Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 158 (2019) 1255–1260

Energy Procedia

www.elsevier.com/locate/procedia

10th International Conference on Applied Energy (ICAE2018), 22-25 August 2018, Hong Kong, China

Determination of the optimal thickness of vertical air channels in double-skin solar façades

Tiantian Zhang, Hongxing Yang *

Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong, China

Abstract

Using interior air layers in building envelopes has become popular in modern building design and construction. With the help of solar radiation, the air natural convection in these air layers provide multiple benefits to the building envelopes. The flow and heat transfer process of the solar driven natural convection in the air channels can greatly influence the performance of these envelopes. This study numerically investigates the flow and heat transfer process in the vertical air channels of double-skin solar façades, and evaluates the influence factors of the temperature and velocity fields, in order to determine the optimal channel thickness. The results show that the flow transition, velocity promotion and temperature increase mainly occur in the near-wall regions. For vertical channels with the height of 2-4m, the thickness of 0.1-0.8m, and the input heat flux of 100-400W/m^2, the flow rate varies between 0.042 kg/s and 0.255kg/s, and the range of the temperature rise is 0.66-14.70℃. Increases in the channel height and the input heat flux may result in a straight increase in the flowrate and the temperature, while the influence of the channel thickness on the flow rate and the temperature rise is limited. For air channels intending to improve ventilation capacity, the channel thickness should not be bigger than 0.6m; while for channels with the purpose of supplying warm air, the thickness should be less than 0.2m.

Keywords: double-skin solar façade; vertical air channel; flow and heat transfer characteristics; optimal thickness

1. Introduction

Building envelopes act as the thermal interfaces between the indoor and outdoor environments, thus can greatly influence the indoor thermal condition and the energy need of air-conditioning systems [1]. As reported, approximately 20–50% of indoor HVAC energy consumption is resulted from the heat gain or loss through building envelopes [2]. Actually, buildings' energy demand is closely linked to exterior envelopes' thermal performances. If the exterior envelopes of a building are not properly designed, the excessive heat fluxes through these structures may result in a significant increase in HVAC energy demand. Therefore, it is important to develop high-performance exterior envelopes to guarantee both low energy consumption and high indoor thermal comfort for a building.

There are several measures to improve the thermal performance of exterior building envelopes, including using high-performance materials in building envelopes, adding insulation to building envelopes and improving envelope structure, etc. [3]. Currently, a widely used structural design strategy is to add interior air channels to building

1876-6102
10.1016/j.egypro.2019.01.415

envelopes. There are many different types of air channel applications in building envelopes, including Trombe walls, solar wall/chimney, double-skin façades, double-skin PV façades [4-8], etc. When the air channel operates in a closed mode, it performs as extra insulation layer for exterior envelopes. When air channel works in an open-ended mode, ventilation channels are formed in between the double-layer envelope structures, and the internal air is warmed up by solar radiation and starts to move upward, which induces air exchange between the channel and the outdoor or indoor environment. Air channels used in these solar façades can provide multiple benefits, including reducing the heating/cooling load and achieving the effects of fresh air preheating, space heating, natural ventilation, passive cooling, etc.

As verified by many literatures, using air channels in exterior building envelopes greatly improves the thermal performance, and provides an energy saving potential in indoor air-conditioning. It is noteworthy that the thermal performances and the overall energy efficiencies of these envelopes are closely related to the flow and heat transfer process of the solar driven natural convection in these air channels. Therefore, it is essential to make clear the flow and heat transfer characteristics of this solar driven natural convection in the air channels, and to investigate the influence factors of the induced air flowrate and the air temperature rise across the channel. This paper numerically investigates the flow and heat transfer process in vertical air channels to evaluate the influence factors of the velocity, temperature, and turbulent kinetic energy field, and to study the influence of geometrical sizes on the flow rate and temperature rise across vertical air channels.

2. Method

2.1. Physical and mathematical model

All air channel involved double-skin solar façades have similar configurations in the structure. Usually, an air channel is reserved in between two building material layers, and a solar absorbing layer made from different materials (massive wall material, metallic plate, absorptive glass, solar air collector, PV cells, etc.) is usually employed to absorb and convert solar radiation into thermal energy to heat internal air. A numerical model based on the conservation equations of mass, momentum and energy is established to simulate the airflow and heat transfer process in vertical air channels of building envelopes. The simplified physical model of the air channel is illustrated in Fig.1.

