Simulation of Thermal Comfort of a Residential House

Masine Md. Tap[1], Haslinda Mohamed Kamar[2], Abdul Kadir Marsono[3] Nazri Kamsah[4] and Khairul Amry Mohd Salimin[5]

[1] **Department of Manufacturing and Industrial Engineering, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Skudai, Johor 81310, Malaysia**

[2] **Department of Thermo-Fluid, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia Skudai, Johor 81310, Malaysia**

[3] **Department of Structural Engineering, Faculty of Civil Engineering, Universiti Teknologi Malaysia Skudai, Johor 81310, Malaysia**

[4] **Department of Thermo-Fluid, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia Skudai, Johor 81310, Malaysia**

[5] **Faculty of Mechanical Engineering, Universiti Teknologi Malaysia Skudai, Johor 81310, Malaysia**

Abstract

In hot and humid climates thermal comfort can become a problem to the occupants of many residential buildings especially when they are not equipped with air-conditioning system. This paper presents outcomes of an ongoing research work to investigate thermal comfort level in a naturally ventilated residential house in Malaysia using computational fluid dynamics (CFD) method. Actual measurements of the temperature distribution, relative humidity and air flow pattern were conducted. CFD simulations on the model of the house allow us to visualize the temperature distribution and air flow pattern and velocity in the house. The thermal comfort in the house was found to be well outside the limits specified by ASHRAE standards. CFD simulation was used to investigate the effects of using a ceiling fan installed in the middle of the hall section and rotating at 150 RPM. It was found that the fan produced swirling flow pattern in the hall section resulting in a more uniform temperature distribution inside the house. However, there is no significant improvement in the thermal comfort level in the house. Results of CFD simulations also show that the use of small extractor fans installed on the front and back walls has no significant effects on the thermal comfort level in the house. Although the mechanical ventilation devices did not help improve the thermal comfort in the house being studied, the CFD simulation results can be used by building designers and engineers to further improved the level of thermal comfort in residential houses in hot and humid climates that are naturally ventilated.

Keywords: *Thermal Comfort in a Residential Building, Computational Fluid Dynamics, Naturally Ventilated Residential House.*

1. Introduction

In hot and humid climates, thermal comfort in both residential and commercial buildings is essential. Most residential buildings are not equipped with air-conditioning systems. Instead they rely mostly on natural ventilation, passive cooling system and mechanical ventilation devices such as ceiling and extractor fans to achieve certain level of thermal comfort. Special attention must be given to the design and installation of these devices so as to optimise their effects to thermal comfort. Moreover, the quality and energy efficiency of these devices varies widely.

It is therefore critical to assess several important building characteristics at the design stage. These include the ability to improve energy efficiency, effects of solar radiation, effects of air flow due to wind and the most importantly the occupant's comfort. To assess the thermal comfort, it is necessary to analyse the air velocity, temperature distribution, and relative humidity of the air in the interior space of the building. Standards that may be used to evaluate thermal comfort are widely available and numerical methods such as computational fluid dynamics (CFD) may be utilized to assist in the analysis.

A CFD software such as Fluent is a useful tool that can be used to create a virtual model of the building interior and simulate air flow, temperature profile and humidity which are directly related to thermal comfort, before the actual construction can be done. Modifications to an existing building can also be simulated using the CFD method prior to any physical renovations.

2. Human Thermal Comfort

Human thermal comfort is defined as the state of mind that expresses satisfaction with the surrounding environment [1]. Maintaining the thermal comfort for occupants of buildings or other enclosures is one of the important goals of the Heat, Ventilation and Air-conditioning (HVAC) design engineers. Thermal comfort in buildings is affected by the transfer of heat energy by conduction, convection, radiation, and evaporative heat loss. Thermal comfort will be maintained when the heat generated by human metabolism is allowed to dissipate, thus a person will maintain thermal equilibrium with his surroundings. Any heat gain or loss beyond this level will generate a sensation of discomfort. The sensation of feeling hot or cold is not just dependent on air temperature alone. Other factors that can affect human thermal comfort are the relative humidity of the ambient air, the air movement pattern and its velocity, radiant heat exchange, metabolic rate of a person and the person's clothing [2].

The thermal comfort condition in a ventilated building may be assessed through the air flow pattern and its velocity, the temperature distribution and the indoor air quality [3]. A good indoor climate will not only make its occupants comfortable but also promote the energy saving and its sustainability [4]. Natural ventilation implies that air is supplied and removed from the indoor space of a building by natural means. The effectiveness of natural ventilation therefore depends very much on the design features of the house [5]. Natural ventilation is usually coupled with the use of mechanical ventilation system such as extractor and ceiling fans to provide better ventilation and thus thermal comfort condition.

