Numerical Simulation on Nozzle Air Supply Performance in Large Space Building

Yong Sun*[1]，Yuming Li[2]，Zhizhong Huang[2]，Yiqun Pan[1]

[1] School of Mechanical Engineering，Tongji University，Shanghai，China
[2] Sino-German College for Applied Sciences，Tongji University，Shanghai，China

ABSTRACT

To study the performance of nozzle air supply in a large space building, this paper builds a simplified computational model of a large exhibition hall applied with vertical nozzle air supply system using the method of computational fluid dynamics (CFD). The Fluent software is used to realize the three-dimensional simulation and obtain the temperature field and velocity field in the occupied zone at nozzles height of 23m and 33m in summer, respectively. Analysis on simulation results of two cases show that the uniformity of temperature and velocity fields in the occupied zone at nozzles height of 23m are better than 33m under the design condition. The paper also contrasts the simulation result to the axial velocity decay by means of classical jet current formula and puts forward some advices to improve the performance of nozzles in large space building.

KEYWORDS：nozzle　large space building　air distribution　CFD

1　INTRODUCTION

For the HVAC system of large space building, the vertical nozzle air supply is used widely and prediction and evaluation of indoor air environment with the air supply system has become one of issue of the modern ventilation and air conditioning engineering research and design. But the indoor thermal environment becomes very complex for the non-isothermal indoor airflow effected by the floating lift and the large spaces in the large space building. [1] So it is scientific and effective through the method of CFD numerical simulation prediction and evaluation and it will be taken more and more seriously.

The objective of this paper is to evaluate performance of nozzle air supply in a large space building using CFD methodology. In this study, a building model of a large exhibition hall with nozzle air supply is constructed to simulate and predict the temperature and velocity distribution in the occupied zone, namely the range considers radiation exchange. The predictions of velocity distribution from the nozzle

* Corresponding author email: sunyongy@126.com

model are compared to the calculation results by means of classical jet current formula.

2 AIR NOZZLE MODEL

To study the effects of the air nozzle characteristics on the geometrical shape and arrangement distance in large exhibition hall, air nozzle models are configured. The geometry for the 2-D CFD model is illustrated in Fig.1, which is built based upon information from the manufacturer and on-site measurements for the nozzle. The cylinder style air supply nozzle has a neck with a diameter of 0.33m, an opening of 0.088m2, and is setup vertically by an offset of 0.3 m from the plane of the ceiling panels. The arrangement distance of every two nozzles in large exhibition hall shown in Fig. 1 is defined as A and B. In this simulation, the value of A and B are both 9m.

