ABSTRAK

Pembangunan di bandar telah menjadi satu kebimbangan dengan peningkatan penduduk. Kualiti udara yang rendah kerap berlaku pada ngarai jalan berpunca daripada kelemahan udara persekitaran dan system ventilasi yang lemah disebabkan konfigurasi bangunan yang kompleks. Melalui kajian ini, kesan aerodinamik pada bumbung di medan aliran yang bergelora dalam street canyon dianalisis untuk mendalami kefahaman kesan elemen kedua pada struktur aliran udara. Kajian ini menggunakan RANS untuk Kenyon yang tetap dengan nisbah W/H = 3, yang dimana W ialah kelebaran jalan dan H mewakili ketinggian bangunan. Lima konfigurasi bumbung yang berbeza disimulasikan untuk memahami hubungan antara bumbung dan ciri-ciri aliran bergelora pada Kenyon jalanan.

ABSTRACT

Urban development has become very frightening with the rise of population. Low air quality levels have been often observed in street canyons due to weak wind environment and poor ventilation caused by complex buildings configurations. Through this study, the aerodynamic effect of overhang on a turbulent flow field within a street canyon is analyzed in order to deepen the understanding on the effect of secondary roughness element on the air flow structure. The study is conducted using RANS simulations for a constant canyon aspect ratio of $W/H = 3$, where W is the street width and H is the height of the building. Five different variables configuration of the overhang were simulated to understand the relation between overhang and the characteristics of the turbulent flow within the street canyon.

DEDICATION

To my beloved mother Mrs. Noraini Binti Kasim and my beloved father Mr. Mohd Zamani Bin Hassan

ACKNOWLEDGEMENT

I would like to thank and appreciate to the people who support me to finish this report. Firstly, special thanks to my final year project supervisor, Puan Noor Saffreena Binti Hamdan because has given guidance and advice to finish my report. Special appreciate also to the other parties that involve and gives extra knowledge and idea to improve my writing and also to the panels that help me to improve my presentation and my report. Lastly, special thanks to my parents that give me encouragement and delivered some moral support while finishing this thesis.

TABLE OF CONTENT

LIST OF TABLE

LIST OF FIGURE

CHAPTER 1

INTRODUCTION

1.1 Background of the study

Nowadays, pollution often occurs in this earth which is done by human being. Pollution that often occurs also increased the number of deaths. It is a major bad impact towards people and environment. Pollution is the introduction of contaminants into a natural environment that causes instability, disorder, harm or discomfort to the ecosystem. Almost all countries familiar with the various types of pollution which is air pollution, water pollution and sound pollution. In this study, air pollution is emphasized for human needs which is human need to breathe fresh air instead of contaminated air.

Regarding on air pollution occurring, there is an information state that amount of pre-native death and premature deaths are increases. United Nations Environment Programme (UNEP) stated, annually, over 1 million amount of pre-native death and premature death happened from 1 billion of people who expose of widespread air pollution which is harmful and loss of life. Up to now, air pollution has become one of the most crucial environmental concerns. Especially in urban areas where the location is enclosed environment. Estimated costs incurred by air pollution in the city is 2 % of Gross Domestic Product (GDP), the region where it is in developed countries. Unfortunately, Malaysia is included in developing countries and contributed for 5 % of the cost incurred for air pollution. Rapidity or the compactness of the city, automatically have been producing street canyon and seems clearly contribute to air pollution. This is due to the increase traffic emissions and reduced natural ventilation. (Vardoulakis, Fisher, Pericleous, & Gonzalez-Flesca, 2003).

Recently, wind driven natural ventilation is familiar among the local developers due to the important for housing development. The wind driven natural ventilation is suitable and recommended for rapid urban housing development. Natural ventilation cooling system can improve indoor air quality and greatly reducing energy costs and negative environmental impacts (Mohammadi et al., 2009). Thus, the residents of the building will earn a healthy and productive environmental. So that, improvement of resident's comfort is highly indispensable. Natural ventilation system assist to remove contamination air through urban canyon. On the other hand, natural ventilation can also be conceived with the concept of a circuit whereby equal consideration is given to supply and exhaust. As stated by Y. Takano, that wind flow plays vital role in human comfort and pollutant dispersion (Takano & Moonen, 2013). Mostly, developing countries are the highest demanding of continuous energy equivalent with enhancement population number. The residential area is third largest contributor to the electricity consumption including air-conditioner 17%, fan 10% and other 2% (Kubota, Jeong, Toe, Ossen, 2011).

