An Application Study on the Simulative Optimization of the Ventilation of the Existing Buildings Based on CFD

Songying Zhao¹, Xiao Si Su², Chen Chen³, Lei Chen⁴*

¹ Institute of Water Resources and Environment, Jilin University, Changchun 130000, China; Municipal and environmental engineering School of Jilin Jianzhu University, Changchun, China 130118

² Institute of Water Resources and Environment, Jilin University, Changchun 130000, China

³ College of Construction Engineering, Jilin University, Changchun 130000, China

⁴ Municipal and environmental engineering School of Jilin Jianzhu University, Changchun 130118, China

Abstract

A good design for the ventilation of buildings is a prerequisite for reducing building energy consumption as well as the most natural means of building energy efficiency. The computational fluid dynamics which is CFD for short is a combination of modern fluid mechanics, numerical mathematics and computer science. The use of CFD in the green building field can more accurately provide the basis for the design of the architectural wind environment. Architects will conduct analysis by linking the architectural technology and the related knowledge with the simulated results. Then multi-program selections and optimization of architectural design will be conducted to achieve the aim of energy efficiency of buildings. In this thesis, according to an existing canteen of a certain university in the north China and the buildings around the canteen, the natural ventilation effect will be analyzed by using CFD. And the energy saving optimization scheme is to be proposed in accordance with the simulation results of ventilation for the planning and design of the canteen and the building group, so as to achieve the best energy saving effect.

Keywords: ventilation of buildings, CFD, optimization scheme

1.1NTRODUCTION

A good design for the ventilation of buildings is a prerequisite for reducing building energy consumption as well as the most natural means of building energy efficiency. Natural ventilation refers to the indoor and outdoor air exchange that does not need special power setting (Chung and Hsu, 2001). The natural ventilation does not consume energy. According to different principles, the natural ventilation can be divided into the one under the wind pressure and the one under the hot embossing. The wind pressure is a static pressure generated when the airflow is blocked by the building. The shape and the spacing of buildings are the main factors that affect the natural ventilation under the wind pressure (Peng, 2013). The hot embossing is caused by the density difference of the air inside and outside the building. The air rises with the small density, vice versa. In this way, the hot air indoors rises and runs away from the windows and other exhaust vents to the outdoors. Meanwhile the cold air flows into the room from below, and forms
a hot and cold air convection inside the room, to reduce the indoor temperature. Natural ventilation under the action of wind pressure and hot embossing can both act on the building alone, but in fact the natural ventilation under the action of wind pressure and the natural ventilation under hot embossing exist simultaneously, to play a role in the ventilation of the buildings. Computational Fluid Dynamics (CFD) is a combination of fluid mechanics, numerical mathematics and computer science. It can accurately provide the basis for the design of the architectural wind environment (Zhan and Xu, 2011). Architects will conduct analysis by combining the architectural technology along with related knowledge with the simulated results. Then multi-program selections and optimization of architectural design will be conducted to achieve the aim of energy efficiency of buildings. In this thesis, CFD is used to analyze a group of existing campus buildings. By analyzing the effect of natural ventilation on a single building and a group of buildings, a modified scheme for existing buildings is proposed, which makes the ventilation of buildings achieve the optimal scheme of the building energy efficiency.

2. THE RELATIONSHIP BETWEEN THE BUILDING VENTILATION AND THE BUILDINGS

2.1 The relationship between the building ventilation and the single building

Air flow will produce a phenomenon of diversion at the corner of the windward side of the building, forming an obvious high-speed wind. The fundamental to weaken the high-speed wind is to optimize the shape of the building’s corner. From the perspective of weakening the airflow, the smoother the corner of the building is, the gentler the changes of the pressure difference between the windward side and the leeward side of the building are, and the weaker the influence of the high speed wind at the corner is. The corners of the windward side of the building should therefore be designed to be the round contours which conform to the aerodynamics. As shown in Fig. 1, which reflects the simulated result of the architectural wind environment, it can be seen that the L-shaped or U-shaped buildings, which can be often seen in the campus, should be designed with the notch facing the dominant wind direction, or the notch forms a small angle with the dominant wind direction when the building is laid out. This type of layout is conducive to reducing the wind shadow area of the building leeward side.

