INVESTIGATION OF WIND LOAD EFFECT ON BUTTERFLY ROOF OF LOW RISE BUILDING USING COMPUTER FLUID DYNAMIC (CFD) SIMULATION

MUHAMMAD MURSYIDI BIN AHMAD SAYUTI

(141538)

SCHOOL OF CIVIL ENGINEERING

UNIVERSITI SAINS MALAYSIA

2022

INVESTIGATION OF WIND LOAD EFFECT ON BUTTERFLY ROOF OF LOW RISE BUILDING USING COMPUTER FLUID DYNAMIC (CFD) SIMULATION

by

MUHAMMAD MURSYIDI BIN AHMAD SAYUTI

This dissertation is submitted to

UNIVERSITI SAINS MALAYSIA

As partial fulfilment of requirement for the degree of

BACHELOR OF ENGINEERING (HONS.)
(CIVIL ENGINEERING)

School of Civil Engineering
Universiti Sains Malaysia

August 2022

Appendix A8

UNIVERSITI SAINS MALAYSIA

SCHOOL OF CIVIL ENGINEERING
ACADEMIC SESSION 2021/2022

FINAL YEAR PROJECT EAA492/6
DISSERTATION ENDORSEMENT FORM

Title: Investigation of Wind Load Effect on Butterfly Roof of Low Rise Building using Computer Fluid Dynamic (CFD) Simulation

Name of Student: Muhammad Mursyidi Bin Ahmad Sayuti

I hereby declare that all corrections and comments made by the supervisor(s) and examiner have been taken into consideration and rectified accordingly.

Signature:

Date: 9th August 2022

Approved by:

(Signature of Supervisor)

Name of Supervisor: Professor Ir. Dr. Taksiah A. Majid

Date : 10th August 2022

Approved by:

(Signature of Examiner)

Name of Examiner: Dr. Mohd Amirul Mohd Snin

Date : 10th August 2022

ACKNOWLEDGEMENT

Alhamdulillah, I'm grateful that I able to complete my research for my final year project. My highest credit goes to my supervisor Professor Ir. Dr. Taksiah A. Majid for help me a lot in completing this research. I would like to thank to my co-supervisor Ir. Dr. Shaharudin Shah Zaini and Madam Junaidah Binti Abdullah for helping me in doing the computational modelling work. Without their assessment, I not able to complete my thesis.

I would like to thanks to my parents Ahmad Sayuti Ismail and Zanariah Ali for providing the moral and financial support. Without their support, I would not have the capability to complete my research. Thankyou also to my sister Nasuhah and Najwa for support me in these studies and providing me guidance throughout this study. May god reward their good deeds with something beautiful.

In the journey to complete the research, I was surround by good people that help me to the modelling and thesis work. They are Taqif, Faiz and Mardhiah that always help to solve my problem in modelling work. Without their help, I would stick in the middle of the journey with all the errors comes out from the modelling work. Conducting the modelling using software giving me a lot of challenge especially on error solving. But, with the help of my Supervisor, Co-supervisor and my member, i able to overcome the errors showed up.

Finally, I would like to thank the computer lab staff, Mr. Abdul Muluk Tajudin and Mr. Mohd Mazlan Kamis@Mahmad for their help and permission given to use the computer lab even in the weekend. Their sacrifice of family time in weekend just to ensure us able to do the modelling work cannot be repaid in dollars. At the end, thankyou again for all the name mentioned above for helping me to complete this research.