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Many researchers have used CFD to analyze thermal comfort in building spaces and investigating the effects of natural and stratified ventilations on the thermal comfort. Some used this technique to improve the efficiency of energy usage for the building. These are very difficult to be carried out by using other methods [6]. Bastide *et.al.* [7] used CFD to predict building energy efficiency. Serra & Semiao [8] used CFD to evaluate and compare between two different ventilation strategies and Stravrakakis *et.al.* [9] used CFD to evaluate the effect of window sizes to thermal comfort and hygiene in buildings.

This paper presents a study on thermal comfort in a single-storey terrace residential house in Malaysia, which is not furnished with an air-conditioning system. The goal of this study is to assess the level of thermal comfort in the house when it is naturally ventilated and when a ceiling fan is used together with the natural ventilation. Fluent CFD software was used to construct a virtual model of the house. CFD simulations were performed to predict the air flow and temperature distributions in the house. The simulation procedure was validated by comparing results of the CFD analysis with data from actual measurement. The effects of a ceiling fan, installed in the hall section of the house and extractor fans installed at the front and rear walls of the house, on the air flow and temperature distributions were also investigated.

3. Thermal Comfort Study of a Residential House

Figure 1 shows a virtual model of the interior regions of a residential single-storey terrace house we used as a case study developed using Fluent CFD software. The house comprises of several regions namely the hall, stack and kitchen. The house is not furnished with air-conditioning system and is naturally ventilated, i.e. both the front and rear doors opened.

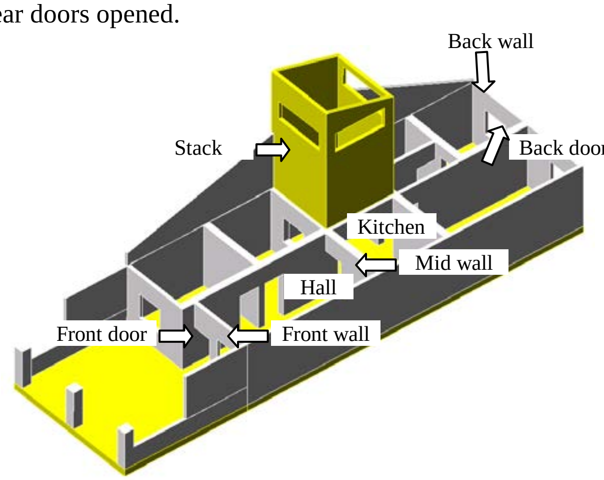

Fig. 1 CFD model of a residential house considered in this study.

To assess the level of thermal comfort in the house, we carried out measurements to acquire the average temperature of the ambient air, the relative humidity of the air and the average velocity of the air inside the house. The data were recorded during a time span from 12AM to 9PM. The data was compared with the upper and lower limits of thermal comfort specified by the ASHRAE standard [1] in order to assess the level of thermal comfort in the house.

3.1 Comfort Conditions with Natural Ventilation

Figure 2 shows variation of the average temperature of the ambient air inside the various sections of the house when the house is naturally ventilated. The air temperature in all sections decreases from about 29°C at 12AM to about 27°C at 6AM. Thereafter, the air temperature increases and reaching the highest value of about 31°C at 6PM. Also shown in Figure 2 are limits of air temperature for acceptable level of thermal comfort, as specified by the ASHRAE standard [1]. It can be seen that at any given time, the average temperature of the ambient air inside the house is well outside the limits of acceptable thermal comfort.

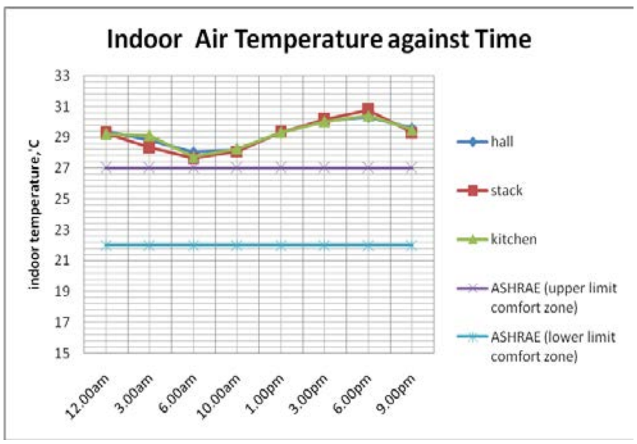

Fig. 2 Variation of average temperature of the air inside the house.