![Figure 1. The simulation of the wind environment of the single building](image)

2.2 The relationship between the building ventilation and the building groups

As for the campus, the opening treatment of the windward interface of buildings is the primary factor to improve wind environment of the campus, and its windward optimization determines the wind environment inside the buildings. The opening of the
windward side of the buildings can effectively increase the rate of flow and the speed of the wind. The appropriate opening of the boundary of the building should be reserved for any type of building layout to facilitate the introduction of wind to provide suitable airflow in the buildings. In the summer and transition season, in order to promote the natural ventilation of buildings, there are mainly two following methods of optimization: First, increase the number of openings. Second, increase the size of openings, as shown in Figure 2. In winter, the number and size of the openings should be in reasonable control of the leading wind direction to avoid the cold air’s breaking into the rooms, taking away the heat inside the buildings, and increasing the building energy consumptions. Meanwhile, the ventilation problem inside the buildings is supposed to be taken into consideration.

![Figure 2. A simulation of the wind environment of the buildings](image)

### 2.3 Wind environment and road planning

Roads form the transportation network on campus, which is typical of the narrow space on campus. Therefore, the ventilation of the main roads in summer and transition season should be fully utilized. When the road is in line with the prevailing wind direction, the circulation of the wind is less hindered. The road, as the ventilated corridor, can strengthen the natural ventilation in the area and can take away the polluted gas around the road, which is beneficial to the heat dissipation of the surrounding buildings in summer. When the angle between the road and the dominant wind direction is larger, the building on both sides of the road has larger wind blocking effect. The wind speed in the pedestrian area is lower, which is beneficial to the windbreak in winter and high pedestrian comfort. When the angle between the roadway and the dominant wind direction is less than 45°, the wind speed can be larger. Moreover, when the building is parallel with the road on both sides of the road, the wind speed is the highest, but the ventilation effect is worse inside the building. When the angle between the direction of the road and the dominant wind direction is 30°----60°, the ventilation effect can be good in the buildings on both sides of the road which are vertical or parallel to the road. Considering the windbreak in winter, it’s good to lay out the windbreak buildings or windbreak trees in the dominant wind direction in winter.

### 2.4 Wind environment and open space

Open space mainly includes public squares, green grounds and flake water. The open space, roads and buildings have a joint impact on the wind environment on campus. As the landscape or traffic nodes, the campus open space also plays the role of guidance for the wind. When the wind reaches the open space, the wind speed and the wind direction get reorganized due to the impact of the buildings, and then get inside the buildings through the road between buildings, forming a good ventilation system. Outdoor open space can buffer the adverse effects of the upwind wind field and promote the natural ventilation of the downwind buildings. In the campus there are many essential open space, rational planning of these open space can ease the effect of buildings’ blocking wind, and enclosure architectural layout is a typical representative of such a situation. When there are high-rise building layouts surrounding the open space, the open space...
can also weaken the high-speed wind generated by high-rise buildings, making the wind speed tend to be uniform and improving the comfort of the wind environment.

3. VENTILATION ENVIRONMENT OPTIMIZATION OF THE EXISTING BUILDINGS

3.1 Basic Situation

Take transforming the natural ventilation of the canteen of a certain college and the ventilation environment effects of the surrounding buildings for example, the existing buildings consists of six buildings. No. 1 is a canteen, No. 2 is a student bathhouse bath, No. 4 is a trade building, and No. 3, 5, 6 are the students’ apartments and other buildings. The plan of the building complex is as shown in Figure 3. The canteen is a three-storey building, which is 16.2 meters high, and covers an area of 1800 square meters. Considering the fact that the building is located in the northern part of China and is in the severe cold area and cannot be ventilated for a long period of time in winter, the CFD simulation of the ventilation environment is only carried out under the influence of the summer monsoon according to the situation of the canteen and other complex. The southwest wind prevails in summer and the wind speed is about 2.5m / s, the rationality of the ventilation environment of buildings is analyzed and optimized.

Figure 3. The plan for the architectural complex

3.2 The mathematical model and the control equation

According to the situation of the ventilation of the architectural complex, the relationship between the physical parameters and the flow parameters can be seen through the theoretical analysis(Zhao, 2015). Set up mathematical models and simplify the equation of the transfer of the flow and the heat.