Most of the residential building designed with overhang or porch. A porch is an external construction to the walls of the main building that allow for sufficient space for a person to comfortably pause before entering or exiting the building. There are various styles and dimensions of porches which depend on the architectural design. This so called secondary urban roughness has made the urban canyon even crowded which will definitely effect the wind driven natural ventilation within it. Thus, it has become a serious concern in designation of residential buildings.

Almost researches have concentrated on the application and the capability of stack ventilation strategy. Typical Malaysian city housing have such various type of porch and the purpose of this study is to introduce the effectiveness of wind driven ventilation system. Traditional automatically approached in investigation of wind effects on past or present buildings. Basically, this investigation is applying flow visualization and measurement using scale model in meteorology. As an example, by applying the wind tunnel equipment. Actually, full-scale measurement is applied to produce data to wind loading in buildings and structure. Besides that, in order to determine the characteristic of atmospheric wind, simulation of natural is strictly needed. Therefore, a typical one-storey terraced house with the variable of presence of the porch and its length are used for the numerical simulation to understand the effect of a porch on ventilation efficiency and the character of turbulent flow within a 2D street canyon. The simulations will be conduct using computational fluid dynamics (CFD) and Reynolds-averaged Navier Stokes (RANS). Computational fluid dynamics (CFD) backed up by the tremendous computing power available today becomes very useful tool for engineers and architects. It allows for accessing the future wind climate in the early stage of design. This is the result of the complexity of the problem as the wind flow zones around the buildings are determined by many parameters and randomness resulting from variations in wind speed, wind direction and the character of the surrounding terrain.

1.2 Problem Statement

Rapid urban development has led to increase pollution levels which often discover in street canyons due to the increase of traffic emissions and reduced natural ventilation. Thus, has significantly affects the levels of comfort of the occupants which cause an upsurge of the consumptions of electricity especially from the air-conditioners and fans. The amount of freshwater usage is often of great concern for electricity generating systems as populations increase and droughts become a concern in the future.

1.3 Objective

- To investigate the aerodynamic effect of overhang on a turbulent flow field within a 2D street canyon using RANS simulations
- To investigate the relation between the configuration of the overhang and the characteristics of turbulent flow within the street canyon
- To analyses the importance of wind driven natural ventilation and the flow pattern

1.4 Scope

The scope of project is to investigate and simulate the aerodynamic effect of overhang on a turbulent flow field within a two-dimensional street canyon. The various configuration of the overhang has become primary in this study considering that the overhang structure is commonly seen in tropical climates for protection from direct solar radiation and rain. The simulation will be performed by CFD method using RANS simulations with standard k- ε equation.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

With increasing awareness towards time regards of the cost and environmental impacts of energy use, natural ventilation has become an increasingly enchanting method for reducing energy use and cost and for providing acceptable both indoor and outdoor environmental quality and maintaining a healthy, comfortable, and productive climate rather than the more prevailing approach of using mechanical ventilation. In favourable climates and buildings types, natural ventilation can be used as an alternative to air-conditioning plants, saving 10%-30% of total energy consumption and more important, eliminate pollutant.

2.2 Types of Ventilation

Basically, people who are working in enclosed area such as offices, shops, factories, hospitals, and laboratories requires a maximum quantity of fresh air purposely for their own healthy environment during working. For safety and healthy environment purposes, the enclosed area must be provided with appropriate ventilation system. In addition, ventilation system has many benefits that human need to know. General ventilation or called ‘dilution’ ventilation is a term used to define the air flow in and out of working area, such as a laboratory space, so that any contaminants are diluted by adding some fresh air. Ventilation system is required because to provide oxygen for breathing in and to remove carbon dioxide during breathing out. Besides that, ventilation system also can remove excess heat and if conditioned, provide heat and keep a comfortable temperature. There is several types of ventilation system which is natural ventilation and mechanical ventilation.

2.2.1 Mechanical Ventilation Effects

Mechanical ventilation is a process of supplying or removing air from indoor area using mechanical system. Usually, this mechanical ventilation system is used in buildings, houses, tunnels, and factories. The major purpose for the system is to remove contaminant air in indoor area. There is several ventilation in buildings such as Demand-Controlled Ventilation (DCV) which to allow the air quality is maintained at the same time, it is able to save energy.

