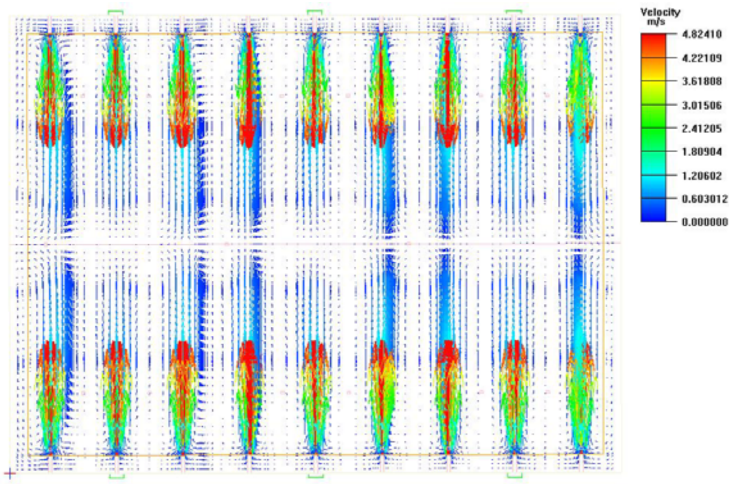
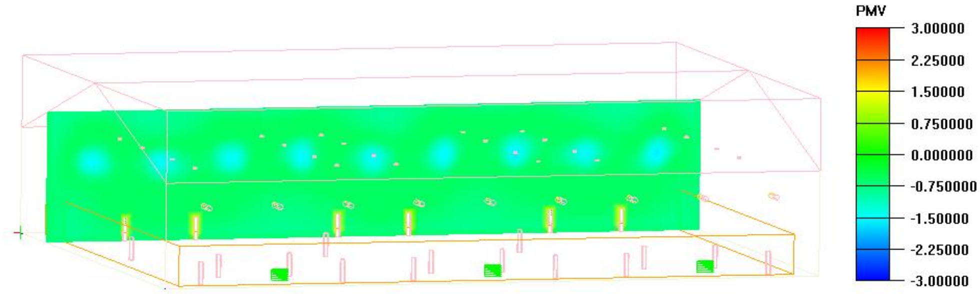


Figure 7. Velocity field of y=5.95m section.

(a)



(b)

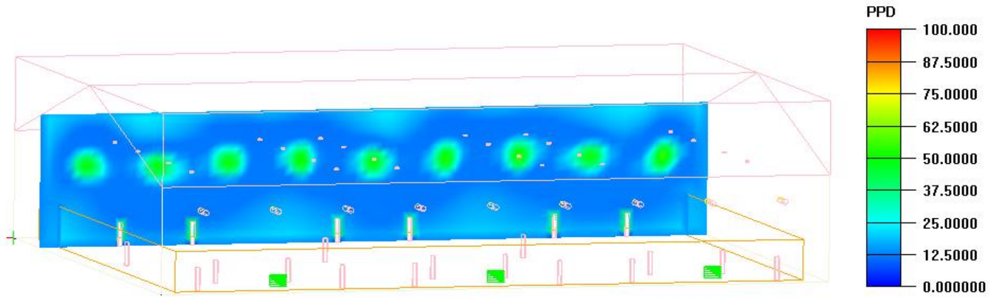
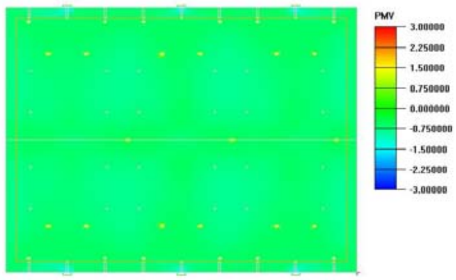


Figure 8. Thermal comfort of z=6.6m section. (a) PMV; (b) PPD.

(a)



(b)

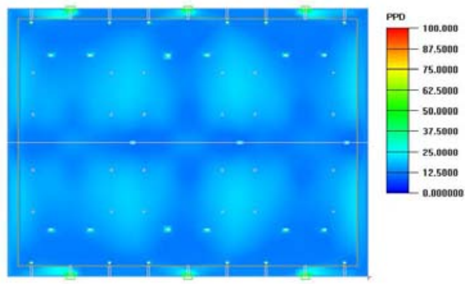


Figure 9. Thermal comfort of y=1.4m section. (a) PMV; (b) PPD.

5 Conclusions

The air distribution design of tennis hall should consider ball movement requirements as well as human thermal comfort requirements. Supplying air with ball spout air diffusers is an appropriate air distribution design scheme for this building type. However, when supplying hot air in winter conditions, floating of hot jet may appear. CFD simulation can predict the indoor airflow and thermal environment well to guide air distribution design and spout regulation. With regard to the case in this article, discharging air horizontally can meet the design requirements. This is because the supply air temperature difference is relatively small, and the floating of hot air has little effect on air distribution. So, supplying air horizontally with ball spouts is recommended in winter for this case. The results can provide some guidance for air distribution design and spouts regulation in winter conditions of air-conditioning systems in similar tennis halls.

6 Acknowledgments

This work was supported by the National Natural Science Foundation of China (Grant No. 51308049), the Postdoctoral Science Foundation of China (Grant No. 2013M532002), and the Construction Scientific Project of Xi’an City (Grant No. SJW201202).

References

[1] Zhao, B., Li, X.T.: *The Difficulty and solution of indoor airflow pattern designing for gymnasium*, Refrigeration and air-conditioning, 2 (2002), 2, 10-14.

[2] Perčić, M., Lenić, K., Trp, A.: *A three-dimensional numerical analysis of complete crossflow heat exchangers with conjugate heat transfer*, Engineering Review, 33 (2013), 1, 23-40

[3] Zhang, H., Yang, S.Y., You S.J.: *CFD simulation for the air distribution in a gymnasium,* Heating Ventilating & Air Conditioning, 38 (2008), 3, 87-90.

[4] Wang, G.S., Lin, B.: *CFD simulation and research on airflow organization in gymnasium competition hall,* Refrigeration, 29 (2010), 3, 62-67.

[5] Zhang, H.Z., Hou, Y., Chen W.N.: *Research on CFD simulation for air distribution in training gymnasium,* Refrigeration Air Conditioning & Electric Power Machinery, 30 (2009), 2, 30-34.

[6] Xie, L.Z.: *Application of motorized nozzle outlets to air conditioning system of Yantai multifunctional gymnasium*, Heating Ventilating & Air Conditioning, 39 (2009), 5, 128-129.

[7] Zhao, B., Ma X.J., Li X.T.: *Outline of Airflow Pattern inside Gymnasium*, Building Energy & Environment, 2 (2002), 5, 26-29.

[8] Beijing Institute of architectural design: *Design code for sports building (JGJ31-2003)*, China Architecture & Building Press, Beijing, 2003.

[9] Ministry of Construction P.R. China, *Design Code for Heating Ventilation and Air Conditioning of Civil buildings (GB50736-2012)*. Beijing: China Architecture & Building Press, 2012.

[10] Tao, W.Q.: *Numerical Heat Transfer*, Xi'an Jiaotong University Press, Xi'an, 2001.

[11] Sui, X.M., Zhang, X., Hu, Y.: *CFD forecast on air distribution in the tennis court and schemes optimizing*, Fluid Machinery, 36 (2008), 3, 58-62.

[12] International Standard Organization for Standardization, *Moderate thermal environments: determination of the PMV and PPD indices and specification of the conditions for thermal comfort (ISO7730)*, Geneva: 2005.