ABSTRAK

Tragedi bencana angin telah menyebabkan banyak kerosakan bumbung di Malaysia. Kerosakkan bumbung berlaku apabila struktur bumbung tidak dapat menahan tekanan angin dan daya angkat yang tinggi. Kajian ini dibuat untuk menentukan taburan tekanan, corak aliran angin, hubungan antara tekanan bersih angin dengan profil bumbung, dan penggunaan overhang pada bumbung. Terdapat banyak kajian yang dilakukan dalam skop ini tetapi kebanyakan kajian hanya tertumpu pada bentuk bumbung tipikal yang dilsebut di dalam MS 1553 2002 seperti bentuk *hip, shed, gable* dan *bumbung rata*. Penyelidikan ini tertumpu kepada bumbung bentuk rama-rama kerana jenis bumbung ini kurang ditemui dalam penyelidikan CFD terdahulu. Parameter utama untuk analisis adalah sudut bumbung dan penggunaan overhang pada struktur bumbung. Model turbulence RNG k-ε digunakan dalam kajian ini kerana model ini mempunyai keupayaan untuk memenuhi objektif kajian ini. Berdasarkan keputusan yang diperolehi, didapati bumbung dengan sudut yang lebih tinggi cenderung mempunyai kesan sedutan yang lebih tinggi dan bumbung dengan sudut yang lebih rendah cenderung mempunyai taburan tekanan yang lebih tinggi terutamanya di bahagian arah angin. Nilai +Cp tertinggi bagi analisis keseluruhan ialah 0.997 yang dialami oleh model bumbung rata di zon A. Nilai -Cp tertinggi yang diperolehi ialah -2.94 bagi model bumbung bentuk rama-rama dengan kecerunan 30°. Akhir sekali, tekanan bersih (-Cp) pada overhang arah angin adalah lebih tinggi daripada overhang di bahagian belakang dan ini menunjukkan bahawa overhang arah angin mempunyai kesan sedutan tertinggi berbanding zon lain.

ABSTRACT

The wind disaster tragedy has caused a lot of roof damage in Malaysia. The roof failure is occurred when the roof structure is not able to retain the high magnitude wind pressure and suction. This study is made to determine the pressure distribution, streamline and net pressure relationship with roof profile, slope and overhang application on the roof. There are a lot of studies conducted in this scope but the most of the study just focus on the typical roof shape that covered in MS 1553 2002 like hip, shed, gable and flat roof. This research is focused on Butterfly shape roof as this roof type is less discovered in the CFD past research. The main parameter for the analysis is the roof slope angle and overhang application on the roof structure. The RNG k-ε turbulence models were used in this study as this model has the capability to fulfil the objective of this study. Based on the result obtained, it was found that the roof with higher slope angle tend to have higher suction effect and the roof with lower slope angle tend to have higher pressure distribution especially in the windward section. The highest +Cp value for overall analysis is 0.997 that experienced by flat roof model in zone A. The highest -Cp value obtained is -2.94 for butterfly shape roof model with 30° slope. Finally, the net pressure (-Cp) at the windward overhang is higher than leeward overhang indicate that the windward overhang has highest suction effect.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER ONE

INTRODUCTION

1.1 Background of Study

Butterfly shape roof is getting popular nowadays in construction industry as it provides high aesthetic value to the house. The ability of this roof type to resist high wind pressure distribution must be investigate to find the best parameter in design the roof. There have been numerous of roof failure tragedy occur at north of peninsular Malaysia due to windstorm (Majid *et al*., 2015). There are few types of windstorms that is cyclone, local windstorm, and downslope windstorm (Yahya *et al.*, 2018). Past storm reports and other related post-disaster studies have revealed a huge loss of and property in both storm area and storm prone area (Roy *et al.*, 2018).

The investigation of the streamline and pressure distribution on the butterfly roof can be done through laboratory work or computer modelling work. Both of the experiment method can produce the approximate result. The laboratory helps the observer to get better observation on the roof reaction when subjected to wind load. In term of cost and time, the wind experiments through modelling are effective approaches to investigate low rise building behavior to get better designs (Pan *et al.*, 2014). This is because the modelling work do not require any physical object, machine or laboratory that require extra cost. This research will be focused more on computer modelling technique to investigate the streamline and wind pressure distribution on butterfly shape roof. Figure 1.1 shows the butterfly roof model.

