Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879

Indoor Air of a Double-Storey Residential House in Malaysia

Open Access

Haslinda Mohamed Kamar[1,*], Nazri Kamsah[1], Kam Jia Liq[1]

[1] Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia.

ARTICLE INFO

Article history:
Received 7 March 2017
Received in revised form 13 April 2017
Accepted 17 April 2017
Available online 27 April 2017

ABSTRACT

This paper presents an investigation on the effects of using solar chimney, gable vent, and the combination of the two natural ventilations on the average air temperature and air-flow condition inside a double-storey house in Malaysia, using computational fluid dynamics (CFD) method. The representative model of the house comprises of a main hall, a kitchen and an upper hall. Both temperature and air velocity boundary conditions were prescribed on the model. Results of the simulation indicates that the average temperature of the air in the house at 1 pm closely matched the measured values. It was found that the average temperature of the air in the house is not so significantly affected by the types of natural ventilation used. Opening the kitchen door causes the air to flow from the main and upper halls towards the kitchen and causing a bottle neck at the pathways. A more uniform air flow is obtained when solar chimneys are used. When gable vents are used, high intensity air flow occurs in the main hall and it spreads uniformly towards the kitchen and upper hall. The air-flow intensity becomes even higher in the main and upper halls when a combination of solar chimney and gable vents are incorporated into the CFD model.

Keywords:
Natural ventilations, Average air temperature, Air-flow conditions, CFD simulation, Solar chimney, Gable vents

1. Introduction

Natural ventilation has attracted a strong growing interest in building sectors because of its potential advantages over mechanical ventilation systems, in terms of energy requirement, economic and environmental benefits [1,2]. Mechanical ventilation systems have undesirable energy implication since they require more electricity to run [3]. Earlier work on natural ventilation mainly concerned with aerodynamic loading [4] and they were carried out in wind tunnel. But with the advancement of computing technology, more complicated studies have been conducted using computational fluid dynamics (CFD) techniques. Nikas *et al*. [5] showed that it is possible to get information about induced velocity and pressure fields for natural cross ventilation using CFD modelling which otherwise are quite difficult to extrapolate from experimental methods.

[*] *Corresponding author.*
E-mail address: haslinda@mail.fkm.utm.my (Haslinda Mohamed Kamar)

Various strategies have been proposed in the literature to enhance buoyancy effect so that adequate air flow rate and a desired level of thermal comfort can be achieved inside a building. One good example is a solar chimney, which is designed to maximize ventilation effect by maximizing solar gain [6]. This creates a sufficient temperature difference between the inside and outside of the building to drives an adequate air flow rate. Solar chimney is a thermo-syphoning air channel in which the principal driving mechanism of air flow is through thermal buoyancy [7]. One can find different variations in solar chimney design, which is affected by a number of factors such as the location, climate, orientation, size of the space to be ventilated and the internal heat gains [8].

Computational methods based on CFD technique have been used by many to predict flow pattern inside the chimney as well as in the space (room) adjoining the solar chimney. The existing CFD models are able to predict velocity and temperature profiles along with other flow characteristics accurately [9-11]. However, they usually do not consider the thermal energy storage in the walls of the building [12]. Nevertheless, the use of CFD modelling in solar chimney study has been increasing. These studies have greatly contributed to the present understanding of the solar chimney.

In this study, we used the CFD method to investigate the effect of using natural ventilations in a double-storey residential terrace house in Malaysia. The natural ventilations considered are solar chimney, gable vents and the combination of the two. The focus of this study is not on the types of ventilation. The main goal of this study is to find out the effect of using these ventilations on the thermal and flow conditions of the air inside the house. For that purpose, the solar chimneys are represented only as simple square openings located on the roof of several sections of the house. The gable vents on the other hand are represented by long rectangular openings on the upper part of several walls of the house. During the simulation, both air velocity and temperature are prescribed on these openings to model the outward air flow from the house.

2. Experimental Setup and Procedure

2.1 Computational Domain

Figure 1 shows a representative model of the house and a computational domain for the CFD simulations. It consists of three sections namely the main hall, the upper hall and the kitchen. There are only four walls that are considered to be exposed to solar radiation. These are the eastern and southern walls of the main hall, northern wall of the kitchen wall, and the eastern wall of the upper halls. Other walls are considered to be insulated and are at the same temperature as the air in the house, which is at 29°C (302K).

