Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 121 (2015) 1475 – 1481

Procedia Engineering

www.elsevier.com/locate/procedia

9th International Symposium on Heating, Ventilation and Air Conditioning (ISHVAC) and the 3rd International Conference on Building Energy and Environment (COBEE)

The Improvement of Ventilation Design in School Buildings Using CFD Simulation

Jiafang Song[a,*], Xiangquan Meng [a]

[a]*Tianjin Polytechnic University, 399 Binshuixi Road, Xiqing District, Tianjin,300387, China*

Abstract

Unlike commercial buildings, many school buildings in China are designed for natural or mechanical ventilation to fulfill thermal comfort without HVAC systems. In hot weather, due to too many students having classes at same time, the thermal conditions in classrooms were so poor to affect student's learning, especially for large lecture rooms. Therefore, it is important to study the ventilation conditions of school buildings and to explore effective means to improve the ventilation design of school buildings. This study was conducted using Fluent CFD software to improve the ventilation performance in one classroom. The field study was carried to collect the actual data of boundary conditions for CFD simulation. In this study, four different ventilation systems were simulated, compared and analyzed. Based on the simulation data, it is found that double rows windows can help to improve the thermal comfort conditions most efficiently, and so do exhaust fans and ceiling fans. But ceiling fans can improve the environment of occupied zones better.

Peer-review under responsibility of the organizing committee of ISHVAC-COBEE 2015

Keywords: School buildings; Fans; CFD simulations; Ventilation design

1. Introduction

Unlike commercial buildings, most school buildings in China are designed for natural or mechanical ventilation to fulfill thermal comfort without HVAC systems. In hot weather, due to too many students having classes at same time, the thermal conditions in classrooms were so poor to affect student's learning, especially for large lecture

* Corresponding author. Tel.: +86-022-83955393; fax: +86-022-83955258.
*E-mail address:*songjiafang@tjpu.edu.cn

Peer-review under responsibility of the organizing committee of ISHVAC-COBEE 2015
doi:10.1016/j.proeng.2015.09.073

rooms. Therefore, it is important to study the ventilation conditions of school buildings and to explore effective means to improve the ventilation design of school buildings.

So far, experimental studies and numerical simulations are two main methods to study the ventilation airflow pattern. Due to the high costing and inevitable experimental errors, numerical simulation has been becoming more popular way to understand the air movement indoors. Computational Fluid Dynamics (CFD) is the most useful numerical method based on solving differential equations of describing fluid motion with the computer reveals the physical law of the flow.

There are many previous studies on ventilation performance in an enclosed space by means of CFD simulation. In 1990, Mokhtarzadeh Dehghan analysized ventilation properties of single-sided rooms using CFD [1]. Ayad utilized CFD to study the ventilation properties for a room with different opening configurations [2]. The CFD model was verified by comparing the results for steady two dimensional flows around a long square cylinder immersed in the atmospheric boundary layer with experimental values. These results provided useful reference to model the computational domain for the atmospheric air flow around the model room. Gan(2000) has numerically predicted the effective depth of fresh air distribution in rooms with single-sided natural ventilation [3]. Eftekhari et al. (2003) have done both experimental and CFD simulation of air flow distribution in and around single-sided naturally ventilated rooms [4]. Hayashi et al.(2002) have analyzed on characteristics of contaminated indoor air ventilation and its application in the evaluation of the effects of contaminant inhalation by a human occupant [5].

2. Methods

Generally, the plan design of classrooms in the school building in China is similar. Therefore, a typical classroom located in Tianjin Polytechnic University, He Dong Disrrict was chosen to be simulated. The CFD software we used in this study is FLUENT. In this study, four cases, which are only one row of left windows opening, two rows of left window opening , two rows of left windows with right exhaust fans and two rows of left windows with top exhaust fans, are respectively provided for the simulation in which boundary conditions used were measured on-site.

The governing equations for turbulent reacting flows are the Navier–Stokes Equations **(**NSE**).** The physical parameters such as air temperature, air pressure and air velocity in the turbulence randomly vary with the time and space. In physical structure, the turbulence is regarded as the flow consisted of diverse eddy of which the size and the axis of rotation are random. In this study, k-ε two-equation model was used in CFD simulation. The recommended value of the coefficients in the model is given in Table 1.