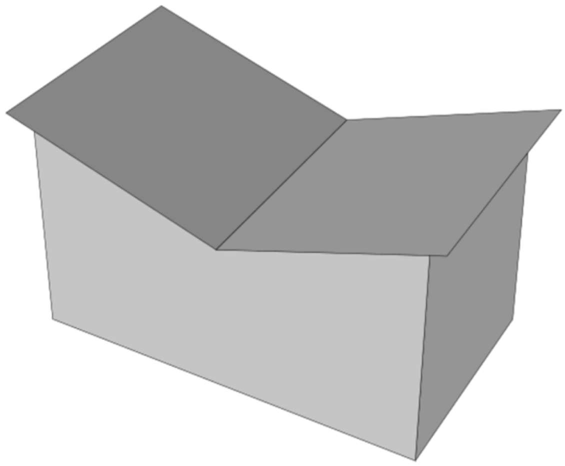


Figure 1.1 Model of butterfly shape roof

The computer modelling technique used in this research is Computer Fluid Dynamic (CFD) simulation method. Computational Fluid Dynamics (CFD) is a multipurpose tool useful to evaluate the aerodynamic force on the roof (Jing *et al.*, 2014). Extreme weather events such as hurricanes, tornadoes, and storm surges have a devastating impact on low-rise buildings, which account for most direct and indirect damage (Kyung *et al.*, 2018 and Shanmugan *et al.*, 2013). Computer modelling work help to evaluate the characteristic of the low rise building with different roof profile and slope subjected to the wind load.

1.2 Problem Statement

The wind disaster event has caused a lot of roof failure tragedy occurred in the affected area. There has been a lack of studies on butterfly roofs with different slope angles and roof profiles subjected to the wind load. Furthermore, some studies by previous researcher like Roy et al. (2012) and Singh et al. (2018) do not consider the roof's overhang as part of the design variable. In addition, the butterfly roof shape is not covered in the MS 1553 2002 Code of Practice on Wind Loading for Building Structure.

The use of butterfly roofs on low-rise buildings in Malaysia increases due to its aesthetic value. Hence, this study will focus on the wind analysis of different roof profiles of low-rise buildings with butterfly-shaped roofs. In addition, the focus of this study is also to evaluate the behaviour of the butterfly roof with different slopes subjected to pressure distribution. This study will be conducted through Computational Fluid Dynamics (CFD) simulation using ANSYS-14 software. This study is important to ensure the most effective design for roofs of low-rise buildings can be obtained to reduce the roof damage incident.

1.3 Objectives

This study is focused on two objectives in study the effect of wind flow on the butterfly roof using CFD simulation namely:

1) To investigate the pressure coefficient distribution and streamline on the butterfly shape roof with different slope angle
2) To evaluate the net pressure acting across the roof overhang

1.4 Scope of Work

This study is focused on numerical experiments through computer fluid dynamics to evaluate the pressure distribution and streamline of low-rise building roofs. The scope of this study is focused on the butterfly roof of a low-rise building. The study on the butterfly shaped roof is carried to determine the characteristic of air flow passing through a butterfly shaped roof.

There are two methods to undergo the wind load analysis that is through lab work and numerical method. These studies use numerical method through computer fluid dynamic (CFD). The computational analysis is made by using ANSYS fluent 14 software to get the streamline, pressure distribution, Cp value, and net pressure value. The Tecplot software is used to display the data of the analysis in a clearer way. The wind analysis numerical model is developed after all of the required parameters obtained from the past study.

In this research, the parameters are the slope angle of the roof and the presence of the roof overhang. The slope angle used in the model analysis are 0°, 10°, 15°, 20°, 25° and 30°. A few of data must be obtained from previous study to be keyed in the ANSYS FLUENT 14 software. The input data is taken from Zaini *et al.* (2019) to develop the actual model. The relevant CFD input parameter is identified from the previous researcher that cover same scope of study but different roof shape (Abdullah *et al.* 2019). The type of the turbulent model that will be used in the study is determined. In this case, the RNG K- ε turbulence model was used for the analysis.