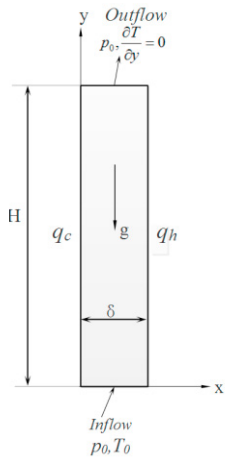

Fig.1 Physical model of airflow and heat transfer in the vertical air channel

The model consists of two vertical plates with the length of H, and the thickness of the structured air channel in between the plates is δ. The left and right plates are heated by constant heat flux q_c and q_h respectively. The flow and heat transfer process is assumed as steady state and two-dimensional natural convection driven by the heat flux. The air density satisfies the boussinesq approximation, and other property parameters are constant with Pr=0.7. The radiative heat transfer is ignored. The governing equations can be expressed as:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

$$\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left[\left(\mu+\mu_t\right)\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\mu+\mu_t\right)\frac{\partial u}{\partial y}\right] \tag{2}$$

$$\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left[\left(\mu+\mu_t\right)\frac{\partial v}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\mu+\mu_t\right)\frac{\partial v}{\partial y}\right]+\left(\rho-\rho_0\right)g \tag{3}$$

$$\rho u\frac{\partial T}{\partial x}+\rho v\frac{\partial T}{\partial y}=\frac{\partial}{\partial x}\left[\left(\frac{\lambda}{c_p}+\frac{\mu_t}{\mathrm{Pr}_t}\right)\frac{\partial T}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\frac{\lambda}{c_p}+\frac{\mu_t}{\mathrm{Pr}_t}\right)\frac{\partial T}{\partial y}\right] \tag{4}$$

In laminar regime, the above equations are solved directly, as the turbulent viscosity (μ_t) is zero due to the flow pattern. In turbulent regime, the turbulent viscosity should be determined by turbulent models. In this study, the standard *k-ε* model is employed.

$$\rho u\frac{\partial k}{\partial x}+\rho v\frac{\partial k}{\partial y}=\frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial y}\right]+G_k+G_b-\rho\varepsilon \tag{5}$$

$$\rho u\frac{\partial \varepsilon}{\partial x}+\rho v\frac{\partial \varepsilon}{\partial y}=\frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial y}\right]+C_{\varepsilon 1}\frac{\varepsilon}{k}\left(G_k+C_{\varepsilon 3}G_b\right)-C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \tag{6}$$

In the above equations, Pr_t, σ_k, σ_ε are Prandtl numbers for *T, k*, and *ε*, the values are 0.9, 1.0 and 1.3 respectively [35]. G_k is the generation rate of kinetic energy; G_b is the buoyancy force; μ_t represents the turbulent viscosity; δ_{ij} is the Kronecker delta; C_u, $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$ are constants with the values of 0.09, 1.44, 1.92 and 1.0 respectively [34]. G_k ,G_b, μ_t and δ_{ij} can be determined as:

$$G_k=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}-\frac{2}{3}\rho k\delta_{ij}\frac{\partial u_i}{\partial x_j} \tag{7}$$

$$G_b=\beta g_i\frac{\mu_t}{Pr_t}\frac{\partial T}{\partial x_i};\ \mu_t=\rho C_\mu\frac{k^2}{\varepsilon};\ \delta_{ij}=\begin{cases}0 & if\ i\neq j\\ 1 & if\ i=j\end{cases} \tag{8}$$

At the inlet, ambient temperature and pressure conditions are applied, and the turbulent kinetic energy and dissipations energy are set to be zero. On the sidewalls, no-slip boundary condition is applied, and uniform heat fluxes are imposed on both sides of the channel; the turbulent kinetic energy vanishes at the wall. At the outlet, ambient pressure is applied and the temperature gradient is set to be zero. The boundary conditions are:

$$\begin{cases}x=0,q=q_c,u=0,v=0,k=0;\\ x=\delta,q=q_h,u=0,v=0,k=0;\\ y=0,p=p_0,T=T_0,k=0,\varepsilon=0;\\ y=H,p=p_0,\dfrac{\partial T}{\partial y}=0.\end{cases} \tag{9}$$

For the thermal boundary conditions of the left and right sidewalls, there is a fixed ratio between q_c and q_h. in this study, this ratio is 0.077 (q_c/q_h=0.077).

