The effect of gap of building on indoor flow field and thermal environment

LI Hao^{1,a}, ZOU Zhijun, WANG Fei

¹School of Environment and Architecture, University of Shanghai for Science and Technology, Shanghai, 200093, China

^aemail: lh19820818@163.com

Keywords: gap of building, indoor flow field, thermal environment

Abstract: In summer and in winter, the outdoor air will flow in room through the gap of building. Which will influence the indoor flow field and thermal environment and cause draft, the human in room will feel uncomfortable. This paper use the method of numerical simulation to research the effect of outdoor air velocity on gap of building on indoor flow field and thermal environment. The results show indoor flow field and thermal environment will be influenced by outdoor air through the gap obviously. With the increase of outdoor air velocity, it will reach greater range.

Introduction

In every construction, the gap (such as on doors and windows) exists inevitably. It was caused by the assemble deviation of door, window or other part of building. And as time goes on, the gap will become large certainly. In summer and in winter, the outdoor air will flow in room through the gaps of building. Which will influence the indoor flow field and thermal environment and cause draft, the human in room will feel uncomfortable. The purpose of this article is to research the flow field and thermal environment in room on different outdoor air velocity through the gap.

Physical Models

Geometrical Model

Fig.1 is the simulation model of gap in a room. The dimension of room is 5000*3000*2500mm. There is an air inlet (800*200mm), an air outlet (300*300mm) and a gap (1000mm height and 10mm width) in the only outer wall. The other wall, ceiling and floor are inner wall.



Fig 1 Simulation model of gap in a room

Turbulence model

Standard k-epsilon model is used to simulate the indoor thermal environment and air flow field.

The K-epsilon model is one of the most common turbulence models. It is a two equation model, which means, it includes two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy.

The first transported variable is turbulent kinetic energy, k. The second transported variable in this case is the turbulent dissipation, ε . It is the variable that determines the scale of the turbulence, whereas the first variable, k, determines the energy in the turbulence.

There are two major formulations of K-epsilon models. That of Launder and Sharma is typically called the "Standard" K-epsilon Model. The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows.

Transport equations for standard k-epsilon model are followed.

$$\frac{\partial(rf)}{\partial t} + div(r\vec{U}f) = div(\Gamma_f grad f) + S$$

The φ in the equation can be velocity, turbulent kinetic energy, turbulence dissipation rate and temperature. For different equation, the specific form is shown in Tab.1.

Equation	φ	Γ_{f}	S _f		
continuity equation	1	0	0		
x-velocity	и	$m_{eff} = m + m_t$	$-\frac{\partial P}{\partial x} + \frac{\partial}{\partial x} \left(\mathbf{m}_{eff} \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mathbf{m}_{eff} \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial z} \left(\mathbf{m}_{eff} \frac{\partial w}{\partial x} \right)$		
y-velocity	v	$m_{eff} = m + m_t$	$-\frac{\partial P}{\partial y} + \frac{\partial}{\partial x} \left(\mathbf{m}_{eff} \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial y} \left(\mathbf{m}_{eff} \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left(\mathbf{m}_{eff} \frac{\partial w}{\partial y} \right)$		
z-velocity	w	$m_{eff} = m + m_t$	$-\frac{\partial P}{\partial z} + \frac{\partial}{\partial x} \left(\mathbf{m}_{eff} \frac{\partial u}{\partial z} \right) + \frac{\partial}{\partial y} \left(\mathbf{m}_{eff} \frac{\partial v}{\partial z} \right) + \frac{\partial}{\partial z} \left(\mathbf{m}_{eff} \frac{\partial w}{\partial z} \right) - \mathbf{rg}$		
turbulent kinetic energy	k	$a_k m_{eff}$	$G_k + G_B - re$		
turbulence dissipation rate	е	$a_{e}m_{e\!f\!f}$	$C_{1e} \frac{e}{k} (G_k + C_{3e} G_B) - C_{2e} r \frac{e^2}{k} - R_e$		
temperature	Т	$\frac{\underline{m}}{\mathrm{Pr}} + \frac{\underline{m}_{t}}{S_{T}}$	S_T		

Tab.1 Th	e specific	form of	transport	equation
IGOII III	o opeenie	101111 01	" and port	equation

Constants in Tab 2.1 are below:

$$G_{k} = \mathbf{m}_{t}S^{2}, \quad S = \sqrt{2S_{ij}S_{ij}}, \quad S_{ij} = \frac{1}{2} \left(\frac{\partial u_{j}}{\partial x_{i}} + \frac{\partial u_{i}}{\partial x_{j}} \right), \quad G_{B} = \mathbf{b}_{T}g\frac{\mathbf{m}_{t}}{\mathbf{s}_{T}}\frac{\partial T}{\partial y}, \quad \mathbf{m}_{t} = \mathbf{r}C_{m}\frac{k^{2}}{e}, \quad C_{m} = 0.0845,$$
$$C_{1e} = 1.42, \quad C_{2e} = 1.68, \quad C_{3e} = \tanh \left| \frac{v}{\sqrt{u^{2} + w^{2}}} \right|, \quad \mathbf{s}_{T} = 0.85, \quad \mathbf{s}_{C} = 0.7$$
$$\mathbf{a}_{k} = \mathbf{a}_{e}, \text{ calculated by } \left| \frac{\mathbf{a} - 1.3929}{\mathbf{a}_{0} - 1.3929} \right|^{0.6321} \left| \frac{\mathbf{a} + 2.3929}{\mathbf{a}_{0} + 2.3929} \right|^{0.3679} = \frac{\mathbf{m}}{\mathbf{m}_{eff}}$$

