STUDY ON THE AIR MOVEMENT CHARACTER IN SOLAR WALL SYSTEM

Y Li, X Duanmu, Y Sun, J Li and H Jia

College of Architecture and Civil Engineering, Beijing University of Technology, Beijing, 100022, China

ABSTRACT

A mathematical model for simulating airflow in solar channel of the insulated Trombe solar wall system is proposed. It is assumed the glazing is isothermal and the solar heat absorbed by the wall is transferred to the air in the channel with a constant flux by natural convection. The mass, momentum and energy conservation equations are discretized and solved using the finite difference control volume method. An experimental study of solar chimney was used to validate the proposed mathematical model. The differences between the predicted results of airflow rate in solar wall and those of measurement data are less than 3.0% when the width of solar wall is 0.2m and wall temperature is lower than 50℃. When the wall is 0.3m wide, theses differences are lower than 5.0% is the wall temperature less than 50℃. Flow and temperature fields are produced and the results are presented in terms of temperature and velocity distribution in various parts of system. The results show that the solar heat gain and channel width are two important parameters affecting the air flow pattern and heat transfer. Further experimental work is needed to refine the model.

KEYWORDS

Solar wall, Building, Natural convection, Modeling

INTRODUCTION

Two ways where solar energy is common used for buildings are solar chimney and solar wall. A solar chimney is one way in which one or more walls of vertical chimney are made transparent by providing glazed walls. Solar chimney is designed to provide ventilation to the building during the day and is located on the top of building. It plays an important role in providing a thermally suitable environment for human comfort in under-developed countries by providing natural ventilation in dwellings. Solar chimney is similar to the classical Trombe solar wall concept (Ormiston et al 1987),. The distinct between them is that wall in solar chimney is assumed to have negligible mass while the Trombe wall has a massive thermal bulk that absorbs solar energy and recirculates warm air for passive heating of the building. There has been extensive use of solar energy for heating buildings by means of storage walls since the works of Trombe were published. A comparatively study of four different configurations of solar wall was finished and reported their main advantages and disadvantages (Zalewski et al. 2002). The standard Trombe wall has the drawback of low thermal resistance which leads to significant losses at night-time of during periods with to sun. A composite solar wall, or a Trombe-Michel wall concept, an insulated Trombe wall etc. have been put forward in order to avoid the shortcoming of Trombe wall (Zrikem and Bilgen, 1987)

There are two major categories in analyzing the solar wall system. First, the natural convection between two parallel plates was concerned when simulating the solar chimney; Second, The heat transfer in the entire system including the solar wall collector and the adjacent room was simulated . A 2.66m high Trombe wall with air gaps varying from 0.10 to 0.35m were studied by flow visualization technology (Akbarzaden 1982). It was concluded that the mode of heat transfer resembled that of turbulent free convection between two single plates and suggested that 0.25 m was an optimum gap. Experimental tests were carried out on a 1:12 small-scale model of the prototype of a solar wall. The experimental results were then used to validate a two-dimensional laminar flow simulation model. The thermal resitance networks methods and empirical formula are used in this study. but it was envisaged that flow predictions could be improved upon by taking account of turbulence and three-dimensional effects as well as employing appropriate boundary conditions (Ong 2003).

The objective of the present study is to apply the technique of computational fluid dynamics (CFD) to simulating air flow and heat transfer in the solar channel of solar wall system. A CFD program was first validated against experimental data for an isothermally heated chimney. Numerical investigation was then carried out into the performance of the glazed solar channel. Effects of geometric parameters such as the channel width, solar heat gain are examined.

MATHEMATICAL MODEL

The Schematic diagram of an insulated wall system solar wall system considered in the present paper is illustrated in Figure 1(Zalewski et al. 2002). The solar wall comprises a transparent outer cover i.e. glazing, a storage wall, a ventilated air layer, and finally an insulation layer. Two vents have been drilled in massive walls. An insulation layer is fit on the back of massive wall in order to increase the thermal resistance of the Trombe wall. This insulation layer blocks off virtually all the supply in summertime. It is therefore no longer essential to fit a solar shield. This solar wall works as follows: the storage wall absorbs part of the solar energy. Solar energy heats up the air inside the chimney. As a result of the difference in air density between the top and bottom of the chimney a natural convection airflow is thermally induced. In this solar wall system, nearly all the energy are transferred to the air by convection. Moreover, the energy collected could be rapidly directed into the room by the air layer (short time lag). When blocking off the circulation of the air, the supply is also stopped, thereby overheating could be avoided.