2.2 Actual average air temperature & humidity

The actual average dry-bulb temperature, wet-bulb temperature and relative humidity of the interior air were determined in the three sections of the house: the main hall, the upper hall and the kitchen, using a sling psychrometer, for every hour beginning from 9 am until 4 pm. In each section, all the data were measured at several locations and then the average values were computed, for every hour. The complete hourly data are shown in Table 1. It is observed that the average dry-bulb temperature of the air is about 30°C and relative humidity is around 73%.

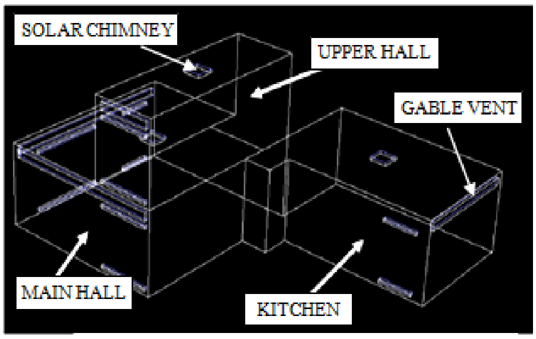

Fig. 1. A representative model of the house considered for the CFD computational domain (rear view)

The air dehumidifier unit is a Bry-Air Fluted Flat Bed (FFB) Series Compact Desiccant Dehumidifier type. The schematic diagram of the air dehumidifier unit is shown in Figure 2. It consists of a solid desiccant rotor, process air inlet and outlet, regeneration air inlet and outlet, an electrical heater and a control unit.

Table 1:
Hourly data for the air inside the house

Location/Time (hour)		9:00	10:00	11:00	12:00	13:00	14:00	15:00	16:00
Hall	Dry bulb (°C)	29.10	29.50	30.02	29.96	30.96	30.70	30.72	30.82
	Wet bulb (°C)	26.00	26.00	26.00	26.14	26.00	25.90	25.72	26.00
	RH (%)	78.4	76.0	73.0	74.2	67.8	68.6	67.4	68.5
Kitchen	Dry bulb (°C)	28.66	29.00	29.1	29.74	30.04	30.00	30.04	30.22
	Wet bulb (°C)	25.66	25.92	26.16	25.86	25.96	25.56	25.52	26.00
	RH (%)	78.9	78.5	79.5	73.7	72.6	70.4	69.9	71.8
Top floor	Dry bulb (°C)	29.00	29.50	30.00	30.17	31.27	31.00	30.93	31.23
	Wet bulb (°C)	25.93	26.00	26.00	26.07	26.43	26.33	26.00	26.60
	RH (%)	78.6	76.0	73.1	72.5	68.7	69.5	67.9	68.9

2.3 Validation of CFD Simulation Procedure

To validate the numerical simulation procedure, a CFD simulation was performed on the model of the house to represent a condition when there are no ventilations. However, a door on the rear wall of the kitchen was left fully opened to generate some air flow within the house. We call this as a "base case" condition. The goal of this simulation is to estimate the average temperature of the air in the various sections of the house and compare them with the actual temperatures measured at 1 pm, when the kitchen door was opened.

Both temperature and air velocity boundary conditions were used. A uniform temperature of 47°C (320 K) was prescribed on the wall of the main hall facing south and the wall of the kitchen facing north. A uniform temperature of 29°C (302 K) was prescribed on the wall of the main hall facing east and the wall of the upper hall facing east. A constant inlet air velocity of 0.1 m/s, at a temperature of 29°C (302 K), was prescribed on all the door seams. These door seams represent the gaps between the doors and the slabs and the clearance between the doors and the walls. The air velocity boundary conditions allow a turbulent analysis to be performed on the computational domain. Turbulent flow analysis using a k-ε model with a 10% turbulent intensity was performed on the CFD model until an acceptable convergence was attained

2.4 Modeling of the Natural Ventilations

Three natural ventilation systems were considered in this study, namely a solar chimney, gable vents and the combination of the two. The solar chimney is a natural-draft device that is used in many passive cooling applications for residential houses. Density of air decreases with increasing temperature. It means that air with higher temperature than ambient air is driven upwards by the buoyancy force. A solar chimney exploits this physical phenomenon and uses solar energy to heat air up. Gable vents are usually placed at the top of the gable on the end of the house. This is to create a draft through the space by having both intake and exhaust vents. While gable vents do increase ventilation they do not offer uniform air flow within the space and their ability to move large amounts of air are limited.