This research will include the roof overhang as part of the roof structure. The roof overhang plays an important role in protecting the house from heavy rain and hot weather conditions. In Malaysia, most of the house or low-rise structure applied roof overhang on roof structure to face extreme weather events. The magnitude of wind load acted on the roof is not affected by increment of overhang length. Thus, in this study the overhang length is set to 0.75m.

1.5 Layout of Dissertation

This study is divided by 5 section that is introduction, literature review, methodology, results and discussion followed by conclusion. The chapter 1 of this study focused on the introduction part. The introduction part contains the background of this research, problem statement, objective and scope of work for the studies.

In chapter 2, the focus is lay on the literature review whereas the studies are made on the previous research. The gap of the previous research is filled by conducting new studies to improve the studies on that topic. There are few of articles by previous researcher related to computer fluid dynamic (CFD), wind tunnel test, roof resistance to high wind distribution and wind disaster is explored.

Chapter 3 focuses more on methodology part where in this part, the workflow and method used along the study is discussed. The method in conducting computational modelling using ANSYS fluent 14 software is described in this part. The CFD input parameter is obtained from previous studies by (Abdullah *et al*. 2019). The input data for model development is taken from Zaini *et al.* (2017).

In chapter 4, the result obtained from the computational analysis is discussed. The justification is made based on the observation on the result obtained. The result is discussed quantitatively and qualitatively.

Finally in chapter 5, the conclusion of this study is made by concluding all of the result obtained based on the objective. The benefit of this study to the engineering field is briefly discussed.

CHAPTER TWO

LITERATURE REVIEW

2.1 Introduction

Lately, the roof failure tragedy due to wind disaster has increased rapidly in Malaysia. The wind disaster is defined as occurrence of an unexpected event that cause a lot of damage to life, property, environment and treasury (Embong, 2013). The study by Majid *et al.* (2019) stated that a lot of the house damage due is occur due to thunderstorm event. The average damage on properties can result in losses ranging from thousands to millions of ringgits (Sitam, 2014). The non-engineered low-rise building is undergoing the most damage as there was lack of technical calculation behind the construction (Zaini *et al.*, 2017).

The structural design and calculation play an important role in ensuring the structure is able to resist high wind load. The basic wind speed at certain area taken from metrological institute must be considered in calculation design of building (Nizamani *et al.*, 2018). By emphasize the technical aspect in designing the low-rise building, the potential of roof fail or any structural damage to occur can be reduced. Thus, the assessment of wind performance and effect to the low rise building need to be improved to improve the building ability to resist high wind load. Figure 2.1 illustrates the typicall roof damage of low rise house that frequently occurred in Malaysia

Figure 2.1 The roof damage in occurred in peninsular Malaysia

Reference: The star, 12th April 2022, Strong winds blow off roofs of 35 houses in Salama, Accessed 30 May 2022

2.2 Low Rise Building

The low rise building is is building with a few stories that can be tall as 3 or 99 meters in height (Diesel, 2019). American Civil Engineer Society (ASCE) defined the low rise building as the building that has roof heigh less than 18m. The development of low rise building has growing rapidly as the demand on better housing (Tsopa *et al.*, 2019). The low rise property is more prefered compare to high rise property as it more convenience and comfortable.

Most of the low rise building in Malaysia is non engineered building especially in rural area (Zaini *et al.*, 2019). The low rise building is exposed to roof failure incident as the low rise building not given strict calculation as the tall building (He *et al.*, 2017). The construction of the non-engineered low rise building is just based on estimation by the contractor. The wind load is seldom being considered in the construction of then low rise building . This lead to occurrence of roof failure tragedy whereas the roof structure is not able to support high wind pressure distribution.