Figure 3 shows variation of the relative humidity of the ambient air inside the house. Also shown in this figure are the limits of relative humidity for acceptable level of thermal comfort, as specified by the ASHRAE standard [1]. The figure shows that the relative humidity of the air fluctuates within the range from about 71% at 10AM to 81% at 3AM. At any given time, the ambient air inside the house is too humid and this condition would cause thermal discomfort to the occupants.

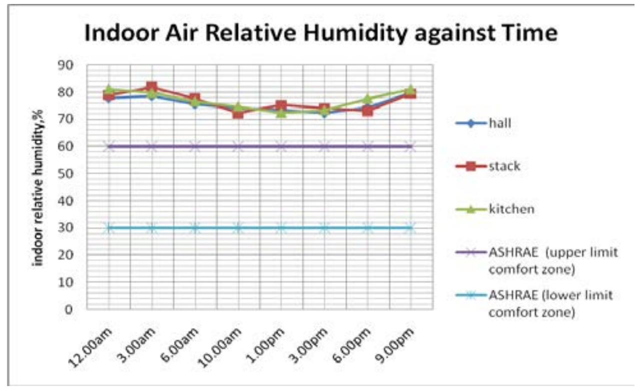

Fig. 3 Variation of relative humidity of the air inside the house.

The variation of average air velocity in various sections of the house is shown in Figure 4. Also shown in this figure are the limits of air velocity for acceptable level of thermal comfort, as specified by ASHRAE standard [1]. It can be seen that the air velocity in the hall and stack sections is fairly constant at about 0.1 m/s from 1PM to 6PM. The air velocity in the kitchen area appears to fluctuate from 0.1 m/s to 0.2 m/s. At any given time, the average velocity of the air inside the house is below the acceptable level of thermal comfort specified by ASHRAE standard [1].

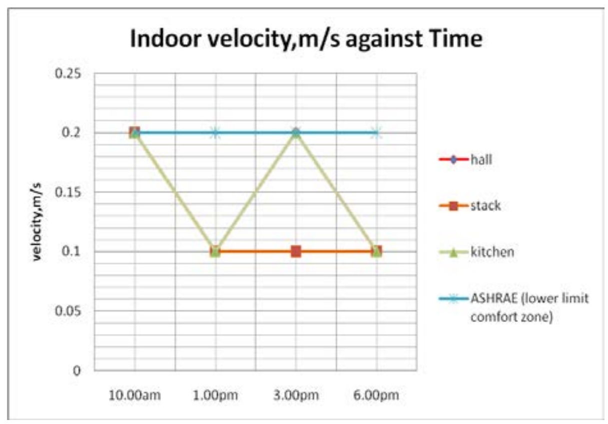

Fig. 4 Variation of average velocity of the air inside the house.

3.2 Comfort Conditions with Combined Natural Ventilation and a Ceiling Fan

The above findings clearly show the thermal comfort in the house is well below a satisfactory level when the house is ventilated naturally. Clearly, some means of increasing the comfort level is required. The simplest way is by the use of a ceiling fan. We then turned on the ceiling fan which was installed in the hall section of the house, during the entire data collection period. After a steady-state condition was established, we repeated the measurements for the average temperature of the ambient air, relative humidity of the air and the average velocity of the air in the various sections of the house. The goal is to investigate the effects of the use of the ceiling fan on these thermal comfort parameters.

Figure 5 shows the variation of average temperature of the air when the ceiling fan was turned on. It can be seen that the air temperature in all regions of the house decreases from about 30°C at 12AM to about 27°C at about 10AM. Thereafter, the temperature increases as more sensible heat is gained by the air, reaching the highest value of about 31°C at 6PM. With the ceiling fan turned on, the temperature of the air inside the house is still outside the thermal comfort limits specified by ASHRAE [1].

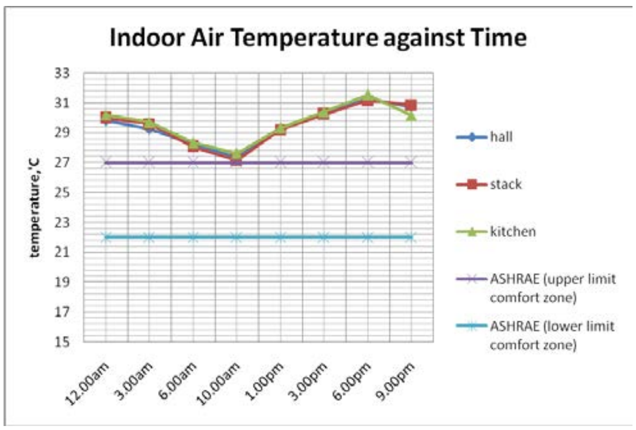

Fig. 5 Variation of average temperature of the air inside the house when the ceiling fan was turned on.