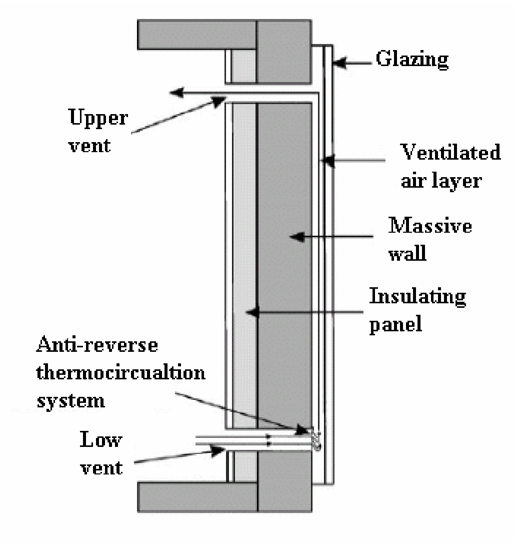


Figure1. Schematic diagram of insulated Trombe wall

To study the natural convection in solar chimney, the flow is assumed to be steady, turbulence and three-dimensional. The Boussinesq approximation is used to account for the density variation. Applications of CFD, based on solving a set of three-dimensional equations derived from conservation laws on mass, momentum and energy are found to be suited for simulating natural convection and ventilation.

The prediction of airflow in the present solar channel is based on the solution of general transport equation (Pantankar 1980):

$$div(\rho\vec{V}\Phi) = div\left[\Gamma_{\Phi} grad(\Phi)\right] + S_{\Phi} \qquad (1)$$

where Φ denotes the dependent variable, ρis the air density, Γ_{Φ} is the diffusion coefficient for variable Φ and S_{Φ} is source terms for variable Φ. Γ_{Φ} and S_{Φ} are given in Table 1(Tao 2001). As for energy equation, the solar radiation is not taken as source term. The effect is considered as giving the heat flux of wall surface.

Table 1: Transport equations for variable Φ in the flow field Figure 1.

Equation	**Φ**	Γ_{Φ}	S_{Φ}
Continuity	**1**	**0**	**0**
U Momentum	**u**	μ_{eff}	$-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left[\mu_{eff}\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[\mu_{eff}\frac{\partial v}{\partial x}\right]+\frac{\partial}{\partial z}\left[\mu_{eff}\frac{\partial w}{\partial x}\right]$
v Momentum	**v**	μ_{eff}	$-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left[\mu_{eff}\frac{\partial u}{\partial y}\right]+\frac{\partial}{\partial y}\left[\mu_{eff}\frac{\partial v}{\partial y}\right]+\frac{\partial}{\partial z}\left[\mu_{eff}\frac{\partial w}{\partial y}\right]+g(\rho-\rho_0)$
W **Momentum**	W	μ_{eff}	$-\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left[\mu_{eff}\frac{\partial u}{\partial z}\right]+\frac{\partial}{\partial y}\left[\mu_{eff}\frac{\partial v}{\partial z}\right]+\frac{\partial}{\partial z}\left[\mu_{eff}\frac{\partial w}{\partial z}\right]$
T Temperature	**T**	$\frac{\mu}{\Pr}+\frac{\mu_t}{\sigma_t}$	**0**
Kinetic energy	k	$\frac{\mu_{eff}}{\sigma_k}$	**G-ρε**
Dissipation rate	**ε**	$\frac{\mu_{eff}}{\sigma_{\varepsilon}}$	$\frac{\varepsilon}{k}(C_1 G - C_2\rho\varepsilon)$

$\mu_{eff} = \mu + \mu_t = \mu + \rho C_{\mu} k^2/\varepsilon$

$C_{\mu} = 0.09$, $C_1 = 1.44$, $C_2 = 1.92$, $\sigma_k = 1.0$, $\sigma_{\varepsilon} = 1.3$, $\sigma_p = 0.7$

The generation term G is defined as follows:

$$G = G_k + G_B = \mu_t\left\{2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial w}{\partial y}\right)^2\right]+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)+\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)^2+\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)^2+g\beta\frac{\mu_t}{\sigma_t}\frac{\partial T}{\partial y}\right\} \qquad (2)$$

The first term on the right side is shear production. Since buoyancy would play an important role in the rising airflow, the production of turbulence due to buoyancy and the effect of thermal stratification on the turbulence dissipation rate are included by the second term on the right side. β is the coefficient of thermal expansion

The finite volume method is used to solve the time-averaged Navier-Stoke equations with a non-uniform network. The diffusion terms and other gradients are discretized using the second-order central difference approximation. The first-order upwind scheme is employed for the convective terms. Standard wall functions are used for the enclosure walls.