2.2. Model validation

To evaluate the model reliability and accuracy, the simulation results obtained from the established model were compared to the experimental data of Yilmaz and Fraser [9]. Fig.2 presents the comparisons between the numerical and experimental results of the turbulent kinetic energy, the velocity and temperature, respectively. For the turbulent kinetic energy profiles illustrated in Fig.2(a), although observable difference can be found between the numerical and experimental results, the changing tendencies are similar. So the established model can provide somehow satisfactory results for the turbulent kinetic energy. As demonstrated in Fig.2(b), the velocity profiles of the numerical results at the channel heights of 0.09m, 1.50m and 2.97m show very good agreements with the experimental results. It can be also found that the established model gave a close predicted results of the temperature field, when compared with the test results, as shown in Fig.2(c). To sum up, the established numerical model and the solution techniques can provide satisfactory prediction results for the temperatures and velocities in the vertical channel, and can show properly the vitiation tendencies of the turbulent kinetic energy.

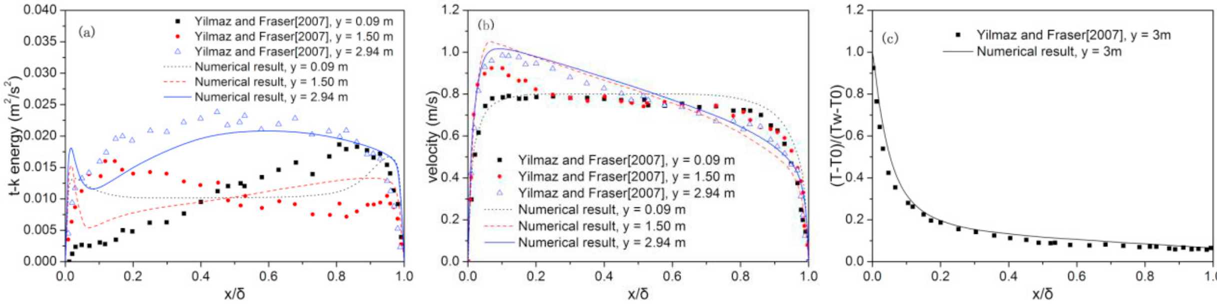

Fig.2 Validation of mathematical model on turbulent kinetic energy (a), velocity (b), and temperature (c) profiles

3. Results and discussions

3.1. The flow and heat transfer process in the channel

Fig.3 illustrates the turbulent kinetic energy, the velocity and the temperature distributions in the channels with different thicknesses, when the channel height is 3m and the heat flux is 200W/m^2.

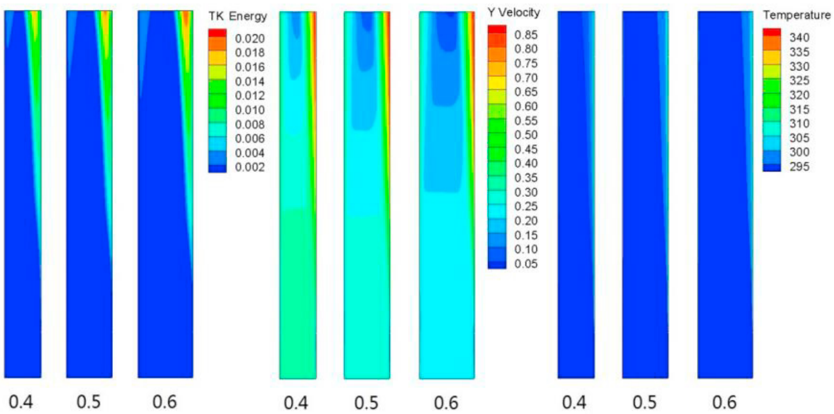

Fig.3 Distributions of the turbulent kinetic energy, velocity and temperature in vertical air cavities with thickness of 0.4-0.6m