The variation of relative humidity of the air inside the house when the ceiling fan was turned-on is shown in Figure 6. It is seen that the relative humidity of the air in all sections of the house fluctuates during the data collection period, reaching the highest value of 80% at 1PM and the lowest value of 65% at 6PM. The relative humidity of the air is also outside the thermal comfort limits specified by ASHRAE standard [1].

The variation of air velocity inside the house with the ceiling fan turned on is shown in Figure 7. It is seen here that the air velocity in the hall section is still above the limit of thermal comfort level specified by ASHRAE standard [1]. However, the air velocity in the stack section appears to be below the comfort level limit. The air velocity in the kitchen section is in the acceptable level of thermal comfort.

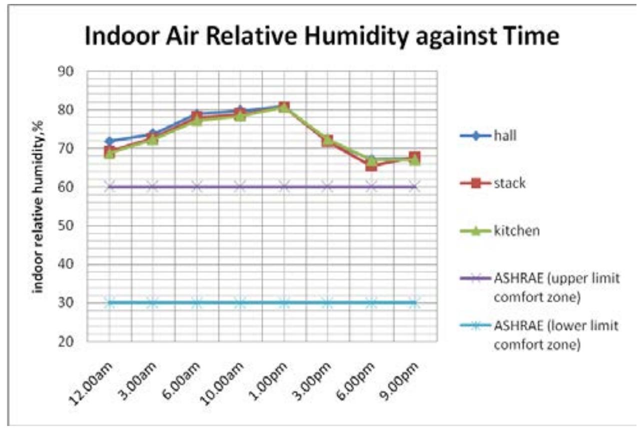

Fig. 6 Variation of relative humidity of the air inside the house when the ceiling fan was turned on.

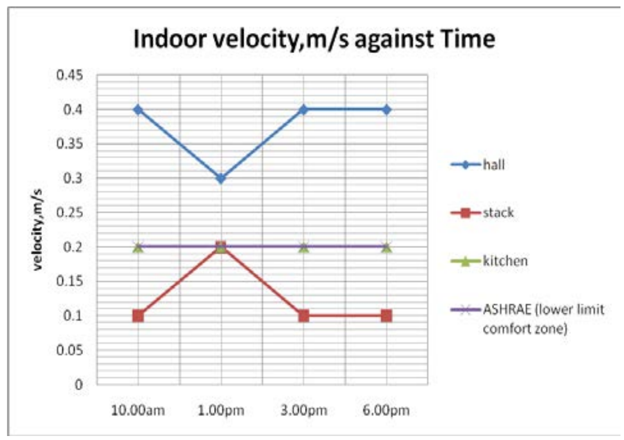

Fig. 7 Variation of air velocity inside the house when the ceiling fan was turned on.

The above results show that although a ceiling fan was turned on during the entire data collection period, the thermal comfort conditions in the house were still outside the recommended acceptable level specified by the ASHRAE standard [1]. Clearly, some other means of mechanical ventilation need to be implemented in order to further improve the thermal comfort conditions inside the naturally ventilated house.

In the next section, we describe the use of computational fluid dynamics (CFD) method to model and simulate the thermal comfort conditions of the house. Our goal is to predict the temperature distribution and velocity profile of the ambient air inside the house for two conditions. First is when the house was naturally ventilated and secondly is when the ceiling fan was turned on. We used the thermal comfort data we collected to verify the CFD modeling and simulation procedure. Once verified, we then used the model to investigate if the thermal comfort conditions in the house could be improved by the use of other mechanical ventilation method.

4. CFD Analysis of the Residential House

Figure 1 shows the virtual model of the residential house we considered in this study. The flowchart shown in Figure 8 summarizes the procedure we employed in performing the CFD modeling and simulating the thermal comfort conditions inside the house.