Table 1. The coefficients in k-ε model

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

The selected classroom with length 11m along south-north direction, width 9m along east-west direction, height 4m, has 6 windows in the south wall. The dimension of windows is 150×120(cm). And there are two doors in the east wall, whose dimension of is 200×140(cm). Using a three-dimensional Cartesian coordinate system, the northeast corner of the classroom is regarded as the origin of coordinates. The classroom model is located in the first octant .While ventilating in CFD simulation, the doors are full open. There are 26 tables totally located in the classroom, in which the dimension of tables located opposite sides of the classroom is 250×37(cm), and the dimension of middle tables is 500×37(cm). The height of all tables is 80(cm). The geometric model was meshed by tetrahedral structure. Node spacing of the grid is 0.2m. The generated grid number is about 60,000.

There are four cases simulated by CFD. The case 1 is to open the lower row of windows. The case 2 is to open both rows of windows. The case 3 is to open both rows of windows and operate exhaust fans located on the east wall. The case 4 is to open both rows of windows and operate ceiling fans. The models of four cases created in CFD are shown in Figure 1. In CFD simulation, the boundary conditions for four cases are same. The outdoor air temperature is set to 300K. The outdoor air velocity was set to 1.22m/s.

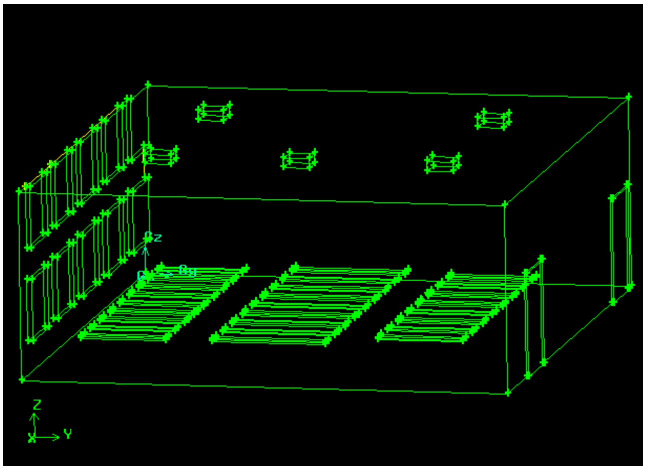

Fig. 1. The 3D drawing of building model.

Due to the operating characteristics of the school building, the whole year of building simulation is divided into three running time. The first stage is February for winter vacation. The second stage is the normal operation stage (January, March to June, and September to December), and the last stage is the summer vacation (July and August).

In order to run simulation smoothly, many parameters related to this building were input in eQUEST. The parameters set in the base case were acquired in field measurement. Computer room equipment load and office equipment load are set to 27 W/SqFt (291.6 W/m^2) and 1.5 W/SqFt (16.2 W/m^2) respectively. Lighting loads in reading area, office and book stacks areas are all set to 0.6 W/SqFt (6.5 W/m^2). Lighting load in the lobby is set to 1.77 W/SqFt(19 W/m^2). Lighting load in the corridor is set to 0.57 W/SqFt(6.2 W/m^2). Indoor design temperature in summer and in winter is set to 24 ℃ and 22 ℃ respectively. The supply air temperature in summer and in winter is set to 13 ℃ and 35 ℃ separately.

In this study, we use the control variables method which often allows the clear identification of cause and effect because only one factor is different at a time, the effect of that single factor can be determined. Some control variables was chosen to be simulated by eQUEST, such as lighting power density, indoor personnel density, summer indoor design temperature and summer air supply temperature. There are five cases in one control variable simulation. The set value in each case is shown in Table 1.

3. Results

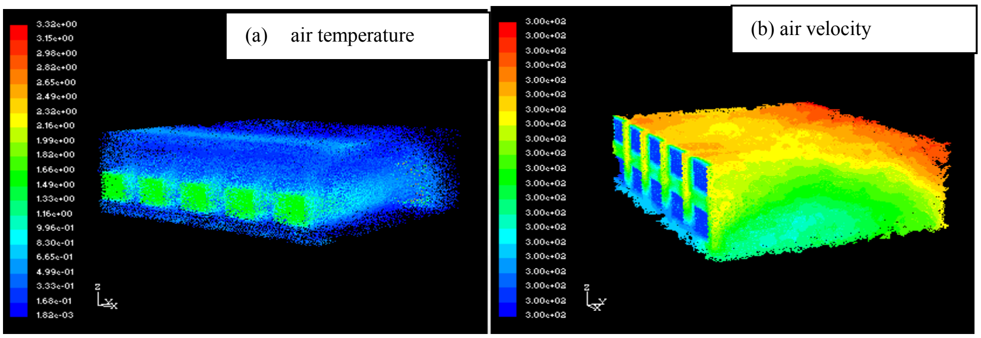

Fig. 2. Air temperature distribution and air velocity distribution for Case 1.

