VOT 74217

# TOWARDS DEVELOPMENT OF TROPICAL SOLAR ARCHITECTURE: THE USE OF SOLAR CHIMNEY AS STACK INDUCED VENTILATION STRATEGY

# KEARAH PEMBANGUNAN SENIBINA SURIA TROPIKA : PENGGUNAAN CEROMBONG SURIA SEBAGAI STRATEGI PENGUDARAAN APUNGAN SEMULA JADI

MOHD. HAMDAN BIN AHMAD
MD RAJEH BIN SALEH
HONG LIM FOO

FACULTY OF BUILT ENVIRONMENT
UNIVERSITI TEKNOLOGI MALAYSIA

2006

VOT 74217

# TOWARDS DEVELOPMENT OF TROPICAL SOLAR ARCHITECTURE: THE USE OF SOLAR CHIMNEY AS STACK INDUCED VENTILATION STRATEGY

# KEARAH PEMBANGUNAN SENIBINA SURIA TROPIKA : PENGGUNAAN CEROMBONG SURIA SEBAGAI STRATEGI PENGUDARAAN APUNGAN SEMULA JADI

MOHD. HAMDAN BIN AHMAD
MD RAJEH BIN SALEH
HONG LIM FOO

UNIVERSITI TEKNOLOGI MALAYSIA

**VOT 74217 TOWARDS DEVELOPMENT OF TROPICAL SOLAR ARCHITECTURE 2004**

# ACKNOWLEDGEMENT

We wish to thank our research partners Agung Murti Nugroho, Chia Sok Ling and Then Jit Hiung for their help in this research from the begining until the report writing. Our gratitude also to Halimah Yahya for her help in managing the accounts. Our sincere thanks also to Dr Dilshan Remaz Ossen for helping with the correction. Last but not least our sincere thanks to Research Management Centre, UTM for the continuous financial support making this research a success.

# ABSTRACT

This research investigates the possibility of using solar chimney as stack induced ventilation strategy in terraced house in Malaysia. The existing terraced house design complies with the minimum requirement of natural ventilation as stipulated in the Malaysian Uniform Building By-Law. Unfortunately, the architectural design solutions do not permit good natural ventilation for thermal comfort. This can be illustrated by the low internal air velocity and high temperature experienced during the day time. The wind effect is also not well captured especially in the single sided ventilation. Solar chimney ventilation has been suggested by many researcher as possible alternative techniques for natural ventilation. The solar chimney study in this research involved both physical modelling and computer simulation using Computational Fluid Dynamics (CFD) technique. The specific software called Flo Vent is used. Validation of CFD FloVent is done by comparing the computer simulation result with the field measurement on site and in experiment existing terraced house. The result of the final model solar chimney prototype shows that it can increase indoor air velocity until 0.8 m/s. The other important factor is that it can continuously induce the flow of natural ventilation regardless of the available wind outside of the house. This effect is significant toward improving the thermal comfort performance in the terraced house through passive natural ventilation.

# ABSTRAK

Penyelidikan ini adalah untuk menilai kemungkinan penggunaan cerombong suria sebagai strategi pengudaraan apung semula jadi dalam rumah teres di Malaysia. Rekabentuk rumah teres didapati menepati keperluan minimum pengudaraan semula jadi yang dikehendaki di dalam Undang-undang Kecil Bangunan Seragam Malaysia. Malangnya reka bentuk senibinanya belum memungkinkan pengudaraan semulajadi yang baik. Keadaan ini ditunjukkan dengan rendahnya halaju angin dan tingginya suhu yang di dalam pada siang hari. Kesan angin juga tidak ditekankan terutama dalam strategi pengudaraan sebelah depan. Pengudaraan dengan cerombong suria telah dicadangkan oleh ramai penyelidik sebagai salah satu kemungkinan di dalam teknik pengudaraan semulajadi. Kajian cerombong suria dalam penyelidikan ini melibatkan permodelan fizik dan simulasi komputer berteknikan pengiraan bendalir dinamik (CFD). Perisian khusus bernama Flo Vent telah digunakan. Validasi perisian Flo Vent telah dilakukan dengan membandingkan dengan hasil pengukuran lapangan dan kajian dalam rumah teres sediaada. Keputusan contoh model cerombong suria telah memberi kesan kepada peningkatan halaju angin dalaman sehingga 0.8 meter per saat. Faktor penting yang lain ialah model ini dapat memberi halaju dalaman secara berterusan tanpa mengira keadaan di luar rumah. Kesan kajian ini cukup besar bagi memperbaiki prestasi keselesaan termal di rumah teres melalui pengudaraan semula jadi.

# TABLE OF CONTENTS

III METHODOLOGY

IV RESULT AND DISSCUSION

IV CONCLUTION AND RECOMMENDATION

# LIST OF TABLES

# LIST OF FIGURES

# LIST OF ABBREVIATIONS

| | | |
|---|---|---|
| ASHRAE | - | American Society of Heating, Refrigerating and Air Conditioning Engineers |
| BCR | - | Bio-climatic roof |
| BFC | - | Body fitted coordinate |
| CAD | - | Computer Aided Design |
| CFD | - | Computational Fluids Dynamic |
| LES | - | Large Eddy Simulation |
| M | - | Material |
| MSW | - | Metallic solar wall |
| PC | - | personal computer |
| PDE's | - | Partial differential equations |
| Re | - | Reynolds |
| RSC | - | Roof solar collector |
| SC | - | Solar chimney |
| SIMPLE | - | Semi-Implicit Method of Pressure-Linked equation |
| SI | - | Sultan Ismail |
| Th | - | Thickness |
| TH | - | Terraced House |
| UTM | - | Universiti Teknologi Malaysia. |
| W | - | Width |

## LIST OF SYMBOLS

$A$ - Surface Area ($m^2$)

$\Delta T$ - Temperature Different (ºC)

$\beta$ - The thermal expansion coefficient (1/K)

$g$ - The acceleration due to gravity (m/s2);

$H$ - Height (m)

$k$-$\varepsilon$ - Turbulent kinetic energy (k)

$l$ - The characteristic length (m)

$L$ - Characteristic length of flow system (m)

$\rho$ - Air density

$P$ - Outdoor air density

$Pv$ - Air pressure

$Q$ - Air flow rate (l/s)

$S_{\varphi}$ - Source

$\Gamma_{\varphi}$ - Exchange coefficient

$U$ - Proposed simulated wind

$\upsilon$ - The viscosity (m2/s)

$\upsilon$ - Kinematics viscosity (u/p)

$\acute{\upsilon}$ - Velocity vector

$v$ - Air Velocity (m/s)

$V$ - Local wind velocity at the upwind building height (m/s)

$\varphi$ - Dependent variable

$Z0$ - The roughness length

# LIST OF APPENDICES

# CHAPTER 1

# INTRODUCTION

## 1.1 Research Background

In tropical climatic regions, passive cooling is one of the most difficult problems to solve. The simplest and the most effective solution for active cooling is by introducing air conditioning. However, such equipment involves high initial and operational costs for installation, energy and maintenance. Therefore, air conditioners are unlikely to be applied widely in building, especially for low income earners. Thus, a passive cooling system is more desirable. Although in Malaysia passive cooling method is a popular cooling strategy adopted in residential building, researches (Hui, 1998; Abdul Razak, 2004) have shown that its natural ventilation performance could not provide good internal thermal comfort. Climate conscious design in the equatorial tropic assumes that air movement is one of the main cures for thermal comfort ills. According to Hui (1998), the indoor air velocity in low rise building ranges between 0.04m/s – 0.47 m/s. The reasons may be due to inappropriate design solutions for indoor air movement or low outdoor air velocity, but these reasons remain to be determined. Recent data from the Malaysian Meteorological Service Department showed that mean outdoor air velocity is between 1 m/s to 1.5 m/s in 10 m high. This velocity will reduce nearer the ground level and sometime can become still air. Thus there is a need to provide alternative to natural ventilation strategy in order to increase the internal air movement within the ventilation comfort range.

Ventilation is one of the important options in providing thermal comfort in buildings (Bansal, 1994). Thus, a passive cooling system is more desirable. To reduce the mechanical cooling energy cost of new building in a hot and humid region, the design should maximize the natural ventilation and minimize the fraction of sun energy absorbed by a dwelling (Khedari J, 1997). Providing adequate natural ventilation would reduce the building cooling load in tropical areas. Two major goals in natural ventilation include provision of sufficient fresh air and satisfactory temperature.  At temperatures below 34°C, which is the average temperature in many hot and humid conditions, air movement might be one of the most useful and least expensive methods to provide a comfortable indoor climate.  The movement of air across human skin creates a cooling sensation causes by heat leaving the skin through convection and by the operation of perspiration.  Air movement at a speed of up to 0.25 m/s can go unnoticed.  The most common way to create air movement without mechanical power is to open a window and allow breezes to blow into a building.  However, the problem with this simple concept is that an open window can admit dust, pollen, and dirt (Khedari J, 2000).

One application of natural ventilation is through stack effect ventilation. One way to increment stack effect ventilation in buildings and, as a consequence, to improve indoor air quality, relies upon the use of solar energy, namely through solar induced ventilation. Ventilation provides cooling by using moving air to carry away heat from the building when the indoor temperature is above the outdoor temperature. Ventilation cooling may use air from either the interior or exterior of the building. The exterior air can be used to remove excessive heat and humidity when outdoor conditions are favorable. The indoor air circulation does not change the interior conditions, only making them more comfortable for the building occupants (Hirunlabh, 1999).

Solar chimney is one of the several available options for achieving solar-induced ventilation (N.K. Bansal, 1994). Solar chimneys are used to enhance air movement in naturally-ventilated buildings. They are similar to conventional chimneys except that the south wall is replaced by a glazing, thus enabling solar energy collection. This leads to an increase in air temperature inside the chimney channel and in the stack effect and, simultaneously, to thermal energy storage in the walls, which can be released later on during periods of small solar irradiation or night periods (Clito Alfonso, 2000).

Studies of solar chimneys involve aerodynamic experiments on buoyancy problems which can be done using both physical modeling and computer modeling methods, the latter based on computational fluid dynamics (CFD). Both methods have advantages and disadvantages. Physical modeling is considered to give results that are easy to check. However, for an experiment which involves various model designs this method can become expensive and time consuming. The computer simulation method, on the other hand, allows easy modification of the design, more precise results in less time. Even though computer simulation has a high initial cost (for its hardware and software), the final cost might be lower than that of the physical experiment since changes in the model designs can be made easily. However, interpretation of CFD data is still difficult especially for new applications, such as in studying natural ventilation. Thus, it is still worth carrying out physical as well as using computer modeling to obtain more reliable interpretations and conclusions from the results (Satwiko, 1994).

## 1.2 The Problem Statement

In hot humid climate, the problem emphasized by the fact that it is important to understand the solar radiation, temperature and wind profile in and outside buildings. Hot humid tropical conditions in Malaysia can result in high temperature, and low air flow contributed further to the decreased in the thermal comfortable condition in the building. Residential buildings especially in low rise terrace housing development are subject to significant cooling requirements due to high intensity of solar radiation received by the roof. Single sided window or opening could not provide good passive ventilation strategy. Alternatively, passive cooling by solar induced ventilation can substantially increase air velocity in the rooms and large energy savings can be achieved. An increase air flow will remove internal heat gain, reduce high temperature in the room and produce psychological cooling. However, this alternative has not been studied.

## 1.3 Research Hypostudy

The hypostudy of this study is that “appropriate” design of solar chimney ventilation will achieve the following:

- Increase temperature different between inside solar chimney and outdoor climate condition.
- Provide maximum air movement (velocity and flow rate) within the range of the thermal ventilation requirement.
- Increase and provide maximum air movement for prediction of the effectiveness of the induced stack ventilation.

The term “appropriate” refers to the best performance of solar chimney configurations which will increase the maximum air movement inside the solar chimney model in order to obtain optimum induced stack ventilation.

## 1.4 Research Questions

The following questions will be addressed in this study:

Q1. Is the use solar chimney ventilation possible in Malaysian climatic condition?
Q2. What is the optimum solar chimney configuration in order to achieve targeted maximum induced stack ventilation?
Q3. What is the proposed solar chimney ventilation model to obtain maximum induced stack ventilation in Malaysia in relation with climate condition elements?

Q4. Does the proposed solar chimney at (Q3) effective to increase induced stack ventilation in residential building?

Q5. What is the limitation of the proposed solar chimney model towards increasing air movement in the residential building?

## 1.5 Research Objective

The main objective of this study is to assess and compare the impact of solar chimney induced ventilation for increasing air movement in Malaysian residential building.

Other specific objectives of the study are as follows:

- To develop solar chimney prototype towards improving indoor air movement in residential building.
- To analyze and establish the effectiveness of the induced stack ventilation technique of the proposed solar chimney

## 1.6 Scope and Limitations

The scope of this study is to evaluate the effectiveness of proposed solar chimney model in Malaysia's residential building. The indoor thermal ventilation aspects are major issue. There are other parameters effecting thermal ventilation, for e.g. air temperature, humidity, air velocity, clothing, metabolic heat production and so forth (Givoni, 1981; Abdul Razak, 2004). Metabolic rate and clothing assumed that by setting the occupant at recommended set value, it will provide the required thermal quality for that space. Mechanical and technical system operating conditions were also kept identical to all experiments tested. A standard single storey terraced house of residential room with a three bedrooms was selected for the experiment.

This study is entirely carried out by using computer simulation program Flo Vent (Version 5) and thus bears the limitations of the simulation tool used. In the following chapter, a review on common research methods used by previous researches and justification for the selection of the present tool will be discussed.

## 1.7 Importance of the Research

The out come of this study is expected to show that the effectiveness of the proposed solar chimney ventilation system will provide the increase in air movement for the induced stack ventilation in building for thermal comfort. The study is also expected to suggest that appropriate design decisions on solar chimney ventilation systems can significantly reduce the heat gain in residential buildings in Malaysia. Apart from an increase in air movement and reduce in air temperature, the use of solar chimney has other benefits on various other aspects as shown in figure 1. The most important aspect is the thermal comfort and energy efficiency. Hence, findings of this study will enable and provide the building designer with wider range of options in selecting appropriate solar induced ventilation strategy for achieving the balance between thermal comfort and energy efficient.

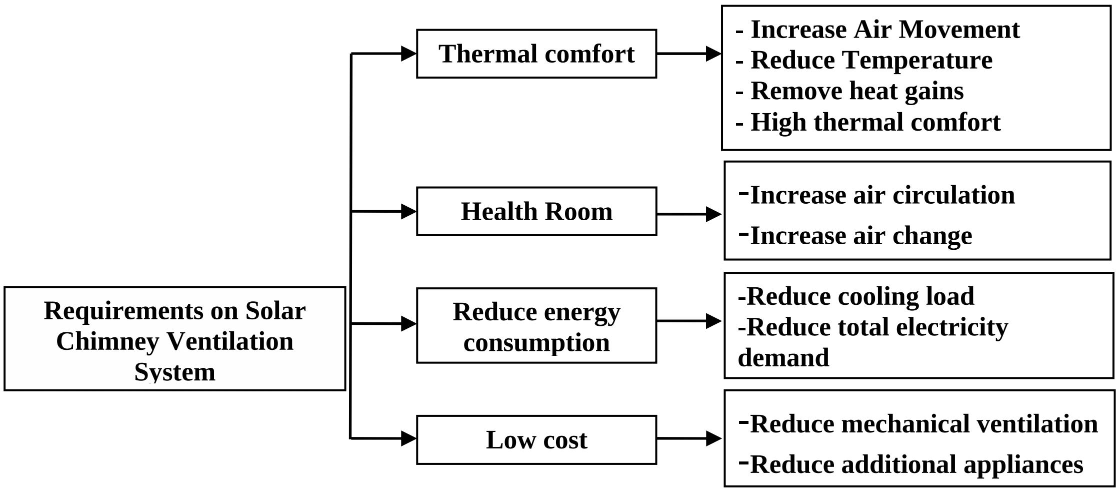


**Figure 1.1:** User requirements for solar chimney ventilation systems

## 1.8 Organization of This Research Report

The study is divided into five chapters as summarized below.

**Chapter one** introduces the main issue of this research. This chapter discusses the research background, problem statements, hypothesis of the study, research questions, objective, scope and limitations of the study, importance of the research and the overall study structure are also presented in this chapter.

**Chapter two** presents the literature review of climate condition and thermal comfort study of Malaysia, solar chimney ventilation and simulation study. This chapter introduces an overview of Malaysian climate in thermal comfort condition and the concept of thermal comfort zone for Malaysia. All aspects of ventilation are discussed in this chapter with the intention of giving a comprehensive review of the solar induced ventilation. The study also covers the concepts and work related to architectural design that affect the stack ventilation especially study on the solar chimney that have been carried out by other researchers. Literature reviews on the work done by previous researchers in solar induced are also deliberated. Finally, the ventilation simulation using Computational Fluid Dynamics in the similar research area is used to study the effect of solar induced ventilation in relation to various design parameters proposed

**Chapter three** discusses the research design and the methodology implemented in solar chimney study. The justification of selecting the methodology for this study is also elaborated. The research investigations conducted in this research were explained, including the experimentation and simulation method. The experimental and simulation design method are discussed separately. A thorough analysis of the raw data collected from each type of study was done and is later discussed in the following chapter 4. Further, development of the base model, procedures, assumptions, limitations, condition and the overall setting-up of the pilot testing and CFD simulation are also described. The reliability and validity of the methods, equipments and simulation procedures are also discussed. The estimation of the air velocity and temperature value for the research is also presented. Finally, the data analysis criterions are discussed, which is used to analyze the results of the experiment.

**Chapter four** presents the results and analysis of the pilot testing, validation, configuration, proposed and application of solar chimney model. The principle findings of the experiment and simulation are also summarized. The results of the research are analyzed as follows by:

- Assessing the pilot testing and the validation of CFD simulation of solar induced ventilation strategies within site climate.
- Assessing the configuration of solar chimney design on the targeted maximum temperature different, air velocity and air flow rate inside the chimney model.
- Assessing the performance of proposed solar chimney ventilation on Malaysian climate condition.
- Assessing the application of proposed prototype of optimum solar chimney model on selected terrace house.

This chapter in general is also divided into four sections. Section one discusses the pilot testing and the CFD simulation on the possibility of the use of stack effect within site climate. The temperature differences, impact of solar radiation on the temperature differences, simulation validation and solar induced estimation are also discussed in order to understand the solar chimney design possibilities within selected climate conditions. Section two discusses the CFD simulation of solar chimney configuration before proposing the best option for solar chimney model. Section three discusses the performance of proposed solar chimney model based on the selected climate data of 2005. Finally, in sub-section three of section four, it discusses the application of proposed solar chimney model employed onto the selected terrace house model. The results obtained from the simulation exercises are presented and analyzed. This includes comparative analysis of the predicted internal air velocity obtained from the proposed design solution with the required internal air velocity for thermal comfort in Malaysian climate. The summary of the major findings is also presented in this chapter.

**Chapter five** concludes the study by summarizing the major findings of the experiment. It also outlines the suggestions for future research on stack ventilation study especially beyond the limitations of this study.

# CHAPTER 2

# LITERATURE REVIEW

## 2.1 Malaysia's Climate Condition

Malaysia lies between 1º and 7º North latitude and 100º and 120º East longitude. Malaysia has two main land areas which is the Peninsular Malaysia and East Malaysia. Being very close to the equator, it naturally experiences an equatorial climate which is characterized by hot and humid condition and heavy rainfall throughout the year with no distinct dry season. It also enjoys abundant sunshine all year round and experiences an almost constant temperature with a yearly mean of between 26ºC and 27ºC. The mean maximum daytime temperature is between 29ºC to 32ºC while the mean minimum temperature is between 22ºC to 24ºC at night in the coastal areas. Because it is surrounded by the sea and receives heavy rainfall throughout the year with an annual average of about 2000mm to 3500mm, its high humidity and heavy cloud cover causes a low yearly diurnal temperature of about 2ºC. Daily diurnal temperature is higher, i.e. between 5ºC to 12ºC (Samirah, 1998). Malaysia falls under the influence of the Southwest Monsoon and the Northeast Monsoon. The Southwest Monsoon originates from Australia and blows across the Sumatera Island and the Straits of Malacca in the months of May to September. During Southwest Monsoon season, the West coast of Malaysia and Sabah and South of Sarawak receives heavy rainfall. The Northeast Monsoon originates from the central Asian continent and blow across the South China Sea through Malaysia to Australia from the months of November to March (Majid, 1996). All area in Malaysia which area faces and exposes to the South China sea will not only receive heavy rainfall during these months but will also receive the strongest winds. Thus the monsoons will bring about more intense rainfall. Generally, Malaysia experiences light winds of variable speed with the minimum wind speeds occurring just before dawn and the maximum, in the afternoon. This pattern is controlled by convection in the surface boundary layer as the sun heats the ground during the day and is cooled by radiation during the night (Exell, .R.H.B. and Fook, C.T. 1985)

### 2.1.1 The Wind Climate of Malaysia

It is essential to know the geographical conditions of Malaysia in order to understand its wind climate. Malaysia is made up of two major sectors: the Peninsula and the eastern sectors (Sarawak and Sabah) in the northern part of the island of Borneo. The Peninsula is bounded by Thailand in the north, separated from the island of Sumatra by the Malacca Street along its western coast, detached from the small island of Singapore by the Teberau street in the south, and separated from the eastern sectors by the South China Sea in the east Peninsular Malaysia, experiences a hot-humid tropical climate with no distinct seasonal variation (Yeang, 1992). The peninsula is narrow and divided into two flat coastal plains by the central mountain ranges. The primary and secondary forest that covers almost three quarters of the land area is important in modifying the climate near the ground by absorbing heat, moderating temperature, giving shade and modifying the wind climate (Yeang, 1992). At macro scale, peninsular Malaysia and the other parts of Southeast Asia are influenced by the major air streams that originate from the North-east and Central Asia, the North Pacific, Australia, the South Indian Ocean and the South Pacific (Takashi, 1993). The air streams pass over Southeast Asia in three main directions and form boundaries. The three boundaries are (as suggested by Majid, 1996):

a) The Northern Equatorial Airstream boundary.
b) The Southern Equatorial Airstream boundary
c) The Combined Airstream boundary

The winds that blow over peninsular Malaysia and other parts of Southeast Asia are related to the above three airstreams and are normally associated with the rainfall in this area (Thomson, 1980). The patterns of air flow created by the airstreams divide the year into three seasons:

a) The north-east Monsoon.

b) The south-west Monsoon.

c) The transitional periods between the monsoons.

**Table 2.1:** Summary of wind flow over peninsular Malaysia (Majid, 1996)

| Duration | Types Of Wind | Affected Area |
|---|---|---|
| November<br>December<br>January<br>February<br>March | north-east Monsoon (Strong, together with heavy rain) | • The whole of peninsular<br>• East coast of peninsular |
| April<br>May | Transitional period<br>(Weak and variable) | • The whole of peninsular |
| June<br>July<br>August<br>September<br>October | south-west monsoon<br>(Not as strong as the north-east)<br>Southerly wind (Light wind) | • Northern part of peninsular<br>• Southern part of peninsular below latitude 5°N |
| October<br>November | Transitional period<br>(Weak and variables) | • The whole of peninsular |

Table 2.1 shows that in the month of November or early December until around March, the strong north-east monsoon arrives in the peninsular with heavy rain especially on the east coast. This is followed in the month of April or May by a transitional period of between half a month and two months with weak and variable winds. In the months of June to September or early October a less strong south-west wind blows over the northern part of the peninsular. However, at about the same time the southern part of the peninsular, especially below latitude 5°N, experiences a light southerly wind. The southwest and southerly winds are never strong and are sometimes overshadow by the land and sea breezes. Finally, in the months of October and early November there is another transitional period of weak and variable wind.

Other important wind phenomena in peninsular Malaysia, especially in the coastal regions, are the land and sea breezes. These winds are developed by the differential heating and cooling over land and sea. The sea breeze begins at about 10 am and blows with the greatest force in the early afternoon and fades out at sunset (Mcliveen, 1992), while the land breeze normally takes over in the late evening and night. The land breeze is never as strong as the sea breeze and is only felt within a range of about 16 km from the shore (Yeang, 1992). However, both of these breezes can reach a maximum average speed of about 3 m/s and are able to overshadow the monsoons in some areas. The prevailing north-easterly winds are too strong to let the land breeze develop along the east coast and the sea breeze along the west coast. Since the south-west monsoons are not as strong as those from the northeast, a reversal situation may also occur, but only for a limited period.

On the whole, the surface winds over peninsular Malaysia are generally mild, with a maximum speed of about 8 m/s and gust speeds of less than 13 m/s (Majid, 1996). The percentage of calm period (periods of no wind) ranges between 20% and 50%, and varies from place to place (Mcliveen, 1992), Local squalls occur, caused by the differences in local topography that disrupt the smooth flow of air streams. Line squalls may accompany a moving air stream and intensify the wind. Both squalls are normally very active from May to August, but may vary from place to place. The most well known line squalls, which normally occur along the west coast of the peninsular, are the "Sumatras" (Majid, 1996).

### 2.1.2 The Solar Radiation in Malaysian Condition

In order to understand the solar radiation distribution, the geographical features of Peninsular Malaysia is briefly described. The land mass is divided into the east and west coasts by the main range running from north to south. The South-west monsoon brings rain and cloud to the west coast from June to early October, while the North-east monsoon brings rain and cloud to the east coast from November to early March. Therefore, the west coast is dry from November to April, while the east coast is dry from March to August. During the dry season, the climate is hot and sunny, with intermittent breaks of cloud formation and hence rainfall in the late afternoons due to convection current. For the purpose of this study, the peninsula will be divided into three regions according to their geographical conditions, namely, Northern Region, East Coast Region and the Southern Region. Alor Setar, Penang, Ipoh, Cameron Highlands, Sitiawan will be referred to as northern towns. Kota Bharu, Kuala Trengganu and Kuantan will be called the east coast towns, while Kuala Lumpur, Malacca, Mersing and Senai, the southern towns (Chuah, 1984).

In Peninsular Malaysia, daily total radiation data are recorded for at least five years since 1975 in three major towns, namely, Kuala Lumpur, Penang and Kota Bharu. These locations represent the three regions of different geographical and climatic conditions. Total radiation data are also recorded in stations in Ipoh, Malacca, Kuala Trengganu (since 1977) and Kuantan (since 1980). Records of radiation data in other towns are scarce. In Sabah and Sarawak the only radiation data available are those from the Kunak estate (Cuah, 1984). The solar radiation received on the ground in most parts of the peninsular is mainly the diffuse radiation component rather than direct radiation. This situation is caused by the continuous presence of clouds in the atmosphere that reflect and scatter the solar radiation. Therefore, the radiation that reaches the ground is normally much diffused, which causes the uncomfortable sky glare. The continuous presence of cloud and water vapour in the atmosphere over peninsular Malaysia also reduces outgoing radiation at night. The mean global radiation in most areas is between 10.5 to 19.0 Mj/m/day (Majid, 1996).

### 2.1.3 Other Related Climatic Elements of Malaysia

#### a. Air temperature

The air temperature of peninsular Malaysia remains almost uniformly high throughoul the year. Based on the records monitored by the Malaysian

Meteorological Service up to 1993, the daily variation of air temperature for most of the months in a year for the major parts of the low land areas is from 22°C to 34°C. The mean maximum temperature during the day is between 31°C and 34°C, and the night mean minimum varies between 22°C and 27°C (Majid, 1992). However, there is a small but noticeable drop in the monthly mean temperatures in the east coast part of the peninsular during the north-east monsoon season. In most parts of the peninsular, the mean animal range is very small and the diurnal range of the temperature is quite narrow, which is typical for a hot-humid tropical climate.

#### b. Relative humidity

The average relative humidity (RH) remains high at about 80% in most areas in peninsular Malaysia. The actual values however may vary from 55% to almost 99%. The characteristic features of peninsular Malaysia, i.e. the steady high temperature and relative humidity, are far from being optimal for physical comfort. It is therefore the air movement from the prevailing winds, and the local land and sea breezes that are the only means of bringing some natural relief to the uncomfortable conditions of this climate (Majid, 1996).

### 2.1.4 Selected Major Town Climatic Condition

Most towns in the Peninsular experience high temperature and humidity throughout the year without remarkable variations. However, there is a seasonal climatic change, which is dominated by the monsoons. The climatic conditions of towns dominated by the monsoons are significantly different between the east and west coast of the Peninsular (Kubota, 2006). Johor Baharu is located in the southern tip of peninsular Malaysia and is physically connected to the neighbouring island of Singapore by a causeway. Economically Johor Baharu is still closely associated with Singapore as some of Malaysia's export and import activities are still carried out through Singapore. The tabulated wind and other related climatic data summarised from the data is gathered from the nearest Senai Meteorological Station (Majid, 1994).

## 2.2 Thermal Comfort Ventilation in Malaysian Condition

Fanger (1970) defined thermal comfort for a person as a condition of mind, which expresses satisfaction with thermal environment. It also can be termed as thermal neutrality for a person (Fanger, 1970; Abdul Razak, 2004). Thermal comfort is affected by two main factors (Fanger, 1970; Abdul Razak, 2004):

- Environmental factor (air temperature, relative humidity, air movement, and radiation.
- Subjective factor (activity, clothing, age, sex, heath condition, food and drink, skin color, human size

In the room, thermal comfort is achieved through the process of removing the heat load in the room when the internal environment is too hot and stuffy. Meanwhile, for human thermal comfort the most important parameter that can provide thermal comfort is the air movement or the air velocity (m/s) that passes the human body rather than airflow rate (m2/s) and air change rate (ACH) (Ansley, 1999, Givoni, 1998, Abdul Razak, 2004). This is known as ventilation for psychological cooling. The airflow and air change rate is more toward providing the healthy living room. Psychological cooling can be achieved through air movement that passes by the surface of human skin. This process of convection and evaporation of sweat secretion helps to cool down the skin surfaces. This will maintain the normal human body temperature at 37°C. Since the velocity of the air inside a building has an effect on air temperature and relative humidity, its presence is very important in human life. However, there are limits with respect to the range of air velocity that is considered acceptable for thermal comfort in a building. The acceptable range of air velocity for thermal comfort especially for Malaysia’s hot and humid condition Md. Rajeh (1989) can be achieved by introducing 1.0 m/s air velocity.

### 2.2.1 Natural Ventilation

Ventilation condition in a building indirectly affects the air temperature and relative humidity in building space. Thus, determine health and comfort of the occupants. Therefore, ventilation may be utilized to ameliorate the thermal condition caused by the air temperature and relative humidity. ASHRAE (1997) defined natural ventilation as an intentional flow of air through open windows, doors, grilles and another planned building envelope penetrations from outside into a building. Its is driven by natural and / or artificial means. Artificial ventilation is usually referred to forced ventilation and natural ventilation is usually referred to cross ventilation.

Forced ventilation according to ASHRAE (1997) is an intentional movement of air into and out of a building using fans and intake exhaust vents or usually being called as mechanical ventilation. Ventilation has three main function in human life (Awbi, 1991; Givoni, 1998; Evans, 1980). They are:

- Ventilation for health
- Ventilation for thermal comfort
- Ventilation for structural cooling

According to Strateen (1967), ventilation function can be categorized into two, which are for health and thermal comfort. The function for structural cooling include as part of a function for thermal comfort (Abdul Razak, 2004). In hot and humid country like Malaysia, the main purpose of ventilation is for health and thermal comfort (Szokolay, 1998). Health requirement is achieved through the exchange of internal air and fresh outdoor air with unpleasant smell removed. For thermal comfort, air movement cools the body and removes the excessive heat. This ventilation process can only occur if there are air movements inside and outside the building

Thermal comfort ventilation is required to provide or improve the comfort conditions of building occupants by means of physiological cooling and is often used in hot and humid climate condition. This is through elimination of the feeling of discomfort due to warmth and skin wetness or sweat by the cooling effect of air movement past the skin surface. The cooling effect is brought about by the accelerated heat dissipation from the body due to increasing the convective heat loss from the body and accelerating the evaporation of sweat from the skin as the air flows past it. Humidity levels however, play significant role in the evaporative capacity of the air. When the humidity is high, the high vapour pressure will prevent evaporation thus restricting the cooling effect unless higher air velocities are introduced. For thermal comfort ventilation, the value of local air velocities in occupied space would be more applicable than ventilation rates or air change rates. It was suggested that indoor air movement required for comfort should not exceed 1.0 m/s as loose papers and light object tend to be disturbed above that velocity. However, it was found that some residents in hot and humid tropics often operate ceiling fans with velocities much higher than 1.0 m/s to improve thermal comfort conditions. Szokolay 1997 rendered an air movement of 1.5 m/s as still being acceptable for hot conditions. Thus, for hot humid climates, it is suggested that provisions should be made to acquire an air movement of at least between 1.0 m/s to 1.5 m/s in order to achieve the desirable comfort condition (Samirah, 1998).

In theory, there are two natural ventilation mechanisms (ASHRAE, 1997). First is by wind pressure and the second is by temperature difference or stack effect. Both mechanisms have the same aim, which is to act as an aid to create air movement and consequently control the indoor air temperature (Abdul Razak, 2004). Natural ventilation may result from air penetration through a variety of unintentional openings in the building envelope, but it also occurs as a result of manual control of building's openings doors and windows. Air is driven in and out of the building as a result of pressure differences across the openings, which are due to combined action of wind and buoyancy-driven forces. Today, natural ventilation is not only regarded as a simple measure to provide fresh air for the occupants, necessary to maintain acceptable air-quality levels, but also as an excellent energy-saving way to reduce the internal cooling load of housing located in the tropics. Depending on ambient conditions, natural ventilation may lead to indoor thermal comfort without mechanical cooling being required (Khedari J, 1999).

The driving mechanism required for the occurrence of wind driven natural ventilation through a building is pressure differences or the presence of pressure gradients across the building. The pressure differences can be induced by two energy sources: wind and the air temperature differences between the inside and outside of a building. Application of natural ventilation using pressure differences created by external wind is widely used in hot and humid climates and during the summer in temperate climates. Natural ventilation utilizing air temperature differences are also known as stack ventilation and thermal buoyancy ventilation (BRE Digest 1994).

- Pressure Differences generated by Wind

When a flow of wind is obstructed by a building, the mean pressure Pv generated at a point in the external surface is given by Bernoulli's equation :

$$Pv=1/2\ \rho V2$$

Where

$\rho$ = outdoor air density as a function of atmospheric pressure, temperature and humidity

V = local wind velocity at the upwind building height

The local wind velocity is governed by the height and the terrain type and can be estimated using data obtained from the nearest meteorological station. Mean values of pressure is used instead of instantaneous pressure as the presence of turbulence in the free wind and that created by the building and that of upstream obstructions causes the surface pressure to fluctuate and vary with time. Also, the computation of infiltration and ventilation rate is more appropriate if mean value are used (Allard and Herlin 1989).

- Pressure Differences generated by Temperature Differences- The Stack Effect

  A vertical pressure difference can be created by temperature differences due to variations in air density since air density varies as the inverse of absolute temperature. The magnitude of the pressure gradient between interior and exterior of a building will depend on the temperature differences between them and the height between the inlet and outlet opening in the building envelope. A pressure difference of about 0.04 Pa per meter of building height is created for each degree of temperature difference between internal and external air temperature.