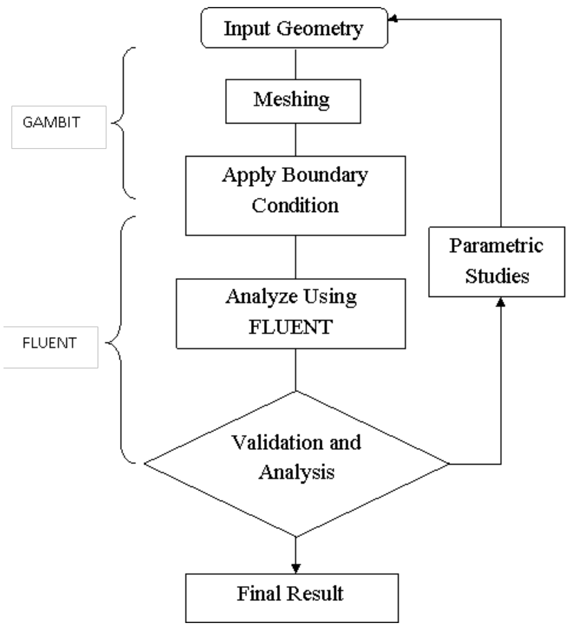

Fig. 8 CFD modeling and simulation procedure.

Three CFD models of the house were developed in this study. The first model represents the house with natural ventilation. The second model represents the house with natural ventilation and furnished with a ceiling fan, installed in the middle of the hall section. The third model represents the house furnished with the ceiling fan and two extractor fans. One extractor fan was installed on the front wall near the door while the other on the real wall, also near the door. The CFD simulations were performed on these models to investigate the distribution of air temperature and air velocity inside the house, under the given ventilation conditions.

4.1 CFD Simulation on Naturally Ventilated House

We first performed CFD simulation on the model which represents the house that is naturally ventilated. After meshing the model, we imposed the boundary conditions (BCs) on the model that represent the natural ventilation conditions, namely with the front door, middle door and back door opened. In addition, there is an opening near the top of the stack, on the front wall to allow outside air to flow into the house. These boundary conditions are shown in Table 1. The values of all parameters were taken from the actual measurement from the house.

Figure 9 shows the distribution of air velocity obtained from the CFD simulation. With natural ventilation, the air is seen to flow in a streamline condition from the front door towards the door on the middle partition wall, with decreasing velocity. It appears that air circulation also occurs in the middle of the hall but there is almost no air flow near wall far from the door. The air flows from the middle door towards the back door also in a streamline condition and with increasing velocity. Uneven flow condition is seen in the stack section and in the kitchen, in the area close to the rear wall.

Table 1 Boundary conditions for the house with natural ventilation.

Types of BCs	Zone	Parameters
Inlet Air Velocity	Front door	Velocity = 0.4 m/s Temperature = 29.6ºC
	Stack	Velocity = 0.4 m/s Temperature = 29.8ºC
Outlet Air Pressure	Back Door	Pressure = 101 kPa Temperature = 29.3ºC
Wall Thermal Conditions	Stack	Heat gain = 10 W/m^2 Temperature = 30ºC
	Front wall	Heat gain = 29.8 W/m^2 Temperature = 29.3ºC
	Back wall	Heat gain = 19.8 W/m^2 Temperature = 29.4ºC

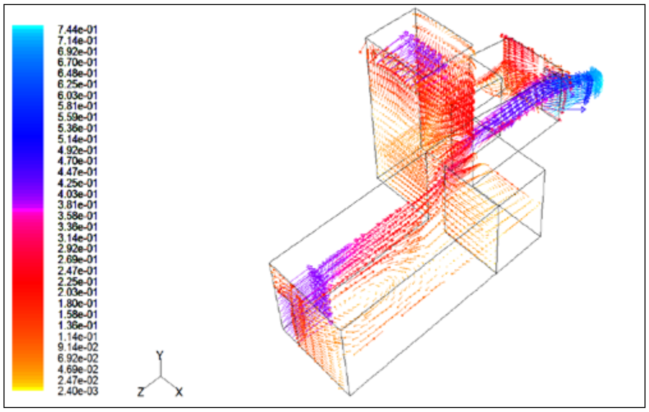

Fig. 9 Distribution of air velocity (in m/s) when the house is naturally ventilated.

Figure 10 shows the contour of temperature distribution on the walls and the air inside the house. The temperature distribution for the air is taken at a height of 1.5 m above the floor level. It can be seen that the air temperature in the hall is almost uniform at about 302K (29°C). However, the air temperature is slightly higher close to the front wall. The air temperature in the kitchen is also quite uniform at about 303K (30°C). There is a significant temperature variation on the front wall surface with the highest temperature is about 305K (32°C). This is due to a high heat gain through this wall. The front wall of the stack and back wall of the house are at a fairly uniform temperature of about 303K (30°C).