Fig. 2(a) shows the air temperature distribution for Case 1. It is obvious that the air temperature distribution in the whole room is uneven. The lowest temperature appears near the windows, and the highest temperature appears the top of the room which is the farthest away the windows. With the increment of height in the room, the indoor temperature increase gradually. The indoor temperature on the top of the room shows higher temperature. It indicates that the ventilation by the windows can decrease the middle and bottom of indoor temperature effectively. It should be take measures of mechanical ventilation away the windows to decrease the temperature.

Fig. 2(b) shows the air velocity distribution for Case 1. It can be seen from the figure that the air velocity entering from the windows is higher. When the air meets tables and chairs etc., the velocity decreases. The velocity around the lower part of the room is higher, while it reaches the highest at the right door. The velocity around the upper part is lower and stable.

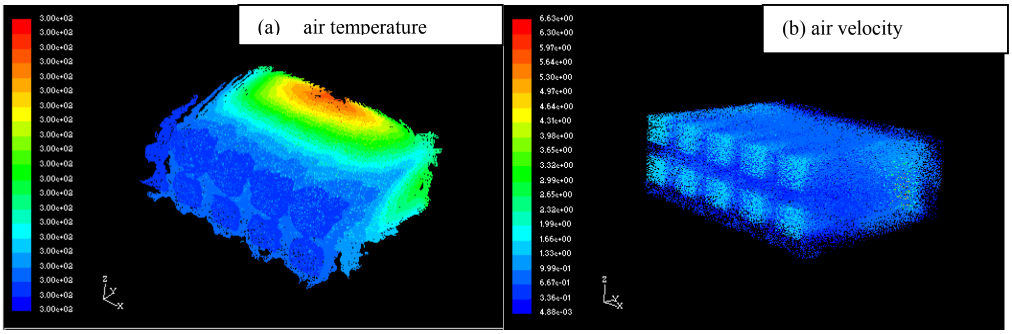

Fig. 3. Air temperature distribution and air velocity distribution for Case 2.

Fig. 3(a) shows the air temperature distribution for Case 2. It can be seen that the air temperature around left side is lower than that around right side. For the left side in the classroom, the air temperature in the upper zone is lower than that in the lower zone. But for the right side in the classroom, the air temperature in the upper and lower zone is higher than that in the middle zone.

Comparing Fig. 2(a) with Fig. 3(a), the air temperature in the upper zone is obviously different. It indicated that opening the upper rows of windows can decrease indoor temperature and affect the air temperature distribution positively, which reduce the higher temperature zones in the classroom.

Fig. 3(b) shows the air velocity distribution for Case 2. It can be found that the air velocity in the whole classroom is distributed evenly. The highest air velocity appears around the opening of both doors. Comparing Fig. 2(b) with Fig. 3(b), it shows that opening the upper windows can make air velocity distribution even. The air velocity of most zones in the classroom satisfied the comfort requirement, which are lower 0.8m/s.

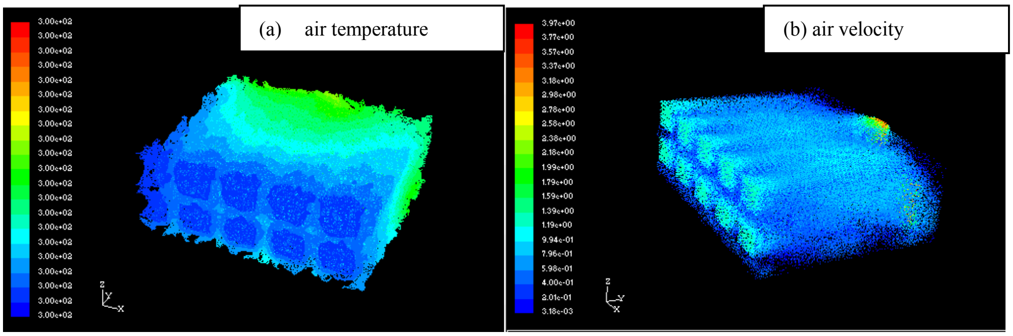

Fig. 4. The monthly energy consumption for different indoor personnel density.