As the figure shows, the turbulent kinetic energy increases with the channel height along the hot sidewall due to the input heat flux. The distributions of the kinetic energy are similar for these three channels. The kinetic energy grows on the both sides of the channel, and reaches its maximum at the outlet of channel. The values near the sidewalls are clearly higher than in other regions. The changing tendencies of the turbulent kinetic energy show that the airflow in the channel translates from laminar to turbulent in the near-wall regions of the hot sidewalls. The velocities near the sidewalls are much higher than in the central regions. Along the sidewall, the velocity keeps increasing with the increase of the height, and at the outlet, the maximal velocity would reach around 0.85m/s for the vertical channels with the thickness from 0.3m to 0.5m. In the central region, the velocity gradually decreases when the height increases. The minimum velocity points locate at the center of the exit. The velocity fields reveal the fact that the velocity improvement of the internal air mainly occurs in the near wall region; while in the central region of a channel, the air velocity gradually decreases with the height. Thus, increasing the channel thickness can help to improve the flowrate through the channel, but when the thickness exceeds a limit, the improvement may not be remarkable. Seen from the temperature contours, the heating effect of the heat flux on the induced airflow mainly occurs in the adjacent regions of the sidewalls. The maximal temperatures at the exit could be increased above 60℃ for all these channels. At the exit, the internal air is not completely heated.

Fig.4 illustrates the changing tendencies of these three parameters at different heights of the channel, when the channel height is 3m, the thickness is 0.4m, and the heat flux is 200W/m^2. The turbulent kinetic energy grows in the vertical channel when the flow and heat transfer progresses. In each channel, the peak value of the turbulent kinetic energy occurs at the exit and locates near the right wall. The values are higher in the near-wall regions than in the central regions. The result indicates that when constant heat fluxes act on the side walls of the channel, the induced flow grows more and more chaotic, especially in the near-wall regions of the side walls, thus the flow pattern translates from laminar to turbulent along the sidewalls. As the velocity curves shows, in the near-wall regions of the right side wall, the velocity increases with the height, while in the central region of the channel, the velocity decreases as the height increases. The exit velocities near the left wall and the right wall can reach up to around 0.35m/s and 0.9m/s respectively. The temperature variation curves at different heights show that the constant input flux at the left and right boundaries can improve the air temperature in the vertical channel, but only the air in adjacent regions of the sidewalls can be warmed. Thus, it can be inferred that the effect of the channel thickness on the air heating has limitation, since the heating occurs only within a very small area near the sidewalls.

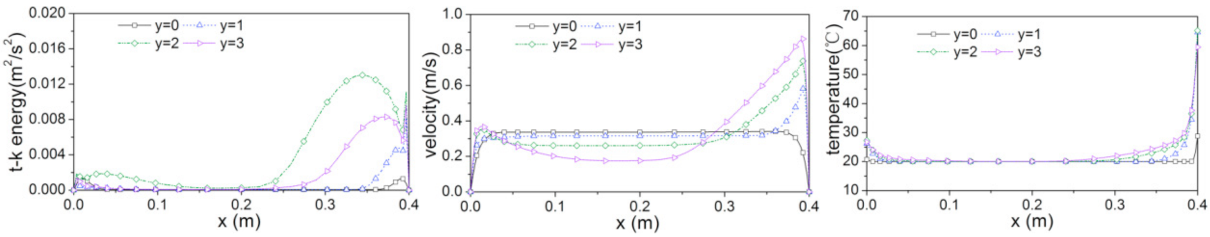

Fig. 4 Distributions of the turbulent kinetic energy, velocity and temperature at different heights of the channel

From the above analysis, the values of the turbulent kinetic energy, the velocity and the temperature in the near-wall regions are much higher than in the central regions of the vertical channel. Therefore, for heat flux driven turbulent natural convection in vertical air channels, the flow transition, velocity and temperature increase mainly occur in the near-wall regions.

3.2. Influence factors of temperature rise and induced flow rate

The main purpose of using air channels in building envelopes is to achieve passive cooling, natural ventilating or space heating effect for these envelopes. Thus, the airflow rate and air temperature rise across the vertical channel are worthy of investigating. The simulation results show that for vertical channels with the height of 2-4m, the thickness of 0.1-0.8m, and the input heat flux of 100-400W/m^2, the flow rate varies between 0.042 kg/s and 0.255kg/s, and range of the temperature rise is 0.66-14.70℃.

Fig.5 illustrates the changing tendencies of the flow rate and temperature rise with the channel height and thickness, with the heat flux of 200W/m^2. When the heat flux and the channel thickness are constant, the increase in the height can result in a rapid improvement in the flow rate and the temperature rise.