The ability of the roof of low rise building to resis high wind load is depend on it geometry , inteference, space and wind direction (Khanduri *et al.*, 1998). If the building was surrounded other building, the building will experience less pressure distribution. If the building is located at open space, it tend to experience higher wind load. Furthermore the terrain characteristc at the low rise building located is also influence the magnitude of wind load subjected to the building (Abdullah *et al.*, 2019).

2.3 Type of Roof House

Different roof type is used in the construction of house and all of roof design has it own pros and contras. All of this roof structure is subjected to wind pressure distribution with certain magnitude (Singh and Roy, 2019). The type of roof play significant role in determine the roof ability to resist high wind pressure distribution. In Malaysia Standard 1553 Code of Practice on Wind Loading for Building Structure, the covered roof type is only flat, hip and gable roof.

Flat roof is the roof type that does not have angle to form the shape of the roof. According to research by Razavi and Sarkar (2021), the flat roof has experienced highest uplift force, overall shear and moment compare to other roof type. The moment effect coefficients on various sections of flat roof buildings is higher than other roof. This is because flat roof geometry has lowest aerodynamic perfomance when subjected to high wind pressure (Prasad *et al.*, 2009).

Hip roof is one of roof geometry type which all the slope is pointing downward to the wall of the building. Hip roof are typically assumed to be more resistant to wind damage under harsh weather conditions compare to gable roof (Stevenson *et al.*, 2018). The geometry features of the hip roof has better aerodynamic for resist the force.

In other hand, gable roof is roof type where two sloping is meet each other at the ridge. For the gable roof , the slope angle played important roles in determine the wind load distribution on the roof. According to Gavanski *et al.* (2013) lower roof slopes are subjected to strong wind loads along the ridge, particularly towards the gable end, when exposed to oblique winds

Despite all of the roof discussed abbove, the butterfly shape roof is getting popular by days in Malaysia. This research will be focused more on butterfly shape roof as theres roof type is less discovered compare to other roof type. In MS 1553 : 2002 Code of Practice on Wind Loading for Building Structure, the butterfly roof is not covered.

The butterfly shape roof comes with a lot of advantage such as it ability to harvest the rain, chanelling the rain water for external use, provide large wall space,and high aesthathic value. Despite of it advantage, the butterfly shape roof has some drawback such as high installation cost, hard to detect damage at roof surface and not suitable in heavy rain area as the load of accumulate rainwater can effect the roof structure. Figure 2.2 shows the practical of butterfy shape roof in Malaysia.

Figure 2.2 House with butterfly roof

Reference: 23 March 2022, Bukit Palma, Cheras, Selangor.

2.4 Method of analysis

The wind load analysis is important to evaluate the ability of the low rise building to reisist high pressure distribution. The wind load around a structure can be explored physically in a wind tunnel or numerically through computer simulations (Deraman *et al.*, 2019). Both of the method is used widely in this scope of study and each method has it own ability. According to Tarbizi *et al.* (2015), the field experimental work has become the main choiche for some researrcher to conduct wind load analysis.

The experimental work is commonly conducted using wind tunnel test. The wind tunnel test is used to investigate the wind characteristic toward the building (Zou *et al.*, 2021). The wind tunnel experiment have been performed in order to provide a more precise reading of the wind pressure (Rechard *et al.*, 2017). In term of it ability, the wind tunnel experiments provide better control over the boundary conditions than other methods. (Hoof *et al.*, 2011).

The computational fluid dynamic (CFD) is type of numerical method to analys the fluid flow and flow characteristic. The computational fluid dynamics (CFD) approach has become to be the most preferred instrument for roof wind field study (Zou *et al.*, 2019). According to Roy *et al.* (2018), the CFD method is used as an alternative to the labaratory work through wind tunnel test.

