

Numerical Study the Performance of Solar Wall System

Yanfeng Li, Xuemei Wang, Xiaoxue Han, Huairong Huang, Na Cai

College of Architecture and Civil Engineering, Beijing University of Technology, Beijing, 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. A good agreement is found between the predicted results of airflow rate and those of measurement in the different conditions. 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, Modelling

1. INTRODUCTION

High energy consumption is one aspect in the renovation of existing building especially in China. In addition to repair, increased comfort, and visual renovation, all projects were designed to save energy. Since the use of solar energy means that significantly less fossil fuel is consumed, solar system should be readily adaptable to existing buildings as well as new buildings. Between 30% and 70% energy savings were resulted from the renovation in many projects. Therefore, renovation presents both special challenges and the opportunity to apply solar technologies to building.

Two ways where solar energy is common used for buildings are solar wall and solar chimney. Solar wall heating with transparent insulation is the most prominent and energy-effective candidate for passive solar wall heating design. 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. 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^[1] were published. Zalewski et al.^[2] have done a comparatively study of four different configurations of solar wall, and reported their main advantages and disadvantages. 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^[3].

There are two major categories in analyzing the solar wall system^[4]. 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^[5].

In the work of Akbarzadeh et al^[6], a 2.66m high Trombe wall with air gaps varying from 0.10 to 0.35m were studied by flow visualization technology. They 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, 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^[7].

Yedder and Bilgen^[4] studied performance of in Trombe wall system including natural convection and conduction. The flow was assumed laminar and two-dimensional, the glazing and massive wall were assumed isothermal. They found that temperature difference between glazing and massive wall and the Rayleigh number were two important parameters affecting heat transfer.

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 carried out into the performance of the glazed

solar channel. Effects of geometric parameters such as the channel width, solar heat gain are also examined.

2. MATHEMATICAL MODEL

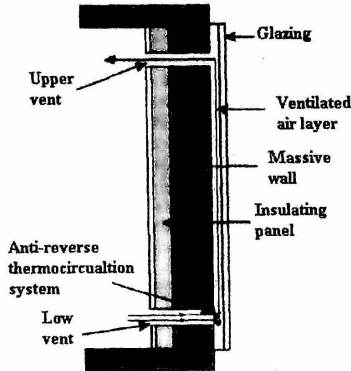


Figure1: Insulated Trombe wall

2.1 Physical model

The Schematic diagram of an insulated wall system solar wall system considered in the present paper is illustrated in Fig. 1^[2]. 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.

2.2 Governing equations

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^[9].

$$\text{div}(\rho \vec{V} \Phi) = \text{div}[\Gamma_{\Phi} \text{grad}(\Phi)] + S_{\Phi} \quad (1)$$

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

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	$\frac{\mu}{Pr} + \frac{\mu}{\sigma_t}$	0
Kinetic energy	k	$\frac{\mu_{eff}}{\sigma_k}$	$G - \rho \epsilon$
Dissipation rate	ϵ	$\frac{\mu_{eff}}{\sigma_{\epsilon}}$	$\frac{\epsilon}{k} (C_1 G - C_2 \rho \epsilon)$

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

$$C_{\mu} = 0.09, \quad C_1 = 1.44, \quad C_2 = 1.92, \quad \sigma_k = 1.0,$$

$$\sigma_{\epsilon} = 1.3, \quad \sigma_p = 0.7$$

where Φ denotes the dependent variable, ρ is 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^[9].

The generation term G is defined as follows:

$$G = G_i + G_s = \mu_i \left[2 \left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial x} \right)^2 + \left(\frac{\partial w}{\partial y} \right)^2 \right] + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)^2 + g \beta \frac{\mu_i}{\sigma_i} \frac{\partial T}{\partial y} \quad (2)$$

The first term on the right-hand side, G_i , is the 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 term G_s .

The above model contains six constants, which are assigned the values recommended by Spalding for turbulent flows^[9]. These values are listed in Table 1. However, these parameters are usually adjusted for particular flows. In Ref. ^[10] it was reported this standard set did not always produce accurate results especially for flows with adverse pressure gradients, curative wall, recirculation and swirl. A test case with $C_\mu = 0.109$, $\sigma_k = 0.614$ was reported in Ref ^[11]. It shows the results for a pure jet were somewhat improved. The spread rates of buoyant plumes were still significant underpredicted. In Ref. ^[12], C_μ and σ_k were taken to the values 0.18 and 0.85 respectively. However, there is no generally accepted standard for choosing the values of these two parameters. In this paper, the values of C_μ and σ_k in the standard k- ϵ model are chosen.