The mathematical model as well as the flow equation of the incompressible fluid adopted in this thesis is

$$\frac{\partial \rho}{\partial t} + \nabla \rho u = 0$$

(1)

The equations of momentum conservation and energy transfer are such as follows:
\[
\begin{aligned}
\frac{\partial}{\partial t} \rho \nu + \nabla \cdot (\rho \nu \nu) &= -\nabla p + \nabla \cdot (\tau) + \rho g + F \\
\frac{\partial}{\partial t} \left( \rho E \right) + \nabla \cdot \left[ \nu (\rho E + p) \right] &= \nabla \cdot \left[ k_{\text{eff}} \nabla t - \sum h_j J_j + \left( \tau_{\text{eff}} \cdot \nu \right) \right]
\end{aligned}
\]  
(2)

In the equation above, \( E \) refers to energy, \( p \) means static pressure, \( \tau \) is the tensor of shear stress, \( p_g \) is the force of the quality, \( F \) is the external force, \( k \) is the effective conductance, \( h_j J_j \) is the component diffusion term, and \( \tau_{\text{eff}} \cdot \nu \) refers to the viscous dissipative term.

The turbulence model uses Model \( k-\varepsilon \):

The turbulent kinetic energy equation (\( k' \)) is:

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

The turbulent dissipative equation (\( \varepsilon' \)) is:

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon} \frac{\varepsilon}{k} (C_k + C_3 \varepsilon G_b) - C_{\varepsilon} \rho \frac{\varepsilon^2}{k}
\]  
(5)

In the equation above, \( G_k \), \( G_b \) refer to the turbulent kinetic energy terms caused by the average velocity gradients and buoyancy respectively. \( C_1 \), \( C_2 \), \( C_3 \) are the empirical constants. \( \sigma_k \) is \( k' \)‘s Prandtl number. And \( \sigma_\varepsilon \) is \( \varepsilon' \)’s Prandtl number.

3.3 The simulation and optimization of the single building

In this study, the single-building canteen is chosen as the research object, and the three-dimensional model is to be established, focusing on the study of the wind environment in the dining area. Considering the comfort, the study surface is made 1.2m above the ground. In this plane, the simulation of the wind environment is conducted, and the results are to be analyzed. Figure 4-a) is the plane plan for the first floor of the canteen, an area of 1800 m², with its height of 5.1m. There is a window where is marked with the letter C in the figure, a door marked with the letter M. Fluent simulation is performed, and the results are shown in Fig. 4-b).

It can be seen from Fig. 4-b) that on the right side of the first floor exists the wind with a high speed. The wind speed reaches 2.6m / s at the inlet, which belongs to the medium strong wind indoors. But the region does not belong to the dining area, which is conducive to the ventilation of the canteen. In other areas in the figure, the wind speed
is all less than 1 m/s, which belong to the quiet wind area. People will not feel the high-speed wind or the air does not flow when having meals, and the wind environment is relatively comfortable. Although there is a whirlpool area of air on the first floor, the comfort of dining will not be affected. Thus, it can be inferred that the effect of ventilation on the first floor is good.

(a) The plane figure of the first floor

(b) The simulative figure of ventilation of the first floor

Figure 4. The plane figure of ventilation of the first floor

Fig. 5-a) is the plane figure of the second floor of the canteen, covering an area of 1,810 m². The height of the building is 5.1m. The simulative results of ventilation environment can be seen in the following figure. Due to the unreasonable window layout, a large swirl area appears near the staircase, and the wind speed in the swirl area is less than 0.5m/s, which is not conducive to the air flow, which will cause air deposition and even form dirty air. It is analyzed the vortex area is near the staircase, the air in this area can get into the third floor or the first floor through the staircase passage. Thus, the air deposition in the vortex area can be resolved. It can be seen from the simulation results that there is the problem of imbalance between the wind speed and the air volume on the second floor. Through analysis, the reasons are that the small number of windows causes a large difference between the wind speed of the entrance and the exit and right angle changes of the building, making the wind at the corner affected. Above all, it is proposed that rounded pillars are provided at the corners to reduce the effect of wind.
Fig. 6 is the simulative figure of ventilation of the third floor. It can be seen from the results that the wind speed is less than 1m/s and a great many vortex areas exist. The low wind speed is not beneficial to the airflow without timely ventilation. The vortex area will make the air silt, and cannot timely send air outdoors. Taking into account that the great number of the dining staff and the intense flow of people, the ventilation of the third floor is unfavorable, which should be optimized.