The CFD has a lot of advantage compare to labarotary work using wind tunnel test. Fisrtly the parameter like boundary condition, reynold number and slope of the roof can be changed easily (Fouad *et al.*, 2018). Then, Numerous studies have been conducted using CFD simulations rather than wind tunnel testing, and the results obtained using CFD simulations are virtually identical to the experimental results (Bhattacharyya *et al.*, 2014). Besides, CFD simulations were shown to be quite successful in determining the unsteady aerodynamic forces acting on the vibrating roof (Ding *et al.*, 2014). In other hands, the numerical method through CFD also saved more cost and time to conduct as it does not require lab work or any physical object. This supported by Canonsburg (2013) that stated CFD decreases both the amount of time and money spent on design and research, while also providing detailed.

In this studies, the focus of using computer fluid dynamic (CFD) is to obtain the pressure coefficent value and streamline of airflow. The wind experiments through modelling are effective approaches to investigate low rise building behaviour to get better designs (Pan *et al.*, 2014). There are few software can be used to conduct the computational fluid dynamic (CFD) analysis such as Ansys Fluent, Autodesk CFD,Solidwork, and CFD Module. This research is focused more on ANSYS FLUENT version 14 software in conducting the CFD analysis.

2.5 Post Experimental Work

The post experimental work is mostly focused on the relation of wind pressure distribution and roof profile. The study by Bienkiewicz and Sun (1992) investigate the pressure distribution acting on roof of low rise building by wind tunnel test. The study considered the effect of wind direction to magnitude of wind pressure acted on roof. Xu *et al.* (1996) studied the wind pressure distribution on the gable roof. The study is conducted using wind tunnel testing method. In this study, the pressure acted and the characteristic of airflow on the gable roof is compared with hip roof. This study only covered the flat roof structure with the presence parapet on roof structure.

The Ginger and Holmes (2003) discussed on the effect of length to span aspect ratio on wind load distribution on roof. The study covered the low-rise building with gable roof and the analysis is conducted through wind tunnel test. The roof angle of each roof is adjusted according to the span and length dimension. The comparison on bending moment and pressure coefficient is made between each model with different span dimension. Then, Tominaga *et al.* (2015) studied the characteristic airflow around the isolated gable roof building. Two method was used for analysis task that is wind tunnel test and computational fluid dynamic (CFD). The wind tunnel analysis is carried for three model with different pitch as a measurement database for time-averaged velocity, turbulent kinetic energy, and pressure coefficient around the building. The result from CFD simulation was compared with the measurement database.

2.6 Post Computation Fluid Dynamic (CFD) Analysis

There are few studies have been conducted using computational fluid dynamic (CFD) before this. The CFD was used to discover the effect of additional roof structure to wind flow characteristic. Balduzzi *et al.* (2012) applied a CFD approach to examine how the direction, size, and height of the structure all affect the wind pressure distribution.

Then, the study by Abdullah *et al.* (2019) find that the variance of terrain height influences the pressure distribution on low rise building using CFD. The pressure coefficient of the wind is increase at higher ground elevation. The wind direction influenced the magnitude of pressure distribution on the structure (Yahya *et al.*, 2019). The negative pressure coefficient increases when the roof of the structure is exposed to oblique pressure.

The study by Deraman *et al.* (2019) discovers that the presence and absent of kitchen house also had direct influence to the wind pressure distribution on low rise structure. The suction magnitude is one of the outputs that produced the difference between two types of roofs. In addition, the presence of roof overhang also influenced the magnitude of pressure coefficient.

The research by Singh *et al.* (2018) is focused only on pyramidal roof to investigate the pressure distribution on house using CFD. This study evaluates two variable parameter which is roof slope angle and wind direction. The output for this study is pressure coefficient, Cp and velocity streamline for different model. Besides, the building opening effect on the pressure distribution also observed in this study.

The research by Majid *et al.* (2019) discovered that the kitchen house and roof overhang has produced different result. The presence of kitchen house had cause positive pressure occur due to change of wind direction. The change of wind direction is occur when the wind hit the kitchen house cause the eddies size become bigger. In term of roof overhang, the length of overhang is not influenced the magnitude of wind pressure distribution on the structure.

