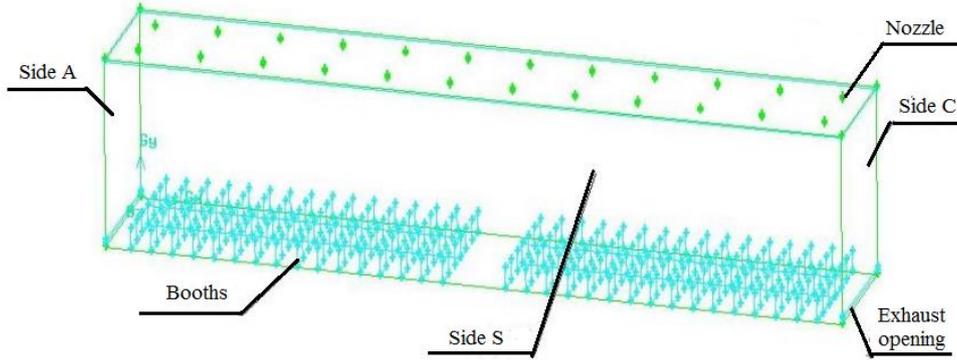


(a)



(b)

**Fig.2** Building model (a) the arrangement of nozzles (b) 3-D room model

**Table 1** Size and parameters of models

	Room size (m)	Nozzle size (m)	Exhaust opening size (m)	Booths size(m)
Case1	106.8*18*23	Diameter D=0.33 Neck height H=0.24	18*0.33	3*3*2.5
Case2	106.8*18*33	Diameter D=0.33 Neck height H=0.24	18*0.33	3*3*2.5

### 3.2 Mathematic model

The air flow of ventilation and air conditioning system is the incompressible fluid stationary flow. The universal form of controlling equation is given by

$$\frac{\partial(\rho\Phi)}{\partial t} + \text{div}(\rho\bar{u}\Phi) = \text{div}(\Gamma \text{grad}\Phi) + S \quad (1)$$

In Esq. (1),  $\Phi$  is the universal variable which on behalf of the  $\bar{u}, v, w, T$  and

other variables;  $u, \Gamma, S$  is respective the velocity vector, the generalized diffusion coefficient and the source term.<sup>[2]</sup>

Esp. (1) will be expressed as mass-conservation equation, momentum-conservation equation and energy-conservation equation respectively when  $\Phi$  equals different values. The specific values of different parameters are shown in Table 2.<sup>[3]</sup>

**Table 2** Governing equation of RNG  $k-\varepsilon$  model

Item	Value	Diffusion coefficient $\Gamma_{\phi}$	Source item $S_{\phi}$
continuity equation	1	0	0
X-velocity	U	$\mu_{eff} = \mu + \mu_t$	$-\frac{\partial P}{\partial x} + \frac{\partial}{\partial x} \left( \mu_{eff} \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_{eff} \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial z} \left( \mu_{eff} \frac{\partial w}{\partial x} \right)$
Y-velocity	V	$\mu_{eff} = \mu + \mu_t$	$-\frac{\partial P}{\partial y} + \frac{\partial}{\partial x} \left( \mu_{eff} \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial y} \left( \mu_{eff} \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_{eff} \frac{\partial w}{\partial y} \right)$
Z-velocity	W	$\mu_{eff} = \mu + \mu_t$	$-\frac{\partial P}{\partial z} + \frac{\partial}{\partial x} \left( \mu_{eff} \frac{\partial u}{\partial z} \right) + \frac{\partial}{\partial y} \left( \mu_{eff} \frac{\partial v}{\partial z} \right) + \frac{\partial}{\partial z} \left( \mu_{eff} \frac{\partial w}{\partial z} \right) - \rho g$
k	K	$\mu + \frac{\mu_t}{\sigma_k}$	$G_k + G_b - \rho \varepsilon$
$\varepsilon$	$\varepsilon$	$\mu + \frac{\mu_t}{\sigma_{\varepsilon}}$	$\rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{v \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b$
temperature	T	$\frac{\mu}{Pr} + \frac{\mu_t}{\sigma_T}$	$S_T$

### 3.3 Numerical calculation

#### 3.3.1 Assumption method

This paper is based on the RNG  $k$ -turbulence model. To simplify the questions, the assumptions are made by the following strategies<sup>[4]</sup>

- 1) The indoor air is incompressible and be conformed to the Boussinesq hypothesis.
- 2) The supply air nozzle parameter at the entrances is uniform; the value of indoor air property is fixed.