The flow rate improves with the increase of the thickness, and the growth rate becomes lower and lower. When the channel height is constant, if the channel thickness is lower than 0.6m, the effect of the thickness increase on the flow rate is quite significant; but when the thickness is bigger than 0.6m, the flow rate improvement is very limited due to the thickness increase. Accordingly, for the vertical channel with the purpose of promoting ventilation effect, the thickness should be no bigger than 0.6m. Similarly, the temperature rise decreases with the increase of the thickness, and the reduction rate becomes lower and lower when the thickness increases. When the thickness exceeds a certain value, the temperature rise caused by thickness increase will be so limited and it can be neglected. For instance, when the heat flux is 200W/m^2, the height is 3m, and the thickness is 0.2m, the temperature rise is only 4.5℃. Considering the return air temperature of a room, the temperature rise may not be sufficient to satisfy the supply air temperature. Thus, for this purpose, the channel thickness should be less than 0.2m.

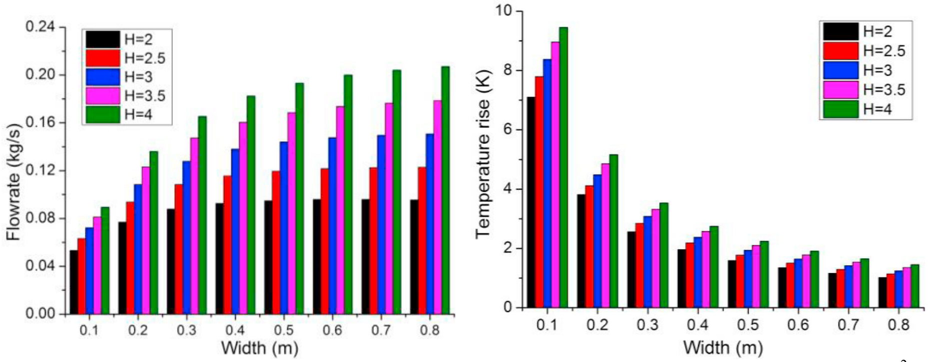

Fig.5 Influence of channel height and thickness on airflow rate and temperature rise (heat flux: 200W/m^2)

4. Conclusion

The study examines the flow and heat transfer characteristics of natural convection in asymmetrically heated vertical air channels of building envelopes. Based on the above results and discussions, the following conclusions have been drawn:

- For heat flux driven turbulent natural convection in vertical air channels, the flow transition, velocity and temperature improvement all occur in the near wall regions.
- The influences of channel height and heat flux on induced flow rate and temperature rise are significant; while the influence of channel width is limited.
- For air channels with the purpose of prompting ventilation effect, the width should not be bigger than 0.6m; for air channels with the purpose of hot air supplying, the width should be less than 0.2m.

Acknowledgements

This research was supported by The Hong Kong Polytechnic University Postdoctoral Fellowships Scheme (G-YW2E).

References

[1] Fang Z, Li N, Li B, et al. The effect of building envelope insulation on cooling energy consumption in summer. Energy & Buildings, 2014, 77(7):197–205.
[2] Yu J, Tian L, Xu X, et al. Evaluation on energy and thermal performance for office building envelope in different climate zones of China. Energy & Buildings, 2015, 86:626-639.
[3] Sozer H. Improving energy efficiency through the design of the building envelope. Building & Environment, 2010, 45(12):2581–2593.
[4] Sadineni S B, Madala S, Boehm R F. Passive building energy savings: A review of building envelope components. Renewable & Sustainable Energy Reviews, 2011, 15(8):3617-3631.
[5] Stevanović S. Optimization of passive solar design strategies: A review. Renewable & Sustainable Energy Reviews, 2013, 25:177-196.
[6] Chan H Y, Riffat S B, Zhu J. Review of passive solar heating and cooling technologies. Renewable & Sustainable Energy Reviews, 2010, 14(2):781–789.
[7] Barbosa S, Ip K. Perspectives of double skin façades for naturally ventilated buildings: A review[J]. Renewable & Sustainable Energy Reviews, 2014, 40:1019-1029.
[8] Zhang T., Tan Y., Yang H., etc. The application of air layers in building envelopes: A review. Applied Energy, 2016,165: 707–734.
[9] T. Yilmaz, S.M. Fraser, Turbulent natural convection in a vertical parallel-plate channel with asymmetric heating, Int. J. Heat Mass Transfer 50 (2007) 2612–2623.