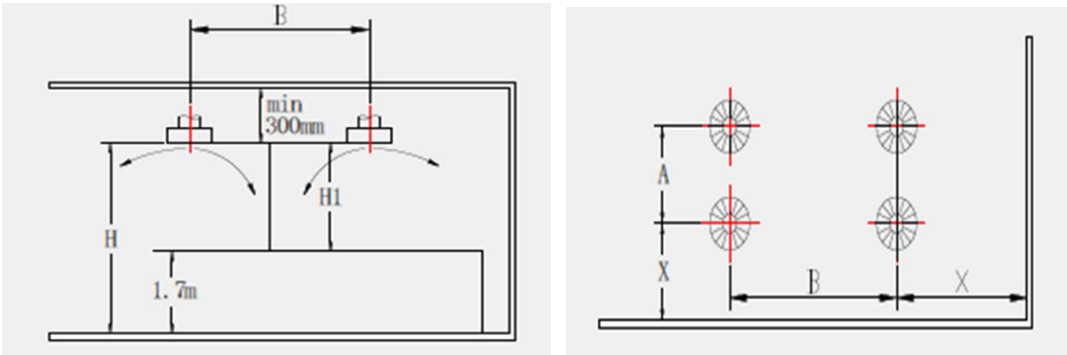


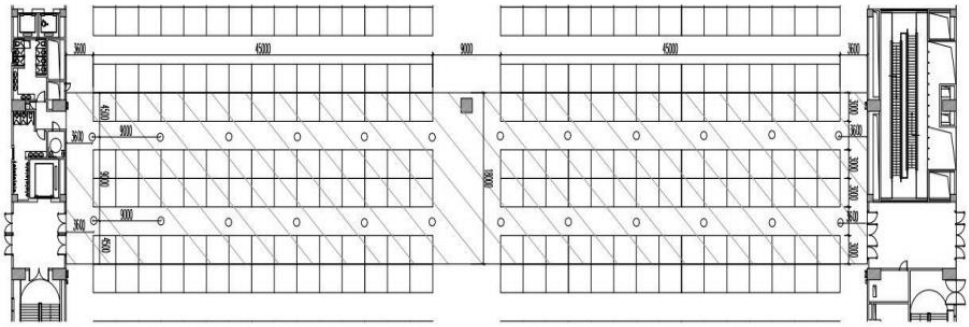
Fig.1 *2-D nozzle model*

The air nozzle turbulent kinetic energy and its dissipation rate are calculated using the Reynolds number based on the velocity in the supply air duct and the diameter of the nozzle.

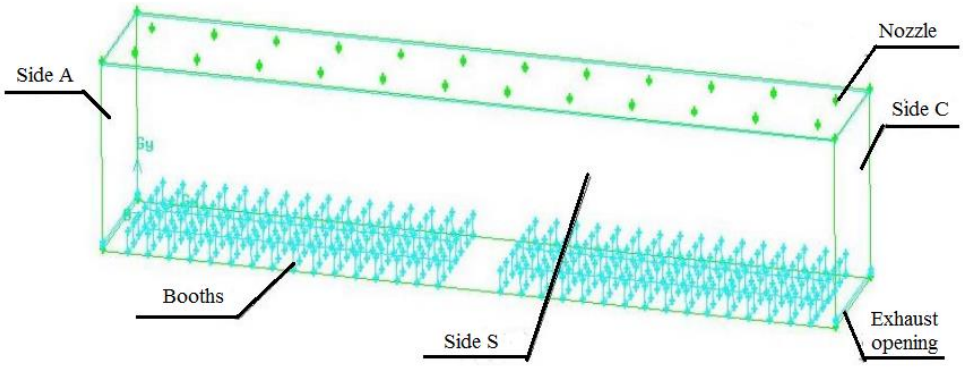
3 NUMERICAL MODEL

3.1 Building model

The research object is a large exhibition hall with high degree of symmetry. In order to evaluate performance of nozzle air supply accurately, the paper built simplified models which building space are 106.8m*18m*23m and 106.8m*18m*33m. 24 nozzles are decorated on the roof of 23m (33m) height uniformly. The established room model showed in Fig.2, the size and specific parameters of main objects presented in Table 1.



(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

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

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

other variables; u , Γ ,S is respective the velocity vector, the generalized diffusion coefficient and the source term.[2]

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

Table 2 *Governing equation of RNG k-ε model*

Item	Value	Diffusion coefficient Γ_ϕ	Source item S_ϕ
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	$\mu + \frac{\mu_t}{\sigma_k}$	$G_k + G_b - \rho\varepsilon$
ε	ε	$\mu + \frac{\mu_t}{\sigma_\varepsilon}$	$\rho C_1 S\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k} C_{3\varepsilon} G_b$
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 [4]

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℃. 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 [5]and thus the pressure outlet condition is preferred. Numerical values of the boundary conditions used for the solution are listed in Table3.