### 2.2.2 Malaysia’s Scenario of Thermal Comfort Ventilation

An overall view of the climatic condition in Malaysia indicates that main factors that affect thermal comfort in this region area solar radiation, high temperature and high humidity. Maintaining the body’s heat balance will require extra effort due to these climatic stresses. Obviously, the primary source of heat gain to the body is direct radiation from the sun. For building occupants, heat gain is contributed by conduction and radiation from the building fabric, hot air, as well as indirect solar radiation through windows and openings (Lim Jee Yauan 1987). Thermal comfort requirement in hot and humid conditions of Malaysia calls for the minimization of heat gain by the building fabric through solar radiation as well as heat gain by the human body while maximizing heat dissipation from the body by ventilation and evaporative cooling. The indoor comfort condition in building in Malaysia comfort to the ASHRAE Summer Comfort Zone described in the ASHRAE Handbook Fundamental (1989). The design condition in an air conditioned space is 24ºC with 60% relative humidity (Guidelines for Energy Efficiency, 1989).

All studies about indoor comfort in hot and humid climate indicated a higher neutral temperature than that predicted by ASHRAE. In fact the neutral temperature found in the field study conducted by Abdul Rahman and Kannan 1997 in naturally ventilated buildings agreed well with Equation of Humpreys's adaptive approach. This indicates that our current indoor temperature standards need to be reviewed and modified.

The most comprehensive study in trying to establish the Malaysian comfort zone was done by Mohd Rajeh Mohd Saleh, 1989. He used Auliciems' neutrality temperature equation to define the comfort temperature of Malaysians and proposed a bioclimatic chart showing the comfort zone for Malaysia based on Olygay's, Arens and Szokolay's (1984) version of the bioclimatic chart. With the width of the comfort zone taken to be 5º C (Szokolay, 1997), thermal comfort temperatures extends approximately about 2.5º C above and below the neutral temperature. Taking the neutral temperature of 26.5ºC in free running building as an illustration, as this research is concentrated on naturally ventilated buildings, the upper limit and lower limit of the comfort zone would then be 29.2ºC and 24.2ºC respectively. This neutral temperature is for conditions without air movement and 50% relative humidity. While Humphrey's equation gives a good approximation of a single comfort temperature in free running buildings, the thermal comfort zone defined using solely this technique does not accommodate the influences of thermal comfort in hot and humid climates. There are 2 methods for accommodating these factors:

- McFarlane (1958) suggested the following adjustments to be made for comfort zones in naturally ventilated buildings for zones less than and greater than 30º latitude : for each 10% increase in relative humidity above 60%, the thermal comfort zone temperatures should be lowered by 0.8ºC, for each 0.15 m/s of air flow past exposed skin up to dry bulb air temperature of 33ºC (mean skin temperature), the thermal comfort zone temperatures should be raised by 0.55ºC.

- Szokolay (1987 and 1997) devised a procedure for plotting the comfort temperature and comfort zone. However, Szokolay's revised version of the effect of air movement on the comfort temperatures are used in the calculation, which allows for the fact there is no cooling effect below 0.25 m/s (Szokolay 1997):

$$dT = 6 \times (v - 0.25) - (v-0.25)2$$

Using this equation, the upper limit of the comfort zone can be extended by providing natural ventilation or some form of air movement. With air movement of 1m/s recommended by most researchers, the upper limit of Malaysia's comfort temperature can be extended by about 4° C to 33.2°C. However, an air flow of 1.5m/s, which is considered still acceptable for hot environment, would be allow the extension of upper temperature limit by 6° C to 35 °C. This cooling effect is very similar to the suggestion given by MacFarlene.

Figure 2.1 shows the proposed minimum air velocity for thermal comfort zone for Malaysia base on a bioclimatic version of the psychometrics chart (Nugroho and Hamdan, 2005).

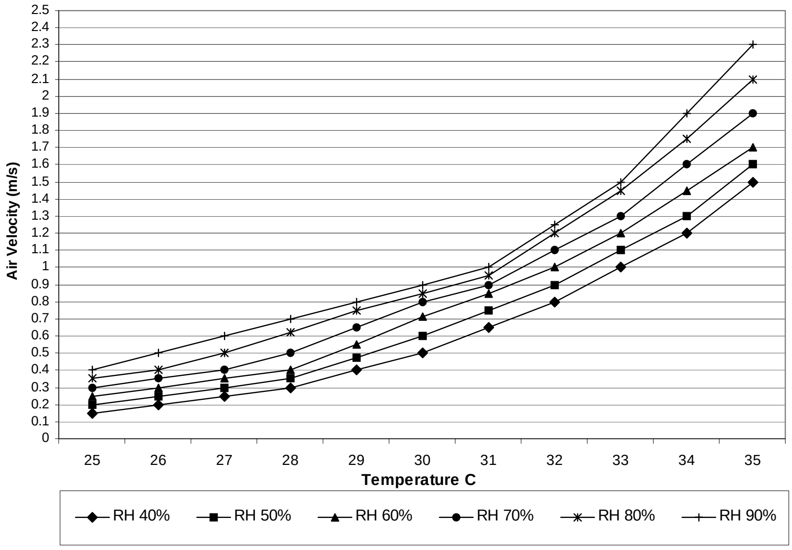


**Figure 2.1:** Minimum Air Velocity Requirements for Thermal Comfort Ventilation

## 2.3 Solar Chimney Ventilation

One of the more promising passive cooling methods for tropical climatic regions is the stack ventilation strategies. Stack ventilation is caused by stack pressure or buoyancy at an opening due to variation in air density as a result of different in temperature across the opening. The same principle can be applied for opening at different heights, the different in pressure between them is due to the vertical gradient (Awbi, 2003). It utilizes solar radiation, which is abundant in these regions, to generate the buoyant flow. However, as currently applied, the induced air movement is insufficient to create physiological cooling. More studies are needed to improve the ventilation performance of this cooling method. Velocities associated with natural convection are relatively small, usually not more than 2 m/s (Mills, 1992).

Stack induced ventilation can be improve by solar induced ventilation. However, in cases where the wind effect is not well captured then solar induced ventilation may be a viable alternative. This strategy relies upon the heating of the building fabric by solar radiation resulting into a greater temperature difference. There are three building element commonly used for this purpose: Trombe Wall, Solar Chimney, and Solar roof (Awbi, 2003).

The first type incorporates glazed element in the wall to absorb solar irradiation into the wall structure. This building has double walls which are combined into a shaft at their upper end. The south facing shaft wall was made from glass. The solar radiation that penetrates the glass heats the inner wall. Eventually, this inner wall heats the air which will rise and induce a flow of fresh air from the openings below (Watson, 1979). Two examples of stack induced ventilation concepts is solar collector and stack height. The former shows one way to amplify stack effect by utilizing solar collectors and increasing the height of the hot air column (stack height). Critical parameters of this design are the stack height and cross sectional area of its inlet and outlet. A massive and high version of this type is needed to generate indoor air velocity as high as 1.0 to 2.0 m/s, which can be achieved easily in an ordinary shallow buildings (with no obstruction at all).

The second form is the solar chimney which has long been known, and applied in vernacular architectural designs. In general, the induced air movement is not used directly to suck indoor air. Instead, it is used for ventilating the building (such as in the double skin building). A stack chimney is usually designed in combination with a wind tower in hot arid climatic regions. In many types of ventilated building, winds are considered to be more important than buoyancy. This is because wind induced ventilation flow is commonly stronger than stack induced flow, in particular, in low rise buildings. A milk house that was built in 1800s is a historical example of a stack chimney application (Satwiko, 1994).

Other method is used in areas with large solar altitude. In this case a large sloping roof is used effectively to collect the solar energy (Awbi, 2004). Another solar roof design called *the Nigerian Solar* Roof was studied by Barozzi (1992) using physical and numerical (Computational Fluid Dynamics Codes) modeling and data from Ife, Nigeria. Two findings were noticeable from this experiment. Firstly, both physical and numerical experiments gave almost identical results. Thus, it showed the potential of CFD Codes to simulate air flow. Secondly, both types of modeling indicated the presence of buoyancy driven ventilation within the model. However, the air speed within the occupant's zone was too low to create physiological cooling. The term *Solar chimney* is used extensively in Barozzi's experiment as the *chimney* shape is quite obvious. In his study the term *Solar Chimney* seems to be more suitable as the chimney takes the form of a roof (Barozzi, 1992).

### 2.3.1 Research of Solar Chimney

In the past decade, solar chimneys have attracted much attention in various investigations. Barrozi et al (1992) modeled a solar chimney-based ventilation system for buildings. The roof of a building performed as a solar chimney to generate air flow and provide cooling for the living room. Experimental tests were carried out on a 1:12 small-scale model of the prototype. Bouchair (1994) showed that for his 1.95 m high and variable width chimney which was electrically heated, the optimum ratio of chimney width/high is 1/10 for maximum air flow rate. If the chimney was too big reverse circulation occurred whereby there was a down-ward flow.

Hirunlabh et al. (1999) studied the performance of a metallic solar wall for natural ventilation of building in Thailand. Theoretical and experimental studies on the natural ventilation of buildings were also carried out by him for four different combinations of height and air gap. Alfonso and Oliveira (2000) compared the behaviour of a solar chimney with a conventional one. They presented a thermal model and transient simulation of a solar chimney by applying a finite difference model to the chimney brick wall assuming unsteady state one-dimensional heat transfer in the direction of the brick wall and not along the flow. Khedari et al (2000) studied the feasibility of using roof and wall to induce ventilation. They showed a significant potential of passive solar ventilation of houses. The Roof Solar Collector could be formed below a heated roof to draw air from the inner spaces of a building. Waewsak et al (2003) carried out another field measurement for the same kind of roof solar collector that outer side of the RSC is replaced by glazing. Ong (2003) has presented a mathematic model and experimental result on a 2 m high solar chimney. He used the matrix method for solving simultaneous equations for heat transfer.

U Drori (2003) studied the induced ventilation on a real-size house based on continuous temperature measurement performed inside and outside the building in summer. Satwiko (2005) found a Solar Wind Generated Roof Ventilation System for low cost dwellings located in high building density urban area. The roof prototype can generate evenly distributed vertical cross ventilation within the occupant’s zone. Bansal (2005) developed a steady sate. Mathematical model for solar chimney system consisting of solar air heater connected to a conventional solar chimney. The estimated effect of the solar chimney was shown to be substantial in promoting natural ventilation for low wind speeds. Table 2.2 gives review on researches of solar chimney ventilation.

**Table 2.2:** Summary of previous research related to solar chimney

| | Climate | | | Applied | | | Configuration | | | | | | Performance | | | | | Tool | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Research | Hot Humid | Hot Arid | Temperate | Wall | Roof | Top of roof | Height | Width | Length | Thickness | Material | Opening | Air velocity | Air flow rate | ACH | Temperature | temp Difference | Full scale | Scale | Simple method | simulation |
| Barrozi, 1992 | * | | | | | * | | | | | * | * | * | | | | | | * | | * |
| Bouchair, 1994 | | | * | * | | | | * | | | | * | | * | | | | | | | * |
| Hirunlabh, 1999 | * | | | * | * | | * | * | | | * | | | * | | | | | * | * | |
| Alfonso, 2000 | | * | | | * | * | * | * | | * | * | | | * | | | | * | | | * |
| Khedari, 2000 | * | | | | * | | | * | * | | | * | * | | * | | * | * | | | |
| Waewsak, 2003 | * | | | | * | | | | | | | | | | * | | * | * | | | |
| KS Ong, 2003 | * | | | * | | | | * | | | * | | * | | | * | | | * | * | |
| U Drori, 2004 | | * | | | * | | | | | | | | | | | * | | * | | | * |
| Satwiko, 2005 | * | | | | * | | | | | | | | * | | | | | | | | * |
| Bansal, 2005 | * | | | * | | | | * | | | * | | * | | | * | | | | * | |

Many researches and experiments on solar chimney have been carried out under different conditions and building element. From review it is found that researches on solar chimney ventilation can be categorize into:

### a. Solar chimney strategy and application

Studies on solar chimney strategies and application have been expressed in terms of conventional solar chimney, roof solar chimney, wall solar chimney and combined solar chimney. Barrozi (1992) and Alfonso (2003) developed solar chimney put on the top of building that can be used to determine the conventional solar chimney. Barrozi (1992) experimented the building prototype in Nigeria consisted of a single chamber. Approximately 5.5 m (length) x 3.5 m (width) x 3.0 m (height) with a corrugated metal roof and fiberboard ceiling were used by him. The experiment was performed with a 1:12 scale model of the prototype, constructed with the same materials where possible. A solar chimney was built into the roof to provide ventilation and cooling. Air flowed in through the windows and was exhausted out through the top of the solar chimney.

Alfonso (2000) indicated the use of solar chimneys in buildings is one way to increment natural ventilation and, as a consequence, to improve indoor air quality. In order to evaluate the contribution of solar chimneys to the natural ventilation of buildings, two chimneys were built, one solar and the other one conventional, each one installed in one compartment of a test cell located in Porto, Portugal.

Khedari (2000), Waewsak (2003), U Drori (2004), and Satwiko (2005) developed a roof solar collector that can be used to determine the roof solar chimney. Khedari's roof solar collector can reduce the mechanical cooling energy cost of new housing built in a hot and humid region, with maximize the natural ventilation and minimize the fraction of sun energy absorbed by a dwelling. The roof solar collector "RSC" used CPAC Monier concrete tiles and gypsum board. Two units of RSC were integrated in the roof structure of the school solar house. The effects of air gap and openings of RSC on the induced air flow rate and thermal comfort were studied experimentally. Waewsak (2003) investigated the performance of a new multi-purpose bio-climatic roof (BCR) developed by The Building Scientific Research Center, Thailand. The innovative functions of this BSRC-BCR are to decrease daily heat gain through the roof fabrics, to induce significant air ventilation rate, which improves the thermal comfort of residents, to ensure appropriate day lighting without any overheating and to act as a roof radiator during nighttime.

U Drori (2004) presented a research experimental induced ventilation of a one-story detached real-size building. The building is a small well-insulated manufactured home. It is located at the Northern edge of the Negev desert in Israel. The flow of air inside the building is induced by a hot element heated by solar radiation. This element is a horizontal metal sheet mounted above the roof of the building and forming a duct connected to the inner space of the building. Heated air flows out of the duct while fresh air is sucked into the manufactured home from the surroundings.

Satwiko (2005) developed of the prototype roof chimney which takes advantages of the abundant solar radiation of hot humid climatic regions to create a buoyancy driven ventilation. This research grows out of a desire to find a Solar-Wind Generated Roof Ventilation System for low-cost dwellings located in high building density urban areas where horizontal air movement is restricted. Solar-Wind Generated Roof Ventilation System was developed as a technique to improve indoor air velocity. It utilizes the available solar radiation and wind to maximize air temperature and wind pressure differences, respectively. A simple residential building type T21 (3x7 m2) located in a high building density area was taken as the case study.

Bouchair (1994), Gan (1997), Hirunlabh (1999), Ong (2003), and Bansal (2005) developed a tromble wall modification to solar collector that can be used to determine the wall solar chimney ventilation. Bouchair (1994) conducted a solar chimney for use in hot arid climates. In this case, the solar chimney walls are made from a high thermal capacity material, allowing a nighttime ventilation strategy to cool occupants within the building. During the daytime, dampers at the entrance and exit of the chimney are closed, allowing the chimney walls to store heat. At night, when the outdoor air is cool, the dampers are opened, and the store heat in the walls is transferred to the air within the chimney, thereby creating a temperature difference between the chimney air and the out doors, and consequent movement of cool air through the building.

Hirunlabh (1999) proposed to utilize a passive system, namely, produced ventilation by a metallic solar wall "MSW” with room side insulation was constructed to investigate heat removal of houses in Bangkok. Room air is removed by ventilation produced by the MSW through openings at the bottom and the top of MSW. The air flow is determined by the inlet area and the square root of the height times the average temperature difference. KS Ong (2003) showed solar induced air ventilation to be provided for by incorporating solar chimneys in building. A solar chimney is one in which one or more walls of a vertical chimney are made transparent by providing glazed wall. Solar energy heats up the air inside the chimney. The solar chimney considered is similar to the Trombe wall except that the wall is assumed to have negligible mass. Bansal (2005) presented the theoretical modeling and experimental validation studies conducted for a small-sized solar chimney, specially having absorber length less than 1m. The additional purpose of conducting this study for small-sized absorber length was to explore the possibility of using small-size window openings as solar chimneys in hot and arid climatic conditions.

### b. Solar chimney and configuration variable

Studies on impact of configuration variable on solar chimney have been expressed in terms of height, length, width, thickness, material and opening. Several studies were concerned with height variable of solar chimney design ventilation. Hirunlabh (1999) found the temperatures increased with increased Metal Solar Wall (MSW) heights. In addition, temperature along the metallic wall height is at a maximum at the middle of the MSW and at minimum near the opening due to the incoming room air at the bottom of the MSW and to the contact with ambient air a the top. Experimental investigations of the performance of the MSW showed that with 1 m height and 03.4 cm gap the MSW would produce optimum natural ventilation. Alfonso (2000) evaluated height parameters of solar chimney for satisfying the needed average flow rate. To evaluate the effect of chimney height on ventilation flow rate, several dimensions of chimney height between 0.5 and 3 m were simulated. Air flow rate increases for higher values of high (H). The Amplitude in air flow rate (differences with between daily maximum and minimum) increases with chimney height. Khedari (1998) showed the impact of length variable for solar chimney ventilation. The air velocity could be increased by increasing the surface area of solar chimney or the number of units of solar chimneys on roof, eastern and western walls.

Several works (Bouchair, 1994; Hirunlabh, 1999; Alfonso, 2000; Khedari, 2000 and Ong, 2003) considering the width solar chimney variable was carried out. Bouchair (1994) completed an important study of a solar chimney for cooling ventilation. The chimney had a fixed height of 2 m with both walls maintained at the same temperature. The influence of the chimney width was investigated. The effects of chimney width on the mass flow rate for inlet heights of 0.1 m and 0.4 m with the chimney wall temperature in the range of 30 to 60 C. The ambient air temperature was maintained constant at 20C. It was found that in the chimney width range 0.1 m – 1 m, there was an optimum chimney width between 0.2 and 0.3 m, which gives maximum ventilation rate. It was also found that this optimum chimney gap is essentially independent on the chimney wall temperature. However, it may be seen that the optimum gap is slightly wider when the chimney inlet becomes higher from 0.1m to 0.4m. The optimum chimney width was approximately one-tenth of the chimney height, or an aspect ratio (H/w) of 10.

Hirunlabh (1999) investigated the relation between air gap and air mass rate in metallic solar wall. The hourly variations of the experimental air mass flow rate produces by the MSW for two different gaps (10-14.5 cm). It can be seen that the mass flow rate increased with increased gap. The maximum average of air mass flow rate during the hot period (10:00-16:00 h) was about 0.015 kg/s. Alfonso (2000) changed the width of solar chimney to evaluate the effect of air flow rate in the room. Variation of chimney width (W) between 1 and 5 m, which correspond to chimney sections between 0.2 and 1 m2 were used. Airflow rate increases linearly with the increase in W. Also, the amplitude in air flow rate increases with chimney width. One can also conclude that the increase in chimney width is more effective than the increase in chimney height, when one seeks to increase solar collection area in order to favour air flow rate.

Khedari (2000) developed two units of Roof Solar Collector, tilted at 25°, and were integrated into the south-facing roof of the single side room solar house model. Their design allows us to test different air gap 8 and 14 cm. The variation of induced air flow rate by RSC with two different air gap spacing was included. It was found that with 14 cm air gap RSC1, the air flow rate is higher than that induced with 8 cm air gap RSC4. Therefore, large air gap is recommended. Ong (2003) investigated the effects of air gap and solar radiation intensity on the performance of different chimneys. In order to verify the theoretical model, experiments were conducted on a 2 m high physical model with air gaps of 0.1, 0.2 and 0.3 m. A solar chimney with a 0.3 m air gap was able to provide 56% more ventilation than one with a 0.1 m air gap. Bansal (2005) investigated the impact of air gap for entry of air in the chimney. An experimental setup was developed as a cubical wooden chamber having a size of 1m x 1m x 1m was made. One of its vertical sides was made like a sliding shutter. This shutter when lifted made opening for the exit of air from the room for its entry into the solar chimney.

In another study by Alfonso (2000) he analyzed the effect of the thickness variable of solar chimney by performing simulations for thicknesses of 5, 10, 15 and 20 cm. Thickness does not change significantly the average flow rates, the maximum being obtained for 10 cm. However, air flow distribution along time has some differences. A higher thickness allows more energy storage in the wall, decreasing the flow rate during sunshine hours and increasing it during the night period. However, thicknesses above 10 cm do not produce a significant difference in the flow rate time distribution. To investigate the effect of insulation thickness on ventilation flow rate, new simulations for a thickness of 0 and 5 cm were performed. The use of insulation increases both daily and night flow rates. A thickness of 5 cm gives results that are similar to the ones of an ideal insulation. Therefore, there is no need of a much higher thickness, for the climatic conditions considered.

The impact of material variable for solar chimney was done by Barrozi (1992), Hirunlabh (1999), Alfonso (2000), Ong (2003) and Bansal (2005. In Barrozi studies (1992), the solar chimney is made of corrugated metal, which has low thermal storage capacity, allowing ventilation to begin as soon as the sun adequately heats the metal. Therefore, this system is best suited to daytime ventilation, it having inadequate thermal capacity to maintain ventilation night. Hirunlabh (1999) developed the Metallic Solar Wall consists of a glass cover, air gap, black metallic plate and insulator made of micro fiber and plywood. The MSW was integrated to the south wall of the house. The MSW was 1 m wide and 1 m high, which is obvious as more radiation was absorbed by MSW. Alfonso (2000) compared between a conventional chimney and a modification of solar chimney. The modification of solar chimney is similar to conventional chimneys except that the south wall is replaced by a glazing. In order to perform experimental tests, an existing test cell was modified. The walls, ceiling and slab are made of concrete, with outside insulation. It is fundamental to use outside insulation in the brick wall, to take advantage of solar gains; if outside insulation is not used, solar assistance efficiency reduces by more than 60%; a thickness of 5 cm is sufficient.

Studies under Ong (2003) has one side of the chimney provided with a glass cover and the other three walls of the chimney form a channel through which the heated air could rise by natural convection. An opening at the bottom of the wall allows room air to enter the channel. The exposed surface of the heat absorbing wall is painted black to increase solar radiation absorption. The heat absorbing wall was made of pre-laminated polyurethane sheet, similar to the other solid walls. A 50 mm thick polystyrene sheet was attached to the back of the heat absorbing wall to provide additional heat insulation. The glass cover and the heat absorbing wall provided an air gap through which air could flow.

Bansal (2005) used on the exposed side of the shutter of wall solar chimney with a 1mm thick aluminum sheet was attached with the help of glue and nails to act as a solar radiation absorber for the setup. Aluminum sheet was painted black with ordinary paint. On the same side, a telescopic frame of matching size was made with a provision of sliding on rollers. The face of this parallel to the sliding shutter was made with float glass. The vertical side of this telescopic part (parallel to the shutter) was made of 4 mm thick float glass. On the vertical face opposite the absorber, an opening was provided for suction of air from the surroundings. All the sides, exposed to ambient, were insulated using 2.5-cm-thick thermocol (EPS) sheets, for prevention of heat transfer. Similarly, on the backside of the absorber that faces inside of the chamber, 5 cm insulation was provided for prevention of flow of heat in this direction. It can be concluded that in hot climatic conditions, when windows are kept closed/ covered for preventing the entry of solar heat, the concept of solar chimney can be utilized by making minor modifications in the existing window design.

Studies about solar chimney opening variables done by several researcher (Barrozi, 1992; Bouchair, 1994; Khedari, 2000). Barrozi (1992) used different window arrangements were obtained by maneuvering the light wooden wall panels, thus allowing the effect of different geometries on the airflow to be examined. A chimney vent is provided at the side of the solar chimney, allowing the exhaust of air. Bouchair (1994) concluded the impact of chimney inlet height (and thus inlet area) for solar chimney performances.The effect of the inlet height on the mass flow rate at two inlet heights: 0.1m and 0.4 m for chimney width in the range of 0.1m to 0.5 m, that at a chimney width of 0.1 m, the mass flow rates were similar at both inlet heights. This suggests that the inlet height at this chimney width had little influence on the ventilation flow rate and the friction loss in the chimney dominated the total pressure loss in the system. However, it can be seen that with an increase in width, friction loss in the chimney was reduced and the influence of the chimney inlet height became more and more significant. Consequently, the friction loss at the inlet became the major contribution to the total pressure loss and determined the mass flow rate through the whole system at the large chimney widths.

Khedari (2000) investigated different free opening vents. Its dimensions are as follows: 2.68 m height, 3.35 m width and 3.45 m length. The solar house has a window and a door with a grill on the northern side. The variation of induced air flow rate by Roof Solar Collector for the different opening vents RSC1, RSC2 and RSC3. It was found that with equal and larger size of free inlet–outlet openings, configuration RSC1, the induced air flow rate was the highest one. Therefore, opening vents have to be of equal size and as large as possible. A higher number of Air Change Rate (ACH) could be done by increasing the surface area of RSC, i.e., increasing the number of RSC units and making use of walls to act similarly. Opening the window and door is less efficient than using solar chimneys, as temperature difference between rooms and ambient was higher than that obtained with solar chimneys.

### c. Solar chimney strategy and performances parameters

The relation between solar chimney and air velocity performance carried out by Barrozi (1992), Khedari (2000), Ong (2003), Satwiko (2005) and Bansal (2005). Barrozi (1992) illustrated the air velocities inside the building were found to be generally very low suggesting that the cooling effect may be less than required. Large flow velocities were predicted along the inclined roof surfaces due to high buoyancy forces. The results showed that the flow pattern in the room was very sensitive to the window geometry, the air flowed in directly along a path from the window inlet to the ceiling opening; while for window geometry increased, the air flowed from the inlet almost vertically along the wall to the ceiling and then through the ceiling opening. Khedari (2000) indicated that the average air velocity along horizontal planes increased with increasing vertical height. Thus, at the living level, around 1 m above floor, there is continuous air motion induced by the buoyancy driven force resulting from the four solar chimney ventilators used here. Regarding thermal comfort, the induced air motion of about 0.04 m/s cannot satisfy occupants as with temperature of about 35–37, a higher air velocity is needed. This air motion could be increased by increasing the number of units of solar chimneys on roof, eastern and western walls and by installing several free openings at the northern facade of room. Ong (2003) showed the higher air velocity through the large air gap of solar chimney, causing greater cooling effects on the heat absorbing wall and glass which then tended to be lower. Air velocity increased with solar radiation and larger air gaps. Air velocities between 0.25 m/s and 0.39 m/s for radiation intensity up to 650 W m were obtained. No reverse air flow circulation was observed even at the large gap of 0.3 m. It could be seen that the solar chimney induced air flow rates of between 0.25 m s and 0.39 m s at air gaps between 0.1 m and 0.3 m, respectively at 650 W/m. Satwiko (2005) found The Solar Wind Generated Roof Ventilation System could generate an indoor air speed range from 0.15 to 0.7 m/s, with all windows closed, and applying the wind speed of 3.53 m/s at 10 m above the ground, combined with solar radiation of 540 W/m2. When the wind was calm (0 m/s), the heat from absorbed solar radiation can create an indoor air velocity of 0.4 m/s. Obtains thermal comfort through physiological cooling. Increasing air movement is the most logical way to passively achieve thermal comfort in a hot humid climate. Bansal (2005) analyzed the small size solar chimney has opened possibilities of utilizing windows as solar chimneys since the flow velocity up to 0.24 m/s has been experimentally recorded. For the three combinations that were experimentally investigated, the highest flow velocity in the chimney was found to be 0.24 m/s.

Several studies about the air mass flow rate performance in solar chimney have been done by Bouchair (1994), Hirunlabh (1999) and Alfonso (2000). In Bouchair’s experiments (1994), a mass balances investigation was also carried out for the mass flow rate through the window (room, inlet), the chimney inlet and the chimney outlet. However, at a chimney width of 0.5m, the measured mass flow rate at the chimney outlet was higher than that through the window and the chimney inlet. Hirunlabh (1999) found that the Metalic Solar Wall with 14.5 cm air gap and 2 m2 of surface area (HxW: 2x1m) produced the highest air mass flow rate of about 0.01-0.02 kg/s. Alfonso (2000) concluded a significant increase in ventilation rate with solar chimneys. Average flow rates for a solar chimney are always higher than the ones for a conventional chimney.

Studies on air change rate performance expressed by Khedari (2000) and Waewsak (2003). Khedari (2000) illustrated the units of Roof Solar Collector produce the average number of induced air change is about 4–5 ACH, which is not sufficient to satisfy complete resident thermal comfort. A higher number of ACH, depending on season, up to 20 is required for houses without any mechanical cooling device. Waewsak (2003) had done a field measurements of the thermal and lighting performance of the BSRC-BCR demonstrated that this innovative bio-climatic roof could reduce heat gain significantly and induce significant air change. With these characteristics, it is expected that the bio-climatic roof can achieve thermal comfort without any mechanical devices during a long period annually where outdoor temperature is not excessive. The rate of induced air change depends mainly on the intensity of solar radiation. The corresponding induced ACH by BSRC-BCR was quite high during the day varying between 10–15 in summer and 5– 8 in the winter. Obviously, this high air change rate induced by BSRC-BCR is due to the fact that more heat is admitted through the transparent tiles. Therefore, the BSRC-BCR can improve indoor thermal comfort considerably. To satisfy resident comfort for different activities and non-air conditioned spaces, a higher ACH is required (about 20–25 ACH). This could be achieved easily by increasing the number of units of BSRC-BCR.

Studies on the temperature parameters on solar chimney performance were done by Hirunlabh (1999), Ong (2003), U Drori (2004) and Bansal (2005). Hirunlabh (1999) illustrated the room temperature during tests was near to ambient air, ensuring human comfort resulting from ventilation produced by Metallic Solar Wall. The MSW can reduce significantly heat gain in house. Ong (2003) evaluated the prediction of the temperatures of the glass glazing and the heat-absorbing wall and also the temperature and velocity of the induced air flow in the chimney. Mean wall temperatures were always higher than mean glass or mean air temperatures. Mean air temperatures were always lower than mean glass temperatures for the range of incident solar radiations encountered. Mean air temperatures and temperature rise increased with solar radiation. It was also found that mean temperatures and temperature rise decreased with air gap depth. U Drori (2004) showed the experimental and numerical results that the air temperature inside the structure typically followed the ambient temperature, indicating that effective induced ventilation has been achieved in the modified structure, while in the closed manufactured home the air temperature was considerably higher than the ambient, especially during the afternoon hours. The results show that effective induced ventilation has been achieved, and the air temperature inside the structure typically followed the ambient temperature. In contrast, when the manufactured home had no openings, the temperature inside it was considerably higher than the ambient, especially during the afternoon hours.

Bansal (2005) indicated the calculated temperature of glass and air in the flow channel tends to be lower than their respective measured values of the small size solar chimney. It has been found that the developed theoretical model slightly under-predicts the temperatures of glass and air in the flow channel. Nevertheless, it can be concluded that the model can predict the performance of such systems well and this approach is even applicable to small-sized solar chimneys. More heat loss from the absorber reduces its temperature and increases the temperature of air. Due to this increase, the temperature difference between glass and air is reducing; hence the heat transfer from glass to air also gets reduced.

Studies on solar chimney and temperature differences were carried out by Khedari (2000) and Waewsak (2003). Khedari (2000) showed the use of Roof Solar Collector to reduce the rate of heat transferred through a ceiling by inducing natural ventilation. The temperature of CPAC Monier, air and gypsum board is changing along with the intensity of solar radiation. When the solar chimney ventilation system was in use, room temperature was near that of the ambient air, indicating a good ability of the solar chimney to reduce house’s heat gain and ensuring thermal comfort. Waewsak (2003) showed the variations of the temperature differences (TD) index for different days in summer and winter, respectively. It can be seen that the TD index of Bio Climatic Roof (BCR) was almost always below or close to neutral (zero) particularly during the hottest midday period (10:00 a.m.–3:00 p.m.). This means that indoor air temperature is distinctly lower than outdoor and BCR has good potential for decreasing heat accumulation and inducing a high ventilation rate. When compared to RSC, BCR performs much better as the daytime index is smaller than that of RSC. It should be pointed out that the TD index is higher than zero in the afternoon (after 4 p.m.) due to the heat gain through the non-insulated west wall (common masonry wall). During night time, both BCR and RSC perform similarly as good radiative coolers. From these results, it can be concluded that the BCR roof design could decrease heat accumulation efficiently year round.

### d. Solar chimney and research tool

Alfonso (2000) showed the use of thermal model can predicts with good accuracy measurements. Originally, it was a one-zone test cell, with a floor area of 12 m2 4x3 m. The experiment used full scale two rooms were equipped with a heating facility, with a precise control of the inside temperature. The exhaust chimneys were fully instrumented with anemometers, thermocouples and flux meters. In this work, results of air flow rate measurements in both compartments with a tracer gas technique are shown, as well as their comparison with theoretical values obtained with a simulation program specially developed for this purpose. A tracer gas technique was used for measurement of air flow rates of both compartments of the test cell, because it enables the evaluation of air exchange rates of buildings in real time. Khedari (2000) introduced a full scale experimental method to measure the air temperature and the air velocity at different points on the Roof Solar Collector units and in the room. The study was conducted using a single-room house of approximately 25 m3 volume. The roof was made by using CPAC Monier concrete tile dark red color. The floor was plywood on grade. The surface area of the RSC unit is considered equal to 1.5 m2 LxW: 1.5x1 m2. The outer side was made by CPAC Monier concrete tiles (33x42x1.5 cm3 ; 4.4 kg/piece ; dark red color while the inner one was made of gypsum board 100x150x0.9 cm3; 1.7 kg). Experimentation started at 8:00 a.m. and ended at 5:00 p.m. by recording data at 30 min intervals.

Waewsak (2003) compared a conventional Khedari's Roof Solar Collector (RSC) with Bio Climatic Roof (BCR). The RSC is composed of CPAC Monier tiles at the outer side, 0.14 m air gap and gypsum board at the inner one. Both units of 1:5 m2 surface area were each integrated into the south facing roof of a single-room solar house of 25 m3 volume. White plywood boards were used to separate the interior space under the two units (BCR and RSC) to make two separate compartments "rooms". Data were recorded every 30 min during 24 h. The experiments were carried out during different days in winter (December 1999) and summer (March–April 2000). U Drori (2004) experimented with a manufactured home a one-story detached real-size building has been studied experimentally and numerically. An extensive experimental study was based on continuous temperature monitoring performed inside and outside the building in summer. Velocity measurements inside the building were performed, as well. Typical experimental runs took from 12 h to a few days. An extensive experimental study included temperature and velocity measurements both in the modified open and the original closed structure, with typical experimental runs from at least one full day to several days long.