In this study, the Boussinesq model, that assumes that density is only a function of temperature through the computational domain, is used for the buoyant force term in the momentum equation. the term $-g(\rho - \rho_0)$ in momentum equation for buoyancy due to temperature difference becomes $-g\beta(T - T_0)$. β is the coefficient of thermal expansion defined as:

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p \quad (3)$$

since $\beta \Delta T \ll 1$ in this study, the Boussinesq approximation is appropriate for this simulation. The reference temperature is taken to be at the supply air temperature. T_0 and ρ_0 are temperature and density of air at inlet respectively.

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. At any step, the solution is considered converged if

$$\frac{|\phi^{n+1}(i,j,k) - \phi^n(i,j,k)|}{\max |\phi^{n+1}(i,j,k)|} < 10^{-3} \quad (4)$$

where n is iteration loop counter

2.3 Boundary conditions

Standard wall functions^[9] are used for the enclosure walls. 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.

2.4 Grid system

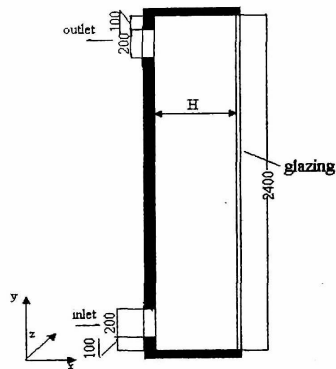


Figure 2: The configuration of solar channel

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 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.

3. RESULT AND DISCUSSION

3.1 Validation of model

Validation of program is performed by comparing the predicted results with experimental

data for natural convection in a solar chimney by Bouchair. The chimney was 2 m high and of variable width. All the wall surface were isothermally heated by electrical heater to temperatures from 30°C to 60°C. The inlet air temperature was controlled at 20 °C. Patterns of air movement were measured with the use of smoke. Detail of the experimental measurement and results were given by Bouchair^[13].

In the experimental study of solar chimney, the air flow rate through a solar chimney is given by

$$Q = C_d A \sqrt{\frac{2\Delta P}{\rho}} \quad (5)$$

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 air flow rate in the conditions where wind effect is not considered are used to validate the proposed model.

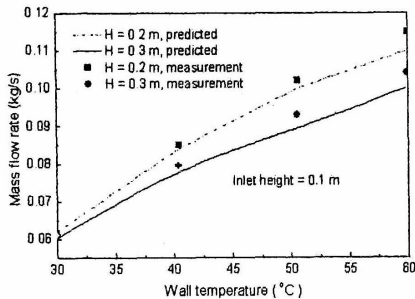


Figure 3: Comparison between the predicted results of mass flow rate and measured data

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 Fig. 3. The inlet height of solar chimney is 0.1m. It can be found that there is a good agreement between the prediction in the simulation by using the numerical model and measurement results. This gives confidence in using the computer code to study the airflow and heat transfer in solar cavities in this study.

3.2 Effects of solar channel width and glazing on airflow pattern

In this study, the inlet air temperature is 20 °C. The glazing inner surface temperature is 25 °C. The inlet air velocity, u , is 0.2 m/s. The wall solar heat gain is calculated using mean solar gain factor. It is assumed the heat flux on the massive wall, $\dot{q}_{\text{wall}}$, is constant and determined by solar radiation and glazing absorptivity. The glazing had an absorptivity for direct solar radiation of 0.2.