The solar chimneys were incorporated into the CFD model by adding square-shaped openings on the roof (at the middle) of the main hall, kitchen and the upper hall. A constant air outlet velocity of 0.3 m/s, at 29°C (302 K) was prescribed on all the solar chimneys while a constant inlet air velocity of similar magnitude was prescribed on all the door seams as the boundary conditions. The same temperature boundary conditions as in the base case were employed in this CFD simulation.

The gable vents were incorporated into the CFD model by introducing thin rectangular-shaped openings on the walls of the house. The width of these openings was made nearly the same as the width of the walls. An inlet gable vent was placed on the eastern wall of the main hall, while outlet gable vents were placed on the southern wall of the main hall, the northern wall of the kitchen and the eastern wall of the upper hall. A constant air outlet velocity of 0.3 m/s, at 29°C (302 K) was prescribed on all the solar chimneys while a constant inlet air velocity of similar magnitude was prescribed on all the door seams as the boundary conditions. The same temperature boundary conditions as in the base case were used in this CFD simulation.

3. Results and Discussion

3.1 Base case conditions

Results of the CFD simulation for the "base case" condition give an average air temperature of about 30.4°C in both the main hall and the kitchen, and about 29.7°C in the upper hall. These values are superimposed on the plots of measured temperature vs. time (within a circle) for the three sections on the house, shown in Figure 2. It can be seen that the average temperatures obtained from the CFD simulation fall within the acceptable range of the measured temperature range. Thus it is safe to say that the CFD model, the boundary conditions used and the turbulent analysis model employed in the simulation are valid and can be further used in the proceeding simulations. The air flow distribution in the house obtained from the CFD simulation for the "base case" conditions is shown in Figure 3. It can be seen that, with the door on the eastern door of the kitchen left opened, the air tends to flow from the main and upper halls towards the kitchen, producing a bottle neck at the pathway connecting the main hall and the kitchen. The air flow is seen fairly uniform in both halls.

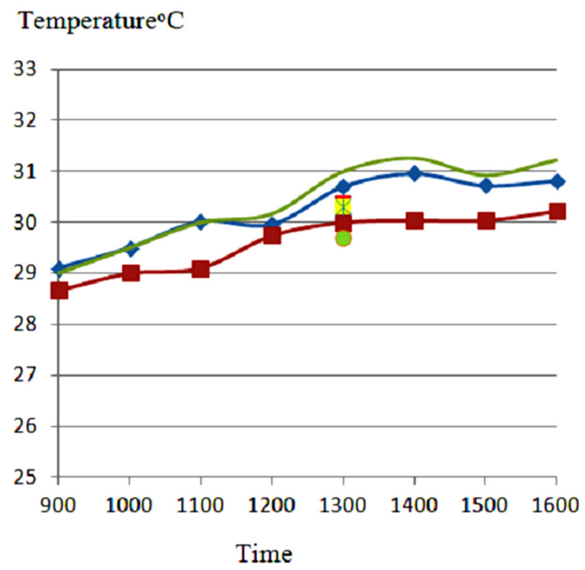

Fig. 2. Comparison between the average air temperatures obtained from the CFD simulation and the measured values

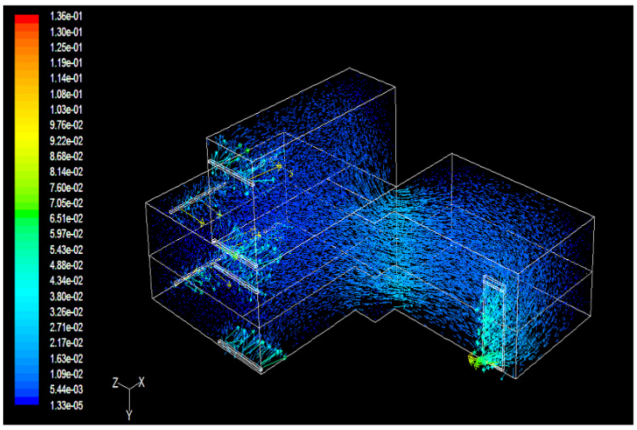

Fig. 3. Air-flow distribution (m/s) inside the house when the door on eastern wall of the kitchen is left opened