Fig. 4(a) shows the air temperature distribution for Case 3. The figure presents that installing the exhaust fans on the right side can decrease the air temperature in the right zone of the classroom. The air temperature distribution in the whole classroom trends even. Comparing with Case 1, it can be found that the high temperature zone in the right upper part disappears. Comparing with Case 2, it can be found that the air temperature in the high temperature zone trends to decrease. Due to the operation of the exhaust fans, the higher temperature zone trends in the classroom to move down. The high temperature zone moves to the right bottom part of the classroom.

Fig. 4(b) shows the air velocity distribution for Case 3. It illustrated that installing exhaust fans has little effect on indoor air velocity. Comparing with Case 2, the indoor air velocity increases a little. The air velocity in the upper part is higher than that in the lower part. The air velocity in the occupied zone is even and satisfied with the comfort requirement.

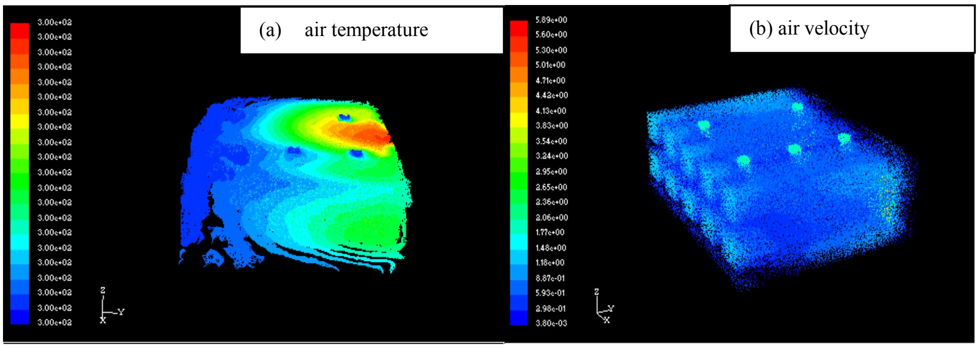

Fig. 5. Air temperature distribution and air velocity distribution for Case 4.

Fig. 5(a) shows the air temperature distribution for Case 4. It can be found that installing the ceiling fans has little effect on the indoor temperature. Fig. 5(b) shows the air velocity distribution for Case 4. It can be found that the air velocity distribution in whole classroom is even.

4. Discussion

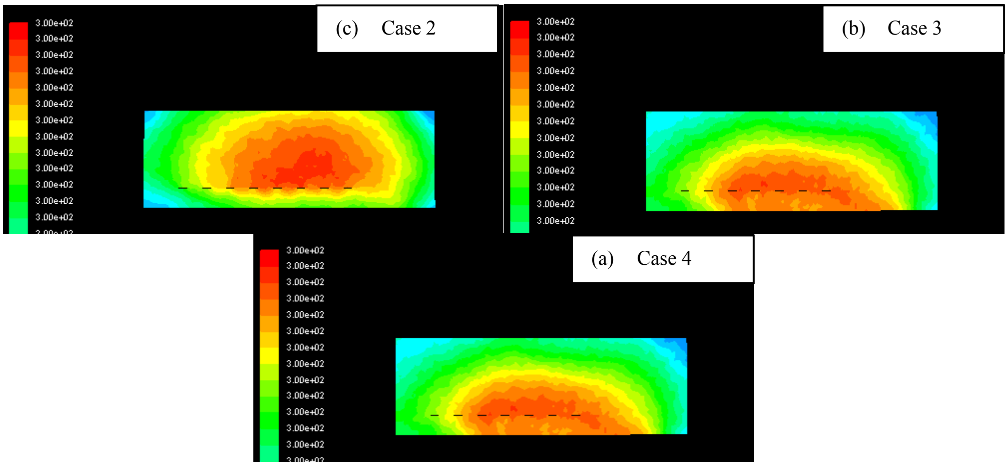

Fig. 6. The relationship between the annual power consumption and lighting power density.

In order to further observe the ceiling fans affection on indoor temperature, we make air temperature cross section on the position of 7.3m in width for Case 2, Case 3 and Case 4. The cross section charts are shown in Fig. 6.

Comparing Fig.6(a) with Fig.6(b), it can be seen that operation of the exhaust fan can decrease the air temperature at this section and the high temperature zone move down obviously. Comparing Figure 6(a) with Figure 6(c), it can be found that installing ceiling fans can decrease indoor air temperature obviously and the range of red high temperature reduces significantly. Base on Fig. 6(b) and Fig.6(c), it shows that the air temperature around the desk for Case 4 is better than that for Case 3.