Where $a_0 = 1.0$

If $m \ll m_{eff}$, $a_k = a_e \approx 1.393$

$$R_e = \frac{C_m r h^3 (1 - h/h_0)}{(1 + bh^3)} \times \frac{e^2}{k}$$

Where h = Sk/e , $h_0 = 4.38$, b = 0.012

Simulation conditions

There are four conditions have been simulated. The difference is the velocity of outdoor air through the gap. Case 1 is zero, Case 2 is 0.5m/s, Case 3 is 1m/s and Case 4 is 2m/s. In simulation, the parameters of air inlet and all walls are listed in Tab.2. Air outlet was a free boundary; the parameters of it were calculated by simulation.

Simulation conditions	Air inlet		Gap		
	Air volume (m ³ /h)	Temperature	Air	Temperature	Ceiling, floor, inner wall and outer wall
		(°C)	velocity	(°C)	
			(m/s)		
Case 1	557	18	0	34.4	thermal insulation
Case 2			0.5		
Case 3			1		
Case 4			2		

Tab.2 Simulation parameters of model

Simulation results

Fig.2~Fig.5 shows the variety of velocity on cross section in Case 1 ~ Case 4. Fig.6 shows the velocity in center of room on X direction. The indoor flow field will be influenced by outdoor air through the gap obviously. With the increase of outdoor air velocity, it will reach greater range. In Case 2, when it is 0.5m/s, the range of influence is 0.5m. In Case 3, it increases to 1m distance. In Case 4, it will affect the supply air from air-condition system.



Fig.2 velocity contour on cross section in Case 1



Fig.3 velocity contour on cross section in Case



Fig.4 velocity contour on cross section in Case 3 Fig.5 velocity contour on cross section in Case 4

Fig.7 shows the temperature in center of room on X direction. In X=5, there is the outer wall. The thermal environment is similar to the result of flow field. The inflection point of temperature in Case 2 is about 0.4m distance from outer wall. In Case 3 and Case 4, it's about 1.3m distance from outer wall. Except the area near the gap, temperature in other area is stable. But due to the higher outdoor air velocity, the more heat will enter the room through the gap, and the temperature will

increase correspondingly. The temperature in Case 2 ~ Case 4 is 18.5°C, 18.9°C and 19.7°C. If want to get the same temperature in Case 2, Case 3 and Case 4, the more supply air volume or lower supply air temperature will be necessary.



Fig.6 The velocity in center of room on X direction Fig.7 The temperature in center of room on X direction

Conclusions

The indoor flow field and thermal environment will be influenced by outdoor air through the gap obviously. With the increase of outdoor air velocity, it will reach greater range.

In the room flow field, in Case 2, the range of influence is 0.5m. In Case 3, it increases to 1m distance. In Case 4, it will affect the supply air from air-condition system.

In thermal environment, the influence range in Case 2~Case 4 is 0.4m, 1.3m and 1.3m. And the temperature will increase correspondingly. The outdoor air velocity increase 2 times, indoor temperature will rise 0.4°C. When it increase 4 times, indoor temperature will rise 1.2°C. Which will influence indoor thermal environment significantly and the more energy consumption will be

need to decrease the temperature. In addition, the larger of gap will result in more energy consumption of air-condition system.

References

[1] Chen, Y., Liu, J., Pei, J., Cao, X., Chen, Q., and Jiang, Y. Experimental and simulation study on the performance of daylighting in an industrial building and its energy saving potential [J], Energy and Buildings, 73(2014), 184-191.

[2] Feng, Z., Long, Z., and Chen, Q. Assessment of various CFD models for predicting airflow and pressure drop through pleated filter system [J], Building and Environment, 75(2014), 132-141.
[3] P.V. Nielsen, C. Topp, M. Sonnichsen, et al., Air distribution in rooms generated by a textile

terminal-comparison with mixing and displacement ventilation[J], ASHRAE Transactions, 8(1) (2005), 733-739.

[4] F. Chen, H. Chen, J. Xie, et al., Characterizing airflow through fabric air dispersion system using a porous media model[J], Energy and Buildings. 43 (2011), 665-670.

[5] Jin, M., Zuo, W., and Chen, Q. Simulating natural ventilation in and around buildings by fast fluid dynamics, Numerical Heat Transfer, Part A: Applications [J], 64(4)(2013), 273-289.
[6] Zuo, W. and Chen, Q. Simulations of air distribution in buildings by FFD on GPU [J], HVAC&R Research, 16(6) (2010), 785-798.

[7] Zhang, Z., Zhang, W., Zhai, Z., and Chen, Q. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: part-2: comparison with experimental data from literature [J], HVAC&R Research, 13(6) (2007), 871-886.

[8] Zhai, Z., Zhang Z., Zhang, W., and Chen, Q. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: part-1: summary of prevalent turbulence models [J], HVAC&R Research, 13(6) (2007), 853-870.

[9] Wang, L. and Chen, Q. Validation of a coupled multizone and CFD program for building airflow and contaminant transport simulations, HVAC&R Research [J], 13(2) (2007), 267-281.