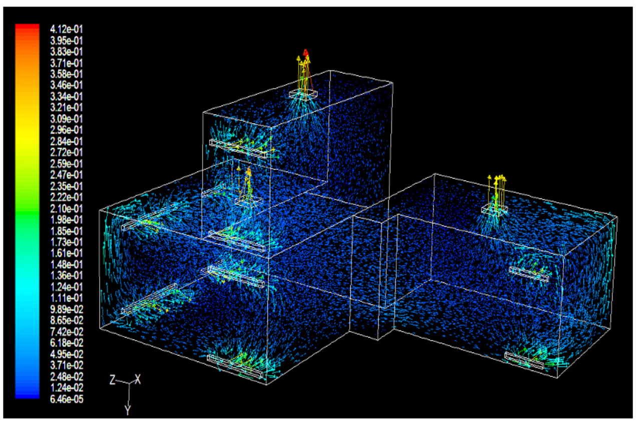

Fig. 4. Air-flow distribution (m/s) inside the house when solar chimney ventilation is used

3.2 The effect of solar chimney ventilation

Results of the CFD simulation when solar chimneys are incorporated into the model give an average air temperature of 302.8 K in the main hall, 302.9 K in the kitchen, and 302.3 K in the upper hall. These are slightly lower than the average air temperature for the base case condition [303.4 K (hall); 303.3 K (kitchen); 302.7 K (upper hall)]. On average, the CFD simulation results indicate that the average air temperature in the house is reduced by about 0.6°C when three solar chimneys were incorporated into the model. This is considered as an insignificant improvement on the average temperature of the air inside the house. The air flow condition in the house when solar chimneys are used is shown in Figure 4. It can be seen that the air flow is fairly uniform in all three sections of the house. Slightly higher air velocity occurs at all the door seams (inward flow) and the solar chimneys (outward flow). A swirling air flow condition can be seen near the northern wall of the kitchen and the southern wall of the main hall.

3.3 The effect of gable vents

Results of the CFD simulation when gable vents are incorporated into the model give the average air temperature of 302.4 K in the main hall and in the kitchen, and 302.2 K in the upper hall. These are slightly lower than the average air temperature for the base case condition [303.4 K (hall); 303.3 K (kitchen); 302.7 K (upper hall)]. On average, the CFD simulation results indicates that the average air temperature in the house is reduced by about 0.8°C when the gable vents were incorporated into the CFD model. This can also be considered as an insignificant improvement on the average temperature of the air inside the house.

The air flow condition in the house when gable vents are used is shown in Figure 5. It can be seen that the air flow is fairly uniform in the main hall and the kitchen but it is less intense in the upper hall section. Higher air velocity condition can be seen at the vicinity of all the gable vents, especially at the inward flow gable vent on the eastern wall of the main hall. No swirling air flow condition can be seen in the figure.

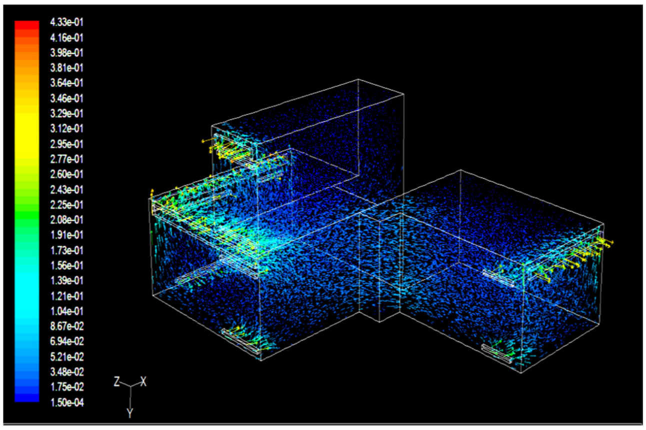

Fig. 5. Air-flow distribution (m/s) inside the house when gable vents are used

3.4 The effect of combined solar chimney ventilation & gable vents

Results of the CFD simulation when a combination of solar chimneys and gable vents are incorporated into the model give an average air temperature of 302.2 K in the main hall, 302.3 K in the kitchen and

302.1 K in the upper hall. When compared to the base case conditions, it is found that the average air temperature is dropped by about 1.1°C, 1.0°C and 0.6°C in the main hall, kitchen and the upper hall, respectively. These are considered a mild reduction in the average air temperature inside the house.