Boundary conditions

In order to satisfy overall conservation of mass, the velocity components are extrapolated from upstream nodal points and then adjusted to the desired airflow rates. At the outlet, a constant mass flux is applied.
At the inlet air temperature is assumed equal to the room air temperature and assumed constant. Warm air leaves the channel at the exit and flows back room. Temperature at the surfaces of glass is assumed to keep isothermal. Resistance to flow due to friction along the surfaces is assumed negligible. The heat flux on the surface of massive wall is assumed constant. Non-slip condition is used for wall surface velocities.

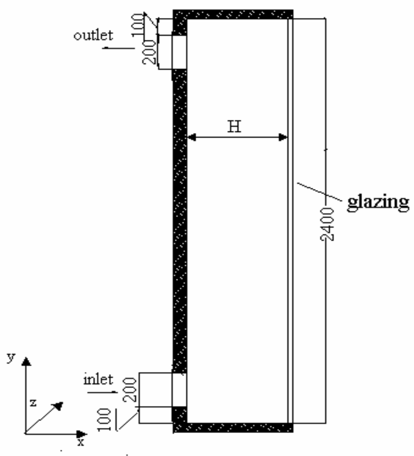


Figure 2. The configuration of solar channel

Figure 2 shows the configuration of solar channel in the insulated Trombe solar wall system. It is 2.4 m tall and of variable width H. The length in z direction is 1.0 m. The inlet and outlet openings are 0.2 m high and 1.0 m wide. The glazing is assumed to face southward. The rest walls are insulated brickwork.
The calculated region is divided into 30, 40 and 20 computation cells along x, y and z direction respectively. The grid independent for CFD are carried out and it is found this grid system is reasonable simulating time consumption and can get good accuracy result. The solution domain is discretized by using a non-uniform mesh with smaller grid spacing near the walls and larger spacing in the interior, which allows the hydrodynamic and thermal boundary layer to be resolved without an excess of nodes.

RESULTS AND DISCUSSIONS

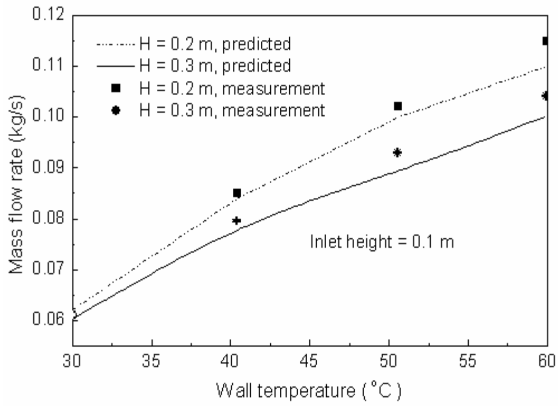


Figure 3. Comparison between the predicted results of mass flow rate and experimental data

Validation of program is performed by comparing the predicted results with experimental data for natural convection in a solar chimney (Bouchair 1994). The chimney was 2 m high and of variable width. All the wall surface were isothermally heated by electrical heater to temperatures from 30℃ to 60 ℃. The inlet air temperature was controlled at 20℃. Patterns of air movement were measured with the use of smoke. Detail of the experimental measurement and results were given by Bouchair(1994). In the experimental study of solar chimney, the air flow rate through a solar chimney is given by

$$Q = C_d A \sqrt{\frac{2\Delta P}{\rho}} \tag{3}$$

Where Q is volume flow rate (m^3/s), C_d is the discharge coefficient. A is the inlet opening area (m^2), and ΔP is the driving pressure due to buoyancy and wind effects. The measurement data of airflow rate in the conditions where wind effect is not considered are used to validate the proposed model.