#### 3.3.2 Boundary condition

No-slip boundary conditions are used along the room walls, the ceiling, and the floor. In the simulation, the second kind boundary condition wall heat transfer coefficient is

given, and it is value from the building design profiles. The velocity inlet type boundary condition is used to model the nozzle. According to the HVAC design condition, the air flow rates of the inlet are 12.6m/s (case of 23m height) and 13.79m/s (case of 33m height). The air supply temperatures of the inlet are both 25°C. At the exhaust opening, there are often reversed flows at the initial iteration stage, the pressure outlet condition offers better stability and convergence in this case <sup>[5]</sup>and thus the pressure outlet condition is preferred. Numerical values of the boundary conditions used for the solution are listed in Table3.

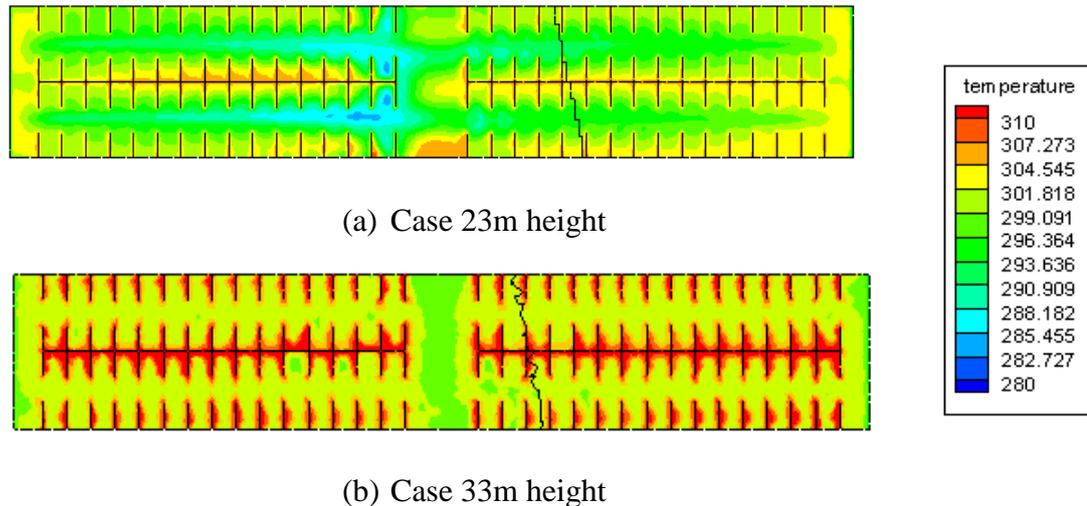
**Table 3** General BCs of CFD simulation

Boundary	Type	Values
Inlet	Velocity-inlet	$v = 12.6\text{m/s}$ (case of 23m height) $v = 13.79\text{m/s}$ (case of 33m height) $t = 20^\circ\text{C}$ $k = 0.12$
Exhaust opening	Pressure-outlet	\
Roof	wall	Heat transfer coefficient $h = 0.7\text{W} / (\text{m}^2 \cdot \text{K})$
Floor	wall	Heat transfer coefficient $h = 1.2\text{W} / (\text{m}^2 \cdot \text{K})$
Side A&C	wall	Heat transfer coefficient $h = 1.0\text{W} / (\text{m}^2 \cdot \text{K})$
Side S	symmetry	\

## 4 RESULTS AND ANALYSIS

### 4.1 Distribution of air temperature and velocity in level Z (height from the underground)=1.7m

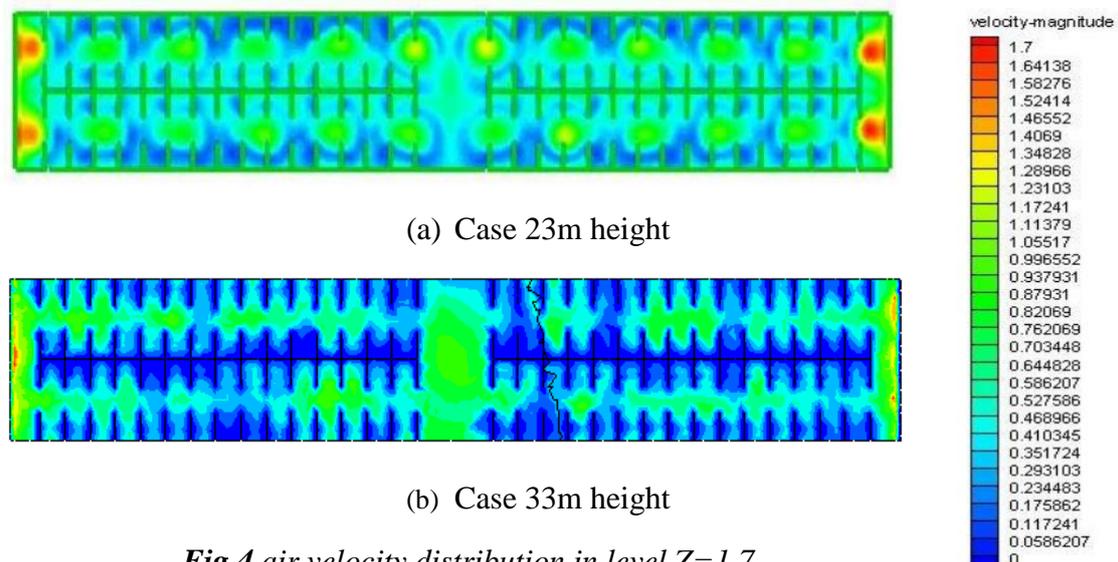
Choose the typical cross-section, Z=1.7m, the distribution of indoor air temperature simulation showed in Fig.3