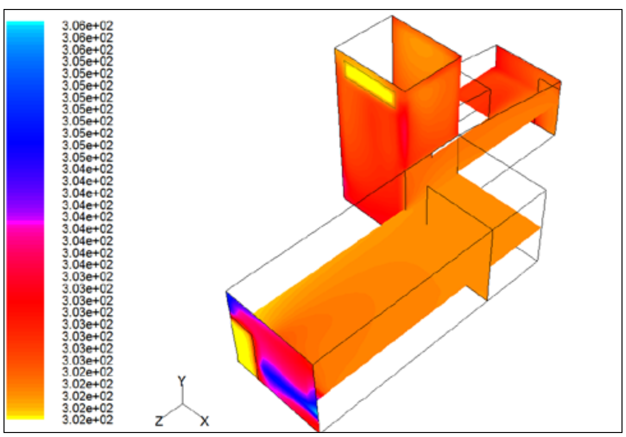

Fig. 10 Temperature (in Kelvin) distribution in the house when it is naturally ventilated.

4.2 CFD Simulation on Naturally Ventilated House with Ceiling Fan

Next, we performed similar CFD simulation on the second model of the house, which represents the naturally ventilated house with the ceiling fan turned on. The goal is to investigate the effects of the ceiling fan on the air temperature and velocity distributions. The boundary conditions prescribed on this model are similar to those given in Table 1. An additional boundary condition was prescribed on the model to represent the ceiling fan that was installed in the middle of the hall and rotating at a speed of 150 RPM.

Figure 11 shows the distribution of air velocity in the house. It can be seen that the ceiling fan had caused swirling air flow in the hall section. The swirling flow intensity appears to be highest in the area closed to the front wall. The air flow velocity increases compared to the natural ventilation condition, where the highest air velocity is about 1.3 m/s near the front wall. Swirling air flow condition can also be seen in the area closed to the middle partition wall. The air is seen to flow in a streamline condition from the door on the partition wall towards the back door, with a uniform velocity. Lower air velocity with uniform pattern occurs in the stack section. Vortex flow condition can also be seen in the kitchen, in the area closed to the wall, near the door.

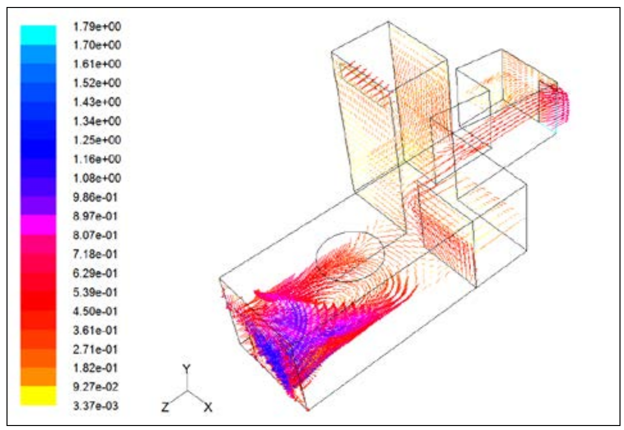

Fig. 11 Distribution of air velocity (in m/s) when the ceiling fan was turned on.

Figure 12 shows the temperature distribution in the house when the ceiling fan was turned on. It is seen that the temperature distribution in the hall appears to be uniform at about 302K (29°C). The temperature in the kitchen also appears to be uniform at about 303K (30°C). There are no more high temperature spots on the front wall when the fan is turned on. The temperature distribution on the back wall and front side of the stack is also uniform at about 303K (30°C).

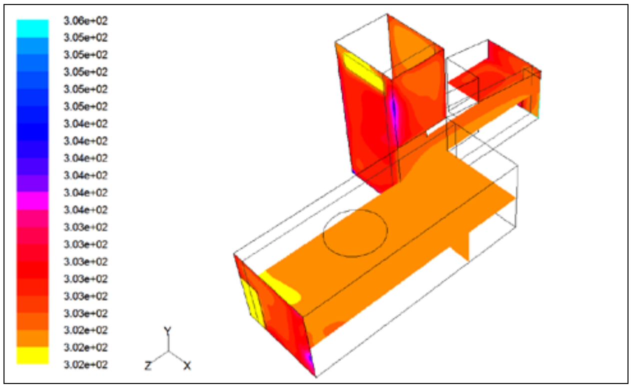

Fig. 12 Temperature (in Kelvin) distribution in the house when the ceiling fan was turned on.

The above observation shows that when the ceiling fan was turned on, vortex and swirling air flow conditions were created in the hall area. The fan also increases the air flow velocity everywhere else inside the house. But the air velocity is much higher than the level specified by ASHRAE standard [1] which could cause discomfort to the occupants. However, the ceiling fan does help to make the temperature distribution in the house to become more uniform, particularly in the hall section.