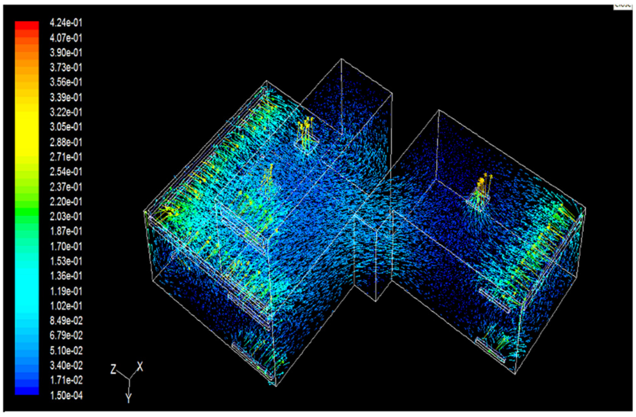

Fig. 6. Air-flow distribution (m/s) inside the house when a combination of solar chimneys and gable vents are used

Figure 6 shows the air flow distribution inside the house when a combination of solar chimneys and gable vents are used. It is seen that high intensity air flow occurs in the main hall and the air appears to move towards the kitchen. The air flow in the upper hall and the kitchen appears to be less intense. Higher air velocity is seen at both the inlet and outlet gable vents. No swirling air flow can be seen from the figure. Also, the air tends to flow toward the solar chimneys located on the ceiling of each section of the house.

4. Conclusion

A CFD simulation method has been used to investigate the effects of several natural ventilations, namely solar chimney, gable vent and the combination of both, on the conditions of the air inside a double-storey residential house in Malaysia. It was found that the average temperatures of the air at various sections of the house, obtained from the CFD simulation for the base case condition, agree quite well with the measured values at 1 pm. The average air temperature drops by about 0.6°C when solar chimneys are used and about 0.8°C when gable vents are incorporated into the CFD analysis. The temperature drops by about 1C when the combination of both ventilations are included in the analysis. When the kitchen door is left opened, the air tend to flow from the main hall and upper hall towards the kitchen. Using solar chimney ventilation results in a more uniform air-flow inside the house. High intensity air flow occurs in the main hall and it spreads uniformly towards the kitchen and upper hall when inlet and outlet gable vents are used. The air-flow intensity becomes even higher in the main and upper halls when a combination of solar chimney and gable vents are incorporated into the computational model.

Acknowledgement

The authors would like to acknowledge the supports from Universiti Teknologi Malaysia and fund provided by Ministry of Higher Education, Malaysia throughout this study under FRGS Vot 4F645.

References

[1] Rahman, Abdul Malek Abdul, Hirda Lailani Khalid, and Yusri Yusup. "Optimizing Wind Power for Energy Efficient Building Design in Tropical Hot-humid Climate of Malaysia." *Journal of Sustainable Development* 4, no. 2 (2011): 217.

[2] Kubota, Tetsu, Doris Toe Hooi Chyee, and Supian Ahmad. "The effects of night ventilation technique on indoor thermal environment for residential buildings in hot-humid climate of Malaysia." *Energy and Buildings* 41, no. 8 (2009): 829-839.

[3] Khanal, Rakesh, and Chengwang Lei. "Solar chimney—A passive strategy for natural ventilation." *Energy and Buildings* 43, no. 8 (2011): 1811-1819.

[4] Linden, Paul F. "The fluid mechanics of natural ventilation." *Annual review of fluid mechanics* 31, no. 1 (1999): 201-238.

[5] Nikas, K-S., N. Nikolopoulos, and A. Nikolopoulos. "Numerical study of a naturally cross-ventilated building." *Energy and Buildings* 42, no. 4 (2010): 422-434.

[6] Bansal, N. K., Rajesh Mathur, and M. S. Bhandari. "Solar chimney for enhanced stack ventilation." (1993).

[7] Gan, Guohui. "Simulation of buoyancy-induced flow in open cavities for natural ventilation." *Energy and buildings* 38, no. 5 (2006): 410-420.

[8] Harris, D. J., and N. Helwig. "Solar chimney and building ventilation." *Applied Energy* 84, no. 2 (2007): 135-146.

[9] Hurnik, M., M. Blaszczok, and Z. Popiolek. "Air distribution measurement in a room with a sidewall jet: a 3D benchmark test for CFD validation." *Building and Environment* 93 (2015): 319-330.

[10] Tong, Zheming, Yujiao Chen, and Ali Malkawi. "Defining the Influence Region in neighborhood-scale CFD simulations for natural ventilation design." *Applied Energy* 182 (2016): 625-633.

[11] Muhsin, Fakhriah, Wardah Fatimah Mohammad Yusoff, Mohd Farid Mohamed, and Abdul Razak Sapian. "CFD modeling of natural ventilation in a void connected to the living units of multi-storey housing for thermal comfort." *Energy and Buildings* 144 (2017): 1-16.

[12] Dimoudi, A. "Solar chimneys in buildings—The state of the art." *Advances in Building Energy Research* 3, no. 1 (2009): 21-44.