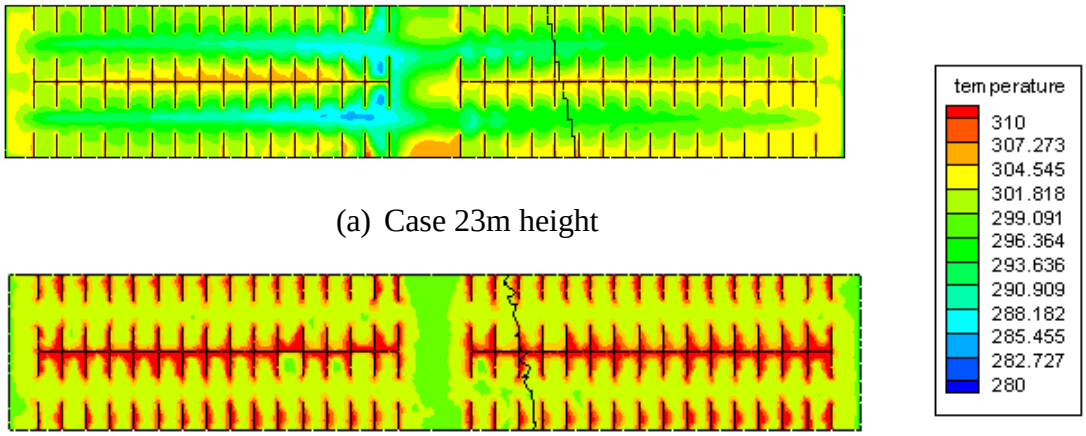
Table 3 *General BCs of CFD simulation*

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) $t = 20℃$ $k = 0.12$
Exhaust opening	Pressure-outlet	\
Roof	wall	Heat transfer coefficient $h = 0.7W/(\text{m}^2 \cdot \text{K})$
Floor	wall	Heat transfer coefficient $h = 1.2W/(\text{m}^2 \cdot \text{K})$
Side A&C	wall	Heat transfer coefficient $h = 1.0W/(\text{m}^2 \cdot \text{K})$
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



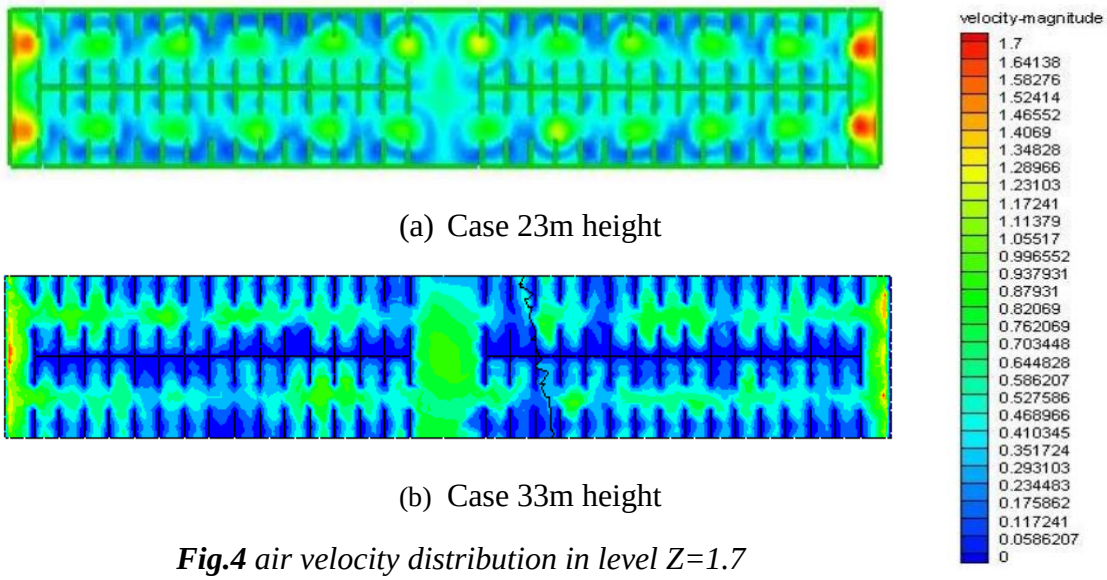
(a) Case 23m height

(b) Case 33m height

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=1.7m is 0.85 ℃.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 1.5m~1.7m, the average temperature of case 23m height is 24.3℃, which meets the building energy sufficiency design standard,[6] whereas, the average temperature of case 33m height is 29.2 ℃. The reason for the higher temperature is that the nozzle air supply cannot meet the cooling demands of lager space.