Research on cold-region tunnel design is becoming urgent because of the vast number of infrastructure construction projects being undertaken for railways and highways in cold regions. The prediction of temperature fields, which are used as references for relevant cold-proofing measures, is an important factor in cold-region tunnel engineering design. Numerous studies have been conducted on temperature fields in cold-region tunnels. Bonacina et al. proposed a finite-difference method for addressing melting and freezing with corresponding phase changes; moreover, the problem formulation could be straightforwardly extended to multidimensional cases.(Bonacina, Comini, Fasano, & Primicerio, 1973).

2.2.2 Natural Ventilation Effects

The application of natural ventilation is more complicated in town compared to interior area, particularly in street canyons due to decreased pollutions and wind velocity. Basically, natural ventilation purposely to replace contaminant air with fresh air without using any mechanical system such as blowers or fans. Hence, natural ventilation is strongly recommended because it can save the energy used by ventilating, heating, and air-conditioning systems in a building. In this study, natural ventilation is crucial to act as spread of pollution within the street canyon. When the wind are applied perpendicular to a street canyon, a vortex generated in the canyon where the wind will rotate and ventilate out of the canyon. Natural ventilation effects are divided into two types which described as below.

a Wind Driven

Wind flow pattern that outside area of the building to exist a dynamic pattern of air movement in the structure. For example, when it is known that wind blows steadily from the south in the evening, by putting an opening space or hole in the wall it can help to take advantage of changes in wind.

b Buoyancy Driven

Buoyancy driven is a system that used or applied a heat rising principle and cool settling to move water through vertical system. This situation is same as a chimney because it can let the hot air and smoke flow out from the building through the roof that is not come to the cooler interior.

Usually, these two methods are used together. An architect and engineer realized that dynamic air movement or humidity of air are not isolated. Combination between dynamic air movement and humidity of air can be used for greater effects. Both two types of ventilation are suggested to create and maintain the air quality. Nevertheless, by creating the air movement, contaminates are eliminated from enclosed area.

2.3 Overhang

Overhang or porch is an external construction to the walls of the main building that allow for sufficient space for a person to comfortably pause before entering or exiting the building. Considering that the overhang structures are commonly seen in tropical climates for protection from direct solar radiation and rain which includes our country, thus this study will be focused on the aerodynamic effect of overhang on a turbulent flow field within a street canyon with various configurations of the overhang. Previous study showed that the flow pattern is greatly affected by the aspect ratio W/H (ratio of street width to the height of the building) (Kellnerova, Kukacka, Jurcakova, Uruba, & Janour, 2011). Furthermore, conducted a wind tunnel experiment on the influence of pitched roofs on the in-canyon flow pattern, which shows that pitched roofs are more effective for enhancing the ventilation within the street canyon compared to wide structures gap.(Louka et al., 2002)

Overhang structures has become one of the secondary urban roughness which affecting the urban canopy flow. A study from Hagishima has proved that an overhang on a building facade greatly intruded the flow pattern around the building (Mohd Faizal Mohamad, Hagishima, & Ikegaya, 2015) Also a study from J.Baker who has conducted a study on the effect of various design roof factors using Large-eddy simulation (LES), concluded that the increase or normalized porch length has modified the in-canyon flow and reduces ventilation performance (Cui, Cai, & J. Baker, 2004). Another investigation of ventilation performance with various arrangements of overhang position and inclination angle conducted by Peren which also performed by Reynolds-Averaged Navier Stokes (RANS), suggested that the secondary urban roughness can significantly interrupted urban ventilation thus raises the necessity for improvement of the modeling of urban canopy layer.(Peren, van Hooff, Ramponi, Blocken, & Leite, 2015).

2.4 Reynolds-Averaged Naviers Stokes (RANS)

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. As of late, the researcher is utilizing Computational Fluid Dynamics (CFD) that gives an option way to calculate the ventilation rate and more detailed airflow dissemination inside and around the isolated structures. The advantages of utilizing CFD is less labor use and low equipment cost in development in turbulence modeling. Furthermore, CFD give a superior data result.

The Reynolds-Averaged Naiver Stokes (RANS) is one of the CFD methods which required less computing time compared to other methods. The RANS equations are primarily used to describe turbulent flows. The RANS modelling can determines time-averaged flow parameters, such as the air velocity and temperature, by using turbulence modelling. These equations can be used with approximations based on knowledge of the properties of flow turbulence to give approximate time-averaged solutions to the Reynolds Navier-Averaged Stokes equations. This study will use the k-epsilon (k-ε) model. The standard k-ε turbulence is the transportation equation that k is known as kinetic energy and ε is dissipation rate. The k-ε model is one of the most common turbulence models, although it doesn't perform well in cases of large adverse pressure gradients. It is a two equation model which includes two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy. Reynolds-Averaged Naiver Stokes equation (RANS) for steady, incompressible, isothermal and turbulent flow written as:

$$\frac{\partial \overline{u_j}}{\partial x_j} = 0$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = - \frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\mu}{\rho}\frac{\partial^2 \overline{u_i}}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j}(\overline{u_i' u_j'}) + g_i$$