Barrozi (1992) developed solar chimney ventilation using a scale model and the result were used to validate a two dimensional, CFD simulation model. The experiment was performed in a sealed laboratory chamber for the conditions of overhead sun and no wind, the scenario placing highest demand upon the ventilation system. Climatic parameters were limited to the ambient outdoor temperature and solar radiation. Relative humidity was assumed to be constant. Several series of tests were taken to asses the reliability and repeatability of their experimental data. A simple solar simulator consisting of four Osram Ultra Vitalux lamps was places above the model to heat its room. Once a steady state condition was achieved, A PC data acquisition system recorded converted data from temperature and velocity sensors. Velocity measurements were limited to the solar chimney outlet region as velocities at other sections were too low to allow accurate measurement. Flow visualization was achieved by injecting smoke near the inlet (window) and recording the smoke trails on film.

Hirunlabh (1999) used a small model of a solar house with 1.57 m height and base area of 2.24x2.34 m. It had one window and one door with an air grille on the north side. A set of type-K thermocouples was used to measure the temperature at several points of the Metallic Solar Wall and at six points inside the room. A hot wire anemometer was used to measure the air velocity at the outlet and inlet vents at several points and an average was calculated. Campbell, portable hybrid and data logger recorder were used to record temperatures. A propeller type anemometer was used to measure the velocity and temperature of ambient air. Studies by Ong (2003) with scale experimental models were carried out outdoors on the roof and the experimental model exposed to both direct and diffuse solar radiation. The solar chimney was oriented facing south and exposed to solar radiation.

Mathematic studies on wall solar chimney use were investigated by using Metal Solar Wall “MSW” done by Hirunlabh (2001). Comparison between numerical study and experimental results showed a good agreement. Therefore the numerical model is valid. Thus, it can be used to evaluate the long term reduction of heat transfer into the habitation. KS Ong (2003) used a mathematical model to simulate the thermal performance of a solar chimney by predicting the surface temperatures of the wall and glass cover and the temperature and induced natural convection flow rate of the air stream in the chimney. More satisfactory qualitative agreement was obtained between experimental and theoretical results. The difference between experimental and predicted instantaneous efficiency values was about 10% in all cases.

Bansal (2005) carried out to investigate the mathematical model for predicting airflow velocity in a solar chimney has been developed through predicting temperature of the absorber, air in flow channel and glass cover. Experimental validation of the model has been done using a solar chimney having less than a 1-m-high absorber. Good agreement between observed and calculated results has been obtained. A mathematical model of the experimental setup was developed for estimating the performance of the solar chimney under different ambient conditions. Validation of the theoretical model with the experimental results was carried out in two steps. The first step was the validation of temperatures at various points and the second one was the validation of the airflow rate.

Barrozi (1992) completed an experimental research with numerical study of a building with a solar chimney. For the numerical simulation, the following simplifications were made in their work: the problem was two dimensional, radiation inside the building was negligible, the flow was steady with laminar flow. Alfonso (2000) tried to obtain Computational Fluid Dynamic based models predict accurately air velocity distribution in the solar chimney, but assuming very simple wall surface conditions: typically, a constant surface temperature and a constant air heat transfer coefficient. The model combines the equations for heat transfer processes in the solar chimney with the equations for natural ventilation flow. However, a constant value leads to calculated flow rates that are within 10% of the measured values. Therefore, the model offers enough confidence to allow its use for different situations and geometries. In another study by U Drori (2004) with numerical studies were performed using the Fluent 4.52 CFD software. Their purpose was to exemplify the flow field typical to the system investigated during the day hours. The temperatures of the heated metal sheet and the ambient air were almost constant for 4–5 h about noon. Since these temperatures determine the system performance, it was assumed that steady ventilation of the structure could be achieved. For this reason, steady-state simulations were performed at this stage. Transient simulations would have required much more computing resources, and will be performed in the future. The research by Satwiko (2005) used computer simulation techniques to calculate and analyze the aerodynamic and thermal performances of SiVATAS. This included the aerodynamics and thermal consequences for any changes in its form and material. A general purpose computational fluid dynamics (CFD-ACE+) program was utilised to explore, analyse and develop a roof model based on its aerodynamics and thermal performance to obtain optimum wind pressure and temperature differences.

Comparisons were made with physical scale models. The CFD program requires inputs representing problem type, flow domain (material, type of flow, etc.), boundary conditions (walls, inlet, outlet, symmetric wall), and calculation method. The problem type is used to activate calculation modules; in this case, Flow, Heat Transfer, and Turbulence modules.

### 2.3.2 Relevant Past Research of Solar Chimney

Previous sections reviewed the literature about solar chimney in order to get a clear understanding of the state-of–art knowledge in the field and identify the areas which had not been covered in the past. The review revealed that research on solar chimney had been focused mainly on four issues: solar chimney strategy and application, impact configuration variable, impact performance parameters, and solar chimney research tool. Many studies in hot humid tropical climate suggested that possibility the use of solar chimney to increase stack ventilation in the building but still not enough for thermal ventilation requirement. Also, little studies are applied of combination of solar chimney type in the building to increase thermal ventilation performance.

Though it was mentioned in many research works and publication that solar chimney configuration variable to increase solar chimney performance, there are many research done but only little research were done on related solar chimney configuration with room configuration. Review also suggests that solar chimney have a significant impact on the thermal ventilation specially for psychological cooling. Few studies have looked into this aspect under different thermal comfort ventilation standard, and with different solar chimney strategies. Further they do not produce holistic performance result for any particular climate and thermal comfort standard. Generally, research tool in solar chimney studies used experiment model, but many numeric and computational method was validation. Validation result shows good agreement with real measurement. However, little research is done by computational tool and no validation done with many experimental models.

Thus, the above reviews suggest that solar chimney strategy on increase air velocity and decrease air temperature has been dealt as separate issues. There is no specific research done to study the relationship between optimal solar chimney configuration and psychological cooling performance. Therefore this next research attempts to focus on the solar chimney configuration to improve stack ventilation for thermal ventilation in tropical condition.

## 2.4 Computational Fluid Dynamic Simulation

The general tools will normally be computationally less efficient than the specialized tools replaced; almost universally this drawback will be outweighed by the gain in man time efficiency. It is expensive to carry out full-scale experiments on natural ventilation. Scale-model studies are useful but scaling parameters have to be identified. Numerical simulations of natural ventilation using Computational Fluid Dynamics (CFD) appeared in the literature, being another possible way of studying the air flow. According W.K. Chow (2004), this technique has been developed and applied for design purposes in the past two decades and is believed to have good potential for studying ventilation, air-conditioning and fire-induced flows in building services engineering. Good reasons for using CFD are (Chow, 2004):

- Graphical presentation of the geometry, temperature fields and velocity vectors. The predicted ‘microscopic’ picture of the thermal environment described by the velocity vector diagram, the temperature and pressure contours are useful for deriving the relevant macroscopic parameters for engineering purposes.
- The time required to get the predicted results is now highly shortened because of the development of efficient computing schemes and high-speed computers.
- Distribution of the pressure coefficients over the building wall can be calculated as demonstrated by Smith et al.

However, there are at least three points of concern in using CFD:

- Turbulence: turbulence models have to be used for solving the set of air-flow equations for practical engineering problems. There are debates on using Reynolds Averaging Navier–Strokes equation (RANS) or large-eddies simulation (LES) . However, empirical parameters appeared in both types of models. The solid-wall boundary conditions have to be considered carefully and whether wall functions of the RANS model can give better results should be investigated. In-depth physical justification for simulating building aerodynamics is required
- Numerical method in solving the set of partial-differential equations: finite difference control volume method is commonly used, though the finite element method appears in the literature.
- Pressure equation: there is no explicit equation on air pressure. Algorithms for solving the set of pressure-velocity linked equations have to be developed

CFD simulation is a technique widely used in studies on wind-induced airflows in buildings. This turbulent flow field is governed by the principles of conservation of mass and momentum. It is recognized that accurate modeling of the wind driven airflow is important to assess the pressure distribution at the surfaces of a building. Extensive work has been done in applying CFD to airflows in and around buildings, by using the k-turbulence modeling method and wind tunnel approach to tackle similar problems (Chow, 2004).

G. Gan (1998) study is to apply the technique of computational fluid dynamics (CFD) to simulating air flow and heat transfer in the proposed solar chimney for heat recovery in naturally-ventilated buildings. A CFD program was first validated against experimental data from the literature for an isothermally heated chimney. Numerical investigation was then carried out into the performance of the glazed solar chimney and good agreement between the prediction and measurement was achieved (Gan, 1998). Comparisons of CFD simulations with wind tunnel experiments found a good agreement between numerical and experimental results. However, Murakami et al. showed some limitations of the k model when compared with rigorous laboratory experiments. Some assumptions used in the k model, suchas isotropic turbulence, are not strictly applicable to flows in and around buildings. In addition, Murakami et al. found that results from laboratory experiments and a more precise turbulence modeling technique, known as LES, were in good agreement. Since LES demands high computing time at present, in this investigation, the CFD program used the k turbulence model. As wind tunnel experiments with high-rise building of this special shape do not exist, we verified the results obtained by the k-turbulence model with the results obtained by the LES turbulence model, as presented in the section on results and analyses. According M.M Sanchez (2002) the assumption of 2-D motion in the solar chimney can be regarded as accurate enough due to geometrical symmetry of tilted channels but it is longer valid in the evaporation zone where 3-D effects cannot be neglected. Heat transfer from plates to air proceeds in a boundary layer regime and makes air lift forces appear. CFD runs are considered as virtual tests where the results allow us to obtain information by means of dimensional analysis and similarity theory, as lab test results do. Therefore, numerical results have been used to achieve important design tools like heat transfer and air mass flow rate correlations (Sanchez, 2002).

# CHAPTER 3

# METHODOLOGY

This research is divided into three main stages. First, is the method of stack induced ventilation; second, CFD simulation; and lastly third, development of simplified model of the Solar Chimney. The methodologies planned for the research are described in this chapter. These methodologies were reviewed from selected literatures and redefined specifically for the purpose of this research.

## 3.1 Method of Solar Induced Ventilation

The solar induced ventilation is buoyancy-driven by the use of a solar air collector and therefore, all equations derived early for buoyancy pressure and flow rate through large opening also apply here.

The main aims of this research are: to estimate the internal temperature difference caused by solar induced at solar chimney model and to predict the internal air velocity of the proposed solar chimney. Consequently, this will determine the performance and effectiveness of the solar induced ventilation at proposed solar chimney in achieving thermal comfort ventilation. Therefore, the appropriate methodology to be used is the two steps method. The first step is to estimate the temperature difference over the internal solar chimney model. The second step is to apply the temperature difference obtained from the sealed building to the openings using flow rate methods. This method is adopted from Awbi (2002).

Various models and tools have been invented to fulfill the different needs in natural ventilation study. Models and methods ranges from a very simple empirical algorithm to calculate airflow rate to a sophisticated computational fluid dynamic (CFD) techniques, in solving the Navier-Stokes equation. Yogou Li (1994) classified the available methods to estimate air flow into three categories. They are analytical, full scale and CFD. Heerwage *et. al.* (2000) classified the available methods into four categories which are field measurement, physical modeling, computer simulation and simplified methods. Wang (1996) categorised the main method for studying natural ventilation due to temperature difference through two main methods. They are physical and virtual experiment methods. The physical experimental method includes scale measurements or field experiment. Meanwhile, virtual experimental method is referred to as CFD modeling base on numerical equation.

### 3.1.1 The Physical Experimental Method

Full-scale measurement can be used to judge the actual performance of a building, a system or a technology. In natural stack ventilation study a scale or field measurement is intended to measure the profile of the wind approaching the site, the temperature distribution and the vectors of wind (Wang, 1996). These techniques can be applied either to an existing or new buildings and to estimate the temperature difference. According to Wouters, *et. a!.* (1998) full scale measurement is the best way to obtain better understanding and evaluate the building performance and its function. It is also the best technique, because it can provide actual experimental condition, this can be used to compare the agreement between the predicted performance and the actual performance.

However, the procedures are very complex and involve comprehensive experimental methodology, experiment equipments, cost, manpower and man-hours. Moreover, the results can be applied only to a specific site and conditions (Wang, 1996). This will limit the contribution of the findings. Due to these reasons, the proposed research is not viable to be carried out using this method.

### a. The Physical Experimental Modeling

When considering the free convection, the Grashof number is always used, which is the ratio of the buoyancy to viscous force.

$$\mathrm{Gr} = g\beta\Delta T l^3 / \upsilon^2$$

where, g is the acceleration due to gravity (m/s2); β the thermal expansion coefficient (1/K); $\Delta T$ the temperature rise (K); $l$ the characteristic length (m); and $\upsilon$ is the viscosity (m2/s). Actually, it is almost impossible to preserve the Grashof number in the reduced scale model and the full-scale building. However, it is found that even for the free convection, as long as the turbulent intensity of the flow is over some value of the Grashof number, the basic characteristic of the flow becomes independent on the Grashof number. For the natural ventilated space, most of the flow region can be regarded as such state (Takashi, 1981). Therefore, substituting the turbulent viscosity $\upsilon_t$ (it is proportional to the product of the local velocity $U$ and length $l$ for the Grashof number,

$$\upsilon_t \ \alpha \ Ul$$

$$(\mathrm{Gr})_t \ \alpha \ g\beta\Delta T l^3 / U^2$$

Considering $\Delta T_M = \Delta T_F$, then

$$U_M / U_F = (l_M / l_F)^{1/2}$$

where, M indicates model and F indicates full-scale. Basically, the Grashof modelling mentioned above can be used to establish relations between the reduced scale model and the full-scale prototype. (Ding, 2004)

### b. The Physical Experimental conditions

During the experimental testing some facts were discovered. These facts could be considered as disadvantages (e.g. the existence of air movement around the model which could confuse the measurement) and advantages (e.g. potential of the model to be improved). They led to the need for a clear experimental procedure and a more flexible design of the models. The former was used to anticipate and minimize the influence of the environmental conditions at the experiment location which could result in a biased experimental result. The latter made it possible to re-configure the model so that an optimum ventilation performance could be obtained (Satwiko, 1994). To realize preferable airflow throughout the building, setting of openings should be reasonably planned. To assess the natural airflow conditions throughout the model, temperature and pressure difference distribution are measured. At the same time, incense sticks are used to judge the airflow direction near the openings. Arrangements of the testing points (instruments) and connected with a data collector to record the data every second (Ding, 2004).

### 3.1.2 The Virtual Experimental Method

Compare to studies using physical models, the results of studies using CFD codes (numerical methods to simulate real conditions) have been regarded with skepticism. As it is a virtual simulation, the CFD codes have been viewed as producing unrealistic results. It has been difficult for these codes to be accepted as imitating real conditions. CFD was originally developed by L.F. Richardson in 1917. Its codes consist of equations, known as Navier-Stokes equations, which are highly non-linear and cannot be solved using simple algebraic methods. As a result, this method has only recently become fully developed, thanks to the new generation of personal computers, which are more powerful, faster, and more economical (on time and labour). These new computers enable experts to solve Navier-Stokes equations iteratively. CFD methods have several advantages over traditional, analytical and physical model measurement methods. A CFD model, compared to a full-scale physical model, is easier to set up, less costly, taking less time to run, while simulating boundary layer wind profiles with greater accuracy and ease (having no building size limitations); it is easier to investigate, parametrically, changes in the building's design, and provides valuable output (eg. detailed air flow patterns around the building). Analytical methods have suffered from severe simplification of assumptions and simplistic designs. Though full-scale physical model measurements provide the most reliable data, they are expensive and difficult (mostly impossible) to do. Nielsen, in 1974, was the first person who used CFD codes to study air motion in rooms. Since then, experts from various countries have vigorously conducted research on the application of their CFD codes to building environmental studies. Among them are Murakami (Japan)

### a. The Virtual Experimental Modeling

In experiments using Virtual Codes, the pre-processing (or preparation phase, i.e. transforming the real model into integer space representation, preparing the grid, etc.) consumes most of the time required. Compared to pre-processing, the calculation process consumes less time. However, once started, the processing step continues to completion with no interaction possible, if for example, the calculation showed a difficulty in convergence. The pre-processing, on the other hand, can be run in an interactive mode which gives two advantages: instant verification of the validity of the command (syntax and logic) and the flexibility to terminate and resume at anytime. The same sequence as in the physical experiment can be illustrated when the model configurations were analyzed using virtual experiment. Unlike in the experiment using physical models, in the experiment using a computer program, the geometry of the model can easily changed by redefining the position of points. Transforming the model with grid strategy is the division of the flow domain into small parts as required by the finite elements method as the basis of CFD calculation. The form of the grid is dictated by the geometry of the flow domain.

Grid strategy is important to achieve an optimal balance (or composition) between the validity of result and the efficiency of the calculation. Small cells and an overall fine grid means that the accuracy of solution is much greater, but costs more in calculation time. In practice, not all parts of the flow domain need to be divided into fine grids. Only in areas where high fluctuations of variables occur the fine grid is required. In general, non uniform grids are preferable to uniform ones. In non uniform grid, a fine grid is applied at locations such as in and around diffusers, heat sources, obstacles, and walls. Hence, intuition or experience is needed to predict the likely air flow pattern so that the finer grids can be located at the right places (Satwiko, 1994). Additional grid points are embedded near the walls, around the openings to enable better resolution in these areas (Ding, 2004).

In the virtual experiment the air flows were assumed to be laminar, the Rayleigh number could be taken anywhere between $10^3$ and $10^6$. When the preliminary tests found that taking $Ra = 10^6$ had resulted in a convergence difficulty, then we can use two solutions. Firstly, a special solution technique called *acceleration factor could* be applied. Secondly, smaller *Ra* numbers could be used. If the preliminary tests found that taking $Ra = 10^s$ could solve the problem (a convergence could be achieved in 4 to 5 iterations) and no acceleration factor was needed. Thus, the reasons why a Rayleigh of $10^5$ was chosen there was a problem of convergence of the CFD calculation if a higher Ra# than $10^s$ was used. A *Ra* $10^5$ was considered tolerable since it is close to the transitional flow range *(Ra* $10^6$ to $10^9$). With this number, a convergence problem could be avoided and a reasonable result could be achieved (Satwiko, 1994).

## b. The Virtual Experimental Condition

The virtual experimental conditions data specify the constant value of certain nodes. These data, together with the grid strategy, form the logical boundaries of the flow domain. Velocity boundary condition data, for example, define the air velocity on surfaces as zero. Thermal boundary condition data define the location and magnitude of heat release in the space from occupants, lighting, and possible heat generating equipment. Boundary condition data are also known as constrained nodal data. There are two types of boundary condition: flux type and nodal point type. Flux types specify conditions such as boundary stresses and heat fluxes. Nodal point types specify boundary conditions directly to nodal points in the mesh; e.g., specified values of velocity, temperature, pressure. Initial conditions data specify initial values for certain nodes. These data are applied to the nodal points in the mesh. For a transient analysis these values are used as the initial conditions at the initial time, while for a steady state analysis they are used as the initial guess for the nonlinear iterative solution method. In general for stack induced ventilation experiment, two boundary conditions were involved, i.e. temperature and velocity boundary conditions. Both were applied to all surfaces of the model. Surface temperatures were taken from the solar radiation calculation. These results were normalized. Velocities at all surfaces were set to zero. This meant velocities at given nodes (which were occupied by the surfaces) were zero. Its value would be calculated by the computer program. It started from zero and increased incrementally until all the equations (which were used in the program) were satisfied. Initial values could be given as a first estimate to speed the calculation (Satwiko, 1994). Fluid property data specified the fluid properties such as density, specific heat, and viscosity. To simplify the calculation, all values were normalized. Using dimensionless values, the density became

$$\rho=(Ra.\mathrm{Pr}^{-1})$$ [5]

and the specific heat became the Prandtl number,

$$C_p = Pr$$

Other properties such as viscosity, thermal conductivity and volume expansion became unity. For the natural ventilation prediction, the flow is considered to be a low level of turbulence and an indoor zero-equation model is used where eddy viscosity is given from an analytical equation without involvement of transport equations. Density variation caused by temperature rise is expressed using Buossinesq approximation, which takes air density as constant and considers the buoyancy influence on air movement by the difference between the local air density and the pressure gradient. The upwind scheme has been used in the calculations (Ding, 2004)

## 3.2 CFD Simulation

However, although it has now been twenty-five years since Nielsen first applied CFD codes to indoor air motion study, issues of validation are still debated. Many research reports have been published, but the majority discusses validating the codes using simple room geometry. Of thirty-two such papers, three (9.4%) discuss purely theoretical aspects (eg. Baker), sixteen (50%) discuss validation processes (eg. Williams), eleven (34.4%) report on the use of CFD codes for practical design purposes (eg. Kolokotroni), and two (6.2%) report on the use of CFD for actual construction projects (eg. Kent). It seems that experts are still unsure about completely relying on CFD codes as a design tool. Moreover, 80% of the papers present advanced mathematics equations, which require a reasonably high skill level in mathematics for understanding. CFD solutions are based on the closure of conservation equations relating to mass, momentum, and energy. As the number of unknowns is larger than the number of equations, more equations are needed for closure. Usually, two equations are use, one concerned with turbulent kinetic energy (k) and another relating to kinetic energy dissipation rate these are popularly known as the $k$-$\varepsilon$ equations. The accuracy of CFD depends very much on the accuracy of the turbulent models. Turbulent modeling expresses Reynolds stress, turbulent heat flux, and turbulent diffusion flux. The accuracy of an iterative solution depends on such variables as grid resolution (i.e., number of grid points) and convergence criterion used. From a computational perspective, interior airflows are complex and generally turbulent. Predicting airflow within buildings is more difficult when buoyancy is involved. In terms of the closure the Partial Differential Equations (PDE), buoyancy terms, being non-linear, are the most difficult to handle. Moreover, improper selection of the reference velocity for scenarios involving natural convection in enclosures can cause considerable numerical problems, and hence, inaccuracy. CFD also overestimates flow rates through windows, especially in buoyancy driven flow. The root of this problem lies in a velocity profile through the opening, not accounted for in the CFD simulation. The $k$-$\varepsilon$ model assumes that eddy viscosity is the same for all Reynolds (Re) stress (isotropic eddy viscosity) and is restricted to flows with high Re. This method has been developed and modified. There are still some different opinions about CFD validity based on concerns about the $k$-$\varepsilon$ methods used by the codes. Shao states that the *k-ε* model is not suitable for complex flow regimes. Conversely, Li asserts that CFD is capable of predicting the complex airflow and heat transfer problems of buildings for engineering purposes. However, it is important for engineering applications to realise the capabilities and limitations of CFD. He also states that the standard *k-ε* model is not suitable for near-wall turbulence and for free re-circulating flows with incompletely developed turbulence. CFD prediction produces detailed information on the distribution of air velocity, temperature, turbulence quantity, contaminant concentration, humidity, and wall surface temperatures, which can be used in airflow design. A study by K.D. Knapmiller using a *k-ε* model found this model to be relatively computationally efficient and stable (compared to the more complicated Reynolds stress model) and reasonably accurate for a wide range of turbulent flows. His position is supported by Awbi. CFD codes (combined with physical modelling) have been used to study the natural ventilation system (large thermal chimney) of the School of Engineering at De Montfort University in Leicester, UK. This is one of the Europe's largest naturally ventilated buildings. Computer simulations were deficient in coping with thermal stratification and the complex internal geometry, which needs a physical model to compensate for missing information.

### 3.2.1 CFD Simulation Procedure

#### a. Modeling Methods

CFD is a modeling technique. CFD modeling is the process of representing a fluid flow problem by mathematical equations. The mathematical equations are phased on the fundamental law of physics. For building application, the parameters of interest would include velocity, pressure, temperature, turbulence intensity and possibly concentration of smoke and contaminations. Therefore, the set of equation in CFD are related to those variables. The equation can be expressed in a common form, shown as follows:

$\text{б}/\text{б}_t\,(\rho\varphi) + \mathit{div}\,(\rho\acute{\upsilon}\varphi - \Gamma_{\varphi}\mathrm{grad}\varphi) = S_{\varphi}$

transient + convection - diffusion = source

Where:

$\acute{\upsilon}$ = velocity vector

$\varphi$ = dependent variable

$\Gamma_{\varphi}$ = exchange coefficient

$S_{\varphi}$ = source

$\rho$ = air density

In general, the equations involved are those for conservation of momentum, which are sometimes referred to as the Navier-Stokes equations, the conservation of mass and the transport equations for turbulent velocity and its scale (Awbi, 1991). In solving these equations, computer calculation technique is used. Due to this process, the method is also known as computational fluid dynamic (Razak, 2002).

Those "conservation laws" may each be expressed in term of partial differential equations (PDE's). Each PDE describes the conservation of one dependent variable within the flow field. This provides a basis for a flow field in CFD model simulation. Usually in wind engineering or building studies the flow field in real situation is turbulent. If a turbulent flow is required in order to represent the real situation then, a "turbulent model" must be used. This will involve solving further equations.

The term "turbulence" is used to describe the apparently random "particles" of fluid, which can occur under certain flow condition. Broadly speaking, the flow which is usually referred to fluid dynamic theory can be described as being either laminar or turbulent, or when between these two states, as transitional (Schlichting, 1979). In fluid dynamic, Reynolds (Re) number is observed to be increasing as a result in an increase in flow velocity, until a certain velocity was reached. At higher velocity, turbulence resists distortion (increase in velocity) to a greater degree than laminar flow. Resistance to flow resulted in the formation of viscous forces and inertial forces. Reynolds number (Re) is a dimensionless number to describe the fluid flow. At low Re number the laminar flow tends to be greater than at higher Re number. However, the tendency for sudden shift from laminar flow to separated flow is greatly increased. Separation causes a great deal more drag than transition from laminar to turbulent flow.

Reynolds number is proportional to (inertia force) - (viscous force) and is used in momentum, heat and mass transfer to account for dynamic similarity. It is normally defined as:

$Re = (U \times L)/ \upsilon$

Where:

U = proposed simulated wind

L = characteristic length of flow system

$\upsilon$ = kinematics viscosity (u/p), air kinematics viscosity lower troposphere = $14 \times I0^{-6}$ m/s

$\rho$ = air density (at = 25°C ambient air temperature, the air density is around $1.225 kg/m^3$)

The aim of turbulence modeling in CFD is to represent the diffusion influence of fine-scale turbulence effects. The turbulence diffusion coefficient is added to the laminar coefficient (laminar viscosity) to yield the combined effect. In turbulent flow the diffusion forces are very much dominated by turbulent mixing. Therefore, the laminar viscosity is usually negligible.

In CFD there are three major turbulence models. They are Large Eddy Simulation (LES), Full Simulation and Reynolds-averaged *(k-ε)* models (Jones and Whittle, 1992 and Murakami and Kato, 1999). In LES model, the Navier-Stokes equations are averaged over a small spatial region. This method enables the simulation of a fluctuating turbulence flow field. The full simulation is the numerical solution of the exact Navier-Stokes equation. The length scale is usually very small. Thus this method requires a very small mesh system and is therefore difficult to apply to analysis of airflow. Reynolds-average models approach is by averaging the Navier-Stokes equations, which governs the fluid motion. The averaging process of the Navier-Stokes equation includes the unknown averages of the products of actuating velocities. Reynolds stress as explained above is one of the fluctuating velocities. In comparison among the three methods discussed, the Reynolds-average model is the most commonly used (Malsiah, 2001; Loomans, 1998 and Murakarni nd Mochida, 1989). It is recommended due to its efficiency and reliability (Jones & Whittle, 1992; Murakami & Kato, 1999 and Satwiko, Locke & 5onn, 1998). Therefore, it is relevant to adopt the LES method of designing the turbulence intensity for this research.

### b. Solution Method

The main process in CFD is to solve the equations that have been mentioned earlier in this section. The solution method of solving the required equations involved is by using the domain of integration of solution domain. The solution domain is actually the region of space within the differential equations are to be solved. The solution approach in CFD is to represent the differential equations in numerical forms, linearise the equations and then solve them using numerical analysis techniques. This approach is known as "discretization method" (Awbi, 1992).

### c. Numerical Procedures

The discretized form of equation can be solved by one of the well-established numerical procedures such as "finite element" method (FEM) or "finite differences" or known as "finite volume" method (FVM) (Awbi, 1991). The selected methods relied on the accuracy of the solution and the computer efficiency. The FVM is more popular then FEM in CFD because it is generally more economical in computational time. The majority of the commercially available CFD computer codes used the FVM solution techniques (Whittle, 1996).

In both methods, the equations are solved on a grid. They are mapped to a computational grid, which forms three dimensional (3-D) volumes or control volumes of space. These grids need to completely fill the enclosure to be modelled. The computational grid is also known as a mesh. The control volume is Renown as a "cells", in FVM or "elements" in FEM. These FVM or FEM models use selected grid types. They are solved in a CFD modelling by an iterative manner. This is to generate field values for all dependent variables. The iterative solution will continue until the imbalance or error in the equations is sufficiently small to be considered negligible. The data obtained from simulation will be used to calculate the values of all dependent variables, such as air velocity, temperature distribution, pressure distribution, concentration of chemical, smoke, etc. for each computation control volumes.

### d. Computational Grid

In CFD application, the computational grid cell defines the solution domain. Number and size of cells represent the level of resolution that the calculation can be achieved. Smaller grid cells are normally defined in areas where large gradient of solution variable are evident. Failure to provide enough mesh in these areas will result in the supply jet or boundary layer flow being insufficiently resolved. Hence, this resulted in an unrealistic local situation. For economic purposes (in term of computing time), it is usually to expend the grid, in the spatial sense, in area remote from those of interest and importance. Figure 3.1 shows the example of the grid layout that is uniform in sizes. The computational grid is having a total of 224 (=7x8x4) cells. Two main types of computational grid can be applied in CFD (Jones and Wittle, 1992). They are rectilinear (Cartesian) grids and curvilinear body fitted coordinate (BFC) grids. The grid is rectilinear (Cartesian) when the cells are formed as rectangles. The advantages of the Cartesian grid are that it is easy for the user to specify and generate the grid. FVM is performed very efficiently using this type of grid. For curvilinear body fitted coordinate (BFC) grids, the x, y and z structures of the Cartesian grid are retained. However, the cells are formed from four-sided (2-D) or six-sided (3-D) elements, which can be distorted to follow irregular boundaries. The BFC forms of the equations are much more complex and computations on these.

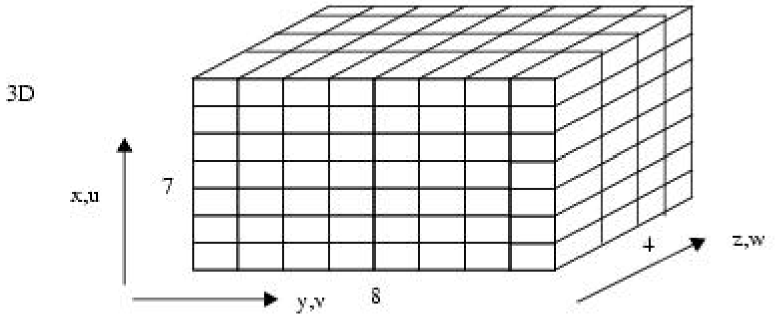


**Figure 3.1:** The example of the cartesian grid layout (Flomerics, 2000)

The most widely used methods in solving the numerical form of differential Ration is SIMPLE (Semi-Implicit Method of Pressure-Linked equation) (Awbi, 1991). Patankar and Spalding developed this method (Jones and Whittle, 1992). FSIMPLE method links velocity to the pressure in order to satisfy continuity. CFD commercial codes of FVM usually employed this method as a pressure if correction algorithm is required. This method uses the staggered grid (Jones and Whittle, 1992). This method enables the velocity component to be corrected using the pressure values at the neighbouring grid nodes. To achieve this, the positions of the velocity components must be displaced from the grid nodes (pressure positions) so that the pressure forces act at the surface of the velocity control volumes.

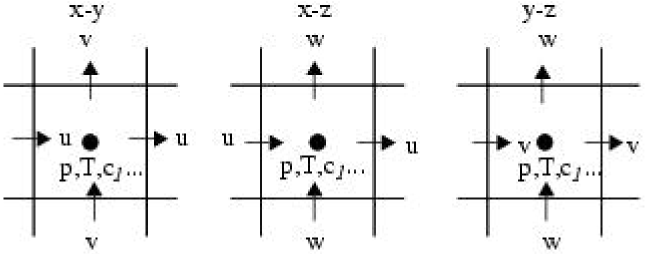


**Figure 3.2:** Simple method. Methods of solving the numerical form of Differential equation Using Staggered Grid (Flomerics, 2000)

### e. CFD Solution Procedures

The CFD method involves an iterative procedure as solution procedures. This involves solving each equation separately for the whole field. This means that in one iteration, the equations are assembled and a solution is obtained for each velocity component (up to three), pressure, temperatures and turbulence quantities. This procedure is repeated until a converged solution is obtained. In general, the CFD solution procedures are summarized by the following example for a 3D simulation of flow and heat transfer:

- Stage 1, initialise the fields of pressure, temperature and velocities,
- Stage 2, increase outer iteration count by stage 1,
- Stage 3, set up coefficients (i.e. the Cs) for temperature field, T,
- Stage 4, solve linearized algebraic equations for the value of T in each cell by performing a number of inner iterations,
- Stage 5, repeat stage 3 and 4 for field variables $u, v, w, k, \epsilon$, and $c_1$, $c_2$, ... for concentrations,
- solve the continuity equations in a similar manner and make any associated adjustments to pressure and velocities,
- check for convergence and return to stage 2 if required.

### f. Selection of Software

The rapid development in computer technology especially with regard to the computer-processing unit (CPU) led to the extension in CFD capabilities. Today, there is a lot of CFD software available. There are two categories of CFD software available for airflow modeling. They are categorised as general purpose CFD software and specific airflow modeling software. General purpose CFD software is usually used in fluid engineering application. The CFD software is used to predict the pipe size, duct size, heating and cooling loads, fire and smoke spread etc. (Jones, 1997). Razak (2002), Malsiah (2001) and Lam and Wong (1999) have investigated some available commercial CFD software specifically for airflow modeling. Table 3.1 shows the summary of the software.