In order to further observe the ceiling fans affection on air velocity, we make air velocity section on the position of 7.3m in width for Case 2, Case 3 and Case 4. The cross section charts are shown in Fig. 7 respectively.

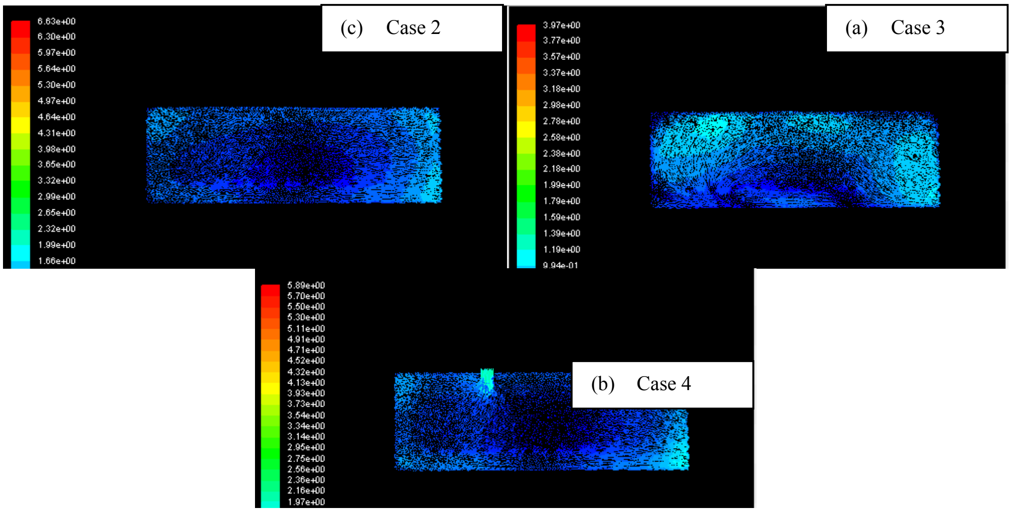

Fig. 7. Air velocity cross section for Case 2, Case 3 and Case 4.

Comparing Figure 7(a) with Figure 7(b), it can be seen that the operation of the exhaust fan increase indoor air velocity obviously, especially in the upper part of the classroom. Comparing Figure 7(a) with Figure 7(c), it can be found that the operation of ceiling fans has little effect on air velocity in the bottom part of the classroom. Base on the three figures, it indicates that installing exhaust fans can affect indoor air velocity distribution most obviously.

5.Conclusions

The boundary condition of the CFD simulation is measured on site. Simultaneously, the questionnaire survey is conducted for three weeks. In the statistical analysis, the good agreement between the investigation and the simulation is obtained: under all kinds of cases, the area in which students feel better generally agrees with the area in which the ventilation is good in the simulation. The students in the area where the ventilation is good feel more comfortable, and they could concentrate on the study while the students in the dead zone hardly act of attention, especially, some of whom feel chest stuffiness. These all support to the result of the simulation well.

The numerical simulation results showed that CFD can be used for predicting ventilation performance of school buildings. The results also indicated that double rows windows can help to improve the thermal comfort conditions most efficiently, and so do exhaust fans and ceiling fans. But ceiling fans can improve the environment of people positions better.

Moreover, the simulation result and the research method could provide basis for improvement and reference for installing fans in new school buildings and the similar architectures designed for natural ventilation. The method integrating simulation and investigation could be used in the similar study.

References

[1] M.R. Mokhtarzadeh-Dehghan, M.M.M.E Telebany, A.J. Reymonds, Transfer rates in single-sided ventilation, Building and Environments 25 (1990) 155–161.
[2] S.S. Ayad, Computational study of natural ventilation, Journal of Wind Engineering and Industrial Aerodynamics 82 (1999) 49–68.
[3] G.H. Gan, Effective depth of fresh air distribution in rooms with single-sided natural ventilation, Energy and Buildings 31 (2000)65–73.
[4] M.M. Eftekhari, L.D. Marjanovic, D.J. Pinnock, Air flow distribution in and around a single sided naturally ventilation room, Building and Environments 38 (2003) 389–397.
[5] T Hayashi, Y. Ishizu, S. Kato, S. Murakami, CFD analysis on characteristics of contaminated indoor air ventilation and its application in the evaluation of the effects of contaminant inhalation by a human occupant, Energy and Buildings 37 (2002) 219–230.