Two turbulence closure equations for standard k-ε turbulence are as below:

$$\frac{\partial k}{\partial t} + \overline{u}_j \frac{\partial k}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{u_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \frac{G_k}{\rho} - \varepsilon$$

$$\frac{\partial \varepsilon}{\partial t} + \overline{u}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{u_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{1}{\rho} C_{\varepsilon 1} G_k \frac{\varepsilon}{k} - C_{\varepsilon 2} \frac{\varepsilon^2}{k}$$

The effect of overhang will be simulated using CFD simulation by using the OpenFOAM 2.0 software. The OpenFOAM is a CFD toolbox that has a broad range of feature to solve the complex fluid flow. This includes the chemical reactions, turbulences and heat transfer, with solid dynamics and electromagnetic. The software capabilities are that it can solve incompressible flows, multiphase flows, combustion, buoyancy-driven flows, compressible flow and other. With this approach, the various configurations of overhang can be easily visualizes for the study of aerodynamic effect.

2.5 Simulation Domain

The simulation will adopted the computational domain. It consists of an idealized 2D street canyon with the structure height, *H* of 25 mm, and a street width, W of 75 mm, and the aspect ratio *W/H* of 5 for all cases. The domain height is set to 6*H* and span wise domain length will keep at 2*H* (Franke, Hellsten, Schlünzen, & Carissimo, 2007) as suggested by Franke and Hellsten.

2.6 Air Flow Analysis

Air flow analysis is very important in order to achieve the objectives of this study. Air flow analysis which includes the identification of velocity vector of the wind at particular points. From the velocity vector analysis, it will lead to the construction of velocity contours and streamlines of the flow which enable to analyze and visualize the behavior of the wind within the street canyon. The results from the study by Zhiqiang Cui, suggest that the ambient flow properties are slightly different in the vertical direction (Cui et al., 2004). This seems to be related to the different ways by which turbulence is generated. Turbulence was generated by small roughness elements over the fetch in the wind-tunnel experiment. The air flow analysis will definitely bring out something which will help the architectures and engineers regards of the designation of overhang especially involving the residential area. A deep understanding on the air flow behavior and good design of the overhang will certainly achieve the fundamental objective of this study which to feature and prioritize the natural ventilation.

2.7 Air Change Per Hour (ACH)

Air changes per hour, or known as air change rate, abbreviated ACH or ac/h, is a measure of the air volume added to or removed from a space which in this study is the canyon divided by the volume of the space. If the air in the space is either uniform or perfectly mixed, air changes per hour is a measure of how many times the air within a defined space is replaced. The higher rate shows that it is reliable as a good ventilator.

CHAPTER 3

METHODOLOGY

3.1 Introduction

In order to perform a simulation of wind flow pattern in this study, OpenFoam software is used to assist the CFD simulation. Basically, OpenFoam is one of CFD's toolbox which have variety of features to solve the complex fluid flow. Before performing or generating the simulation, the author has identified and created a basic steps from the beginning until end. It is contain three main process which are pre-processor, solver, and post-processor. In those steps are including five elements which are modelling, meshing gneration, solver, validation, and data analysis. The flowchart that has been constructed illustrating all the methodology of the aerodynamic effect of the overhang on a turbulent flow field within a 2D street canyon with variety configuration of overhang.