(a) Case 23m height

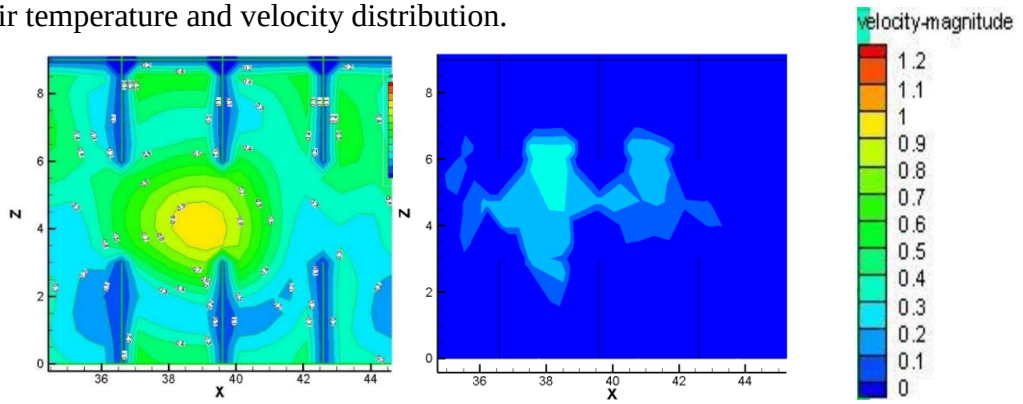
(b) Case 33m height

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 1.7m, the air flow rate is very small, is about 0.35m/s, which meets the design specifications of the indoor comfort of air conditioning of the exhibition hall in summer.

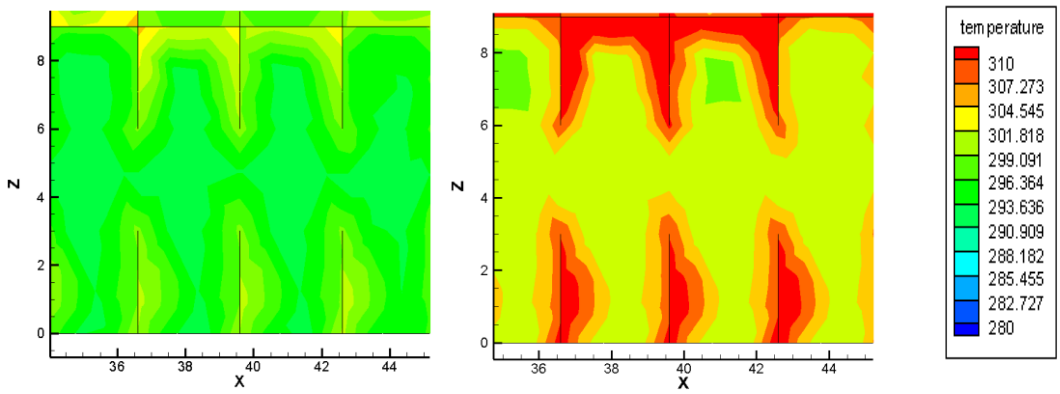
4.2 Air velocity distribution of the squared zone under the center of a nozzle

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.



(a) Case 23m height (b) Case 33m height

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



(a) Case 23m height (b) Case 33m height

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℃. 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,[7]

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

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

where, v_x is the axial velocity, v_s is the supply velocity, $d_s = 0.33m$, $x = 21.8m$, $t_n = 25℃$, $K_P = 6.2$.

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℃, 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 effiency of public buildings. (GB 50189-2005)

[7] Zhao Rongyi. 1998. Concise Air-conditioning Design Manual[M]. China Building Industry Press,1998.