**Fig. 3** air temperature distribution in level  $Z=1.7$

As can be seen from the figures, the temperature is in blocks in horizontal direction, and the maximum vertical temperature difference in the horizontal plane of  $Z=1.7$  m is  $0.85\text{ }^{\circ}\text{C}$ . The temperature in the exhibition area is lower than others for the heat flux cannot be transferred well under the stagnant air condition.

Compared these two cases under different building height, in the region of the main activities, namely the range of  $1.5\text{m}\sim 1.7\text{m}$ , the average temperature of case 23m height is  $24.3\text{ }^{\circ}\text{C}$ , which meets the building energy sufficiency design standard,<sup>[6]</sup> whereas, the average temperature of case 33m height is  $29.2\text{ }^{\circ}\text{C}$ . The reason for the higher temperature is that the nozzle air supply cannot meet the cooling demands of larger space.



**Fig.4** air velocity distribution in level  $Z=1.7$

As can be seen from Fig.4 the flow rate variation is very small in the horizontal plane while it is larger in the outlet. For the case 23m height, in the occupied zone, namely the range of  $1.7\text{m}$ , the air flow rate is very small, is about  $0.35\text{m/s}$ , which meets the design specifications of the indoor comfort of air conditioning of the exhibition hall in summer.

#### 4.2 Air velocity distribution of the squared zone under the center of a nozzle

In order to evaluate the performance of the air nozzle, this paper chooses a squared zone about 9m on a side under the center of a nozzle in the occupied zone for analyzing the air temperature and velocity distribution.