4.3 CFD Simulation on Naturally Ventilated House with Ceiling and Extractor Fans

Finally we perform CFD simulation on the third model of the house. This model represents the naturally ventilated house furnished with a ceiling fan and two extractor fans which are installed on the front and back wall, near the top of the doors. We anticipated that the extractor fans will improve the air flow and circulation conditions inside the house, thus improving the temperature distribution. We prescribed the same boundary conditions as those for the second model. In addition, we introduced extractor fans by prescribing little openings on the front and back walls, through which air at a pressure of 101 kPa and temperature of 29°C flows with a velocity of 1.4 m/s.

Figure 13 shows the distribution of air velocity inside the house when the extractor fans were turned on along with the ceiling fan. It can be observed that the extractor fan installed on the front wall does not seem to have significant effects on the flow condition and velocity of the air inside the hall section. The amount of air and velocity of the air blown by this fan are probably too small to have any significant effects on the swirling flow condition of the air created by the ceiling fan. Nevertheless the extractor fan installed on the back wall appears to affect the air flow condition in the stack and kitchen sections.

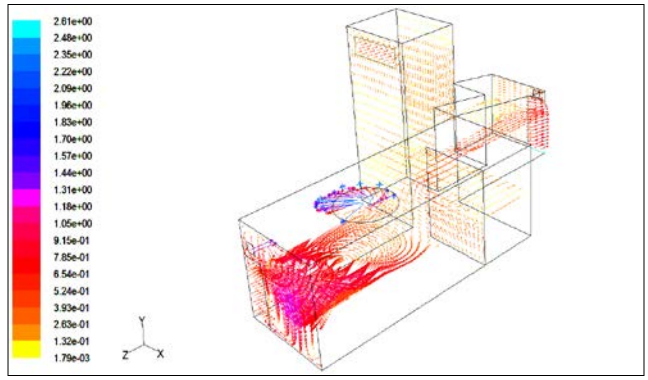

Fig.13 Distribution of air velocity inside the house when the ceiling fan is used along with the extractor fans.

Figure 14 shows the distribution of air temperature inside the house, on a plane 1.5 m above the floor. It can be seen that the temperature distribution inside the hall section remains uniform at about 302K (29°C) and appears to be unaffected by the air blown by the extractor fan on the front door. The temperature distribution on the front and back wall appears to the more uniform with a couple of high temperature spots occur near the corners of the house. High temperature spot can also be seen at the bottom corner of the stack. The front wall, back wall and stack wall show higher temperature than the air inside the house due to the heat gain from the surrounding.

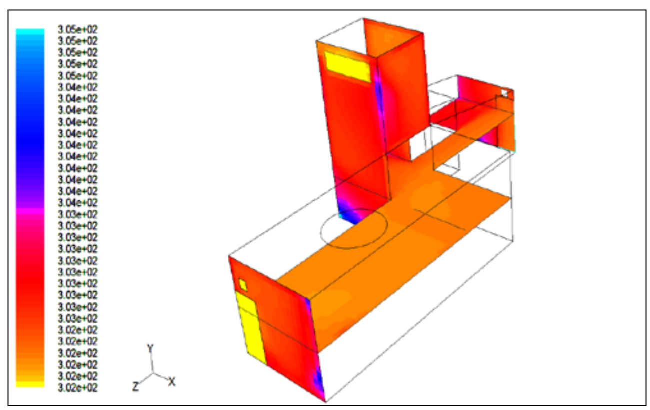

Figure 14 Distribution of air temperature inside the house when the ceiling fan is used along with the extractor fans.

The above finding suggests that the use of small extractor fans installed on the front and back walls closed to the door has almost negligible effects on the temperature distribution in the house. The flow condition and velocity of the air in the hall section are also unaffected by the extractor fans. Both parameters are still outside the range of thermal comfort level specified by ASHRAE [1].

5. Conclusion

A thermal comfort study on a naturally ventilated single-storey terrace residential house was presented. Actual measurements on the air temperature, relative humidity and air flow velocity show that the thermal comfort in the house is well outside the recommended limits specified by the ASHRAE standard. CFD simulations were conducted on the representative models of the house. Using this method, we were able to observe the temperature distribution and air flow conditions inside the house when it was naturally ventilated. The use of a ceiling fan, installed in the middle of the hall section and rotating at 150 RPM, produces a swirling flow of the air in the hall section. The distribution of air temperature becomes more uniform almost everywhere inside the house. However, the level of thermal comfort in the house still remains outside the recommended level. Results of CFD analysis also show that the use of two extractor fans, installed on the front and back wall respectively, has insignificant effects on the temperature distribution as well as air flow pattern inside the house. The research will continue to find the effective methods of improving the thermal comfort in the house.