**Table 3.1:** Computer Software for Ventilation and Air Quality Analysis Source: Lam and Wong (1999), Sapian (2004)

| Name | Software Developer | Description | Output |
|---|---|---|---|
| 1. COMIS | Lawrence Berkeley Laboratory, Building 90, Room 3074, Berkeley, CA 94720. | - Based on multi-zone network model.<br>- Coupled to a pollution migration model<br>- Incorporation of a statistical algorithm to compute Cp values at any point of the building envelope (COSMIS)<br>- Incorporation of a simple HV AC module for the consideration of HVAC effects.<br>- Rudimentary Graphical User Interface (GUI) | - Air change rate for each zone<br>- Ventilation and infiltration rale (mechanical & natural)<br>- Rate and direction of airflow through individual openings<br>- Pattern of flow between zones<br>- Internal room pressure<br>- Pollution concentration for each zone<br>- Pollution flow between zones and between inside and outside of bull dings |
| 2. CONTAM | National Institute of Standards and Technology (N1ST), BR/A 313, Washington DC<br>20234. | | |
| 3. PHEONICS | Concentration, Heal amd Momentum Limited, Bakery House, 40 High Street, Wimbledon Village, London SW19 | - Computational Fluid Dynamics (CFD) Models.<br>- Normally requires the use of Pre-processor for geometry definition and<br>- Post-processor for processing the data output<br>- Rudimentary Graphical User Interface (GUI) | Room air flow<br>Airflow in large enclosures<br>Air change efficiency<br>Pollution disposal pattern<br>Pollution removal effectiveness<br>Temperature distribution<br>Air velocity distribution<br>Turbulence distribution<br>Pressure distribution<br>Fire and smoke movement<br>Airflow around building. |
| 4. FLUENT | Fluent Inc. USA. Worldwide Corporate Headquarters, Central Resource Park, 10 Cavendish Court, | | |
| 5.FLOVENT | Flomerics Limited. S 1 Bridge Road, Hampton Court, Surrey KT89HH, UK. | | |

Since this research is intended to study the solar induced and to estimate the air velocity inside solar chimney model, then the temperature differences values needs to be converted into values for input data in the stack effect method in order to predict the internal air velocity. Therefore, in selecting the appropriate software certain criteria of the research parameters should be met by the software. In general, the software should be able to;

- Act as a virtual atmospheric boundary layer real condition.
- Represent the turbulent characteristics of airflow. In CFD it is usually represented through the *k-ε* method.
- Estimate the pressure distribution at the surface of the building.
- Provide with the Atmospheric Boundary Layer (ABLJ generator so that the required wind profile can be simulated.
- Construct the required building configurations easily.

Due to the limitation of the study time and cost, the software should also be able to fulfill some other factors such as;

- The software should be able to be installed in a personal computer (PC) using user-friendly operating system such as windows base application. Some software used complicated operating system such as DOS as an operating system (Malsiah, 2001).
- The user interface should be intuitive and easy to navigate around. For example, in building a model, the drawing should be easily edited like the CAD software
- The post-processing facilities should enable the user to communicate his results easily and effectively
- The most important factor is that the software should be user-friendly. Technical support after sales also should be able to provide training, assistance and support efficiently
- In term of cost, the software should be economical. The cost for the software license period should be within the study budget and period.

The above criteria and factors led to the selection of FloVent version 3,1 software developed by Flomerics Ltd. UK. Flomerics Pte. Ltd. (Singapore) distributes the software for South East Asia, The description of the software is explained in Abdul Razak (2002); Baskaran, A. and Stathopoulos, T., (1989) and (1992); Murakami, S. and Mochida, A., (1989) and Murakami, S, *et.* |; *al,,* (1999); reported that this software is proven reliable and capable to simulate stack airflow and estimate the air velocity.

### 3.2.2 CFD Simulation Condition

This section explains the setting-up and procedures of conducting the CFD simulation. The simulation conditions used are also deliberated in this section.

#### a. Position of Solar Chimney Model in the Overall Domain Solution

Using FloVent version 5.1 programs, the simulation models are created directly using the drawing board window provided with this software. This is to give more accurate representations of the buildings. The sizes and the dimensions of the models involved in this simulation are described in section 3.3. Like CAD software, the building created in this CFD software using 1:1 scale. The model was placed inside an overall domain solution size of 40 m x 40 m x 20 m high. The position of the model inside the overall domain solution was at 20 m from x-plane, 20 m from z-plane and 1 m from y-plane. Figure 3.3 shows the overall domain solution and the position of the model inside the overall domain solution. Figure 3.3 can also be considered as an overall setting-up.

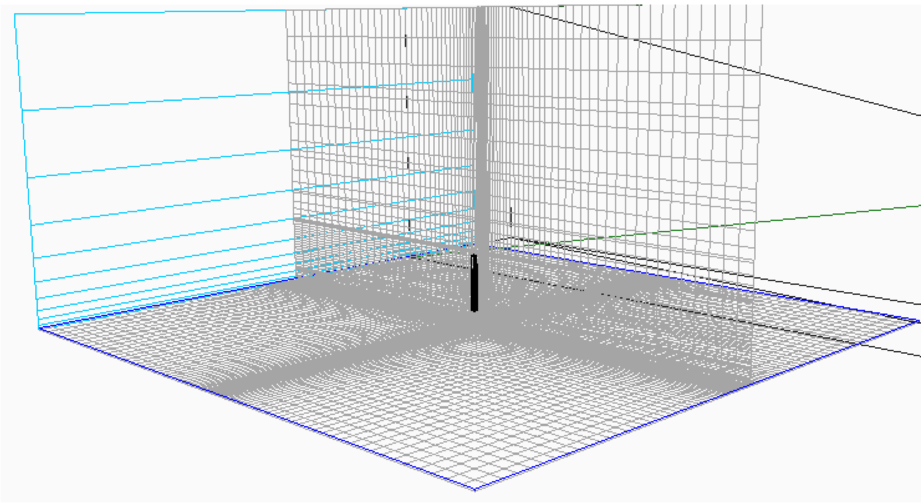


**Figure 3.3:** Overall simulation modeling setting up for solar chimney model

### b. Monitor Point

The main objective of the simulation is to estimate the air velocity inside of the solar chimney model. Therefore, monitor point represents the velocity tapping point was placed at the mid point of both inlet, middle of model and outlet openings. The exact location of the monitor point determines the accuracy of the predicted internal air velocity. The total monitor point used in solar chimney models is 3 numbers (figure 3.4).

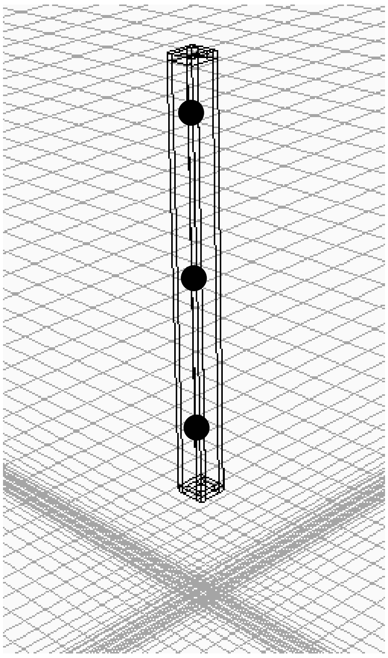


**Figure 3.4:** The position of monitor point in relation to the position field measurement and size of model

### c. Grid System

The simulation used Cartesian type of grid system. Figure 3.42 shows the grid setting for the solar chimney model, where the grid was defined x, y and z direction. The x-direction grid was further defined into 3 grid constrain, the y direction into 3 grid constrain and the z-direction into 3 grid constrain as shown figures 3.5. Each grid constrains form a cell. The total number of cell produced from this system grid is 48 numbers (x-direction) x 40 numbers (y-direction) x 48 numbers (z-direction). This gives a total of 92.160 cells all together. Therefore, 92.160 control volumes carry out the numerical solution for this building. Grid distribution is very important as it affects the numerical accuracy. In determining the number of grid, start with a coarse grid and refine it in region of; complex flow, high gradients and particular interest. The more detailed the grid, the more trade-off between accuracy and computing time required.

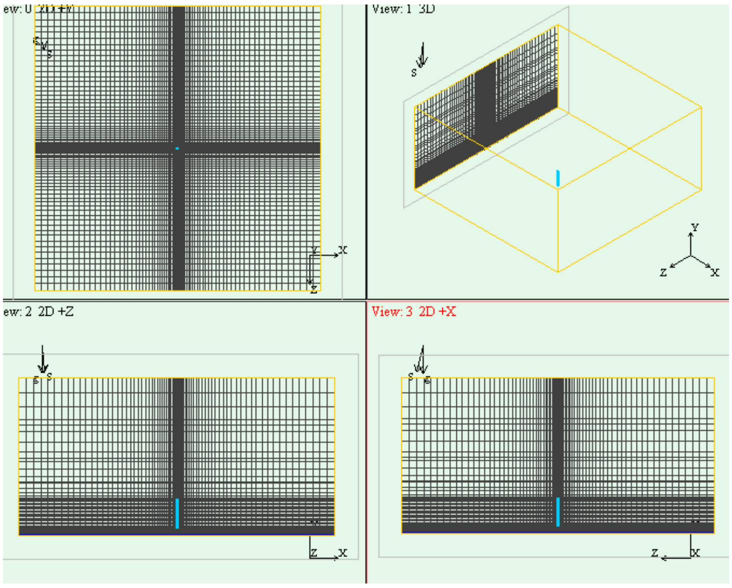
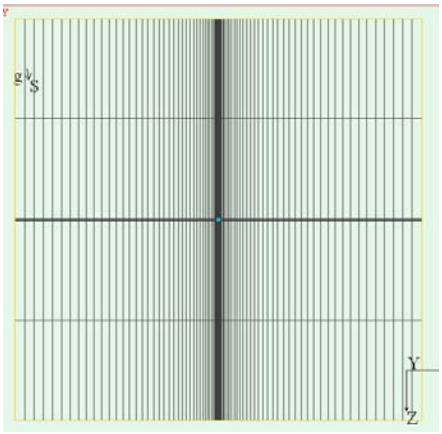


**Figure 3.5:** The grid setting for basic of solar chimney model in the Flo Vent

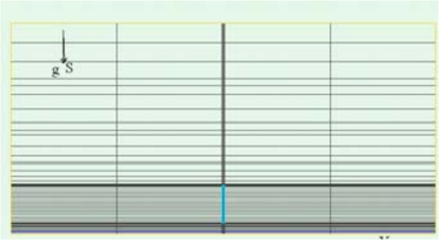


Grid constrain #1:
- Start location: 0 m
- End location: 19.875 m
- Nos. of grid: 20
- Type of grid distribution: decrease

Grid constrain #2:
- Start location: 20.125 m
- End location: 40 m
- Nos. of grid: 20
- Type of grid distribution: increase

Grid constrain #3:
- Start location: 19.875 m
- End location: 20.125 m
- Nos. of grid: 8
- Type of grid distribution: uniform

**Figure 3.6:** The grid setting for basic of solar chimney model at x-direction

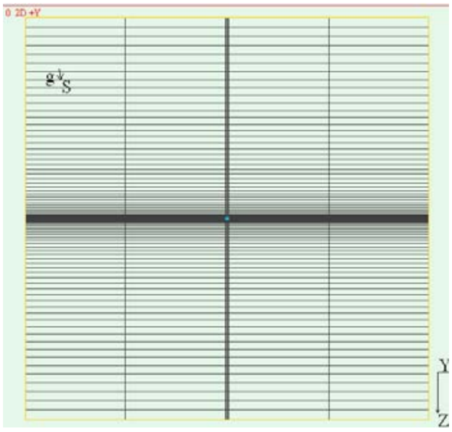


Grid constrain #1:
- Start location: 0.01 m
- End location: 1 m
- Nos. of grid: 8
- Type of grid distribution: decrease

Grid constrain #2:
- Start location: 1 m
- End location: 4.5 m
- Nos. of grid: 17
- Type of grid distribution: uniform

Grid constrain #3:
- Start location: 4.5 m
- End location: 20 m
- Nos. of grid: 14
- Type of grid distribution: increase

**Figure 3.7:** The grid setting for basic of solar chimney model at y-direction



Grid constrain #1:
- Start location: 0 m
- End location: 19.875 m
- Nos. of grid: 20
- Type of grid distribution: decrease

Grid constrain #2:
- Start location: 20.125 m
- End location: 40 m
- Nos. of grid: 20
- Type of grid distribution: increase

Grid constrain #3:
- Start location: 19.875 m
- End location: 20.125 m
- Nos. of grid: 8
- Type of grid distribution: uniform

**Figure 3.8:** The grid setting for basic of solar chimney model at z-direction

### d. Boundary Condition

In this research, the major variables are site, wind data or reference wind speed, wind profile, solar radiation and atmospheric boundary layer characteristics. The summary of the site conditions for Johor Bahru sub urban hot and humid climate that is used in the simulation are;

- The location of the study is Universiti Teknologi Malaysia.
- The latitude is 1° 08' N and the longitude is 104° 42' E of Greenwich.
- The height above sea level is 37.8 m.

- The nearest meteorological station is located at Sultan Ismail (SI) airport known as Senai meteorological station, approximately 20 km from centre of Johor Bahru.
- The average ground characteristics of on site weather station area are considered as a combination of flat terrain and low-rise buildings. This gives the empirical exponent (a) value of 0.22, the roughness length (Z0) value of 0.25 m and the gradient height (Zg) value of 370 m (ASCE, 1999).

Due to its geographical location, Johor Bahru receives winds from almost all directions. However, the northerly winds that prevail from November until April, and the southerly winds that prevail from around May to September, are the main prevailing winds. The secondary winds come from the north-east during the months of November to April and from the south-west in the months of May to September. The mean surface wind fluctuates between 0 m/s and 1.43m/s. The mean maximum surface wind fluctuates between 13.4m/s and 19.0m/s. The percentage of calm period is about 31.6%, and 48.1%. Figure 3.9 shows the annual wind rose for Johor Bahru which exhibits the intensity and various percentage frequencies of wind speed and direction. The occurrence percentages of the northerly and southerly winds are amongst the highest. The northeasterlies and northwesterlies are the secondary winds (Kubota, 2006).

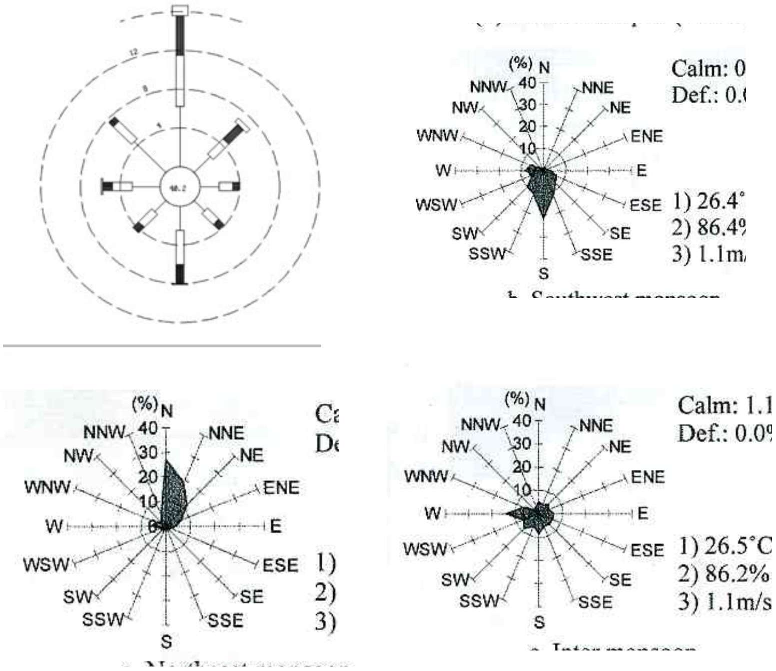


**Figure 3.9:** Wind rose and climate summary in Johor Bahru 1988-2002 (Kubota, 2006)

On the whole, the winds progressively pick up in speed during the day. Starting from about 7:00 am in the morning, the speed increases and reaches its peak at about 2:00 pm, and then starts decreasing until it reaches the lowest steady speed at about 10:00 pm. From 10:00 pm to about 7:00 am, the wind speeds are relatively low or calm (Malsiah, 1998). In Johor Bahru, the mean wind velocity in the northeast monsoon period (1.8m/s) is slightly higher than that of Kuala Lumpur, and the wind velocities both in the southwest monsoon period and the inter monsoon period (l.lm/s) are a little lower than those of Kuala Lumpur (Kubota, 2006). However, the data was taken at 17.4 m high (height of anemometer head above ground). In this research, the international standard reference height for mean wind at 10 m above ground is used. Therefore, the wind speed has to be corrected to a wind speed that refers to the international standard reference height. The Atmospheric Boundary Layer ABL generator was downloaded from FloVent website. The ABL generator provided by FloVent used Log Law model to create the required wind profile- Johor Bahru can be described as a sub urban with high crops and residential suburban; therefore, the roughness length ($Z_0$) equivalent to 0.25 as proposed by ASCE is used. Table 3.2 shows ASCE (1999) ABL layer characteristics for different terrain roughness.

**Table 3.2:** ABL characterictics for different terrain roughness (ASCE, 1999)

| Class | Terrain Description | Zo (m) | $\alpha$ | *Iu* (%) | Exp. | Z(m) |
|---|---|---|---|---|---|---|
| 1 | Open sea, fetch at least 5 km | 0.0002 | 0.10 | 9.2 | D | 215 |
| 2 | Mud flats, snow; no vegetation, no obstacles | 0.005 | 0.13 | 13.2 | | |
| 3 | Qpen flat terrain; grass, few isolated obstacles | 0.03 | 0.15 | 17.2 | C | 275 |
| 4 | Low crops; occasional large obstacles | 0.1 | 0.18 | 27.1 | | |
| 5 | High crops; scattered obstacles, residential suburban | 0.25 | 0.22 | 27.1 | B | 370 |
| 6 | Parkland, bushes; numerous obstacles | 0.5 | 0.29 | 33.4 | | |
| 7 | Regular large obstacle coverage (dense spacing of low buildings, forest | 1.0-2.0 | 0.33 | 43.4 | A | 460 |
| 8 | City centre with high and low-rise buildings | $\geq$2.0 | 0.40~0.67 | | | |

Correction can be done using the log law equation and the ABL characteristic is selected from table 3.2 Consequently the mean wind speed of UTM sub urban terrain condition can be predicted using the same log law equation.

Log Law model:

$V_z = V_{ref} [\log (Z / Zo) / \log (Z_{ref} / Zo)]$

Where:

$V_z$ = the mean wind speed at height (gradient wind)
$V_{ref}$ = the mean wind speed at some reference height
$Z_{ref}$ = the reference height
$Z$ = the height for which the wind speed $V_z$ is computed (gradient height)
$Zo$ = Roughness length or log layer constant

According to the description given by ASCE (1999) as shown in table 3.2, Senai meteorological station ABL characteristics can be described as open terrain (grass and few isolated obstacles). Therefore, its mean wind speed exponent (a) can be assumed as 0.15, the roughness length ($Z_0$) is 0.03 and the gradient height (Z) is 275 m. Using the same log law equation, gradient wind for UTM can be obtained. Before that, UTM ABL characteristic has to be determined. Referring to table 3.2, UTM terrain condition can be described as an area with high crops; scattered obstacles, residential suburban. Observation shows that although UTM is approximately ± 6 km from Senai, this area is denser compared with Senai. Therefore, its mean wind speed exponent ($\alpha$) can be assumed to be as 0.22, with the gradient height of 370 m and the roughness length ($Z_0$) of 0.25 (ASCE, 1999). Correction of Senai mean wind speed at 10 m reference height determines UTM mean wind speed. It is predicted that UTM mean wind speed at 3 m reference height is shown in figure 3.11. This value is actually within the ranges that have been obtained from the solar chimney model simulation

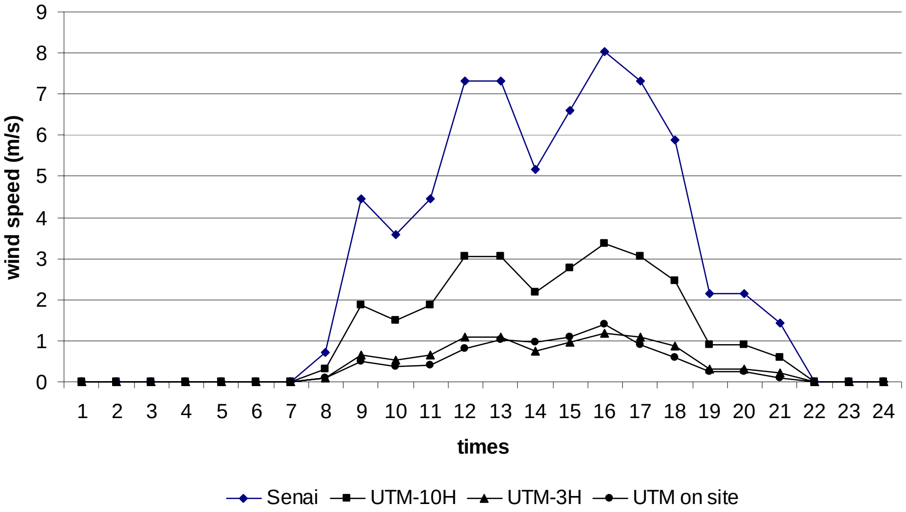


**Figure 3.10:** Wind speed on 21 March of Senai Meteorological Station weather file data and on site weather data (2005)

Figure 3.10 shows the results of total monthly and daily solar radiation calculated for data measured at the Senai Meteorological Station (SMS). Comparison between the respective days with monthly data indicates same pattern for total solar radiation. However, the Senai Meteorological Station indicated higher solar radiation intensity on 21 March at 12:00 pm (916 W/m$^2$).

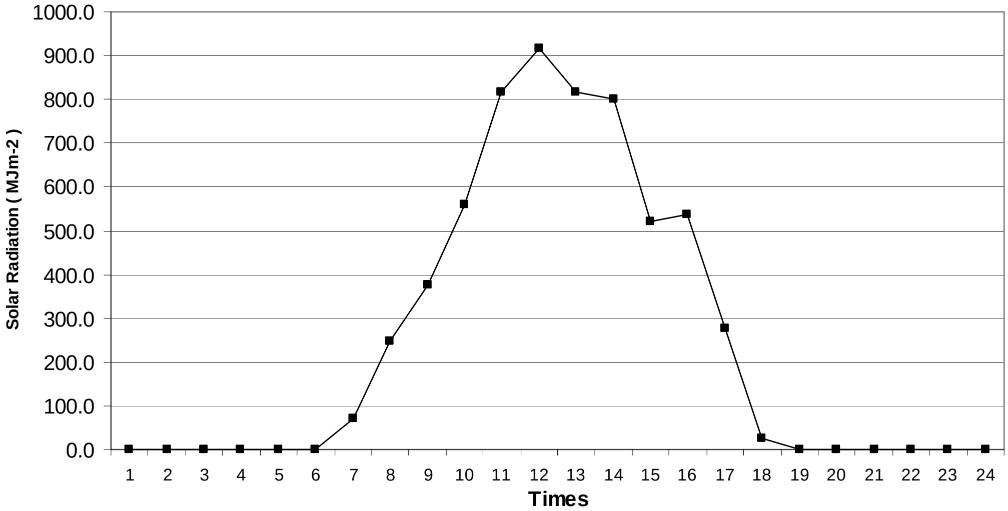


**Figure 3.11:** Solar Radiation intensity on 21 March of Senai Meteorological Station weather file data and on site weather data (2005) (Latitude: 1.8$^{0,}$ Longitude: +104$^0$ & Time zone: +7)

As discussed in section 3.3.1.1 the *k-ε* method of designing the turbulence intensity for the simulation is adopted. In this CFD software the *k-ε* model define the turbulent viscosity at each point from two additional variables that characterise the local state of turbulence, viz;

- the kinetic energy of turbulence *(k)*
- its rate of dissipation (*ε*)

In the *k-ε* model adopted by this FloVent version 5.1 software, the turbulence kinetic energy and the dissipation rate of turbulence are automatically set by the program based on average inlet velocity values as 0.1 J/kg and 0.l W/kg respectively. These automatic boundary settings are usually suitable because the generation of turbulence within the calculation domain is usually sufficiently high to make precise settings of the inlet values inconsequential (Flomeric, 2000). Other defaulted basic references values that are used throughout the simulation are;

- Datum pressure, this variable is set as a gauge pressure. It is used to calculate the absolute; pressure values for use in variable density models. For most calculations, the default; value of standard atmospheric pressure of 1 arm is appropriate (Flomeric, 2000).
- Air temperature, temperature around 33.2 °C was used as a constant temperature outside the enclosure. Air temperature around 33.2 °C was selected based on the mean air temperature for respectively day (21 March 2005) at 12:00 pm. This temperature is used by default for fluid temperature at the inflow boundaries and reference temperature in the buoyancy force.

### 3.2.3 CFD Simulation Analysis

In general, for simulation analysis and assessment, these research principles apply: qualitative analysis, quantitative analysis, comparison with previous studies, comparison with real world conditions and statistical (Satwiko, 1998). Firstly, the qualitative analysis for secondary problems is made. These include issues that should be accounted for, but should not be the major consideration. Some research is full of philosophy and not be easily linked to building science scenarios. To determine the relevance of a philosophical thought to building science, a logical qualitative analysis is conducted. This usually precedes a quantitative analysis. Second, the quantitative analysis for simple problems is conducted. People demand different indoor air conditions to keep comfortable, depending on their activities. In many cases, recreation and behaviour are culturally determined. This kind of relationship is discussed quantitatively and checked using the ASHRAE Thermal Comfort Program. Third, comparison with previous studies is sourced in research reports for complicated numerical problems. Computational fluid dynamics codes involve complex mathematics. Since this research focuses on the application of computational fluid dynamics codes, any issues raised by the numerical are referred to the relevant experts or, if appropriate, compared to results found in other research reports. Fourth, the comparison with real world conditions is made. Rather than simply adjusting the computer simulation to imitate real conditions, it quickly identifies suspicious or *strange* results that may indicate the existence of flaws. These flaws can be caused by various problems from false data input to improper computer programming configuration. Last, Statistical analysis is used particularly for interpreting weather data. In this research Excel version XP software is used.

To calibrate the program, the results of experiments using CFD-Flo Vent were compared to pilot testing experiments. The reliability of the results was determined, and input adjusted to produce results reflecting real situations. The graphic comparison used Excel XP software. The air temperature and velocity calibrations use, velocities and non-dimensionalised temperature as parameters. The solar chimney calibration compared two sets of data: pilot testing data and CFD-Flovent results from eight different experiments. The tolerance range can be derived from Selvam's report. He confidently states that his CFD experiment has a good agreement with the field data. Selvam allows up to 7% deviation for the average windward between his CFD calculation and the real condition. In a separate case study, another CFD expert, Shao, accepts 20% tolerance for a *good agreement* between his CFD codes' Cp results and field data. Expecting a complete match between CFD results and field data is not only difficult but also misleading. Airflows around and inside buildings are turbulent and always changing with time. Therefore, any measurements of airflow variables are generally noted as average values. CFD programs, on the other hand, tend to calculate the flow based on a particular set of steady state condition. Thus, a certain degree of deviation (between CFD results and the field data) can be tolerated (Satwiko, 1998).

The analysis of the study is based on the output data obtained from the simulation for the tested solar chimney configurations. The output results were obtained in two forms: the configurationally results for the selected time and the hourly values for the designated year. The configurationally results by end users are analyzed for the following performance variables:

i. Temperature Differences
ii. Air velocity
iii. Air Flow Rate

The suggested air velocity standard for indoor residential buildings is 0.25-1.5 m/s and it is used as a bench mark in describing the air velocity requirement of the configurationally tested solar chimney models. The analysis of each tested solar chimney configuration models will be evaluated with the maximal performance variables values for base-case model (pilot testing). Also, all the performance variables were correlated with solar chimney geometry ratio (SCGR) of the tested solar chimney models. The hourly results were obtained for the following performance variables:

iv. Air velocity
v. Air Flow Rate
vi. Air Temperature

The hourly data for respective days (21 March, 22 June, 24 September and 21 December) were chosen three times within hours chosen (8:00, 12:00, and 16:00) for analysis. The selected time was based on different position of the sun within the activities schedule. The results of the air velocity and air flow rate were combined into a single graph on respective dates and orientations. Likewise, the air velocity and air temperature were also illustrated in a single graph on respective dates and orientations. This is to get a better understanding of the influence of solar chimney over and on the performance variables. The maximum, minimum and mean activity plane air velocity, air flow rate and air temperature are used to describe general performance of the models tested.

Design variables and criterions for evaluation of data were determined from the literature review presented in chapter three. The overall assessments of results of the simulation were analyzed as in table 3.3.

**Table 3.3:** Data analysis indicators and their interpretation

| | **Data Analysis** | **Interpretation and Performance Variables** |
|---|---|---|
| 1 | Assess the impact of solar chimney on the use solar radiation as stack induced ventilation in Malaysian Condition | - Solar radiation<br>- Temperature Difference |
| 2 | Assess the impact of solar chimney configuration on target maximal air velocity | - Air velocity<br>- Air flow rate |
| 3 | Assess the impact of the use solar chimney on residential building | - Indoor air temperature and velocity with solar chimney<br>- Indoor air temperature and velocity without solar chimney |

Most results will be presented and interpreted graphically. Evaluation during interpretation is guided by logic informed by analysis and assessment. Any unexplained results should encourage repetition of the experiments. Graphical presentations, plots and linear probes, are used to analyse results. Plots are defined in vertical, horizontal sections (cutting planes), or any defined surfaces. A linear probe shows the values at points (locations) in a line.

- Vector plots. These plots show the airflow patterns around and within the models, including wake formations and stagnation locations.
- Velocity (V) plots and linear probes. Velocity plots allow for easy visual detection of low and high air velocity as well as stagnant locations. The velocity variable is used in the comfort equations to calculate the comfort level of a given location.
- Temperature (T) plots and linear probes. Temperature plots make the study of air temperature distribution easy. These plots show warmer and cooler locations within the house caused by radiation and convection heat transfer from the warm roofs.

## 3.3 Development of Simplified Solar Chimney Model

The simplified solar chimney model used in the CFD simulation is developed from the basic model of the pilot testing, which is identify through inventory exercise as discussed in chapter 4. From the basic and modified, two sets of simplified configurations with seven modifications (by introducing solar chimney configuration) of solar chimney basic model are derived. The development of the simplified model is described in the following sections.

### 3.3.1 The Pilot Testing of Solar Chimney Simulation Model

The basic simplified pilot testing shown in figure 3.12 is a typical configuration with overall size of 0.15m x 0.15 m x 3.5m high. This size is to represent 1 cylindrical PVC pipe on pilot testing. The diameter of pipe is 0.15 m and the height of pipe is 3.5 m. The different between the simulation model and pilot testing is the section shape. The simulation consists of rectangular section while the pilot testing model consists of cylindrical shape.

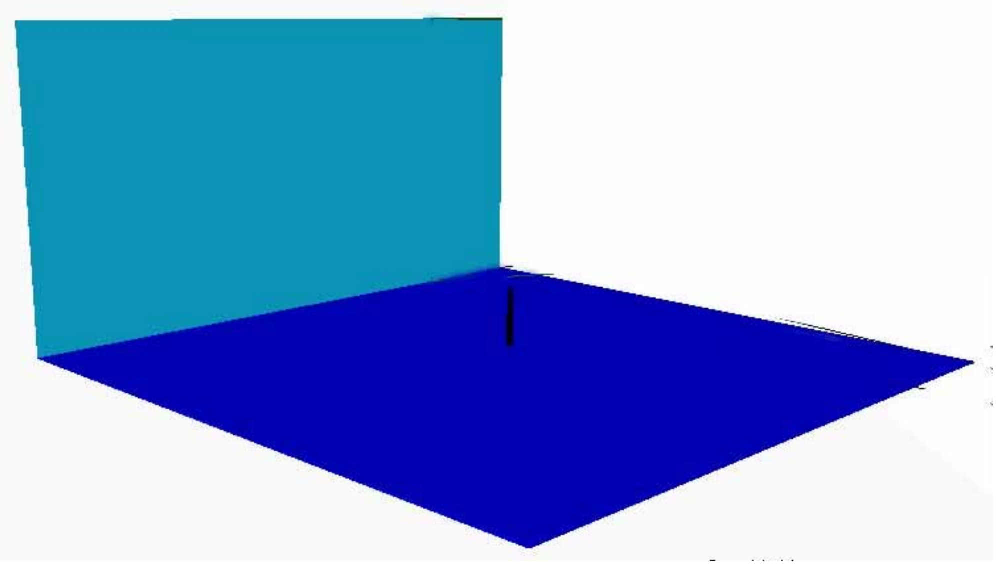


**Figure 3.12:** The pilot testing model in the Flo Vent

### 3.3.2 The Modified of Solar Chimney Configuration

The modified solar chimney configurations are extension of the pilot testing model described in section 3.3.1. In this stage, the basic solar chimney models are modified physically into five alternative modifications. The modification is by introducing geometry and material configuration.

#### a. Size of height

In this study, the maximum limit of the height is assumed as 200% of the building height between the floor and top of the roof. The aperture above the height of solar chimney model is assumed to be effective in induced stack ventilation, while the area below the window has no effect on air movement on the sitting plane. However, when considering the effect of chimney height, several sizes were simulated: chimney height between 1 m, 3.5 m, 6 m and 9 m.

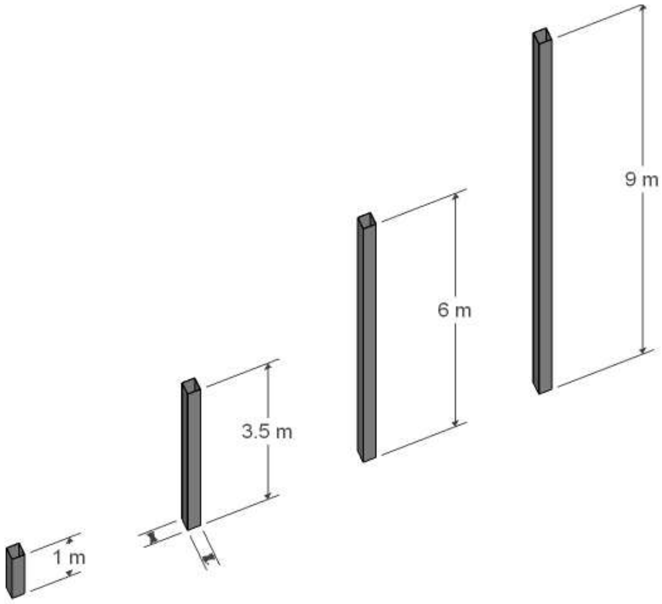


**Figure 3.13:** The size modification of height of solar chimney model in the Flo Vent

### b. Size of width

In the general, the width size is equivalent to the size of height of the solar chimney. According to Bouchair, ratio between width and height is 1:10. Therefore the solar chimney width and solar chimney height are assumed to be equal. The width extends from 0.05 m of the basic height and length to the other and upward to the maximum width size (1m). Hence, the size of the width is 0.05m, 0.15m, 0.3m, 0.6m and 1m in width for 3.5m high and 0.15m length (Figure 3.14).