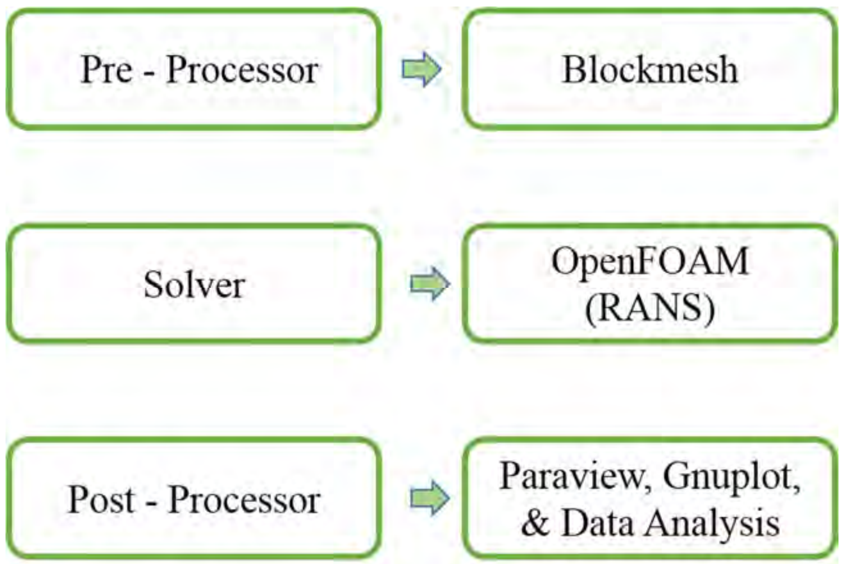

Figure 3.1: Main Process of Computational Fluid Dynamic Simulation

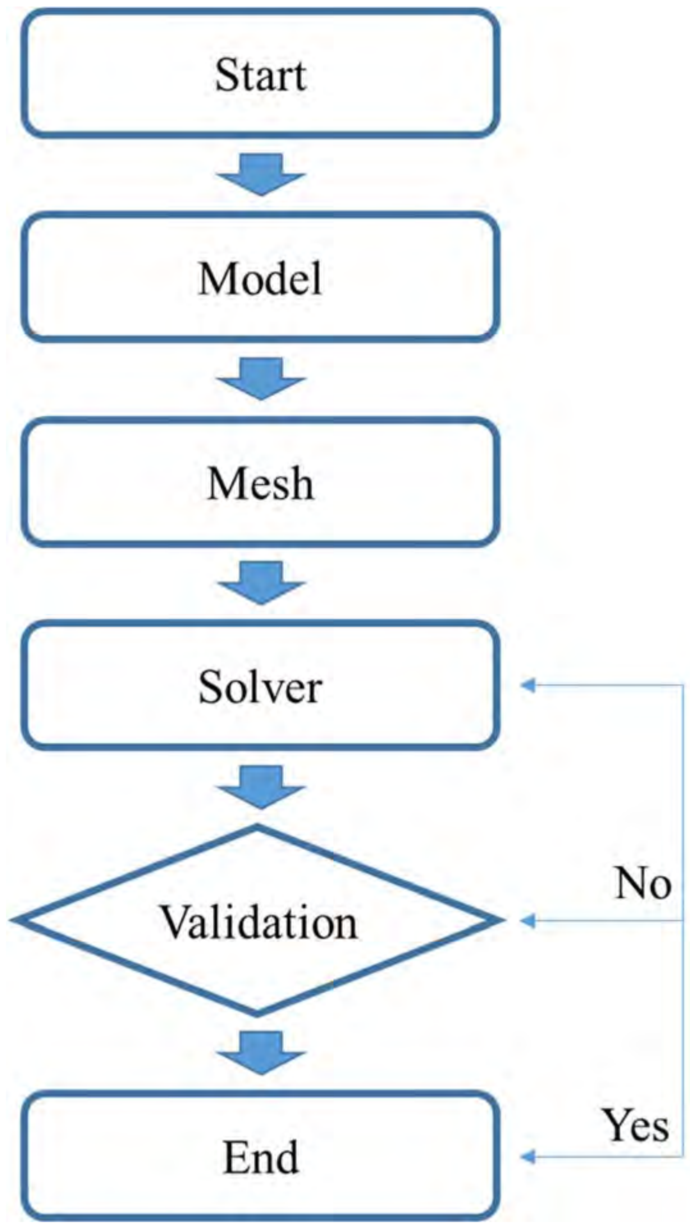

Figure 3.2: The Flow Chart in Methodology

3.2 Pre Processor

Preprocessing is the initial step towards the simulations. In this process the geometry and the physical modelling is defined. Boundary conditions are also defined which involves specifying the fluid behavior and properties at the boundaries of the building. During the meshing generation, the amount of the cells and its size are also calculated.

3.2.1 Model

After the establishment of the literature reviews which to elucidate the research direction, a model of the simulation needs to be build. In this circumstance, because of concentrating on the street canyon, the modeling or demonstrating will be slightly complex as the author needs to ascertain the vertices of every nodes, the inlet surface and outlet surface before assemble them in the blockMesh and export the coding into paraView which embedded with the OpenFoam in order to view the model. In a nutshell, instead of doing the model by using the CATIA software, the author needs to use coding to build all five cases. Figure 3.3 below illustrates the simulation domain and all the five cases.