According to the analysis, two optimizing schemes will be proposed, one is to open the skylight and the other is to set the fan in the three-layer whirlpool of the larger airflow. The natural ventilation and the mechanical ventilation will work together to keep the indoor air circulating and clean.
If three skylights are open on the roof, the position and the effect of the simulated ventilation are shown in Figure 7. Compared with Figure 6, after the opening of the skylights, the wind speed on the third floor is more than 1 m/s, which is significantly improved than before. This is conducive to the rapid flow of air, but the vortex areas do not reduce. There will also exist air deposition in the whirlpool area. It can be seen from this that open skylights on the third floor will have some effect on the ventilation, but not ideal. Therefore, at the same time of opening skylight, the exhaust devices should be installed, which is the best way to solve the problem.

3.4 The simulation of ventilation and optimization of the buildings

Fig. 8 is a simulation of the wind environment of the buildings. It can be seen from the results that there are two ventilation problems around the canteen. One is the partial wind speed is too large between Canteen One and Apartment Six, more than 10m / s. This strong wind results in the unacceptable discomfort for outdoor passers-by. Through analysis, the reasons for this local strong wind are the building layout of Canteen One and Apartment Six, forming a narrow space effect which causes the rapid wind speed in this range. The other problem is that there is the vortex area of airflow among Canteen One, Bathhouse Two and Commercial Building Four and the wind speed is too slow, and the air does not flow, resulting in poor air quality, and pollutants are not easy to spread. The reasons may be that the layout of these two buildings is too narrow, and they are located on the sharp corners of the building, resulting in a large pressure on the front
and back tuyeres. It is not conducive to the airflow, which helps to form a whirlpool area of airflow.

**Figure 8.** The simulative figure of ventilation of buildings

**Figure 9.** The simulative figure of the optimized ventilation of buildings

According to the problems above, two optimized schemes are proposed here. First, a series of high and dense vegetation is to be planted between Canteen One and Apartment Six, so that the vegetation can be used to block the airflow at the narrow space between the two buildings, reducing the wind speed between the two buildings, and controlling the wind speed in a reasonable range. Second, the sharp-angled contour of the windward of Canteen One should be changed into smooth one, to make the wind at the leeward more even to speed up the flow of wind, and drive the airflow, and finally to make the vortex area of airflow disappear. Figure 9 shows the optimized ventilation simulation of the buildings. It can be seen from the figure that the optimized simulation of ventilation environment of the buildings is good. Where the wind speed was rapid, with the vegetation covered, the wind speed reduces to 5m / s, and the vortex of airflow also between Canteen One and Commercial Building Four disappeared.

4. CONCLUSION

A good design for buildings’ ventilation is a prerequisite for reducing building energy consumption, and the most natural means of achieving building energy efficiency. In this thesis, CFD is used to simulate the wind environment of the single building of canteen and the surrounding buildings in the university in the north of China, analyzing the existing problems and proposing the solutions to optimizing the design scheme. The following conclusions can be drawn from the research:
The installation of windows of a single building should be paid attention. Try to form the draught, which can promote the indoor airflow and timely updates.

The layout of the indoor windows of the single building should be reasonable. The dominant direction of wind should be taken into consideration to prevent too many air intakes with more or fewer outlets, resulting in uneven wind flow and too rapid or too slow wind speed, all of which are not conducive to ventilation.

The layout of buildings should be reasonable to prevent forming a narrow space effect due to the big distance between the two buildings or forming a whirlpool area due to the small distance.

5. ACKNOWLEDGMENTS

This work was supported by the National Natural Science Foundation of China (Grant No.51606084); Ji Lin Sheng Jiao Yu Ting Science and Technology Research [2016] (Grant No. 149); Ji Lin Sheng Ke Ji Ting QingNian Science and Technology Research [2016] (Grant No. 20160520028JH); Zhu Jian Bu Science and Technology Research [2016] (Grant No. 2016-K1-30).

6. REFERENCES