Two channels with width of 0.2 and 0.3 m are studied. For the same channel, two conditions where $\dot{q}_{\text{wall}}$ are 280 and 140 W/m^2 are calculated respectively. Figures 4 to 5 show the flow pattern and

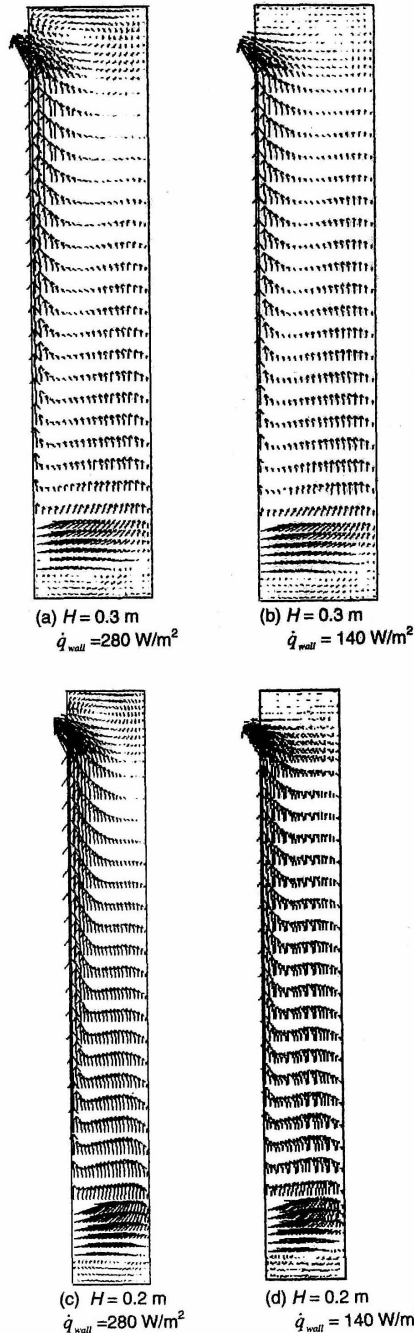


Figure 4: Airflow pattern in different solar channels air temperature field in two solar channels. For the 0.3 m wide solar channel it can be seen clearly in Figs. (4a) and (4b) that there is circulate flow at the top of solar channel when $\dot{q}_{wall}$ is 280 W/m². This means there is a larger air 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. This is a larger airflow velocity is needed for passing the same mass flow rate with a narrower wall passage because of mass conservation in the solar channel. It can be also found when channel width is 0.2 m that the airflow velocity increases when the massive heat flux is larger.

As mentioned earlier, heat flux on the massive wall, $\dot{q}_{wall}$, is determined by solar radiation and glazing absorptivity. Therefore, the effect of glazing on the predicted airflow condition depended on the total solar irradiance. For a vertical wall with single glazing, the mean solar gain factor is 0.64. The mean solar irradiance on a vertical south surface between 8:00 and 16:00 h sun time on December 21 at 45° north latitude is 438 W/m². The corresponding wall solar heat gain is thus 280 W/m². As for double glazing the mean solar gain factor is about 0.76, $\dot{q}_{wall}$ is about 333 W/m² correspondingly. The larger wall heat flux would be benefit for heating the air in the channel. That means double glazing is better than the single glazing for meeting the requirement of warming room air in winter.

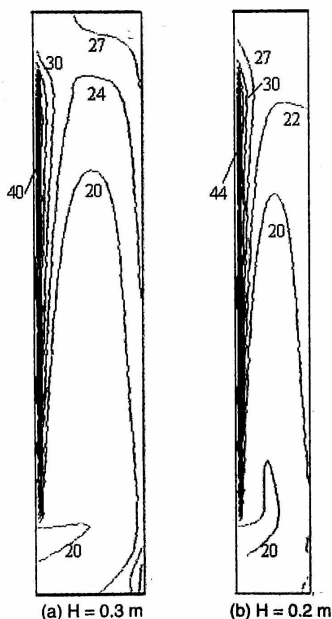


Figure 5: Temperature field in the solar channel (°C)

The temperature field in two channels with 280 W/m² wall heat flux are illustrated in Fig. 5. 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 °C while the value is 44 °C in the 0.2 wide channel. This is because the higher airflow velocity leads to a larger quantity of heat transfer and thereby a higher temperature near the massive wall surface. In the channel air becomes warmer and warmer when moving from the bottom to the exit. Detailed heat transfer in the channel will be discussed in the following.