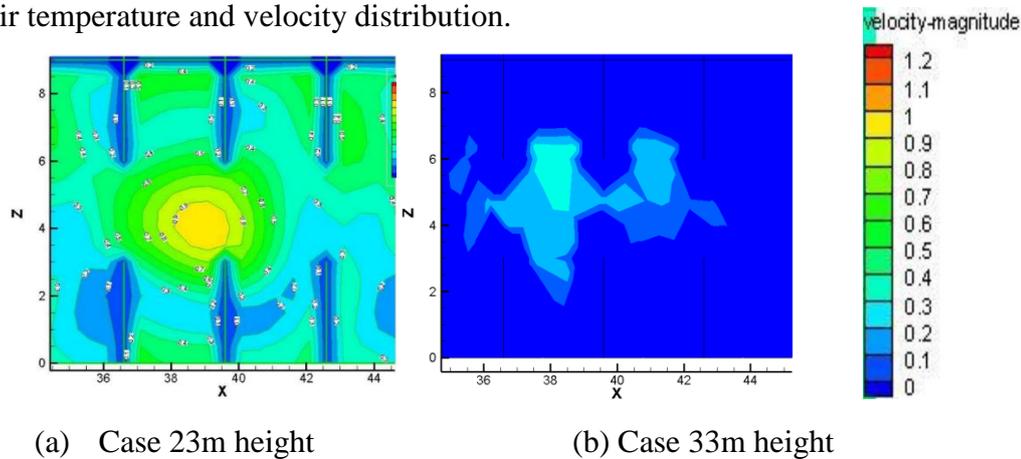


Fig. 5 air velocity distribution of the squared zone under the center of nozzle

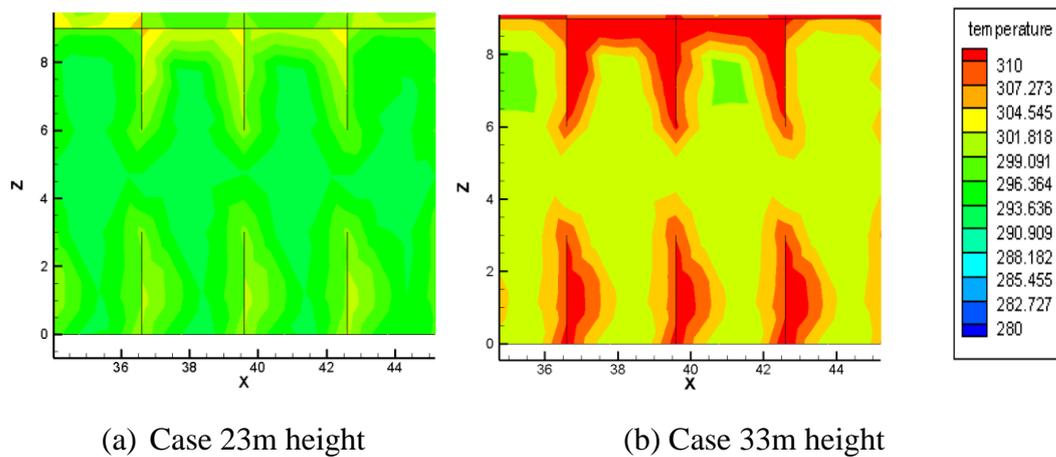


Fig. 6 air temperature distribution of the squared zone under the center of nozzle

As shown in figures, for the case 23m height, the maximum air velocity is 1.04m/s and the maximum air temperature is 25.6°C. And with the rising distance from the center, the air velocity tends to be reductive as well as the air temperature distribution in this zone.

Fig.5 &6 illustrates the air velocity of case 33m height is lower than the case 23m height and the air temperature is higher. We can conclude from the simulation results that for the case 33m height, the nozzle under the current air supply condition cannot reach the effective air delivery distance.

## 5 CURRENT FORMULA CALCULATION RESULTS OF AIR VELOCITY

A classical jet current formula for calculating the axial velocity decay has been put forward to study the performance of nozzle air flow. This paper uses the current formula to calculate the air velocity of the zone under the center of nozzles for case 23m height and compares the simulation results to the calculation results.

The classical jet current formula of the axial velocity decay is given as,<sup>[7]</sup>

$$\frac{v_x}{v_s} = K_p \frac{d_s}{x} \left[ 1 + 1.9 \frac{A_r}{K_p} \left( \frac{x}{d_s} \right)^2 \right]^{1/3}$$

$$A_r = \frac{g \Delta t_s d_s}{v_s^2 (t_n + 273)}$$

where,  $v_x$  is the axial velocity,  $v_s$  is the supply velocity,  $d_s = 0.33m$ ,  $x = 21.8m$ ,

$t_n = 25^\circ\text{C}$ ,  $K_p = 6.2$ .

The calculation velocity is 1.12m/s, which is a little larger than the simulation results and the whole tendency is consistent.

## 6 CONCLUSION

6.1 The velocity and air temperature field are simulated and analyzed through the numerical simulation of indoor thermal environment in a large space building with nozzle air supply system. The numerical simulation results provide a reference basis for reasonably supply and design to the air temperature, velocity parameters and indoor air distribution in air conditioning room.

6.2 Compared to the simulation results in this research, for case 23m height, in the occupied zone, the average air velocity is 0.35m/s and average temperature is 24.3°C, which conform to the indoor comfort design specification requirements in summer air conditioning.

6.3 The building height is a key element for effecting the performance of nozzle in large space building. For higher space building, increasing the air supply velocity of nozzle may be an effective measure to satisfy the ventilation requirement.

## REFERENCE

- [1] Hu Dingke, Rong Xiancheng, and Luo Yong, Numerical simulation and thermal comfort analysis of indoor air distribution in large space buildings [J]. Journal of HVAC, 2006, 36(5)
- [2] Tao Wenquan. 2001. Numerical heat transfer[M]. Xian: Xian Jiaotong University Press, 2001.
- [3] H.K. Versteeg and W. Malalasekera, An Introduction to Computational, Fluid Dynamics: the Finite Volume Method. Pearson Education Ltd., (2nd Edition).
- [4] Zhang Zhiqiang, Wang Zhaojun, Lian Yueming. 2004. Residential building indoor thermal environment numerical simulation research[J]. Building Energy & Environment, 10(23).
- [5] Fluent Inc. Fluent 6 user manual. 6-58-72, 10-1-102. 2001
- [6] Design standard for energy efficiency of public buildings. (GB 50189-2005)
- [7] Zhao Rongyi. 1998. Concise Air-conditioning Design Manual[M]. China Building Industry Press, 1998.