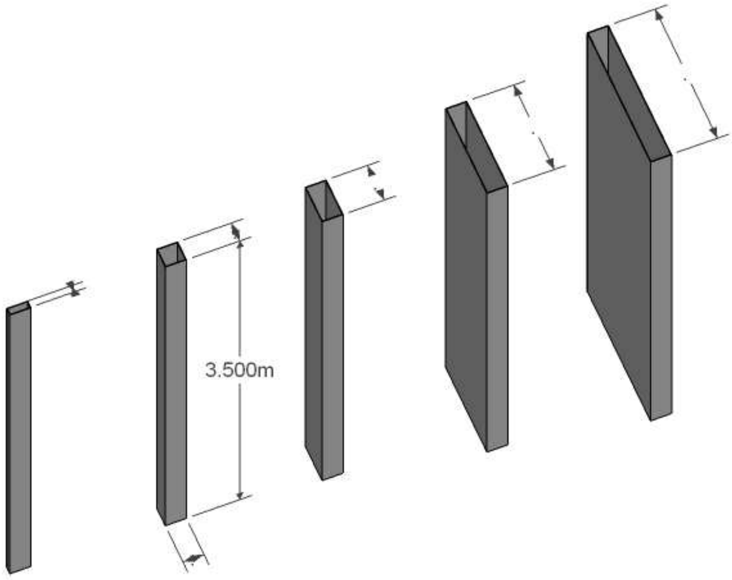


**Figure 3.14:** The size modification of width of solar chimney model in the Flo Vent

### c. Size of length

The length of solar chimney is independent variable in this study. The main purpose of this study is to determine the optimum length of solar chimney in terms of increase airflow rate and achieving target air velocity level in order to obtain psychological cooling. The length of the solar chimney are as follow 0.05 m, 0.15 m, 0.5 m, 1 m, 1.5 m, 3m

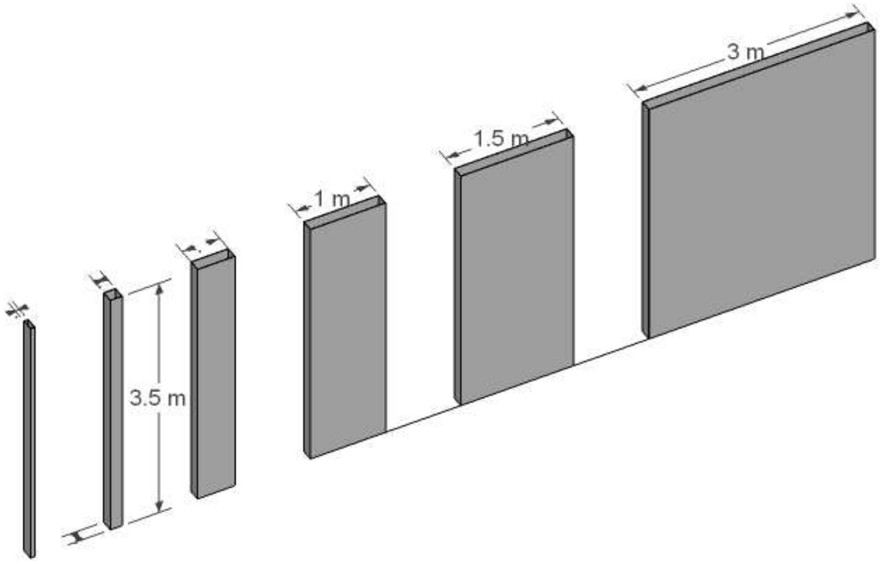


**Figure 3.15:** The size modification of length of solar chimney model in the Flo Vent

### d. Size of thickness

The base solar chimney model and the modified configurations with different thickness will be used to investigate the objectives of the study. Further, the characteristics of the models will be determined based on the types of thickness variables to be investigated and the study procedure. The thickness of solar chimney is 0.002m, 0.005m, 0.01m, 0.05m, 0.15m

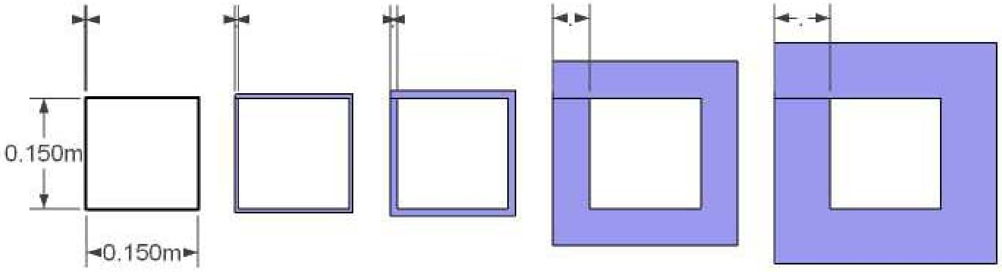


**Figure 3.16:** The size modification of thickness of solar chimney model in the Flo Vent

### e. Variety of material

The basic solar chimney construction is 0.005m thick PVC material with external black surface color. The conductivity value is about 0.16 W/(m K) and specific heat is about 1004 J/(kg K), which is a different to the value of common brick wall, aluminum, glass and brick-glass. The conductivity value for the brick wall is about 0.84 W/(m K). Specific heat from the brick wall is 800 J/(kg K). The aluminum and glass U values are about 2.0 $W/m^2K$ and 0.5 $W/m^2K$ respectively. The properties of the specific heat of aluminum and glazing are as follows: 913 J/(kg K) and 836 J/(kg K).

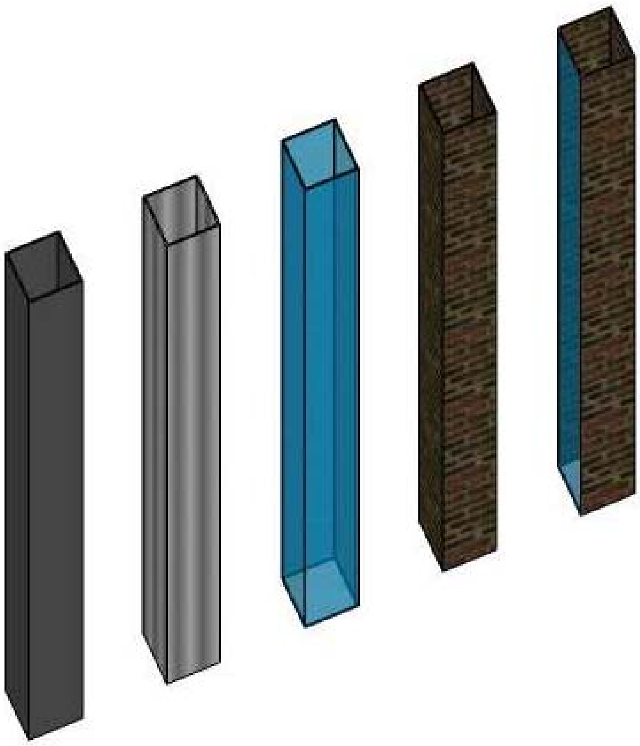


**Figure 3.17:** The modification of material of solar chimney model in the Flo Vent

# CHAPTER 5

# CONCLUSION

The findings of the research have been presented and discussed in the previous chapter. This final chapter will conclude the overall findings of the thesis. The application of the research findings are also discussed in relation to the aims and objectives of the thesis as set in Chapter 1. Finally, further work related to this study will be suggested in this chapter in order to strengthen and compliment this thesis.

## 5.1 Review of Thesis Objectives and Research Questions

As stated in Chapter 1, the main aim of this thesis was to assess and compare the impact of solar chimney induce ventilation for increasing air movement in Malaysia residential building. in hot and humid climates. This objective was achieved by using the Flovent 5.1 Computational Fluids Dynamic computer simulation program. Other specific objectives of the study are as follows:

- To develop solar chimney to improve indoor air movement in residential building.
- To analyze and established the effectiveness of induced stack ventilation technique of solar chimney

The hypothesis of this study is that a proposed design of solar chimney ventilation will achieve the following:

- Increase temperature different between inside solar chimney and outdoor climate condition.

- Provide maximum air movement (velocity and flow rate) at thermal ventilation requirement.
- Thus increase and provide maximum air movement for predict the effectiveness induced stack ventilation.

The term “proposed design” refers to best performance of solar chimney configuration which will increase maximum air movement inside the solar chimney model to obtain induced stack ventilation.

The following questions were addressed in order to achieve the main objectives of the thesis:

1. Does the use solar chimney ventilation still possible in Malaysia condition?
2. What is relation solar chimney configuration to achieve target for maximum induced stack ventilation?
3. What is the proposed solar chimney ventilation model to obtain maximum induced stack ventilation in Malaysia and relation with climate condition elements?
4. Does the effective proposed solar chimney at (Q3) increase induced stack ventilation in residential building?
5. What is the limitation of the proposed solar chimney model to increase air movement in the residential building?

## 5.2 Thesis Conclusion

This section attempts to conclude the research by summarizing the major findings of the research and answering the research questions as stated. They are as follows:

### 5.2.1 Vertical Solar Chimney and On Site Climate Condition

a. The introduction of pilot testing as basic model of vertical solar chimney affected the vertical distribution of temperature at top, middle and bottom pipe, significantly induced air velocity
b. The temperature difference on the black pipe pilot testing received the higher compare than white pipe. The results were compared on each hour times, while the white pipe received 13%, 18%, 17%, and 6% less than black pipe at 10:00, 12:00, 14:00, 16:00 and 18:00 hours
c. Influence of solar radiation on different times indicated that in the noon has significant impact on the temperature difference than in the morning and afternoon under tropical sky condition. Therefore, it is important to consider the temperature difference in solar chimney design decisions especially in the intensity of solar radiation related with sky conditions.
d. Simulations of the pilot testing were developed to predict the air velocity on the pilot testing pipe on similar condition. The results can be use to validate pilot testing at early design state. The validation result showed a good agreement between the measurement and simulation. Therefore, it is give confidence in using simulation to study air movement for next stage

### 5.2.2 Vertical Solar Chimney Configuration and Maximum Air Movement

a. Increase in chimney height had more impact on the amount of the air velocity and air flow rate received inside chimney. Hence, the results indicated that use of 3.5 m selected chimney height could increase more than 100% of the air velocity on the 1 m chimney height. Therefore determining the chimney height based on air velocity may result in higher chimney highs; as well this might induce beneficial air velocity into the chimney.

b. Application of chimney width of 0.15 m, 0.3 m and 1 m on modification of basic model indicated the maximum air velocity when the chimney width is 0.15 m. Differently, the modification of the chimney width indicated maximum air flow rate induction when chimney width is 1 m compared to the other models. Influence of chimney width of 1 m indicated that the increasing chimney width have significant impact on the air flow rate than on the air velocity.

c. Application of the chimney length affected the maximum air velocity and air flow rate on different modifications. Increase of chimney length until 3 m reduced the air velocity and increase air flow rate. The use 0.15 m chimney length obtained maximum air velocity.

d. The overall finding indicates that the use 0.005 m of chimney thickness is significant enough to maintain the maximum internal air velocity and air flow rate at chimney model. However, further investigation should be done to increase the potential of this thickness design approach.

e. According to table 5.1, the air velocity and air flow rate values indicated higher by the use aluminum material compared to another material. Considering the material attributes to develop chimney model; therefore aluminum is required to achieve maximum air movement. The reason can be stated as due to the influence of absorption solar radiation is high on aluminum material compared to the another material.

f. As discussed in chapter 4, the proposed solar chimney model for the following performance variables were experimented; chimney height chimney width, chimney length chimney thickness and material. The study indicated that the width and length of solar chimney model as main modification to achieve the maximum air velocity and air flow rate. Thus, the finding suggested several solar chimney geometry models for respective width (0.15 m, 0.3 m and 1 m) and length (0.15 m, 1.5m and 3 m) variables. The selected model for performance on different conditions was determined by the air velocity and air flow rate induction. The results obtained for model A1 showed the maximum air velocity and model A9 illustrated the maximum air flow rate. Hence, it can be concluded, that for a solar chimney model, the model A1, A7 and A9 can be used to determine the appropriate chimney geometry configurations at early design stage.

### 5.2.3 Vertical Solar Chimney Geometry and Air Movement Performance

a. The introduction of selected model at specific geometry model A1, A7 and A9 affected the air movement values especially at reference solar times. Generally, the A9 of solar chimney to receive the maximum air velocity and air flow rate induction that flow through the chimney pipe. As a result, this increase section area of selected model (A9) induces the maximum air velocity and air flow rate at 12:00 hours. Increase of solar radiation follow the solar times and also impact the induction values of the air movement.

b. Application of the solar chimney model A9 on the north-south and east-west orientations resulted in better air movement performance than model A1 and A7. This implies that the increasing chimney width and length increased the air velocity and air flow rate induction. These results can be combined to obtain the optimum solar chimney model. This is mainly due to the high air movement induction for adequate natural ventilation level obtained on model A9.

c. Also the results indicated that average of air velocity and air movement induction between minimum and maximum value are higher on the model A9 than model A1 and model A7 on all respectively days. Table 5.1 illustrated that the air movement induction on model A1, A7 and A9

d. The investigation of the selected model (A1, A7 and A9) showed that solar chimney model A9 was selected model for application in terraced house. The performance of solar chimney model A9 on respective conditions were given as a induction value of air movement:
    - On respectively solar times : air flow rate induction 1.5 m3/s and air velocity induction 0.4 m/s

- On respectively orientation : air flow rate induction 1.3 m3/s and air velocity induction 0.4 m/s
- On respectively days: air flow rate induction 0.8 m3/s and air velocity induction 0.25 m/s

**Table 5.1:** Influence of selected geometry models for air velocity and air flow rate on proposed solar chimney model.

| Configuration | | Air velocity induction (m/s) between maximum-minimum | | | Air flow rate induction (m3/s) between maximum-minimum | | | Optimum Result |
|---|---|---|---|---|---|---|---|---|
| | | A1 | A7 | A9 | A1 | A7 | A9 | |
| Solar times | | 0.3 | 0.35 | 0.4 | 0.05 | 0.2 | 1.5 | v :A9<br>Q :A9 |
| Orientation | North-south | 0.3 | 0.35 | 0.4 | 0.05 | 0.2 | 1.5 | v :A9<br>Q :A9 |
| | East-west | 0.25 | 0.4 | 0.4 | 0.05 | 0.3 | 1.25 | |
| Sun Path | 21 March | 0.3 | 0.35 | 0.4 | 0.05 | 0.2 | 1.5 | v :A9<br>Q :A9 |
| | 22 June | 0.3 | 0.3 | 0.25 | 0.05 | 0.2 | 0.5 | |
| | 24 September | 0.15 | 0.25 | 0.25 | 0.05 | 0.2 | 1 | |
| | 21 December | 0.15 | 0.1 | 0.05 | 0.05 | 0.15 | 0.25 | |

### 5.2.4 Vertical Solar Chimney and Optimum Building Thermal Ventilation

a. The simulation results of the field study in respectively days indicates that the predicted internal air velocity very low or less than 0.1 m/s. This is below the minimum internal air velocity for thermal comfort which is 0.25 m/s. In the case of the single storey terraced house, the lowest internal air velocity on 21 March 2005.
b. Observations on the internal air movement level revealed that terraced house with solar chimney more natural ventilation than terraced house without solar chimney for all respectively conditions (table 5.4). The results showed that the optimum conditions received the maximum air velocity at 12:00 hour solar times, on the north-south orientation and on 21 March 2005.
c. Optimum the solar times condition was obtained for the maximum

induction air velocity and reduction air temperature following selected condition (21 March and east-west orientation):

- Temperature reduction : 12:00 hour
- Air velocity induction : 12:00 hour
- Air flow rate induction : 12:00 hour

d. The maximum psychological cooling was obtained on the north-south orientation that showed the maximum temperature reduction and air velocity induction. This indicates when considering only the indoor air movement and temperature, the north-south orientation can improve thermal comfort ventilation. All orientations illustrated provide natural ventilation requirement under Malaysian conditions compared to the field study without vertical solar chimney.

e. The relationship between the thermal ventilation and the respectively days condition were determined based on the assumptions of the maximum air temperature reduction and air velocity induction on the selected days. Thus, the optimum day suggested that the maximum temperature reduction and air velocity induction on respective orientations and solar times to following respectively days:

- Temperature reduction: 21 December
- Air velocity induction: 21 March

**Table 5.2:** Influence of solar chimney condition for temperature reduction and air velocity induction on application of solar chimney basic model.

| **Configuration** | | **Temperature reduction °C (between maximum-minimum)** | **Air velocity induction m/s (between maximum-minimum)** | **Optimum Result** |
|---|---|---|---|---|
| Solar times | 8:00 | 0.07 | 0.2 | Δ T: at 12:00<br>v :at 12:00 |
| | 10:00 | 0.23 | 1.4 | |
| | 12:00 | 0.4 | 2.6 | |
| | 14:00 | 0.29 | 2.2 | |
| | 16:00 | 0.16 | 0.1 | |
| Orientation | North-south | 0.9 | 0.25 | Δ T: north-south<br>v : north south |
| | East-west | 0.9 | 0.21 | |
| Sun Path | 21 March | 0.9 | 0.21 | Δ T: 21 Dec<br>v :21 March |
| | 22 June | 0.14 | 0.2 | |
| | 24 September | 0.7 | 0.18 | |
| | 21 December | 1.39 | 0.15 | |

## 5.3 Suggestions for Further Research

This research has revealed two significant findings. Firstly, the introduction of vertical solar chimney at terraced house is significantly affected the stack ventilation. Thus, it can maintain the preferable internal air velocity required for thermal comfort (1.0 m/s the most favourable air velocity for thermal comfort) at activity level of selected room. Secondly, with single side ventilation (similar cases for terraced house bedroom in the planning guideline) minimum air velocity inside of the room is created. As a result the internal air velocity performance of the single storey terraced house is very low. Some of the area did not even achieved 0.1 m/s of internal air velocity. However, the introduction of vertical solar chimney at (specific position) of the roof increases the indoor air velocity.

This study has suggested how a simple vertical solar chimney strategy can be effectively used to reduce of the air temperature and increase internal air movement. In other words little had been known about the relationship between the vertical solar chimney and thermal ventilation in hot humid tropical climate. Therefore, the vertical solar chimney design strategies require a rethinking in solar induced ventilation. However, several areas of study need further investigation, to develop the knowledge of the use vertical solar chimney strategies in Malaysia and regions with similar climates. Therefore, it is recommended that future research could look further into this area in order to strengthen and compliment this research. The following are some suggestions:

a. Investigation on the effectiveness of the solar chimney geometry. Apart from the width and length of the 3.5 m high solar chimney, the other factors need to be investigated are; the ratio between high, width and length of vertical solar chimney for selected condition (solar times, orientation and sun path).
b. Investigation on the effectiveness of material of the solar chimney for improvement air velocity. Effectiveness of the solar chimney also depends on the material as they affect on solar radiation absorption. Therefore, these aspects need to be studied in terms of value of

conductivity, density, specific heat etc. It may also contribute to the aesthetics of the solar chimney.

c. Further investigations are required to determine the effects of the solar chimney strategy on different room size on various building forms.
d. In terms of thermal ventilation, the influence of humidity on air velocity to be explored. In hot and humid tropics, high humidity creates discomforts in buildings. Therefore, information on relationship between solar chimney strategies, air velocity, temperature and humidity are very important.
e. Investigation of other different the solar chimney and their influence on improve the stack ventilation. A detail study should be carried out to look into the impact of combination the solar chimney with building elements (roof, wall, opening).
f. Further studies need to be carried out to develop a method to define solar chimney by considering the total heat transfer. In hot and humid tropics influence of solar intensity on thermal effects are significant. Therefore, considering the total heat transfer may be an important aspect in determining different solar chimney strategies. Studies on heat transfer properties can be used to develop a design method to determine different solar induced strategies for the tropics.
g. Further study and analysis on existing and new single storey terraced house typology should be carried out to give a better indication on the indoor air velocity distribution in relation to the natural ventilation and thermal comfort performance. Hence, a better comparison on the performance can be carried out.
h. Further investigation on the effect of surrounding building or urban proximity condition should be carried out using CFD simulation. This is to predict the actual condition of the wind speed distribution in the terraced house and the effect to the natural ventilation and thermal comfort performance at the urban situation.

Finally, it can be acknowledged that this work is among the pioneer works done on the study of vertical solar chimney ventilation at terraced house especially in

Malaysian hot and humid tropical climate. It is just a small contribution by the researcher towards understanding the effect of vertical solar chimney design on the natural ventilation and thermal comfort condition of the built environment. It is hoped that future development of terraced house with vertical solar chimney will be more beneficial to the user.

# BIBLIOGRAPHY

Abdul Majid Ismail (1996). Wind Driven Natural Ventilation in High-Rise Office Building With Special Reference To The Hot-Humid Climate of Malaysia, University of Wales College of Cardiff, PhD, Thesis,

Alfonso, Clito (2000). Solar Chimneys: Simulation and Experiment, Energy and Buildings, Vol. 32, Pergamon Press., pp. 71–79.

ASHRAE Handbook Fundamentals (1989). American Society of Heating and Air Conditioning Engineers, Inc. Atlanta.

Awbi, Hazim B. (2003). Ventilation of Building, Spon Press, London.

Bansal, N.K, R. Mathur, M.S. Bhandari (1994). A Study of Solar Chimney Assisted Wind Tower System for Natural Ventilation in Buildings, Building and Environment, Vol. 29, 4, Pergamon Press., pp. 495-500.

Barozzi G.S., Imbabi M.S.-E., Nobile E., Sousa A.C.M. (1992). Physical and Numerical delling of a Solar Chimney-based Ventilation System for Buildings. Building and Environment, Vol.27, No.4, Oxford: Pergamon Press, pp.433-45.

Bouchair (1994). Solar Chimney for Promoting Cooling Ventilation in Southern Algeria, Building Services Engineering Research and Technology, Vol.15, pp.81-93.

Drori, U, G.Zizkind (2004). Induced ventilation of a one-story real size building, Energy and Building. Vol 36.

Evans M (1980). Housing Climate and Comfort, The Architectural Press Ltd., London,

Flomerics Limited (2000). “Flo Vent User’s Guide”

Gan, G, A. (1998). Numerical Study of Solar Chimney for Natural Ventilation of Buildings with Heat Recovery, Applied Thermal Engineering, Vol. 18, Oxford: Pergamon Press , p 1171-1187

Hamdan, Mohd. and Nugroho, Agung Murti (2005). Towards Development of Tropical Solar Architecture: The Use of Solar Chimney as Stack Induced Ventilation Strategy, Proceeding World Renewable Energy Regional Congress and Exhibition, Jakarta

Hui Seng Kin (1998). Natural Ventilation of Low Cost Dwellings in The Hot Humid Tropics Malaysia, Master Thesis

Hunt, G.R., Linden, P.F. (1999). The Fluid Mechanics of Natural Ventilation—Displacement Ventilation by Buoyancy-Driven Flows Assisted by Wind. Energy Environ. Vol. 34, pp. 707–720

Khedari J, Weerapong Mansirisub, Sompong Chaima, Naris Pratinthong, Jongjit Hirunlabh (2000). Field Measurements of Performance of Roof Solar Collector, Energy and Buildings, Elsevier Science Ltd. ,Vol 31, pp 171–178.

Khedari J. (2000). Field Measurements of Performance of Roof Solar Collector, Energy and Buildings, Elsevier Science Ltd. ,Vol 31, pp 171–178.

K.S. Ong , C.C. Chow (2003). Performance of a Solar Chimney, Solar Energy, Elsevier Science Ltd. ,Vol. 74, pp 1–17.

Malsiah Hamid (2001). Keberkesanan Pengudaraan Semualajadi pada Rekabentuk Bilik Darjah Sekolah Piawai di Malaysia, Tesis Sarjana, UTM

Mills A.F. (1992). Heat Trans Boston: Irwin., p.293.

M Rajeh (1989). Natural Ventilation in Terrace Housing of Malaysia : Effect of Air Well on Air Flow and Air Velocity, University of Queensland, Master Thesis.

Nugroho, Agung Murti and Hamdan, Mohd. (2005). Possibility to use Solar Induced Ventilation Strategies in Tropical Conditions by Computational Fluid Dynamic Simulation, Proceeding International Seminar Sustainable Environmental Architecture 6, Bandung Indonesia,

Ossen, Dilsan Remaz (2005). Optimum Overhang Geometry for Controlling Penestration Heat Gain and Building Energy Saving in Tropics, PHd Thesis, Universiti Teknologi Malaysia

Sapian Abdul Razak (2004). Possibilities of using Void to Improve Natural Cross Ventilation in High-rise Low Cost Residential Building, Thesis.

Satwiko, Prasasto (1994). The Development and Modelling of a Natural Ventilation System for Indonesian Conditions, Thesis Master of Building Science, Victoria University of Wellington.

Satwiko, Prasasto (2005). Solar-Wind Generated Roof Ventilation System (SiVATAS) for a Warm-Humid Climate, International Journal of Veentilation, Vol 3 No 3, 2005

Waewsak. J, Hirunlabh, Khedari, U.C.Shin (2003). Performance Evaluation of the BSRC multi purpose Bio-Climatic Roof, Building and Environment, Vol 38.

Watson D., ed. (1979). Energy Conservation through Building Design. An Architectural Record Book, New York: McGraw Hill Book Company., p.34.

# APPENDIX A

## TOWARDS DEVELOPMENT OF TROPICAL SOLAR ARCHITECTURE: THE USE OF SOLAR CHIMNEY AS STACK INDUCED VENTILATION STRATEGY

by
Dr. Mohd Hamdan bin Ahmad
Agung Murti Nugroho
Dilshan Remaz Ossen

Departement of Architecture Faculty Built Environment
UNIVERSITI TEKNOLOGI MALAYSIA
81310 Skudai, Johor Bahru, Malaysia

This paper is presented at the 2nd World Renewable Energy Regional Congress and Exhibition, Jakarta, Indonesia, April, 2005.

### Abstract

The approaches in "*Renewable Energy*" are diversifying. We need to make shifts in our perceptions, our accepted models of thinking, making and doing, to "*Renewable Energy"* with our built environment in response to the tropical context. The climatic conditions of these regions are characterized by high air temperatures, high relative humidity's and very low wind speeds, which make the environmental conditions uncomfortable. The use of solar chimneys in buildings is one way to increment natural ventilation and, as a consequence, to improve indoor air quality. In this context, this paper presents the first stage of full development on solar chimney as stack induced ventilation strategy. This research uses a cylindrical PVC pipe to a solar chimney in low cost residential building. The performance of the chimney was evaluated by predicting the temperatures along the pipe. The effects of air gap and solar radiation intensity on the performance of different chimneys were investigated. In order to verify the theoretical model, experiments were conducted on black and white pipe with each 13 feet heights. This black and white colored pipes were used to understand the effect of color on temperature difference along the pipes. The results indicated a temperature different of 5-6°C before top and bottom . This results encourage research to develop the solar chimney in tropical condition.

Key word : Solar Energy, Tropical Climate, Solar Chimney

### A. Introduction

In terms of passive cooling, tropical climatic regions the most difficult problem to solve. The simplest and the most effective solution for active cooling is to introduce air conditioners. However, such equipment involves high initial and operational costs for installation, power and maintenance. Therefore air conditioners are unlikely to be applied widely, in particular, for residential building. Thus, a passive cooling system is more desirable. Although in Malaysia, passive cooling method is a popular cooling strategy adopted in residential building, researches (Pan, 1997; Tan, 1997; Jones, 1993; Zulkifli, 1991; Hui, 1998; Abdul Razak, 2004) have shown that its natural ventilation performance could not provide internal thermal comfort. Climate conscious design in the equatorial tropic assumes that air movement is one of the main cure fure thermal comfort ills.

According to Hui (1998), the indoor air velocity in low rise building 0.04m/s – 0.47 m/s (Hui, 1998). The reasons may be due to inappropriate design solutions for indoor air movement or low outdoor air velocity remains to be determined. However, recent data from the Malaysian Meteorogical Service Development showed that mean outdoor air velocity is between 1 m/s to 1.5 m/s. In theory, there are two natural ventilation mechanisms (ASHRAE, 1997). First is by wind pressure and the second is by temperature difference or stack effect. Both mechanisms have the same aim, which is to act as an aid to create air movement and consequently control the indoor air temperature (Sapian, 2004).

Natural ventilation may result from air penetration through a variety of unintentional openings in the building envelope, but it also occurs as a result of manual control of building’s openings doors and windows. Air is driven in and out of the building as a result of pressure differences across the openings, which are due to the combined action of wind and buoyancy-driven forces. Today, natural ventilation is not only regarded as a simple measure to provide fresh air for the occupants, necessary to maintain acceptable air-quality levels, but also as an excellent energy-saving way to reduce the internal cooling load of housing located in the tropics. Depending on ambient conditions, natural ventilation may lead to indoor thermal comfort without mechanical cooling being required.

One of the more promising passive cooling methods for tropical climatic regions is the solar chimney (Satwiko, 1994). This method employs buoyancy driven ventilation to introduce physiological cooling. It utilizes solar radiation, which is abundant in these regions, to generate the buoyant flow. However, as currently applied, the induced air movement is insufficient to create physiological cooling. More studies are needed to improve the ventilation performance of this cooling method. The basic form of the solar chimney has long been known, and applied in vernacular architectural designs. In general, the induced air movement is not used directly to suck indoor air. Instead, it is used for ventilating the building (such as in the double skin building). A stack chimney is usually designed in combination with a wind tower in hot arid climatic regions. In many types of ventilated building, winds are considered to be more important than buoyancy. This is because wind induced ventilation flow is commonly stronger than stack induced flow, in particular, in low rise buildings. Stack induced ventilation (or buoyancy driven ventilation) is, so far, insufficient to create physiological cooling in warm humid climatic regions (tropical climate). Velocities associated with natural convection are relatively small, usually not more than 2 m/s (Mills, 1992). A milk house that was built in 1800s is a historical example of a stack chimney application in . This building has double walls which are combined into a shaft at their upper end. The south facing shaft wall was made from glass. The solar radiation that penetrates the glass heats the inner wall. Eventually, this inner wall heats the air which will rise and induce a flow of fresh air from the openings below (Watson, 1979). Two examples of stack chimney concepts is solar collector and stack height. The former shows one way to amplify stack effect by utilizing solar collectors and increasing the height of the hot air column (stack height). Critical parameters of this design are the stack height and cross sectional area of its inlet and outlet. A massive and high version of this type is needed to generate indoor air velocity as high as 1.0 to 2.0 m/s, which can be achieved easily in ordinary shallow buildings (with no obstruction at all). An attic that was designed to create a green house effect and eliminate the chimney. It heats a much greater volume of air compared to the previous design. Thus, in terms of air changes, it is likely to have a better performance (Baker, 1994). Another solar chimney design called *the Nigerian Solar* Roof. It was studied by Barozzi et. al., using physical and numerical (Computational Fluid Dynamics Codes) modeling and data from Ife, Nigeria. Two experimental findings were noticeable. Firstly, both physical and numerical experiments gave almost identical results. Thus, it showed the potential of CFD Codes to simulate air flow. Secondly, both types of modeling indicated the presence of buoyancy driven ventilation within the model. However, the air speed within the occupant's zone was too low to create physiological cooling. The term *Solar chimney* is used extensively in Barozzi's article as the *chimney* shape is quite obvious. On the other hand, in this thesis the term *Solar chimney* seems to be more suitable as the chimney takes the form of a roof (Barozzi, 1992).

Air movement created by the stack effect is usually too weak to achieve physiological cooling. It is less than the recommended air speed range for cooling of 0. 15 to 1. 5 m/s in tropical condition (Satwiko, 1994). It can be seen that two means are available for improving air movement: firstly, by increasing the air volume (stack height) and secondly, by increasing the air temperature difference. Meanwhile, the indoor air temperature has to be kept low. All the above designs involve stack effect. However, in terms of construction (complexity, technology, etc.) and material (cost, durability, availability, etc.) these designs are not suitable for wide application in low cost housing in tropical countries.

Studies of solar chimneys involve aerodynamic experiments on buoyancy problems which can be done using both physical modeling and computer modeling methods, the latter based on computational fluid dynamics (CFD). Both methods have advantages and disadvantages. Physical modeling is considered to give results that are easy to check. However, for an experiment which involves various model designs this method can become expensive and time consuming. The computer simulation method, on the other hand, allows easy modification of the design, more precise results in less time. Even though computer simulation has a high initial cost (for its hardware and software), the final cost might be lower than that of the physical experiment since changes in the model designs can be made easily.

The objective of this study is to present a experimental model to :

- understand the temperature different and distribution along the chimney
- understand the impact color in temperature distribution.

## B. Basic Theory of Stack Effect

Stack effect pressures are generated by differences in air density with temperature, i.e. hot air rise and cold sinks. The air within a building during the wintertime acts like a bubble of hot air in a sea of cold air. In the summer time the situation is reserved, although air temperature differences area usually less.

The density of dry air, $\rho a$, varies with temperature. The greater the height of a column of air, the greater the potential difference in pressure if that column is at a different temperature. The pressure difference generated by a column of air h meters high with temperature difference between indoor and outdoor air at standard temperature and pressure is approximately:

$$\Delta P = 3465 \cdot \Delta h \cdot \left(\frac{1}{T_o} - \frac{1}{T_i}\right) \text{ [Pa]}$$

where To and Ti are the outdoor and indoor temperature respectively, (in Kelvin = Celcius+273)

For example, if the air in a one meter high cylinder, open at the bottom and containing room temperature air (20°C) is taken into the outdoors at the temperature of -10°C, an outward pressure of the 1.34 Pa would act at the top (Figure 2). The pressure at the bottom must be zero since it is connected to the outdoors. The horizontal plane at which the pressure equals the outdoor pressure (i.e. the difference is zero) is called the Neutral Pressure Plane (NPP).

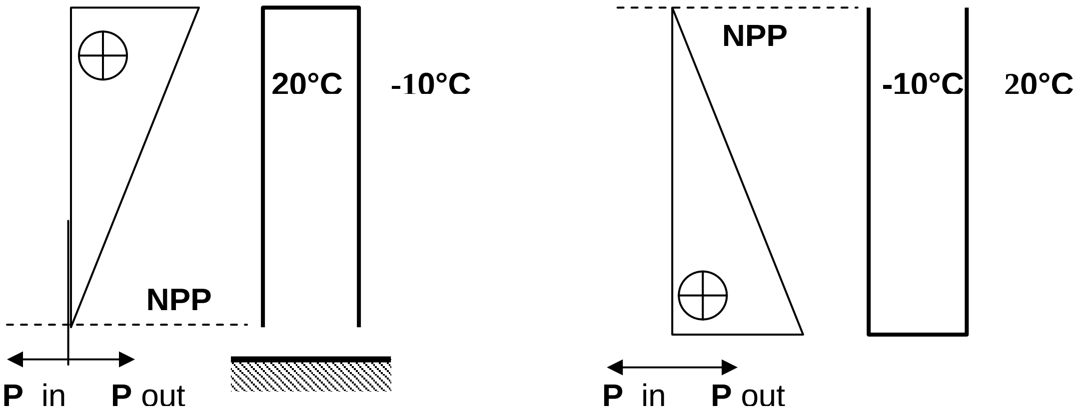


Figure 1. Pressure in open-ended cylinder due to bouyancy

If the cylinder remained outdoors, the air it contained would slowly cool down to the exterior temperature and no pressure difference would exist. If the cylinder were then inverted and brought back indoors, the pressure at the closed end of the cylinder would again be 1.34 Pa acting outward as the cold air fell downward relative to the indoor air.