Acknowledgments

The authors would like to acknowledge Universiti Teknologi Malaysia for supporting this research through the Research University Grant scheme.

References

[1] American Society of Heating, Refrigerating and Air-Condition Engineers, PIA ANSI/ASHRAE 55-2004, Atlanta, GA: ASHRAE, 2004.

[2] S. Mohamed, and K. Srinavin, “Thermal environment effects on construction workers’ productivity”, Work Study, Vol. 51, No. 6, 2002, pp. 297-302.

[3] A. Stamou, and I Katsiris, “Verification of a CFD model for indoor airflow and heat transfer”, Building and Environment, Vol 41, 2006, pp. 1171-1181.

[4] J.F. Nicol, and M.A. Humphreys, “Adaptive thermal comfort and sustainable thermal standards for buildings”, Energy and Buildings, Vol 34, 2002, pp. 5563-572.

[5] A. T. Balasbaneh, “Hot climate air flow study and effect of stack ventilation in residential building”, MSc Thesis, Faculty of Civil Engineering, Universiti Teknologi Malaysia, Johor Bahru, Malaysia, 2010.

[6] N. Djongyang, R. Tchinda, and D Njomo, “Thermal Comfort: A review paper”, Renewable and Sustainable Energy Reviews, Vol. 14, 2010, pp. 2626-2640.

[7] A. Bastide, P. Lauret, F. Garde, and H. Boyer, “Building energy efficiency and thermal comfort in tropical climates: Presentation of a numerical approach for predicting the percentage of well-ventilated living spaces in buildings using natural ventilation”, Energy and Buildings, Vol. 38, 2006, pp.1093-1103.

[8] N. Serra, and V. Semiao, “Comparing displacement ventilation and mixing ventilation as HVAC strategies through CFD”, Engineering Computation: International Journal for Computer-Aided Engineering and Software, Vol. 26, No. 8, 2009, pp. 950-971.

[9] G.M. Stavrakakis, D.P. Karadimou, P.L. Zervas, H. Sarimveis, N.C. Markatos, “Selection of window sizes for optimizing occupational comfort and hygiene based on computational fluid dynamics and neural networks”, Building and Environment, Vol. 46, 2011, pp. 298-314.

Masine Md. Tap, received her Bachelor’s Degree in Mechanical Engineering from Universiti Teknologi Malaysia in 1986, MPhil in Computer Aided Engineering from Herriot-Watt University, United Kingdom in 1989 and PhD. from Dundee University, United Kingdom in 1999. She is now an associate professor in the Faculty of Mechanical Engineering, Universiti Teknologi Malaysia. Her areas of interest are industrial engineering, work design and operations research. She is a co-inventor for Industrial Building System (IBS) product at national and international level.

Haslinda Mohamed Kamar, received her Bachelor’s Degree in Mechanical Engineering from University of Glasgow, Scotland in 1993, Master and PhD. in Mechanical Engineering from University Teknologi Malaysia in 1997 and 2009, respectively. She is now a senior lecturer in the Faculty of Mechanical Engineering, Universiti Teknologi Malaysia. Her areas of interest are automotive air-conditioning system, thermal comfort & energy efficiency in hot climates, and natural ventilation as passive cooling strategy in buildings.

Abdul Kadir Marsono, received his Bachelor of Engineering (civil) from Universiti Technologi Malaysia in 1985, Master of Philosophy for Heriot University in 1989 in the decipline of structural engineering and PhD in Structural Engineering from Dundee University in the year of 2000. Currently he is an associate professor in structural engineering, taught Information technology and tall building system analysis and design. He carry out research in non-linear analysis of reinforced concrete of tall buildings and industrialized building system (IBS) as well as sustainable product design for civil engineering. He is a principal patent inventor for Industrial Building System

(IBS) product at national and international level.

Nazri Kamsah received his Bachelor's Degree in Mechanical Engineering from University of Sunderland, United Kingdom in 1983, Masters of Engineering from Universiti Teknologi Malaysia in 1988, and PhD in Mechanical Engineering from University of New Hampshire, USA in 2001. He is currently a senior lecturer in the Faculty of Mechanical Engineering, Universiti Teknologi Malaysia. His areas of interest include computational solid mechanics modeling and simulation, thermal management in microelectronics, thermal comfort & energy efficiency in hot climates, natural ventilation as passive cooling strategy in buildings and computational fluid dynamics (CFD) modeling and simulation.

Khairul Amry Mohd Salimin, received his Bachelor's Degree in Mechanical Engineering from Universiti Teknologi Malaysia in 2011.