The predicted and measured mass flow rates per unit length of the chimney for channels with width of 0.2 and 0.3 m are shown in Figure 3. The inlet height of solar chimney is 0.1m. The differences between the predicted results of airflow rate in solar wall and those of measurement data are less than 3.0% when the width of solar wall is 0.2m and wall temperature is lower than 50℃. When the wall is 0.3m wide, theses differences are lower than 5.0% is the wall temperature less than 50℃. This gives confidence in using the computer code to study the airflow and heat transfer in solar cavities in this study.

Since the inlet air velocity is given in this study, it is important to calculate the quantity of heat transferred to airflow in channel. The useful heat transfer to moving air stream can be written as (Ong 2003):

$$\dot{q} = \frac{\dot{m}c_f\left(T_f - T_{f,i}\right)}{\gamma HL} \tag{4}$$

where $\dot{q}$ is heat flux transferred to moving air stream. γ is a constant in mean temperature approximation, Hirunlabh et al took a value of $\gamma = 0.75$ form their experimental observation, $\dot{m}$ is mass flow rate, c_f is air specific heat, L is section length at z direction and is 1 m. T_f is mean air temperature which can be approximated from experimental observation.

$$T_f = \gamma T_{f,i} + (1-\gamma)T_{f,o} \tag{5}$$

where $T_{f,i}$ and $T_{f,o}$ are temperatures of air at inlet and outlet respectively.

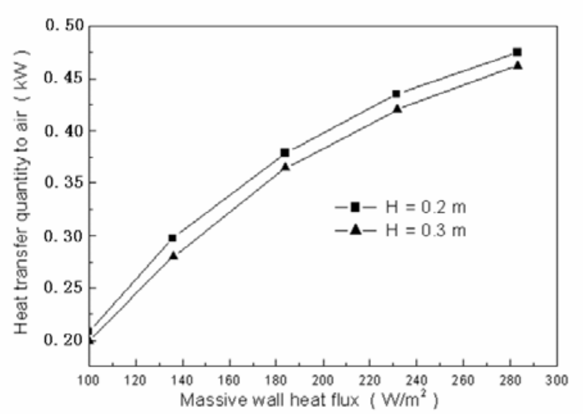
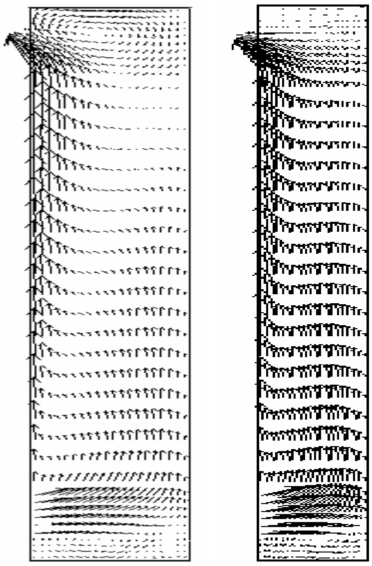


Figure 4. Quantity of heat transfer to air in channel

In this paper, the inlet air temperature is 20 ℃. The glazing inner surface temperature is 25 ℃. The inlet air velocity, u, is 0.2 m/s. The wall solar heat gain is calculated from the mean total solar irradiance and mean solar gain factor. For example, the mean solar irradiance on a vertical south surface is 400 W/m^2, as for a vertical wall with double glazing whose mean solar gain factor is 0.64 (Hirunlabh et al 1999), the corresponding wall solar heat gain is 256 W/m^2. For the wall, only glazing facing side has a constant heat flux the other side is taken as adiabatic. Two channels with width of 0.2 and 0.3 m are studied. For the same channel, two conditions where $\dot{q}_{wall}$ are 280 and 140 W/m^2 are calculated respectively. The radiation between the back wall and glass is considered as the six-flux thermal radiation model is selected in CFD program.