3.3 The quantity of heat transferred to air in channel

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^[7]:

$$\dot{q} = \frac{\dot{m} c_f (T_f - T_{f,i})}{\gamma H L} \quad (6)$$

where $\dot{q}$ is heat flux transferred to moving air stream. γ is a constant in mean temperature approximation, Hirunlabh et al.^[14] took a value of $\gamma = 0.75$ from 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.

$$T_f = \gamma T_{f,i} + (1 - \gamma) T_{f,o} \quad (7)$$

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

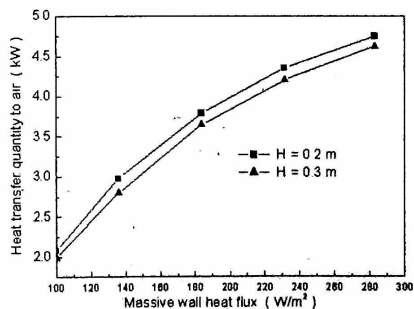


Figure 6: Quantity of heat transfer to air in channel

The quantities of heat transfer to moving air in the channel under different conditions are illustrated in Fig.6. It can be seen that heat transfer quantity is 4.75 kW for $\dot{q}_{wall} = 280$ W/m² while it is 3.02 kW for $\dot{q}_{wall} = 140$ W/m² 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 Fig.6. 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 and it would influence the heat transfer. This research will be done in the future. Further experimental work is needed to guide the simulation.

4. CONCLUSION

The thermal performance of solar channel in the insulated Trombe wall collector system is studied. A mathematical model for solar chimney is proposed. 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 double glazing is better than the single glazing for meet the requirement of warming room air. The present study has shown that the computer program developed can be used for the prediction of buoyant airflow in solar channel. Further experimental work is needed to refine the model.

ACKNOWLEDGEMENTS

This paper is supported by the Scientific Research Foundation for the Excellent Youth and Middle-aged person in Beijing.

REFERENCES

- [1] Trome, F. 1974. *Maisons Solaires Techniques de l'Ingénieur* 3 C777. German.
- [2] Zalewski, L, Lassue, S, Duthoit, B & Butez, M. 2002. Study of Solar Walls-Validating a Simulation Model. *Building and Environment*, Vol.37, pp 109-121. France
- [3] Zrikem, Z, Bilgen, E. 1987. Theoretical Study of a Composite Trombe-Michanel Wall Solar Collector System. *Solar Energy*, Vol.39, pp409-419. U.K.
- [4] Yedder, R.B, Bilgen, E. 1991. Natural Convection and Conduction in Trombe Wall Systems. *International Journal of Heat and Mass Transfer*. Vol.34, No.5, pp 1237-1247. Canada.
- [5] 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*. Vol.29, pp 869-877. U.K.
- [6] Akbarzaden, A, Charters, W.W & Lesslie, D.A. 1982. Thermocirculation Characteristics of a Trombe Wall Passive Test Cell. *Solar Energy*. Vol.28, No.6, pp 461-468. U.S.A.
- [7] Ong, K.S. 2003. A Mathematical model of a Solar Chimney. *Renewable Energy*. Vol.23. 1047-1060. Malaysia.
- [8] Pantankar, S.V. 1980. *Numerical Heat Transfer and Fluid Flow*, Hemisphere McGraw-Hill, New York, U.S.A.
- [9] Tao, W.Q. 2001. *Numerical Heat Transfer*, 2nd Edition, Xi'an Jiaotong University Press Xi'an. China.
- [10] Yang, K.T, Lloyd, J.R & Kanury, A.M. 1984. Modeling of Turbulent Buoyant Flows in Aircraft Cabins. *Combustion Science and Technology*. Vol.39. pp.107-118. China
- [11] Markatos, N.C, Malin, M.R. 1982. Mathematical Modeling of Buoyancy-induced Smoke Flow in Enclosures. *International Journal of Heat and Mass Transfer*. Vol.25, No.1. pp 63-75. U.K.
- [12] Nam, S, Robert, G.B. 1993. Numerical Simulations of Thermal Plumes. *Fire Safety Journal*. Vol.21. pp 231-256. U.S.A.
- [13] Bouchair, A, 1994. Solar Chimney for Promoting Cooling Ventilation in South Algeria. *Building Service Engineering Research and Technology*. Vol.15. pp 81-93. Algeria.
- [14] 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*. 1999. Vol.18. pp109-119. U.S.A