In the above examples, no flow occurred because no flow path was provided. If an open ended cylinder containing room temperature air where used, any temperature difference between the cylinder and the surrounding air would cause flow, and the warm air would be immediately removed and replaced with cool outdoor air. However, if a heating coil were added to the cylinder to maintain the air temperature at 20°C, airflow in the bottom would be heated. This is analogous to a heated building. Friction would slow air flow and result in a constant pressure drop along the height of the cylinder. Note that the NPP would now be located at mid height and that air flow is involved (Figure 3). Obviously, the less air flowing through this cylinder the less heat energy required to maintain the interior of the cylinder at 20°C.

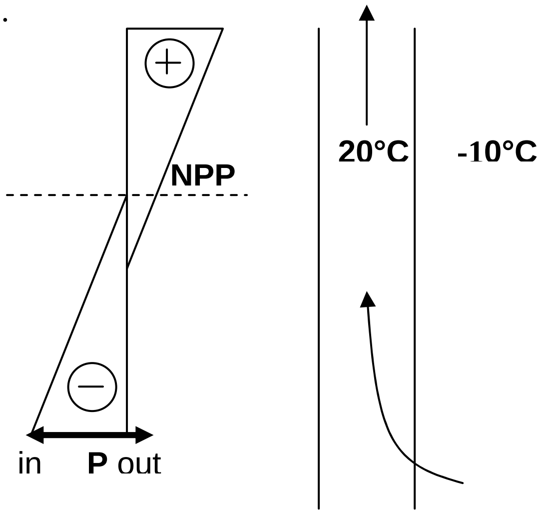
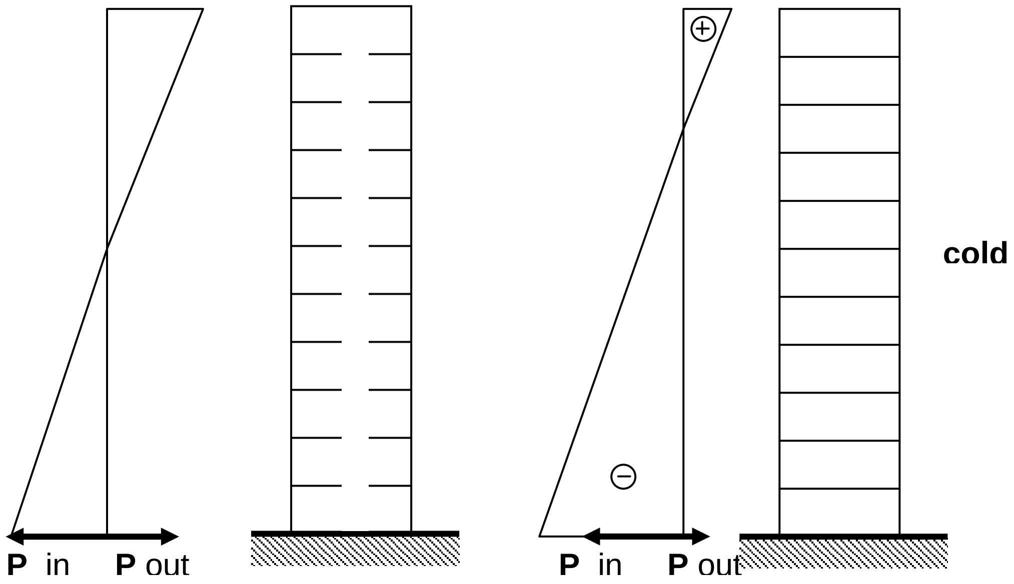


Figure 2. Flow through a heated cylinder or a building



Floors leakier than walls building acts like a perforated tube

Top leakier than bottom NPP rises

Figure 3. Stack effect in real building

In warm climate and during warm weather, stack effect reserves and air is often drawn in at the top and pushed out at the bottom. Infiltration of warm moist air in warm weather can cause as many problems as exfiltration of warm moist air in the winter (Straube 2001).

## C. Research Design Solar Chimney Development

### a. Stage of Research

The Solar Chimney project is initiated at Universiti Teknologi Malaysia in 2004 to investigate the opportunities to apply solar chimney to achieve indoor thermal comforts in tropical climatic conditions. The research work consists of

Stage One : A literature review is conducted to get a clear understanding of the state-of-the art knowledge in the field. This will enable to orientate our research work towards the areas which had not been covered in the past. This will include understanding the theoretical back ground, solar chimney design considerations and factors, user needs and response and finally typology of solar chimney used in building industry.

Stage Two. Development of actual scale solar chimney for initial experiments to understand the different parameters and their impact on stack ventilation. The experiment includes designing an alternative solar chimney in actual scale, analysis on the alternative design, finalizing the design and computes the construction drawings.

Stage Three: Studies of the impact of solar chimney on Indoor Climate using computer modeling and simulation.

Stage Four: Implementation of prototype full scale model in selected building type. Indoor Climate monitoring and data collection for further analysis.

Stage Five : Comparative Studies between computer model and physical model to get an effective results and evaluation base on the performance of the solar chimney design.

Stage Six: Suggestions and recommendations to improve the design of solar chimney and development of design guideline for solar chimney in hot humid climates.

This study completes stage two of the solar chimney project. The duration for this stage is two months and the measurements were taken for one month duration.

### b. Experimental investigation in initial test model

*Model test*

The two experimental solar chimney are 13 ft high and 0.75 ft diameter cylindrical PVC vertical pipe, supported structurally by timber framework. A schematic illustration of the features of this design is shown in Figure 4. An opening at the top and bottom of the pipe which air to enter the channel. The resultant "greenhouse effect" is expected to keep chimney temperatures above ambient temperatures consistently. The models were black and white surface colors respectively. A different color surface was used to exaggerate the internal temperature difference along the solar chimney. It is not envisaged that selective surfaces would be employed in practice. The accurate monitoring of the outdoor air velocities and air velocity within the chimney is difficult. Therefore air temperature is used to evaluate the performance of the solar chimney.

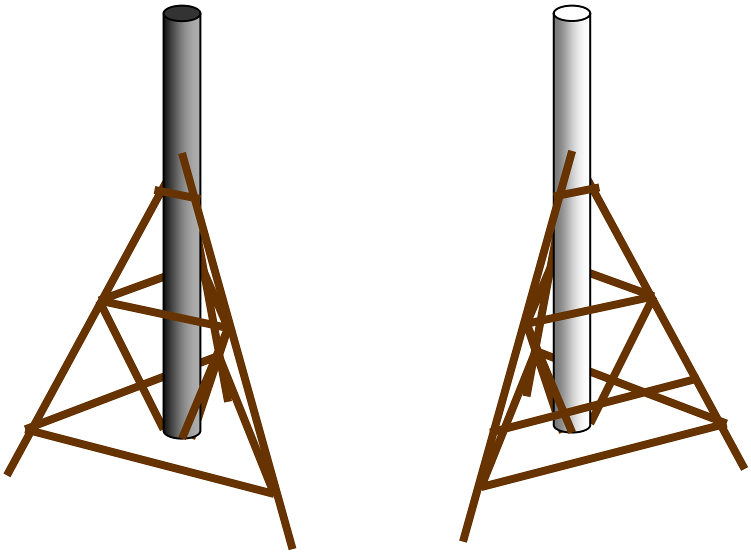


Figure 4. two initial test model : black pipe and white pipe

*Measurement tool*

The tool used to measure the temperature and humidity is the Dickson temperature and humidity data logger. Dickson data logger have dual channel sensors record and can store 2 months of humidity and temperature data with accurate and lightweight. Dickson data logger can store thousand of sample points and specifications represented in table 1.

Table 1. Specifications and system requirement Dickson data loggers

| Humidity specifications | |
|---|---|
| Range | 0-95% RH |
| Accuracy | +/- 2 % RH |
| Temperature Specifications | |
| Range | -40° to 176°F |
| Accuracy | +/- 1.8°F  +/- 1°C |
| General Specifications (16.256 per channel | |
| Data Storage | 32.512 sample points |
| Interval | 10 sec to 24 hrs |
| Power (included) | Lithium Cell |
| Battery life | 5 years |
| Dimension (LWH) | 3.1” x 2.1” x 0.9” |

*Procedure*

Duration set data logger in every 5 minutes from 10.00 Am to 18.00 PM. There are two kind measurement: first, measurement of different temperature for two color pipe (black and white pipe) and second, measurement of distribution temperature in black pipe. Data logger position for first measurement put in three point each one pipe and one point in outdoor. The point in each pipe on the bottom, middle, and top (figure 5). Data logger position for second measurement put in six point in black pipe and one point in outdoor (figure 6).

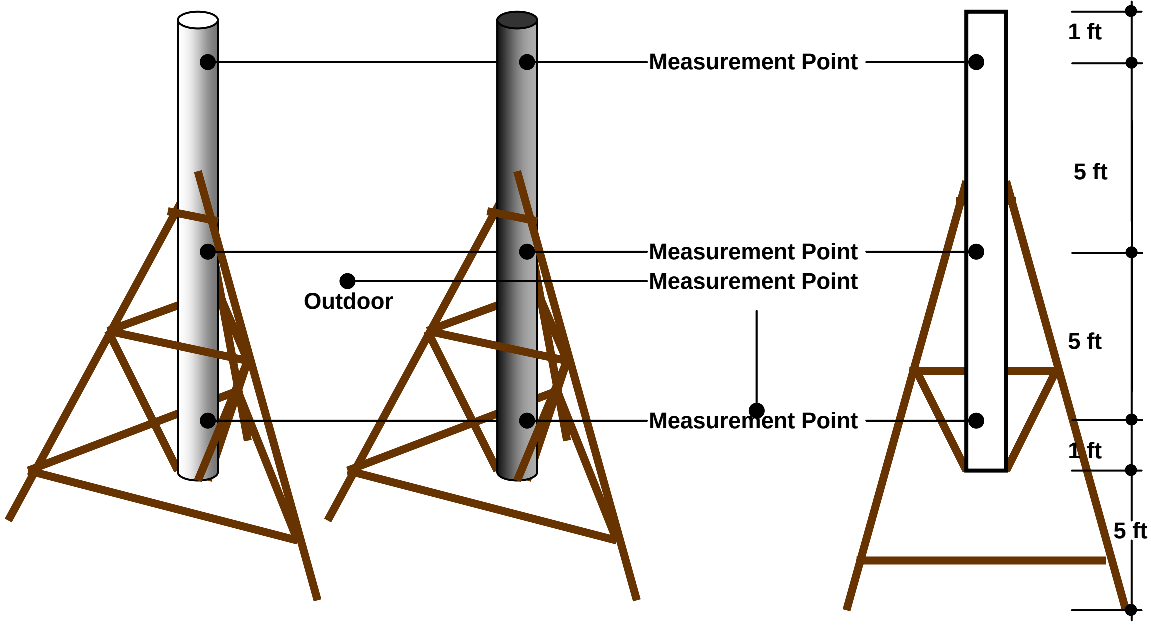


Figure 5. Data logger position to measurement different temperature in white pipe, black pipe and outdoor

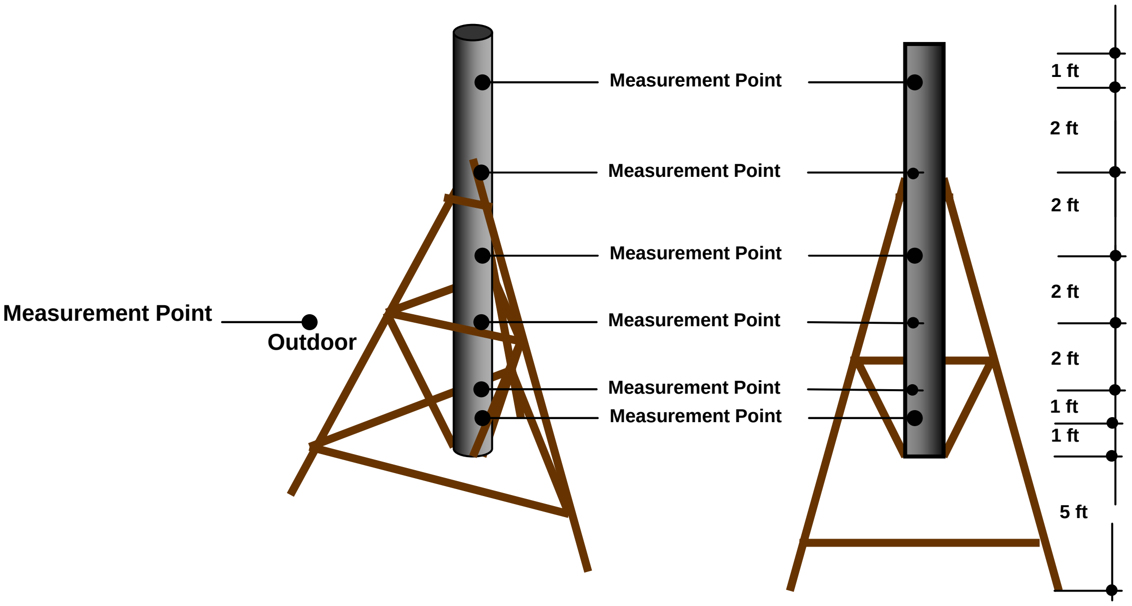


Figure 6. Data logger position to measurement distribution temperature in black pipe

## c. Result and discussion for initial test model

*Site Temperature and Humidity*

A typical diurnal variation of the mean chimney air temperature against the outdoor temperature is illustrated in Figure 7. It can be observed that the chimney air temperatures were significantly above the outdoor in 10.00AM until 18.00 PM. Peak ambient temperature (outdoor temperature) of 30°-36°C, this condition base ventilation thermal comfort need 0.55 m/s air velocity (ASHRAE)

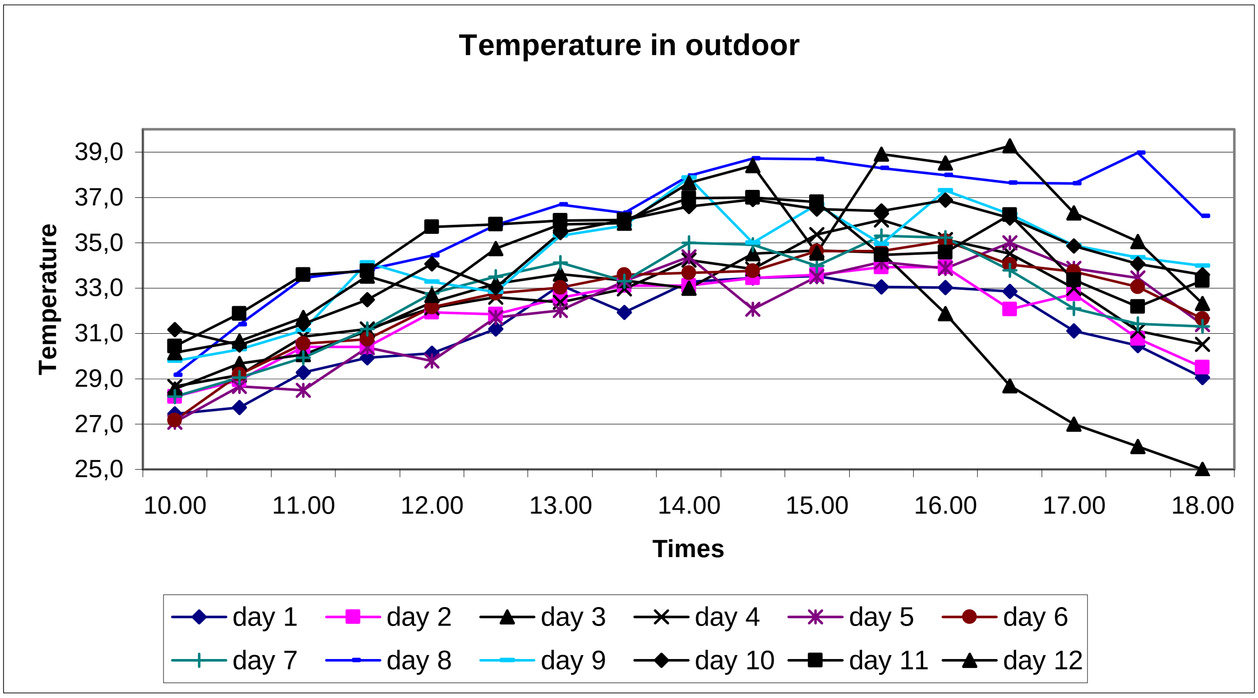
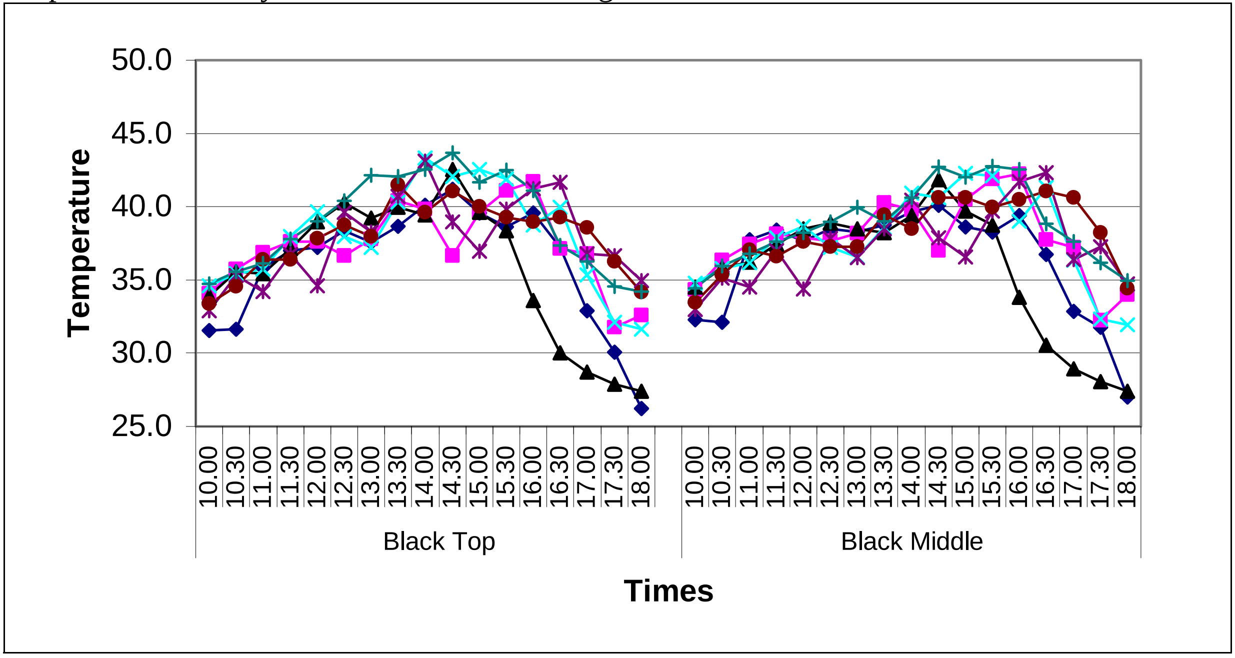


Figure 7. Temperature and humidity average in outdoor site

*Temperature different*

Preliminary investigations were conducted to determine the air temperature different at position of outdoor, bottom, middle and top in black and white pipe. Figure 8 showed that the heat absorbing PVC pipe temperature was highest in black pipe cases. The black pipe temperature was generally higher than the air temperature in white pipe. Air temperature in black pipe increasing start in 10.00AM and peak in 13.00 PM. Air temperature decrease but still high from 14.00 PM until 17.00 PM. The different temperature in black pipe for black top-black bottom 5°C and black top-outdoor 9°C. Effectiveness for time to large different temperature in 12.00PM until 16 PM for black top-outdoor. The measurement in day 3 showed the cloudly impact for decreasing different temperature. The heat absorbing black pipe exhibited a greater influence on the air temperature different for the narrower chimney. For the diameter chimney pipe, 0.75 ft, the air temperature distribution across the air gap depth was relatively flat in the central core region.



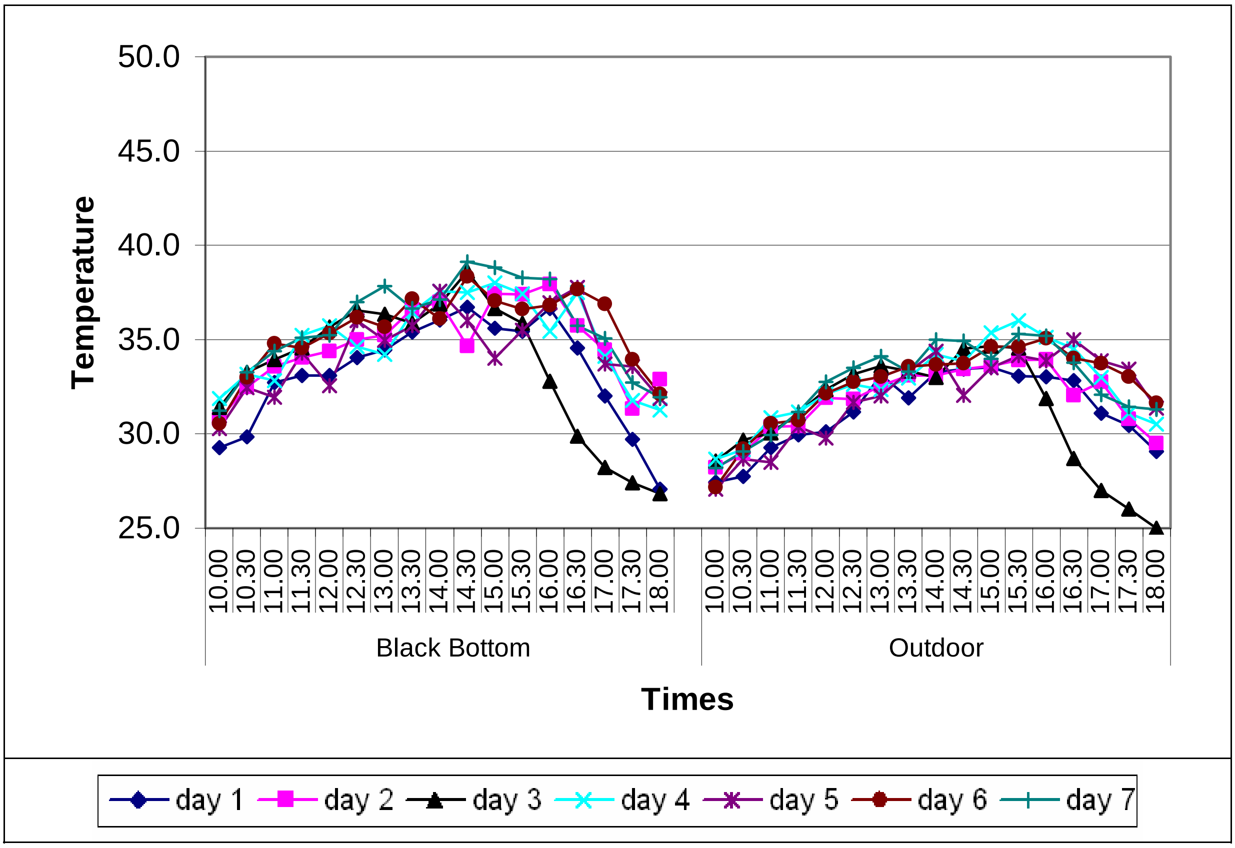
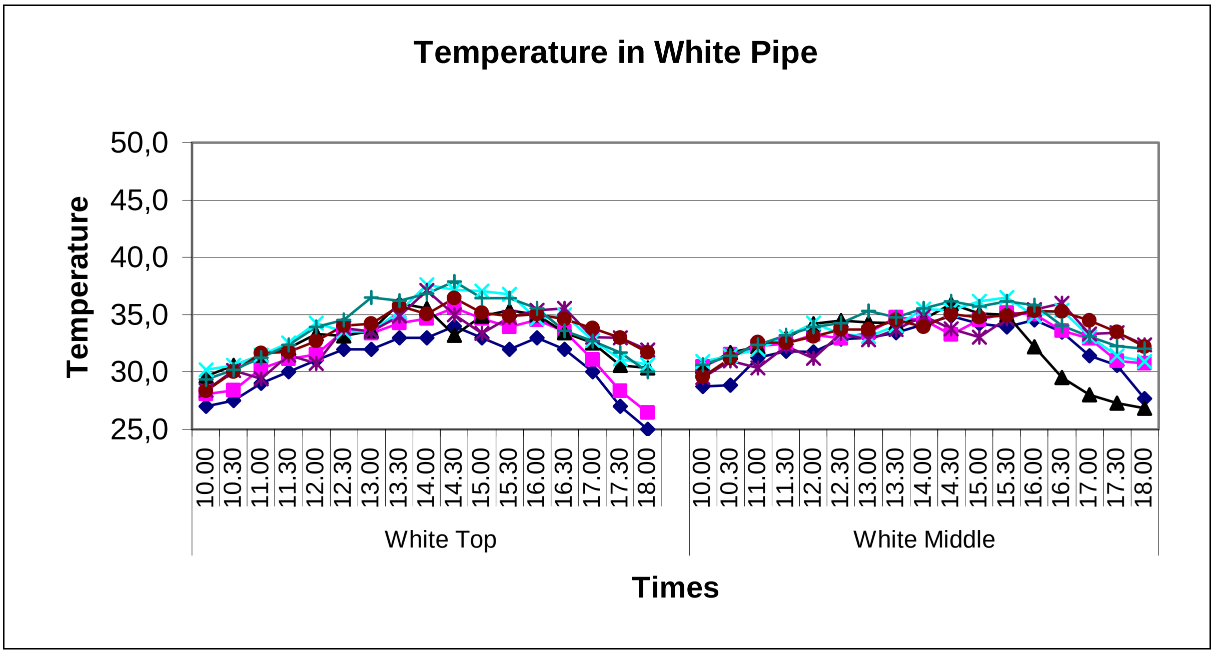


Figure 8. Temperature in black pipe

Figure 9 showed that the heat absorbing white PVC pipe. The white pipe temperature was generally lowest than the air temperature in black pipe but more high than outdoor. Air temperature in white pipe increasing start in 10.00AM until 16.00 PM. Air temperature in white top lowest than white bottom in 10.00 AM-13.00 PM and 16.00PM-18.00PM.



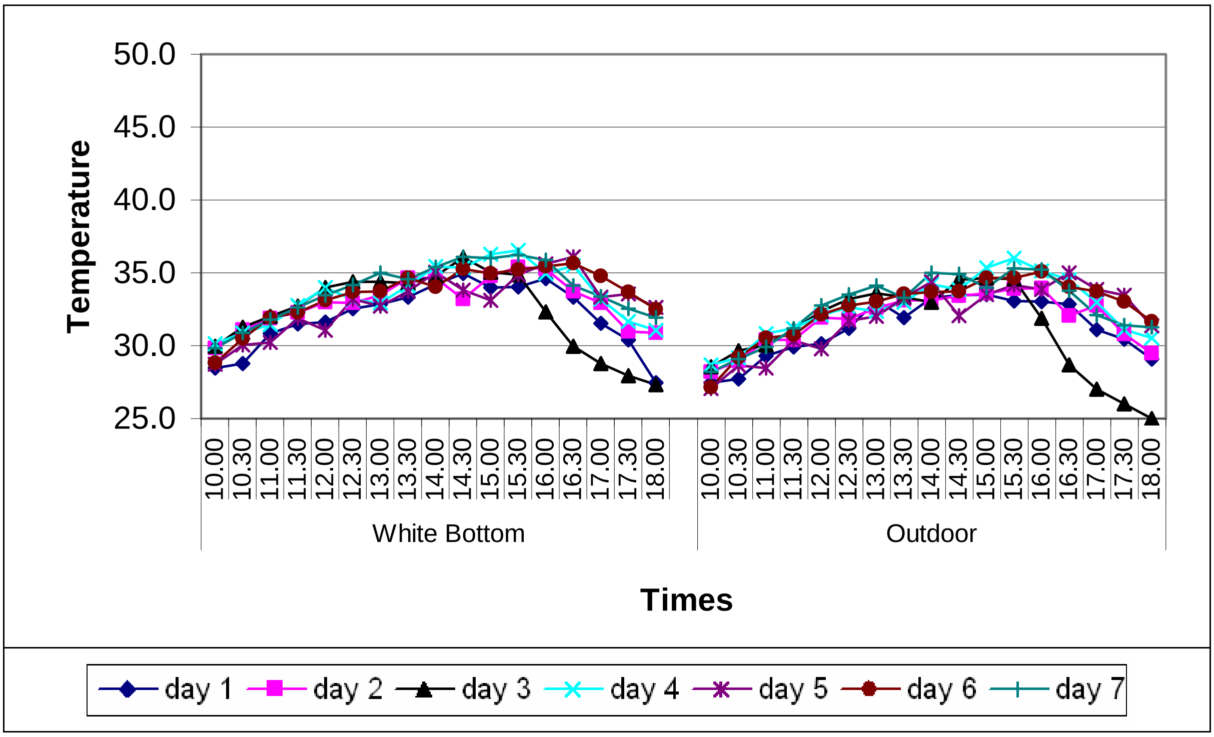
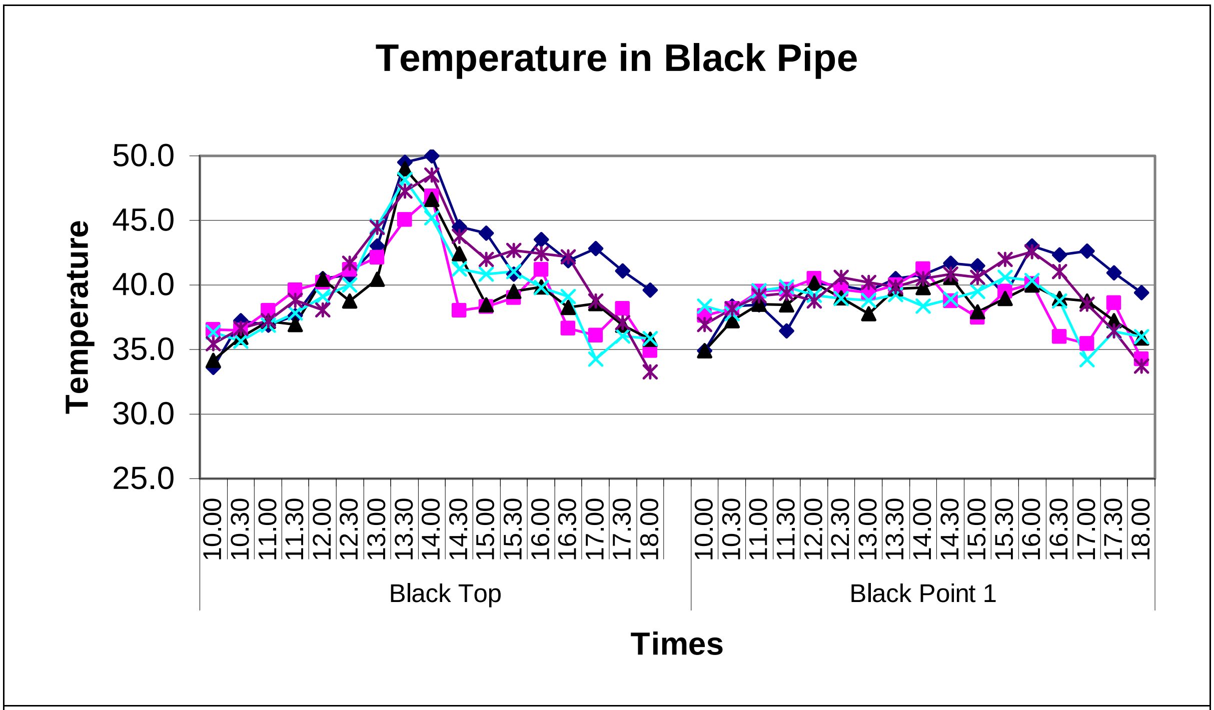


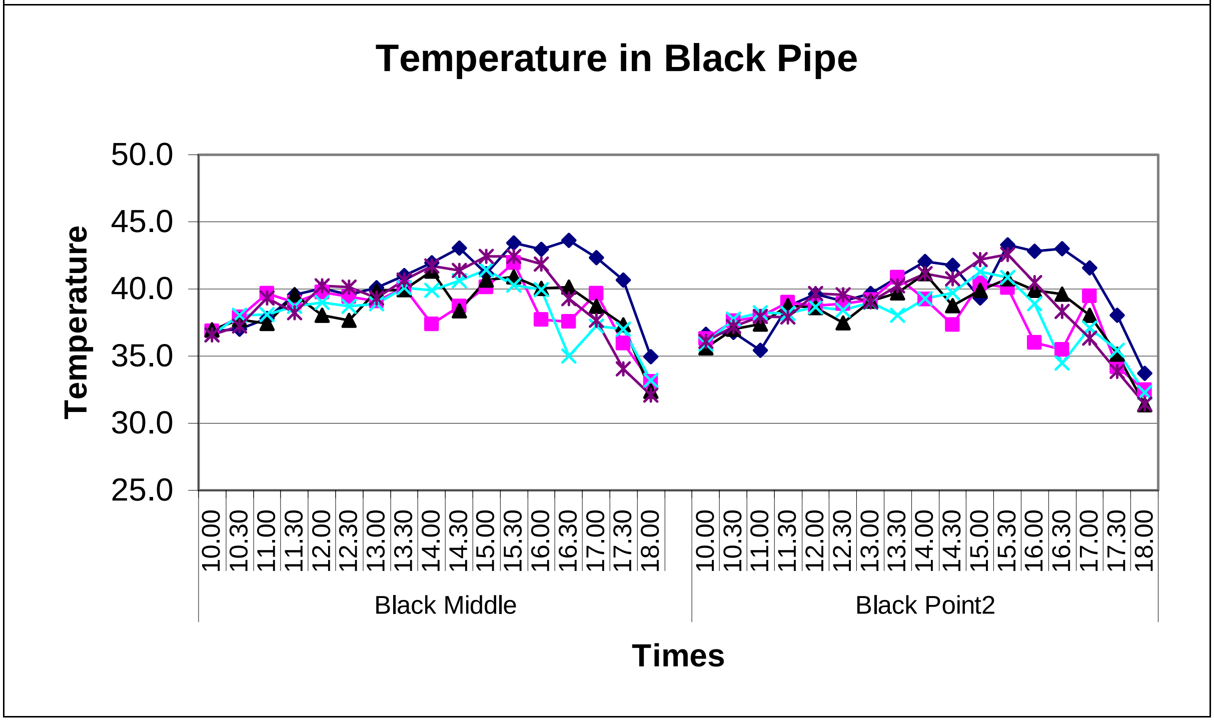
Figure 9. Temperature in White pipe

Generally, temperature different in black and white pipe is 5°C -10°C for compare in black top and white bottom. The effective time to this condition in 12.00PM-16.00PM. Half of black pipe potential to absorbs heat gain base no large different temperature for black top and black middle no

*Temperature distribution*

The temperature distribution in black pipe is illustrated in Figure 10. The point of black top pipe have highes temperature and lowest in point of black bottom. Temperature distribution depend on the time and area. The significant different temperature started in point 2 for 10.00 AM -13.00 PM. Posisition of point 2 in 4 feet from bottom or 1/3 high pipe.