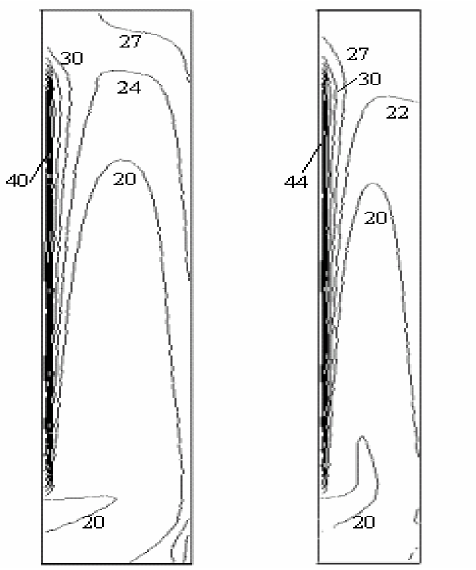
The quantities of heat transfer to moving air in the channel under different conditions are illustrated in Figure 4. It can be seen that heat transfer quantity is 0.475 kW for $\dot{q}_{wall}$ = 280 W/m^2 while it is 0.302 kW for $\dot{q}_{wall}$ = 140 W/m^2 when the channel width is 0.2 m. The higher the massive wall heat flux, more heat would be transferred to moving air. At low wall heat flux, more heat transfer quantity would be reduced for the same decrease degree of wall heat flux for the same channel conditions. The large reduction for lower heat flux is partly due to the flow reversal near the glazing. The effect of channel width on heat transfer quantity to air is also illustrated in Figure 4. It can be seen for the same wall heat flux, the heat transfer quantity in narrow channel is larger than in channel of larger width. The reason is that the air mass flux is constant and the air velocity in narrow channel is higher. The lower velocity would result in a lower heat transfer coefficient for the convection on massive wall surface. However, there should be an optimum channel width because the very narrow channel could lead to the plug flow where the mass flow rate $\dot{m}$ in Eq.4 is very small and it would influence the heat transfer.

Figures 5 shows the flow pattern in two solar channels. For the 0.3 m wide solar channel it can be seen clearly that there is circulate flow at the top of solar channel This means there is a larger air rising velocity in this condition. As for the channel of 0.2 m wide, there is a higher velocity at the cross section compared with that of 0.3 m wide channel with the same inlet air velocity. The reason is that a larger airflow velocity is needed for passing the same mass flow rare with a narrower passage. The temperature field in two channels with 280 W/m^2 wall heat flux are illustrated in Figure 6. It can be seen that temperature of air near massive wall is higher than those of air in the channel centre and that near glazing. The highest air temperature for 0.3m wide channel is 40 ℃ while the value is 44 ℃ in the 0.2 wide channel. This is because the higher airflow velocity leads to a lager quantity of heat transfer and thereby a higher temperature near the massive wall surface.



(a) H = 0.3 m (b) H = 0.2 m

Figure 5 Airflow pattern in solar channels



(a) H = 0.3 m (b) H = 0.2 m

Figure 6. Temperature field in solar channels

CONCLUSION

A mathematical model for solar chimney is proposed and an experimental study of solar chimney is used to validate the proposed mathematical model. A good agreement is found between the predicted results of airflow rate and those of measurement in the different conditions. The predicted heat transfer rate increases with channel and massive wall surface heat flux. The performance of a glazed solar chimney is influenced by the channel width as well as solar heat gains. The present study has shown that the computer program developed can be used for the prediction of buoyant airflow in solar channel.

ACKNOWLEDGEMENTS

This paper is supported by Funding Project for Academic Human Resources Development in Institutions of Higher Learning Under the Jurisdiction of Beijing Municipality

REFERENCES

Akbarzaden A, Charters WW & Lesslie DA. 1982. “Thermocirculation characteristics of a Trombe Wall passive test cell,” Solar Energy. 28(6):461-468.

Bouchair, A, 1994. “Solar chimney for promoting cooling ventilation in south Algeria,”Building Service Engineering Research and Technology, 15(1):81-93.

Hirunlabh, J, Kongduang, W, Namprakai, J & Khedari J. 1999. Study of Natural Ventilation of House by a Metallic Solar Wall Under Tropic Climate. Renewable Energy. 18:109-119. .

Ong, KS. 2003. “A mathematical model of a solar chimney,” Renewable Energy, l23: 1047-1060.

Ormiston, S.J, Raithby, D.D & Hollands K.G. 1986. Numerical Prediction of Natural Convection in Trombe Wall System. International Journal of Heat and Mass Transfer. 29:869-877.

Pantankar SV. 1980, Numerical Heat Transfer and Fluid Flow, Hemisphere McGraw-Hill.

Tao WQ. 2001. Numerical Heat Transfer, *2nd Edition*, Xi’an Jiaotong University Press.

Zalewski L, Lassue S, Duthoit B & Butez M. 2002. “Study of Solar Walls-Validating a Simulation Model,” Building and Environment, 37(2):109-121.

Zrikem Z, Bilgen E. 1987. “Theoretical Study of a Composite Trombe-Michanel Wall Solar Collector System,” Solar Energy, 39(6):409-419.