Temperature in Black Pipe
Temperature
50.0
45.0
40.0
35.0
30.0
25.0
10.00
10.30
11.00
11.30
12.00
12.30
13.00
13.30
14.00
14.30
15.00
15.30
16.00
16.30
17.00
17.30
18.00
Black Top
Black Point 1
Times


Temperature in Black Pipe
Temperature
50.0
45.0
40.0
35.0
30.0
25.0
10.00
10.30
11.00
11.30
12.00
12.30
13.00
13.30
14.00
14.30
15.00
15.30
16.00
16.30
17.00
17.30
18.00
Black Middle
Black Point2
Times

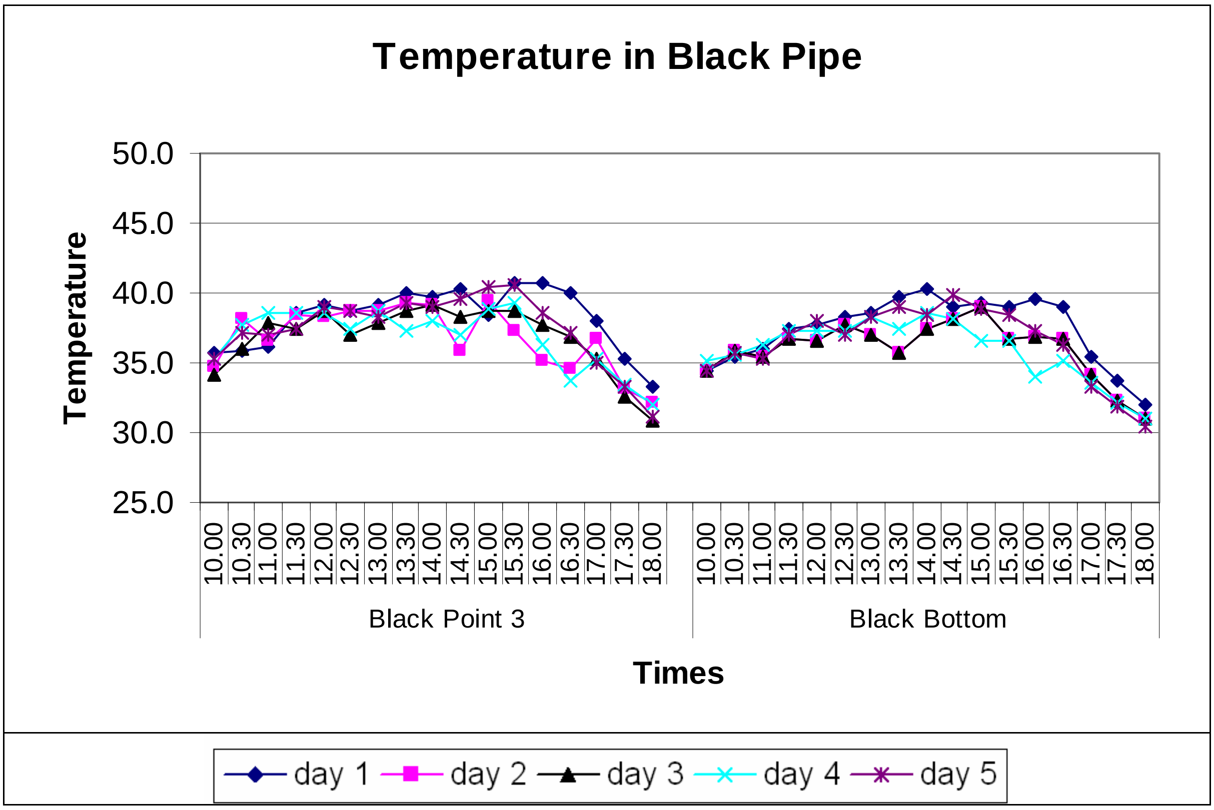


Figure 10. Temperature distribution in black pipe

**D. Conclusion**

The results obtained from the experimental solar chimney have illustrated that solar chimneys if designed properly can maintain chimney air temperatures consistently above the outdoor temperature which would enhance the desired buoyancy-induced air flow through the chimney. The desired performance was achieved with the solar "greenhouse" chimney studied. Better performance was obtained with a solar radiation absorbing surface within the chimney.

The study of the different color surface solar chimney has shown promising results. It is possible to create a maximum air flow of black pipe. The incorporation of a combined black and white color solar chimney to a pipe can be increase different temperature in the top and bottom. To obtain such an air flow rate the pipe should be extended the height and width pipe.

This air velocity could be increased by increasing the surface area of solar chimney. Finally, it is our opinion that natural ventilation seems to be feasible and viable and the opportunity to development solar chimney use low material for low cost residential building for renewable energy. This, of course, awaits a large scale testing in order to see how effective such a ventilation system would be in a real building.

Reference

(1) ASHRAE Handbook Fundamentals, American Society of Heating and Air Conditioning Engineers, Inc. Atlanta, 1989

(2) Baker E., Floro C.J., Gostelow J.P. and McCaffrey J.J., Solar Heating and Cooling Systems. Design for Australian Condditio Sidney: Pergamon Press., p.126. , 1994

(3) Barozzi G.S., Imbabi M.S.-E., Nobile E., Sousa A.C.M., Physical and Numerical delling of a Solar Chimney-based Ventilation Sy stem for Buildings. Building and Environment, Vol.27, No.4, October, Oxford: Pergamon Press, pp.433-45. , 1992

(4) Hui Seng Kin, Natural Ventilation of Low Cost Dwellings in The Hot Humid Tropics Malaysia, Thesis, 1998

(5) K.S. Ong , C.C. Chow, Performance of a Solar Chimney, Solar Energy 74 1–17, 2003

(6) Mills A.F, Heat Trans Boston: Irwin., p.293., 1992

(7) Sapian Abdul Razak, Possibilities of using Void to Improve Natural Cross Ventilation in High-rise Low Cost Residential Building, Thesis, 2004

(8) Satwiko, Prasasto, The Development and Modelling of a Natural Ventilation System for Indonesian Conditions, Thesis Master of Building Science, Victoria University of Wellington, 1994

(9) Straube, J.F., “Air Flow Control in Building Enclosures: More than Just Air Barriers”, Proc.of 8 th Bldg Sci.&Tech.Conf., Toronto, Feb.22-23, 2001, pp.282-302.

(10) Watson D., ed., Energy Conservation through Building Design. An Architectural Record Book, New York: McGraw Hill Book Company., p.34. ,1979

# APPENDIX B

# POSSIBILITY TO USE SOLAR INDUCED VENTILATION STRATEGIES IN TROPICAL CONDITIONS BY COMPUTATIONAL FLUID DYNAMIC SIMULATION

**Agung Murti Nugroho**
**Mohd Hamdan bin Ahmad**

Makmal Sain Bangunan
Departement of Architecture Faculty Built Environment
UNIVERSITI TEKNOLOGI MALAYSIA
81310 Skudai, Johor Bahru, Malaysia

This paper is presented at the 6th Sustainable Environmental Architecture, at Institut Teknologi Bandung, Bandung, Indonesia, September, 2005.

## Abstract

The climatic conditions of the tropical regions are characterized by high air temperatures, high relative humidity's and very low wind speeds, which make the environmental conditions uncomfortable. The use of solar roof chimneys in buildings is one way to increase natural ventilation and, as a consequence, improve indoor air quality. The present study evaluates the stack induced ventilation strategies performance on experimental room model in Malaysian condition. The use of solar induced ventilation strategies in building was investigated using CFD FloVent technique. Validation of CFD Flovent was done by comparing the pilot testing. The effect of solar chimney, solar wall, and solar roof were simulated in order to determine the best option for a tropical ventilation strategy. The simulations were performed on selected model of trombe wall, solar chimney, and solar roof. The results showed the solar chimney can increase air velocity in the room but also increase heat gain in the room. The results also indicated that solar roof reduced the heat gain but resulted in low air velocity. Use of solar wall can increase air velocity in the room depending on the orientation of the solar wall. Based on the above finding a combined strategy was developed to increase stack induced ventilation.

Key word : Solar induced ventilation, Solar Chimney, Solar Roof, Trombe Wall

## A. INTRODUCTION

In tropical climatic regions passive cooling is one of the most difficult problems to solve. The simplest and the most effective solution for active cooling is to introduce air conditioning. However, such equipment involves high initial and operational costs for installation, energy and maintenance. Therefore air conditioners are unlikely to be applied widely, in particular, for residential building. Thus, a passive cooling system is more desirable. Although in Malaysia, passive cooling method is a popular cooling strategy adopted in residential building, researches (Pan, 1997; Tan, 1997; Jones, 1993; Zulkifli, 1991; Hui, 1998; Abdul Razak, 2004) have shown that its natural ventilation performance could not provide internal thermal comfort. Climate conscious design in the equatorial tropic assumes that air movement is one of the main cure for thermal comfort ills. According to Hui (1998), the indoor air velocity in low rise building range between 0.04m/s – 0.47 m/s. The reasons may be due to inappropriate design solutions for indoor air movement or low outdoor air velocity remains to be determined. However, recent data from the Malaysian Meteorogical Service Department showed that mean outdoor air velocity is between 1 m/s to 1.5 m/s.

In theory, there are two natural ventilation mechanisms (ASHRAE, 1997). First is by wind pressure and the second is by temperature difference or stack effect. Both mechanisms have the same aim, which is to act as an aid to create air movement and consequently control the indoor air temperature (Abdul Razak, 2004). Natural ventilation may result from air penetration through a variety of unintentional openings in the building envelope, but it also occurs as a result of manual control of building's openings doors and windows. Air is driven in and out of the building as a result of pressure differences across the openings, which are due to combined action of wind and buoyancy-driven forces. Today, natural ventilation is not only regarded as a simple measure to provide fresh air for the occupants, necessary to maintain acceptable air-quality levels, but also as an excellent energy-saving way to reduce the internal cooling load of housing located in the tropics. Depending on ambient conditions, natural ventilation may lead to indoor thermal comfort without mechanical cooling being required.

## B. NATURAL STACK VENTILATION

One of the more promising passive cooling methods for tropical climatic regions is the stack ventilation strategies. Stack ventilation is caused by stack pressure or buoyancy at an opening due to variation in air density as a result of different in temperature across the opening. The same principle can be applied for opening at different height, the different in pressure between them is due to the vertical gradient (Awbi,2004). It utilizes solar radiation, which is abundant in these regions, to generate the buoyant flow. However, as currently applied, the induced air movement is insufficient to create physiological cooling. More studies are needed to improve the ventilation performance of this cooling method. Velocities associated with natural convection are relatively small, usually not more than 2 m/s (Mills, 1992).

Stack induced ventilation can be improve by solar induced ventilation. However, in cases where the wind effect is not well captured then solar–induced ventilation may be a viable alternative. This strategy relies upon the heating of the building fabric by solar radiation resulting into a greater temperature difference. There are three building element commonly and used for this purpose : Trombe Wall, Solar Chimney, and Solar roof (Awbi,2004).

The first type incorporates glazed element in the wall to absorb solar irradiation into the wall structure. This building has double walls which are combined into a shaft at their upper end. The south facing shaft wall was made from glass. The solar radiation that penetrates the glass heats the inner wall. Eventually, this inner wall heats the air which will rise and induce a flow of fresh air from the openings below (Watson, 1979). Two examples of stack induced ventilation concepts is solar collector and stack height. The former shows one way to amplify stack effect by utilizing solar collectors and increasing the height of the hot air column (stack height). Critical parameters of this design are the stack height and cross sectional area of its inlet and outlet. A massive and high version of this type is needed to generate indoor air velocity as high as 1.0 to 2.0 m/s, which can be achieved easily in an ordinary shallow buildings (with no obstruction at all).

The second, form is the solar chimney which has long been known, and applied in vernacular architectural designs. In general, the induced air movement is not used directly to suck indoor air. Instead, it is used for ventilating the building (such as in the double skin building). A stack chimney is usually designed in combination with a wind tower in hot arid climatic regions. In many types of ventilated building, winds are considered to be more important than buoyancy. This is because wind induced ventilation flow is commonly stronger than stack induced flow, in particular, in low rise buildings. A milk house that was built in 1800s is a historical example of a stack chimney application (Satwiko, 1993).

Other method is used in areas with large solar altitude. In this case a large sloping roof is used effectively to collect the solar energy (Awbi, 2004). Another solar roof design called *the Nigerian Solar* Roof was studied by Barozzi et. al., using physical and numerical (Computational Fluid Dynamics Codes) modeling and data from Ife, Nigeria. Two findings were noticeable from this experiment. Firstly, both physical and numerical experiments gave almost identical results. Thus, it showed the potential of CFD Codes to simulate air flow. Secondly, both types of modeling indicated the presence of buoyancy driven ventilation within the model. However, the air speed within the occupant's zone was too low to create physiological cooling. The term *Solar chimney* is used extensively in Barozzi's experiment as the *chimney* shape is quite obvious. In his study the term *Solar chimney* seems to be more suitable as the chimney takes the form of a roof (Barozzi, 1992).

## D. ISSUES OF RESEACH OBJECTIVES

Air movement created by the stack effect is usually not adequate to achieve physiological cooling. It is less than the recommended air speed range for cooling of 0. 15 to 1. 5 m/s in tropical condition (Satwiko, 1994). It can be seen that two means are available for improving air movement: firstly, by increasing the air volume (stack height) and secondly, by increasing the air temperature difference. The indoor air temperature has to be kept low. All the above designs involve stack effect. However, in terms of construction (complexity, technology, etc.) and material (cost, durability, availability, etc.) these designs are not suitable for wide application in low cost housing in tropical countries.

Studies of solar-induced ventilation involve aerodynamic experiments on buoyancy problems which can be done using both physical modeling and computer modeling methods, while based on computational fluid dynamics (CFD). Both methods have advantages and disadvantages. Physical modeling is considered to give results that are easy to check. However, for an experiment which involves various model designs this method can become expensive and time consuming. The computer simulation method, on the other hand, allows easy modification of the design with more precise results in less time. Even though computer simulation has a high initial cost (for its hardware and software), the final cost might be lower than that of the physical experiment since changes in the model designs can be made easily.

The objective of this study is to :

- understand the possibility of using solar-induced ventilation in experimental room under Malaysian wind condition
- understand the performance of variety of solar-induced ventilation to increase stack induced ventilation

## E. METHODOLOGY

### a. Model experiments

A pilot testing using three models were simulated for solar induced ventilation study. To simplify the discussion, only the pilot testing and study simulation model are discussed. Figure 1. shows the pilot testing model and models simulation. The chimney pvc pipe in the pilot testing was 12 feet high and 0.5 feet diameter. The solar induced ventilation models are based from basic size experimental room (20 feet length x 12 feet width x 10 feet high) and uses metal partition.

### b. Tool

Several tools were used to obtain data on temperature, humidity, wind velocity, wind direction, and solar radiation in the area of the study. Air temperature and humidity data were measured using data logger. CFD Flo Vent were used to validate pilot testing model and simulation of the solar-induced ventilation model. Flovent is the most widely used software for modeling engineering fluid flows due to its robustness, accuracy, and user friendliness. Modeling was performed for three dimensional domain due to the large space involved. The computational grid is recommended to be set of maximum size. Therefore in following calculations the model, is basically divided into a grid of 0.5 m x 0.5 m x 0.5 m control volume. Additional grid points are embedded in the part of chimney, roof, walls, around the inlets and outlets.

### c. Procedure

In the pilot testing the data logger is set for every 5 minutes from 10.00 am to 18.00 pm. Data logger were positioned of three points on each pipe and one point outdoor (figure 1). In the CFD simulation, following boundary condition area used: the material and thickness are set as same as the base models; Inlets and outlets are set as the same as ambient temperature; climatic condition are set to the site climatic conditions.

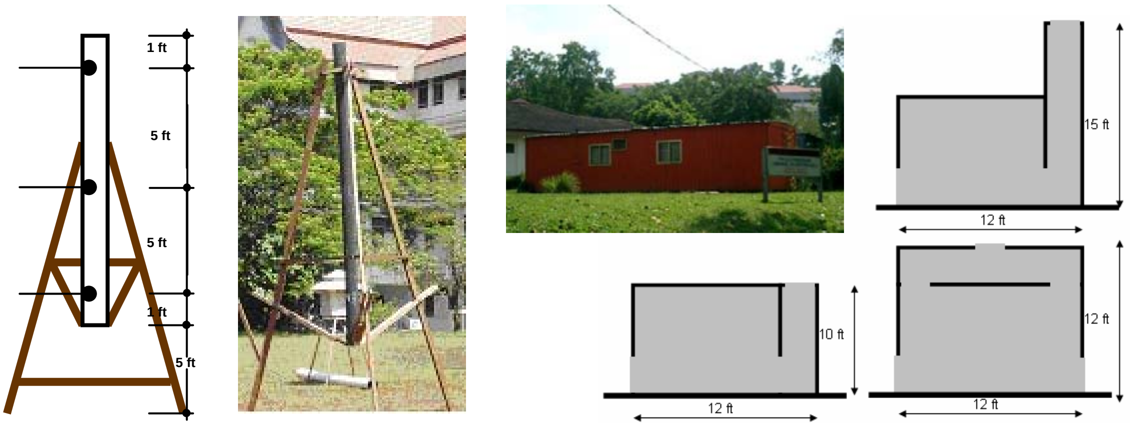


Figure 1. The models experiment (left) and pilot testing model for CFD model simulation (right)

## C. RESULT AND DISCUSSION

a. Validation

Validation of the program was performed by comparing the measurement of pilot testing with the CFD simulation. Figure 2 shows the comparison of measurement and simulation result. It shows that the agreement between the measurement and simulation is generally good. The average difference between the measurement and simulation for ambient temperature was 0%; for black bottom was about 3%; the maximum difference was 8% for the cavity 10 am of Black Top pipe. This gives confidence in using the computer code to study the air flow and temperature.

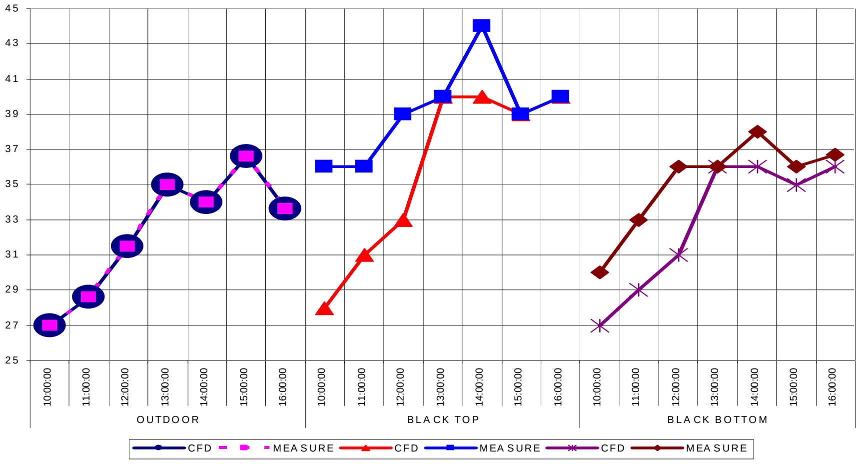


Figure 2. Comparing measurement and CFD simulation of pilot testing

b. Solar Induced Ventilation Simulation

Climatic data from weather station obtained from the pilot testing were used. The specific day chosen for the simulation and assumed as the hottest date of the year 21 march was. Times chosen 12.00 noon. The solar induced ventilation simulations were performed on selected model of trombe wall, solar chimney and solar roof. Figure 3 shows the simulation result for air velocity and temperature profile.

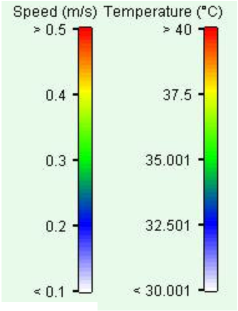
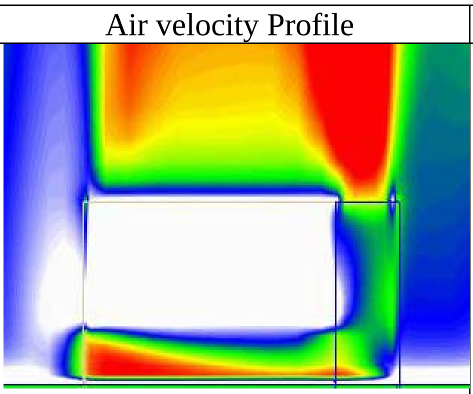
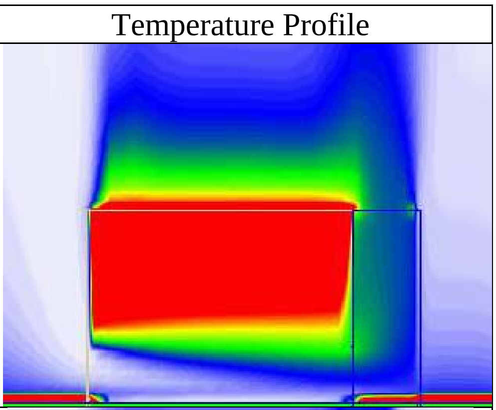
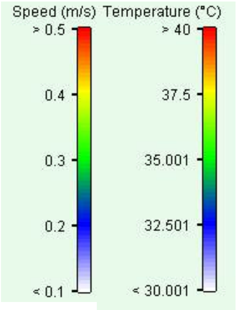
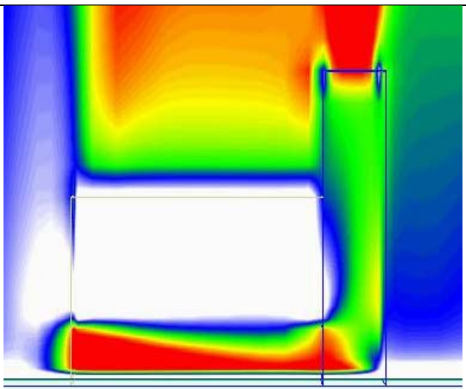
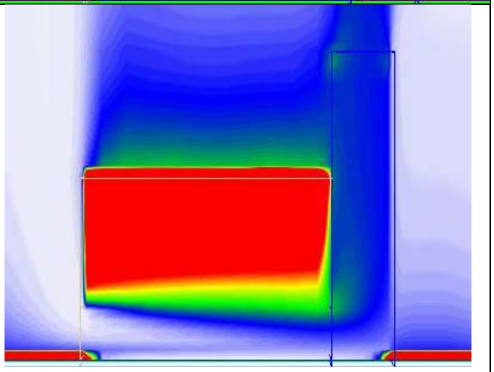
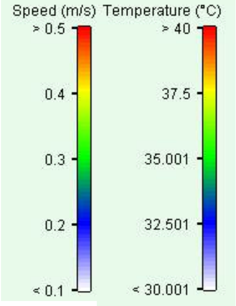
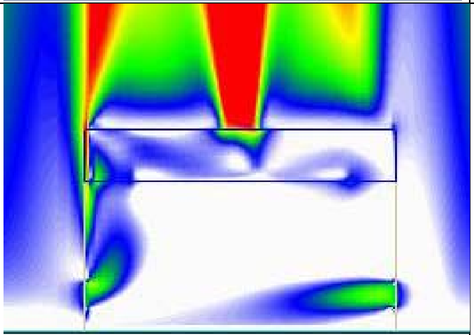
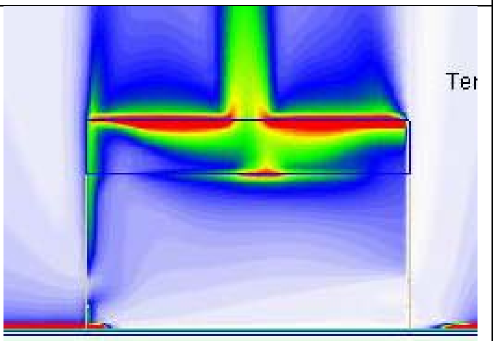
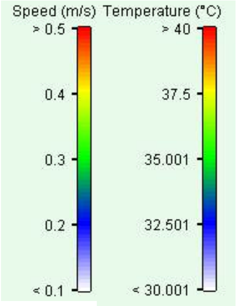
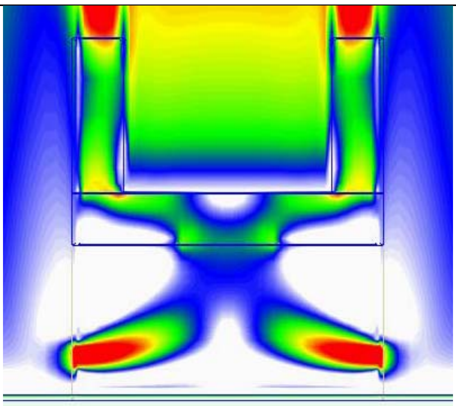
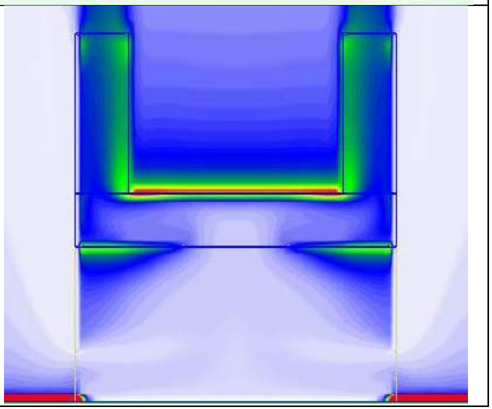
| Model | Air velocity Profile | Temperature Profile |
|---|---|---|
| Trombel Wall<br><br> |  |  |
| Solar Chimney<br><br> |  |  |
| Solar Roof<br><br> |  |  |
| Solar Roof Chimney<br><br> |  |  |

Figure 3. CFD simulation result of Solar Induced Ventilation

c. Air Flow Profile

Air velocity on the tromble wall model and solar chimney model can increase air flow up to 0.5 m/s while air velocity in solar roof model until 0.3m/s. The position of 0.5 m/s air velocity in the tromble wall and solar chimney model depend on the opening position. The position of 0.3m/s air velocity in the solar roof is only around the inlet of the room. The result indicated that stack induced performance of tromble wall and solar chimney were better than solar roof. Base of size air velocity zone, solar chimney model broader than tromble wall model.

d. Temperature Profile

Temperature profile on the tromble wall and solar chimney models can reach until 40°C (mostly in the room). These condition is uncomfortable temperature in the room. On the solar roof model, the double roof prevented the heat gain in the room.

e. Modification model

The above results showed that the use of solar chimney was able to increase air velocity while solar roof indicated less heat gains. By combining the solar chimney and solar roof the results indicated a better performance of air velocity and temperature reduction in the experimental room model

## D. CONCLUSION

The results showed that the solar chimney can increase air velocity in the room but also increase heat gain in the room. The results also indicated that solar roof reduced the heat gain but resulted in low air velocity. Use of solar wall can increase air velocity in the room depending on the orientation of the solar wall. Modification with combine solar chimney and solar roof will be use to improve the induced ventilation and further investigation will be done.

## D. REFERENCE

(1) ASHRAE Handbook Fundamentals, American Society of Heating and Air Conditioning Engineers, Inc. Atlanta, 1989

(2) Awbi, Hazim B., Ventilation of Building, Spon Press, London, 2003

(3) Baker E., Floro C.J., Gostelow J.P. and McCaffrey J.J., Solar Heating and Cooling Systems. Design for Australian Condditio Sidney: Pergamon Press., p.126. , 1994

(4) Barozzi G.S., Imbabi M.S.-E., Nobile E., Sousa A.C.M., Physical and Numerical delling of a Solar Chimney-based Ventilation Sy stem for Buildings. Building and Environment, Vol.27, No.4, October, Oxford: Pergamon Press, pp.433-45. , 1992

(5) Hui Seng Kin, Natural Ventilation of Low Cost Dwellings in The Hot Humid Tropics Malaysia, Thesis, 1998

(6) K.S. Ong , C.C. Chow, Performance of a Solar Chimney, Solar Energy 74 1–17, 2003

(7) Mills A.F, Heat Trans Boston: Irwin., p.293., 1992

(8) Sapian Abdul Razak, Possibilities of using Void to Improve Natural Cross Ventilation in High-rise Low Cost Residential Building, Thesis, 2004

(9) Satwiko, Prasasto, The Development and Modelling of a Natural Ventilation System for Indonesian Conditions, Thesis Master of Building Science, Victoria University of Wellington, 1994

(10) Watson D., ed., Energy Conservation through Building Design. An Architectural Record Book, New York: McGraw Hill Book Company., p.34. ,1979

# APPENDIX C

# EVALUATION OF PARAMETRICS FOR THE DEVELOPMENT OF VERTICAL SOLAR CHIMNEY VENTILATION IN HOT AND HUMID CLIMATE

*Agung Murti Nugroho, Mohd Hamdan Ahmad, Then Jit Hiung*
*PhD Student, Associate Professor, Master Student*
*Department of Architecture, Faculty of Built Environment*
*Universiti Teknologi Malaysia, 81310, Sekudai, Johor*
*MALAYSIA*

This paper is presented at the 2nd International Network for Tropical Architecture conference, at Christian Wacana University, Jogjakarta, from 3 to 5 April, 2006.

**Abstract:** In terms of housing passive cooling design, tropical climatic regions present the most difficult problem to solve. A good dwelling design can keep the indoor environment favorable and comfortable during most of the year without the use of any mechanical devices. This can be accomplished by various techniques such as the use of radiant barrier, insulation materials, and natural ventilation. Depending on ambient conditions, natural ventilation may lead to indoor thermal comfort without mechanical cooling. However, in cases where the wind effect is not well captured especially in single side ventilation, then solar–induced ventilation may be a viable alternative. Solar induced ventilation standing involved temperature difference experiments which can be done using both physical modeling and computer simulation (Computational Fluid Dynamics). Solar induced, especially vertical solar chimney ventilation combined air movement and solar radiation simulation. They have different input data which depend on the climatic data of the selected location. This paper evaluates the parametric study strategies in pilot testing, terrace house model and previous research model by simulation and experiment for solar induced ventilation in tropical condition. Comparison of the results of simulations and experiments illustrate a good agreement between numerical and experimental results. These results encourage further research to develop the vertical solar chimney suitable for tropical condition.

**Key word:** tropical climate, parametric study and vertical solar chimney

## 1. Introduction

In tropical climatic regions passive cooling is one of the most difficult problems to solve. The simplest and the most effective solution for active cooling is to introduce air conditioning. However, such equipment involves high initial and operational costs for installation, energy and maintenance. Therefore air conditioners are unlikely to be applied widely, particularly in residential building. Thus, passive cooling system is more desirable. Although in Malaysia, passive cooling is a popular cooling strategy adopted in residential buildings researcher (Pan, 1997; Tan, 1997; Jones, 1993; Zulkifli, 1991; Hui, 1998; Abdul Razak, 2004) have shown that its natural ventilation performance could not provide internal thermal comfort. Climate conscious design in the equatorial tropic assumes that air movement is one of the main cure for thermal comfort ills. According to Hui (1998), the indoor air velocity in low rise building range between 0.04m/s – 0.47 m/s. The reasons may be due to inappropriate design solutions for indoor air movement or low outdoor air velocity. However, recent data from the Malaysian Meteorogical Service Department showed that mean outdoor air velocity is between 1 m/s to 1.5 m/s.

One of the more promising passive cooling methods for tropical climatic regions is the stack ventilation strategies. Stack ventilation is caused by stack pressure or buoyancy at an opening due to variation in air density as a result of difference in temperature across the opening. The same principle can be applied for opening at different height, where the difference in pressure between them is due to the vertical gradient (Awbi, 2004). It utilizes solar radiation, which is abundant in these regions, to generate the buoyant flow. However, as currently applied, the induced air movement is insufficient to create physiological cooling. More studies are needed to improve the ventilation performance of this cooling method. Velocities associated with natural convection are relatively small, usually not more than 2 m/s (Mills, 1992). Stack induced ventilation can be improve by solar induced ventilation. However, in cases where the wind effect is not well captured then solar–induced ventilation may be a viable alternative. This strategy relies on heating of the building fabric by solar radiation resulting into a greater temperature difference. There are three building elements commonly used for this purpose: Tromble Wall, Solar Chimney, and Solar roof (Awbi, 2004).

The first type incorporates glazed element in the wall to absorb solar irradiation into the wall structure. This building has double walls which are combined into a shaft at their upper end. The south facing shaft wall was made from glass. The solar radiation that penetrates the glass heats the inner wall. Eventually, this inner wall heats the air which will rise and induce a flow of fresh air from the openings below (Watson, 1979). Two examples of stack induced ventilation concepts is solar collector and stack height. The former shows one way to amplify stack effect by utilizing solar collectors and increasing the height of the hot air column (stack height). Critical parameters of this design are the stack height and cross sectional area of its inlet and outlet. A massive and high version of this type is needed to generate indoor air velocity as high as 1.0 to 2.0 m/s, which can be achieved easily in an ordinary shallow buildings (with no obstruction at all). The second, form is the solar chimney which has long been known, and applied in vernacular architectural design. In general, the induced air movement is not used directly to suck indoor air. Instead, it is used for ventilating the building (such as in the double skin building). A stack chimney is usually designed in combination with a wind tower in hot arid climatic regions. In many types of ventilated building, winds are considered to be more important than buoyancy. This is because wind induced ventilation flow is commonly stronger than stack induced flow, in particular, low rise buildings. A milk house that was built in 1800s is a historical example of a stack chimney application

(Satwiko, 1993). The other method is used in areas with large solar altitude. In this case a large sloping roof is used effectively to collect the solar energy (Awbi, 2004). Another solar roof design called *the Nigerian Solar* Roof was studied by Barozzi, using physical and numerical (Computational Fluid Dynamics Codes) modeling and data from Ife, Nigeria. Two findings were noticeable from these experiments. Firstly, both physical and numerical experiments gave almost identical results. Thus, it showed the potential of CFD Codes to simulate air flow. Secondly, both types of modeling indicated the presence of buoyancy driven ventilation within the model. However, the air speed within the occupant's zone was too low to create physiological cooling. The term *Solar chimney* is used extensively in Barozzi's experiment as the *chimney* shape is quite obvious. In his study the term *Solar chimney* seems to be more suitable as the chimney takes the form of a roof (Barozzi, 1992).

## 2. Objectives

Air movement created by the stack effect is usually not adequate to achieve physiological cooling. It is less than the recommended air speed range for cooling of 0.15 to 1.5 m/s in tropical condition (Satwiko, 1994). It can be seen that two means are available for improving air movement: firstly, by increasing the air volume (stack height) and secondly, by increasing the air temperature difference. The indoor air temperature has to be kept low. All the above designs involve stack effect. However, in terms of construction (complexity, technology, etc.) and material (cost, durability, availability, etc.) these designs are not suitable for wide application in low cost housing in tropical countries. Studies of solar chimney ventilation involve temperature difference experiments which can be done using both physical modeling and computer simulation (Computational Fluid Dynamics). Solar induced, especially vertical solar chimney combined air movement and solar radiation simulation. They have different input data which depend on the climatic data of the selected location. This paper evaluates the parametric study in initial test, field measurement and terrace house model by simulation and experiment for solar induced ventilation in tropical condition. These results encourage further research to develop the vertical solar chimney suitable for tropical condition. The objective of this study is to:

- Evaluate the possibility and limitation of solar chimney ventilation parametric design in initial test and terrace house model under Malaysia’s climate condition
- Evaluate the optimum performance of solar chimney ventilation parametric design to increase indoor air velocity.

## 3. Methodology

### 3.1. Climate data and simulation program

Climatic data for Malaysia’s condition, in particular, Johor Bahru, was collected from Senai Weather Station. Wind speed calculation using Power law concept obtained by using this technique was done by Ismail (1996), Kin (1998) and Hamid (2001), and will be used as an illustration for the calculation of Malaysia’s proposed wind profile. Solar radiation, temperature and humidity were obtained from

DOE weather file data (Ossen 2005). The simulations for this study are done using a general three dimensional computational fluid dynamics model. For the stack effect prediction, density variation caused by temperature rise, which takes air density as constant and considers the buoyancy on air movement by the difference between the local air density and the pressure gradient. The upwind scheme has been used in the calculation (Hunt, 1999). The simulation tool used to simulate the temperature and air velocity distribution is the CFD Flo Vent. Flovent is the most widely used software for modeling engineering fluid flows due to its robustness, accuracy, and user friendliness (Flomerics, 2000).

### 3.2. Pilot testing model

A pilot testing using one model was measured and simulated for solar induced ventilation study. Figure 1 shows the pilot testing model and the simulation model. The chimney pvc pipe in the pilot testing was 12 feet high and 0.5 feet in diameter, supported structurally by timber framework. The models were black surface colors respectively. Data loggers were positioned at three different points on each pipe and another at outdoor (figure 1). In the CFD simulation, the following boundary condition area used: the material and thickness of the chimney are based on the base model, while the climatic condition is set similar to the site climatic conditions (Nugroho, 2005).

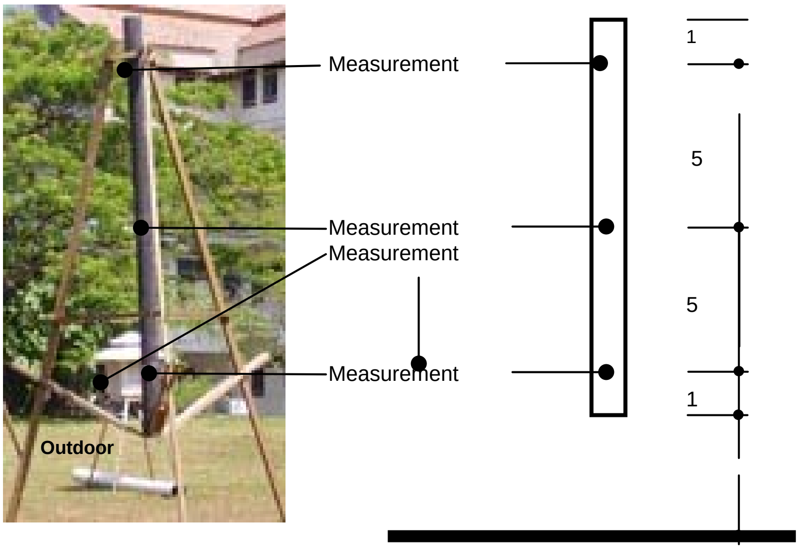


Figure 1: Pilot testing model

### 3.3. Terrace house model

A field study of terrace house has been carried out by Moh Najib Ibrahim (1989). The field study is located in Taman Tun Aminah, 15 km from Johor Bahru and surrounded by housing estates. The layout of terrace house is similar to the typical low cost housing. They consist of 2 bedrooms and 1 living room. The bed room dimension is 3 m (height), 3.5 m (depth), 3 m (length) and has single side window. Under the same prescribed experiment conditions, the results have been validated by experimental and simulation models. The purpose of the experiment and simulation is to evaluate of the room parametric study.

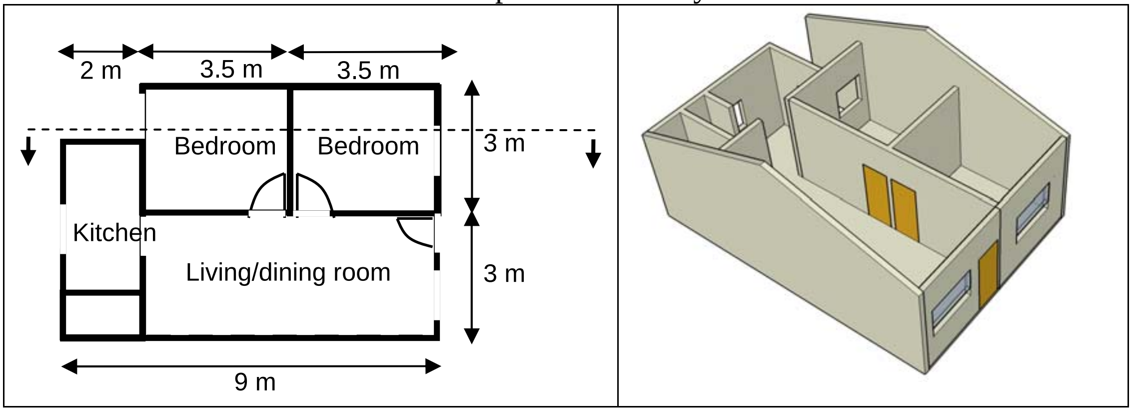
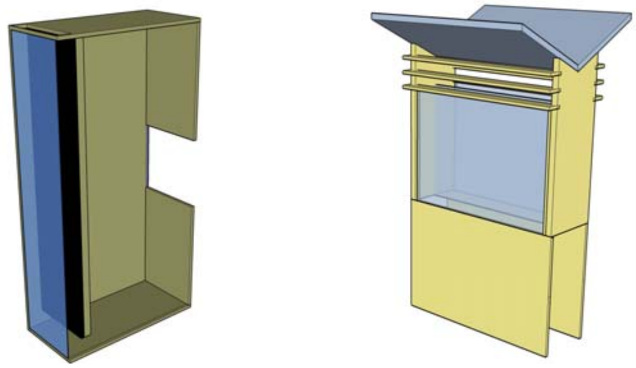


Figure 2: Terrace house model

### 3.4. K.S. Ong Experiment and Optimum Vertical Solar Chimney Model

The simulation is based on KS Ong’s experimental results and previous research conducted for evaluating solar chimney design parameters. The additional purpose of conducting this study was to explore the optimum design parameter of vertical solar chimney in hot and humid climatic conditions. Ong (2003) has presented a mathematical model of solar chimney using the matrix method for solving simultaneous equations for heat transfer and experimental results on a 2-m-high solar chimney in Malaysia.



KS Ong Experiment Optimum VSC Model

Figure 3: K.S. Ong experiment model and optimum vertical solar chimney model

### 3.5. Vertical solar chimney model in terrace house

In order to achieve optimum vertical solar chimney, a basic terrace house model was modified. The vertical solar chimney model was placed in between two bedrooms, at a height of 2 m above floor level (figure.4). Climatic data for simulation model under the same field experiment.

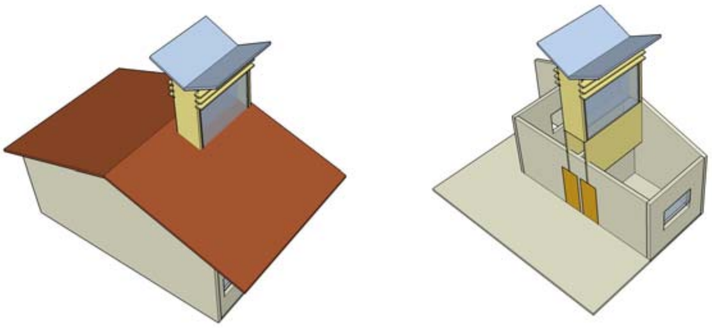


Figure 4: Application of vertical solar chimney model in terrace house

## 4. Result and Discussion

### 4.1. Pilot testing experimental and simulation.

The experiment tool used to measure the temperature and humidity is the Dickson temperature and humidity data logger. Dickson data logger has dual channel sensors which can record and store humidity and temperature data for two months, accurately and it is also lightweight. Dickson data logger can store thousand of sample points and specifications requirement. In the pilot testing the data logger is set for every 5 minutes from 10.00 am to 18.00 pm. Evaluation of the pilot testing and simulation design parameter was performed by comparing the measurement of pilot testing with the CFD simulation. Figure 5 shows the comparison of the results from the measurement and simulation. It shows that the agreement between the measurement and simulation is generally good. The average difference between the measurement and simulation for black bottom was about 3%; the maximum difference was 8%, which is recorded at 10 hr for black top pipe, while there are no differences for ambient temperature (Nugroho, 2005). Base on the simulation result, the average air velocity inside the pipe until 0.1 m/s. Figure 6 shows the distribution of air velocity in the time of day. Maximum air velocity is recorded at 13 hr and this shows the possibility of using stack effect in Malaysia's climate condition.

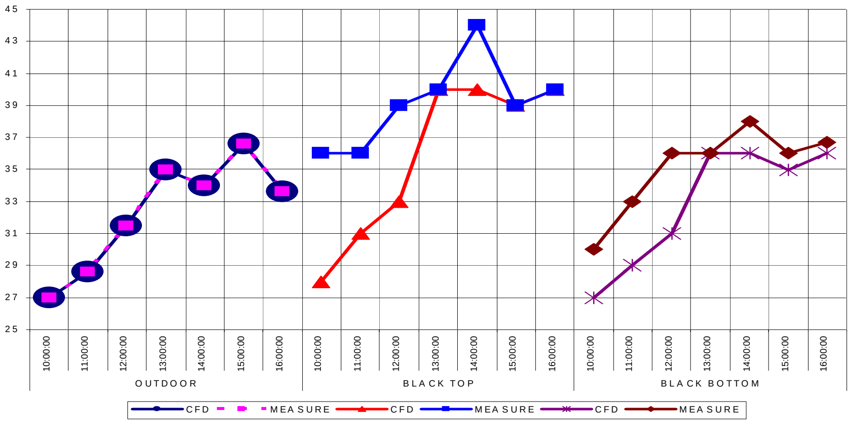


Figure 5: Comparing experiment and simulation of pilot testing

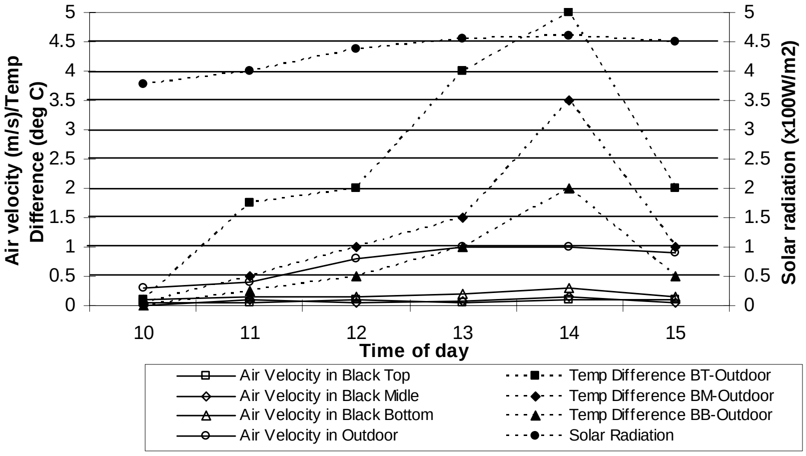


Figure 6: Air velocity and temperature difference of pilot testing simulation

## 4.2. Field Study Experiment and Simulation

The study is based on continuous air velocity, temperature and humidity measurements performed inside the building. Indoor climate analyzer was used for collecting the data. Data was taken at the center of the bed room, 110 cm above floor level (ASHRAE 1992). Figure 7 shows the results from experiment and simulation. They are in close agreement and deviations are within 1% of the calculated temperature and 2 % of the calculated air velocity. Figure 8 shows the effect of single side room ventilation is indicating lower air velocity than outdoor wind speed. The air velocity of less than 0.1 m/s mean insufficient for thermal comfort psychological cooling (Rajeh, 1989). The experiment and simulation results showed the limitation of field design parameters using single side ventilation.

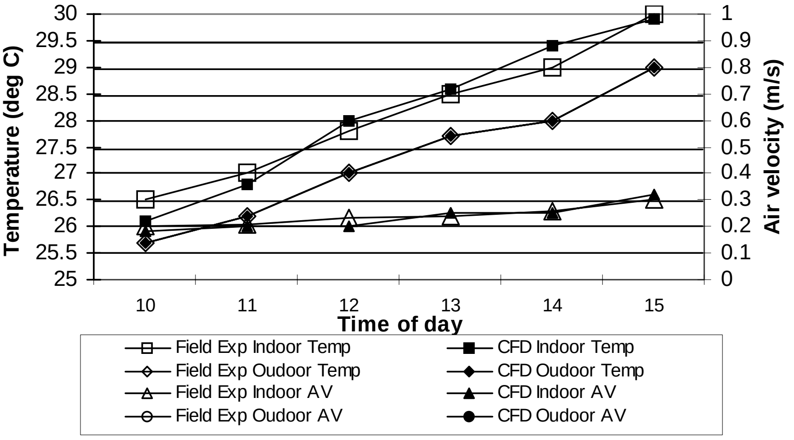
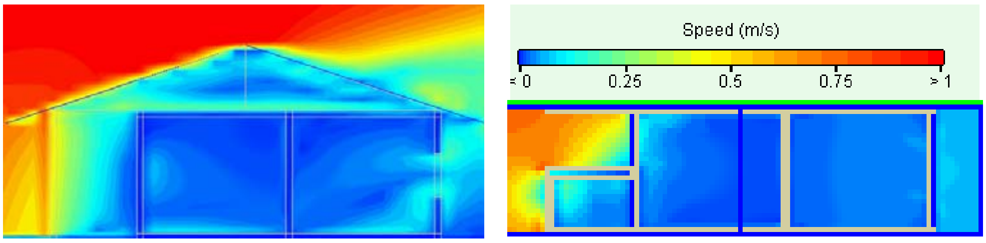


Figure 7: Comparing experiment and simulation of field study



Section

Plan

Figure 8: Air velocity pattern of Terrace House Model in Section and Plan

### 4.3. Previous solar chimney research and simulation

Barrozi (1992) performed flow visualization studies on Nigeria solar chimney. The prototype is of 0.25 m (height), 0.1 m (width) and 0.5 m (length), using aluminum material. The experiment result indicated a low air velocity (less than 0.1 m/s). Bouchair (1994) showed that for his 1.95 m high and 0.2 m – 1 m width chimney which was electrically heated, the optimum ratio for chimney length /gap width of 5/10 can achieve maximum air velocity. If the chimney was too big, reverse circulation occurred whereby there was a down-ward flow of air via the center of the duct. They concluded that 0.1 m/s is the average air velocity between two single glasses. Alfonso (2000) studied the performance of a thermal model and validated it with the help of the tracer gas technique for a solar chimney attached to a room of 12m2 floor areas having a brick wall and a concrete roof. They presented the solar chimney model on a 0.6 m (width), 1m (length) and different chimney height (0.5 m- 3 m). Their results showed that there was a significant increase in air velocity (0.3 m/s) with the varying solar chimney heights. Bansal (1994) developed a steady state mathematical model for a solar chimney system consisting of a solar air heater connected to a conventional chimney. The chimney model was 0.75 m high, 1 m long and having glass wall materials. Their investigation indicated that 0.15 m/s air velocity is encountered during their experiment for two different combinations of air gap (0.1 m and 0.3 m). Among others, theoretical and experimental studies on natural ventilation of buildings were also carried out by Ong (2003). The study has presented a mathematical model of solar chimney using matrix method for solving simultaneous equations for heat transfer. The experiment was carried out outdoor on a one-sided double glass-wall. The experimental models are 2 m high, 1 m wide and varying chimney gaps of 0.1 to 0.3 m. They concluded that increase in air velocity (0.3m/s) of inlet solar chimney is relative to solar chimney width. Figure 9 shows the minimum difference between KS Ong's experimental model and simulation where the minimum difference was about 1% and the maximum difference was 9%. The agreement between the measurement and simulation is generally good (Khedari, 2000). Based on previous research, simulation is conducted to evaluate solar chimney design parameters (figure10) in order to find the optimum design (3 m high, 3 m long, 0.5 m wide, glass material).

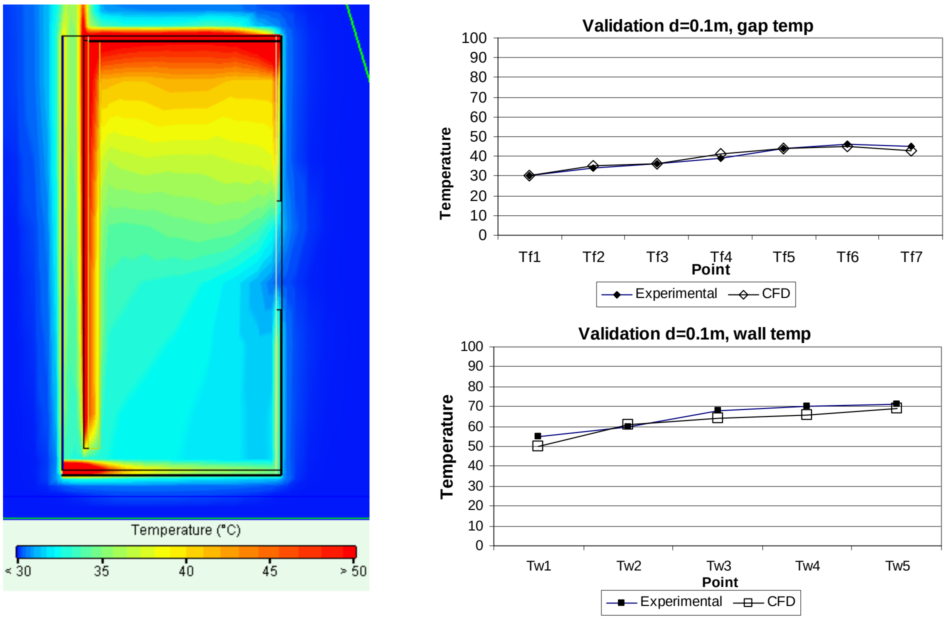


Figure 9: Comparing experiment and simulation of KS Ong model

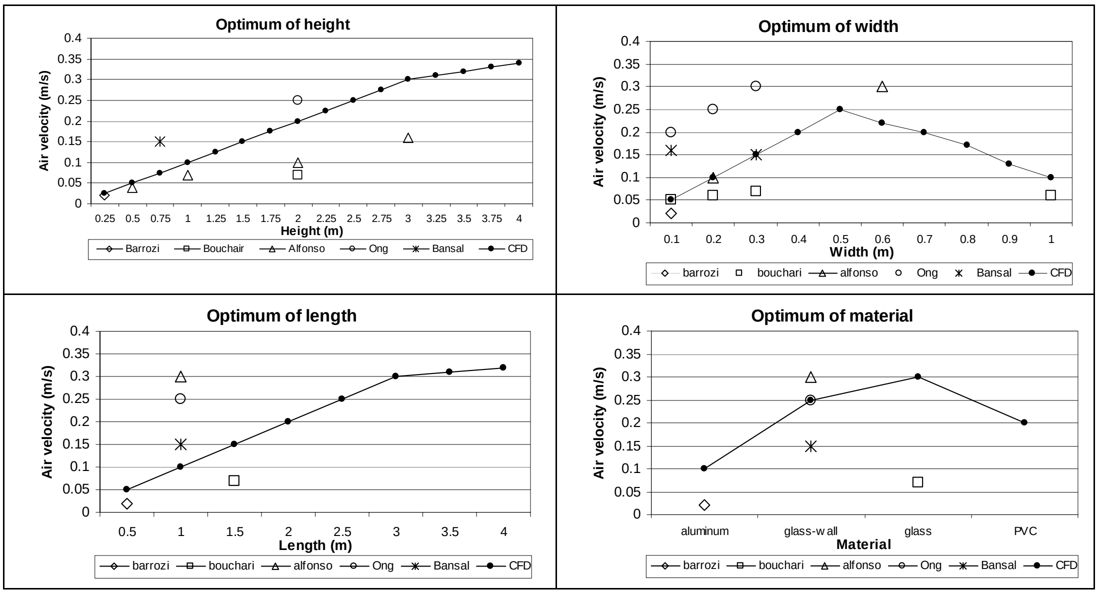
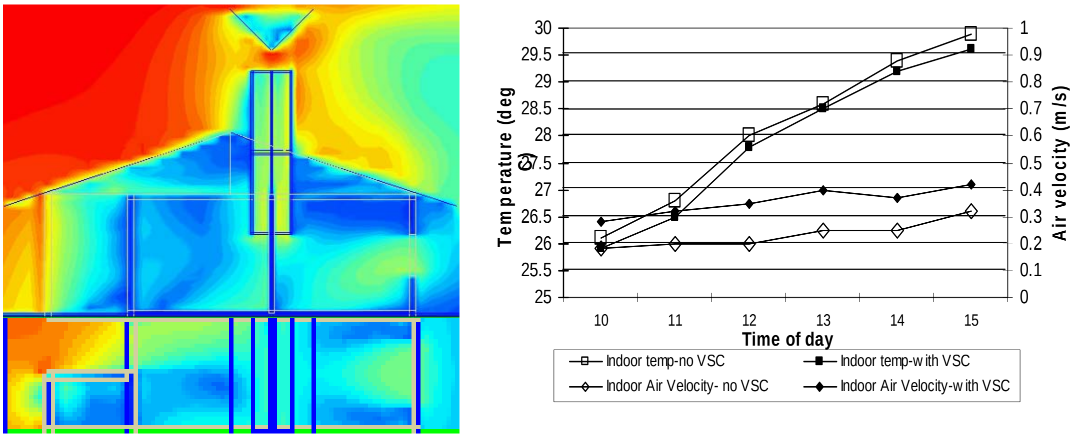


Figure 10: The optimum of vertical solar chimney base on previous research

## 4.4. Evaluation of Vertical Solar Chimney Parameter

It is found that in a terrace house using vertical solar chimney, air ventilation is induced by the solar chimney. Results can be more significant by increasing air velocity (0.15m/s) as well as cooling of temperature. The average temperature in the room ranges between 26˚C and 29.5˚C during 10 hr – 15 hr, which is about 0.5˚C lower than a normal terrace house without solar chimney.



Plan and Section

Temperature and Air Velocity at 10 h – 15 h

Figure 11: Performances of parametric study of vertical solar chimney in terrace house model

## 5. Conclusion

The computational fluid dynamics simulation evaluates the solar induced ventilation in terrace house model. We have reached the following conclusions:

- Air velocity without considering the wind effect is influenced by climate parameters (solar radiation and ambient temperature). High solar radiation and temperature differences may create air flow inside the chimney pipe for pilot testing.
- Low indoor air velocity and high ambient temperature may create unwanted negative ventilation by limiting room design parameters (single side ) in terrace house
- Optimum design parameters (height, width, length and material) of a vertical solar chimney can be deduced by comparing simulation results and results based on previous research.
- The performances of design parameters proposed in this study, if incorporated in terrace house model, is expected to create a reasonable indoor air velocity for human comfort, besides contribution to energy efficient and environmentally friendly options.

## References

Alfonso, Clito.2000. Solar Chimneys: Simulation and Experiment, Energy and Buildings, Vol. 32, Pergamon Press., pp. 71–79.

ASHRAE Handbook Fundamentals, 1989. American Society of Heating and Air Conditioning Engineers, Inc. Atlanta.

Awbi, Hazim B., 2003. Ventilation of Building, Spon Press, London.

Bansal, N.K, R. Mathur, M.S. Bhandari, 1994. A Study of Solar Chimney Assisted Wind Tower System for Natural Ventilation in Buildings, Building and Environment, Vol. 29, 4, Pergamon Press., pp. 495-500.

UNIVERSITI TEKNOLOGI MALAYSIA
Invention Disclosure

This form contains disclosure of invention or copyrightable material and should be submitted in confidence to The Secretariat, Intellectual Property Committee, Research Management Centre (RMC), Universiti Teknologi Malaysia, 81310 UTM Skudai.

1. Type of Material : Invention ☐ Copyright ☐

2. Title of Invention or Copyright : ______________________________
______________________________

3. Inventor(s) Full Name Department/Institute/ Approximate
Centre/Unit %
Contribution

3.1 Principal ______________________________

3.2 Associates ______________________________
______________________________

3.3 Others ______________________________

4. Identify sources and estimate % of support (materials, facilities, salaries) contributing to the development of the invention :

Government Funds ( IRPA, grants and /or contract) _____ %
UTM-RMC Funds _____ %
Other Institution (s) :
Name : ______________________________ _____ %
Other source : ______________________________ _____ %

5. If developed with Government Funds :

Has invention been reported to granting agency ? Yes ☐ No ☐
Agency Name : __________________________ Report Date: ________
Grant Number : __________________________

Has notification been made to the relevant Government Agency for retention of rights to invention ?
Yes ☐ No ☐

Please attach copies of any correspondence with any Government agency related to disclosure of invention or rights to invention.

6. If developed with other funds (industry sponsor, foundation grant ,etc.) :
Has the invention been reported to the sponsor? Yes ☐ No ☐

Source name : __________________________ Report Date : _________
Please attach copies of relevant correspondence with the sponsor.

7. Record of Invention :

7.1 Invention was first conceived on or about (Date) : ____ / ____/ _____

7.2 An oral disclosure has been made :

(Name)__________________________________________ on (Date) ___ / ___ / ___

(Name) __________________________________________ on (Date) ___ / ___ / ___

7.3 First sketch or drawing was made on (Date) : ____ / ____ / _____ disclosed to and understood by (Name ) : ______________________________ on (Date) : ___ / __ / __

That document is now located at: ____________________________________________

7.4 First written description was completed on (Date) : ____ / ____ / ____ and that document is now located at : _________________________________________________

7.5 First reduction to practice was successfully tested on (Date) : ____ / ____ / ____ and the records of that test are now located at ______________________________________

7.6 First publication disclosing the invention was dated (Date ) : ____ / ____ / ____

8. List companies or individuals with whom you may have discussed this project and append copies, showing dates, of all correspondence relating to their interest.

(Name) __________________________________ (Correspondence Date) : ___ / ___ / ___

(Name) __________________________________ (Correspondence Date) : ___ / ___/ ____

If you have communicated, via telephone, with any additional companies, please list the company names, giving dates, and append a brief summary of your conversations.

(Name) __________________________________ (Communication Date ) : ___ / ___/ ___

Summary of conversation: __________________________________________________

(Note : Valuable rights to inventions may be lost if disclosed to outside parties unless signed Confidentiality Agreement is obtained)

9. Has invention or components thereof been described in a draft of an article or lecture ? If so, please attach copies of drafts of abstracts, manuscripts, or reprints and give proposed presentation and/or publication dates.

(Type & Title of Draft) : ___________________________________________________

(Proposed Presentation Date) : ____ / ____ / ____

(Note : Premature disclosure of invention in lectures, articles, etc. may result in loss of all right to obtain patent)

10. Briefly outline your views regarding potential commercial application of your invention :

10.1 List potential licencees or manufacturers or companies active in this field.

Name : ____________________________________________________________

Name : ____________________________________________________________

Name : ____________________________________________________________

10.2What is potential market ?

______________________________________________________________________

______________________________________________________________________

______________________________________________________________________

______________________________________________________________________

______________________________________________________________________

______________________________________________________________________

10.3Estimate commercial market in RM and USD : RM _______ USD _______

11. List independent referees with expertise in the area of the invention with whom we may communicate for additional information (with your approval) :

(Name, address, contact nos. ) : ____________________________________________

(Name, address, contact nos.) : ____________________________________________

12. Please append a full description of the invention which should include the following :

12.1 Drawings, diagrams, figures, flowcharts, sketches etc. which illustrate the invention.
12.2 Chemical structural form (if the invention is a new chemical compound).
12.3 List of equivalents which can be substituted for the invention or for components of the invention.
12.4 Reprints of articles or patents describing inventions, methods etc. similar to the one described in this disclosure.
12.5 Describe why your product or process is sufficiently novel compared to those already available to warrant patentability.

---

| | |
|---|---|
| Principal Inventor : ___________________<br>*(Signature)* | Dean/Director : _______________________<br>of Faculty/Centre/Institute *(Signature)* |
| Name : | Name : |
| Address : | Address : |
| Telephone / Fax : | Telephone / Fax : |
| Date : ____ / ____ / ____ | Date : ____ / ____ / ____ |

---

UNIVERSITI TEKNOLOGI MALAYSIA RETAINS TITLE TO ALL INVENTIONS AND PATENTS AS PROVIDED FOR IN THE INTELLECTUAL PROPERTY POLICY.

If you have any enquiry about this set of forms, or intellectual property protection in general, please contact Deputy Director (IP Secretariat), Research Management Centre, UTM 81310 Skudai at 07-5502382 or e-mail td2@rmc.utm.my.

Lampiran 20

UTM/RMC/F/0024 (1998)

# UNIVERSITI TEKNOLOGI MALAYSIA

## BORANG PENGESAHAN
## LAPORAN AKHIR PENYELIDIKAN

TAJUK PROJEK : TOWARDS DEVELOPMENT OF TROPICAL SOLAR ARCHITECTURE: THE USE OF SOLAR CHIMNEY AS STACK INDUCED VENTILATION STRATEGY

Saya: MOHD. HAMDAN BIN AHMAD

Mengaku membenarkan **Laporan Akhir Penyelidikan** ini disimpan di Perpustakaan Universiti Teknologi Malaysia dengan syarat-syarat kegunaan seperti berikut :

1. Laporan Akhir Penyelidikan ini adalah hakmilik Universiti Teknologi Malaysia.
2. Perpustakaan Universiti Teknologi Malaysia dibenarkan membuat salinan untuk tujuan rujukan sahaja.
3. Perpustakaan dibenarkan membuat penjualan salinan Laporan Akhir Penyelidikan ini bagi kategori TIDAK TERHAD.
4. * Sila tandakan ( / )

| | | |
|---|---|---|
| ☐ | SULIT | (Mengandungi maklumat yang berdarjah keselamatan atau Kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972). |
| ☐ | TERHAD | (Mengandungi maklumat TERHAD yang telah ditentukan oleh Organisasi/badan di mana penyelidikan dijalankan). |
| ☐ | TIDAK TERHAD | |

______________________________

TANDATANGAN KETUA PENYELIDIK

MOHD. HAMDAN BIN AHMAD

Nama & Cop Ketua Penyelidik

Tarikh : ____________________

**CATATAN :** * *Jika Laporan Akhir Penyelidikan ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh laporan ini perlu dikelaskan sebagai SULIT dan TERHAD.*

## UNIVERSITI TEKNOLOGI MALAYSIA
**Research Management Centre**

## PRELIMINARY IP SCREENING & TECHNOLOGY ASSESSMENT FORM
*(To be completed by Project Leader submission of Final Report to RMC or whenever IP protection arrangement is required)*

1. **PROJECT TITLE IDENTIFICATION** :

TOWARDS DEVELOPMENT OF TROPICAL SOLAR ARCHITECTURE THE USE OF SOLAR CHIMNEY

Vote No: 74217

2. **PROJECT LEADER** :

Name : MOHD HAMDAN BIN AHMAD
Address : Fakulti of Built Environment, Universiti Teknologi Malaysia
Tel : 075530611 Fax : 075566155 e-mail :b-hamdan@utm.my

3. **DIRECT OUTPUT OF PROJECT** *(Please tick where applicable)*

| Secientific Research | Applied Research | Product/Process Development |
|---|---|---|
| Algorithm | Method/Technique | Product / Component |
| Structure | Demonstration / Prototype | Process |
| Data | | Software |
| ☐ | ☐ | ☑ |
| Other, please specify | Other, please specify | Other, please specify |
| ☐ ____________ | ☑ ____________ | ☐ ____________ |
| ☐ ____________ | ____________ | ☐ ____________ |
| ☐ ____________ | ☐ ____________ | ☐ ____________ |

4. **INTELLECTUAL PROPERTY** *(Please tick where applicable)*

☐ Not patentable
☐ Patent search required
☐ Patent search completed and clean
☑ Invention remains confidential
☐ publications pending
☐ No prior claims to the technology

☐ Technology protected by patents
☑ Patent pending
☐ Monograph available
☐ Inventor technology champion No
☐ Inventor team player
☐ Industrial partner indentified

## 5. LIST OF EQUIPMENT BOUGHT USING THIS VOT

1. Computer and UPS
2. Printer
3. temperature/humidity data loggers
4. weather stations (mini)
5. air flow TA 45

## 6. STATEMENT OF ACCOUNT

| | | |
|---|---|---|
| a) | APPROVED FUNDING | RM : 118,000.00 |
| b) | TOTAL SPENDING | RM : 118,000.00 |
| c) | BALANCE | RM : 0 |

## 7. TECHNICAL DESCRIPTION AND PERSPECTIVE

*Please tick an executive summary of the new technology product, process, etc., describing how it works. Include brief analysis that compares it with competitive technology and signals the one that it may replace. Identify potential technology user group and the strategic means for exploitation*.

a) Technology Description

Solar chimney to induced air ventilation as passive strategy to replace air conditioning unit. Reduce cooling load and low energy used in terraced house

b) Market Potential

In residential development especially for low cost terrace housing

c) Commercialisation Strategies

Technology

Further study required by installing the prototype on the existing terrace house and the result will provide scientific evidence towards commercialisation of the product

## 8. RESEARCH PERFORMANCE EVALUATION

a) FACULTY RESEARCH COORDINATOR

| | Excellent | Very Good | Good | Satisfactory | Fair | Weak |
|---|---|---|---|---|---|---|
| Research Status | ( ) | ( ) | ( ) | ( ) | ( ) | ( ) |
| Spending | ( ) | ( ) | ( ) | ( ) | ( ) | ( ) |
| Overall Status | ( ) | ( ) | ( ) | ( ) | ( ) | ( ) |

Comment/Recommendations :

______________________________________________

______________________________________________

______________________________________________

______________________________________________

______________________________________________

b) RMC EVALUATION

| | | | | | | |
|---|---|---|---|---|---|---|
| Research Status | ( ) | ( ) | ( ) | ( ) | ( ) | ( ) |
| Spending | ( ) | ( ) | ( ) | ( ) | ( ) | ( ) |
| Overall Status | ( ) | ( ) | ( ) | ( ) | ( ) | ( ) |
| | Excellent | Very Good | Good | Satisfactory | Fair | Weak |

Comments :-

____________________________________________________________

____________________________________________________________

____________________________________________________________

____________________________________________________________

____________________________________________________________

____________________________________________________________

Recommendations :

☐ Needs further research

☐ Patent application recommended

☐ Market without patent

☐ No tangible product. Report to be filed as reference

---

UT

## UNIVERSITI TEKNOLOGI MALAYSIA
**Research Management Centre**

## PRELIMINARY IP SCREENING & TECHNOLOGY ASSESSMENT F

*(To be completed by Project Leader submission of Final Report to RMC or whenever IP protection arran*

1. **PROJECT TITLE IDENTIFICATION** :

TOWARDS DEVELOPMENT OF TROPICAL SOLAR ARCHITECTURE THE U

CHIMNEY Vote

2. **PROJECT LEADER** :

Name : MOHD HAMDAN BIN AHMAD