The study by Abdullah *et al.* (2019) covered the investigation of airflow characteristic of rural house model located on different terrain height. The terrain height relationship with the pressure coefficient and streamline is determined in this study. However, the focus on this research is limited to low rise building is rural area. This is because most of the roof failure tragedy is occurred at rural area as the structure is built without engineering calculation. The gable roof with slope angle of 22º is used in this study for all low-rise building model.

Singh and Roy (2021) investigate behaviour of conical roof structure subjected to wind load. The studied is carried by computational fluid dynamic (CFD) method using Ansys fluent software. Different slope angle (20º, 25º, 30º, 35º,40º) is applied for the conical roof model to determine which angle has optimum high wind load resistant ability. The finite volume method is used in numerical solution to run the analysis in this study.

2.7 Summary of Post Studies

There was a lot of study has been conducted through the years related to roof failure tragedy due to wind disaster. Each of the study covered many parameters related to this research including roof geometry, method of analysis, software used and the roof profile. Table 2.1 summarizes the related past study together with method used and findings of the studies. There are two common methods, namely Wind Tunnel Test (WTT) and Computer Fluid Dynamics (CFD).

Table 2.1 Summary of Post Studies

Author(s)/ Year	Method	Findings
Bienkiewicz and Sun (1992)	WTT	• Highest suction on the roof of a low-rise structure occurred at the windward edges. • The present of parapet on roof structure affect the roof overall suction. High parapet structure tends to produce low suction • Wind direction of 235° produced the highest suction coefficient
Xu *et al.* (1996)	WTT	• The wind suction occurrence is variance on different part of gabble roof • The roof configuration and types of roof sheeting influenced the roof damage • Under same wind pressure magnitude, the hip roof suffers more damage than gabble roof at cladding part.
Ginger and Holmes (2003)	WTT	• The combination of a steep roof pitch and a big horizontal aspect ratio (length/span) caused high negative pressures on the leeward roof slope • High average of negative pressure was observed on leeward of the roof, wall and windward of the roof • The inner part of the building undergoes the lowest magnitude of pressure coefficient.
Roy *et al.* (2012)	CFD	• For the pyramid shape roof with slope lower than 30°, the suction is occurred at whole surface. • Building model with slope15 to 20° has higher resistance to wind disaster than another roof angle • Suction on the roof faces has risen dramatically if the window of the building is opened.

Tominaga *et al.* (2015)	WTT & CFD	• The RNG K-ε model is the best turbulence model as a result from sensitivity analysis. • The slope angle influenced the separation flow at the edge. The increase of roof angle lead to larger separation flow at roof pitches
Majid *et al.* (2019)	CFD	• The additional structure on low rise building directly affect the airflow characteristic on that building • The eddies recirculation is produced if the wind direction change and longer roof overhang will increase eddies recirculation size. • The overhang length does not contribute to huge influence on wind pressure magnitude.
Singh *et al.* (2018)	CFD	• The roof with pyramid geometry undergoes less suction compare to other roof type. • The roof that exposed to wind direction experienced maximum section but it can be reduced by increase the slope angle of the roof • Suction increases on leeward units and decreases on windward units over the centre section of the roofing due angle increment
Abdullah *et al.* (2019)	CFD	• The different terrain height influenced the pressure distribution on the roof • The steeper terrain produced experience highest suction • The magnitude of pressure coefficient is not increase with terrain slope increment • The magnitude of suction increased with terrain slope increment
Deraman *et al.* (2020)	CFD	• The present of kitchen house effect the wind flow characteristic of the house that cause different pressure distribution on each part of house • The roof overhang, roof profile and kitchen position effect the value of pressure distribution • The kitchen house helps to reduce the potential of dangerous effect from wind disaster as it reduces the wind pressure.
Singh and Roy (2021)	CFD	• The outcome of the CFD analysis using k-ε is precise compared to other turbulence model. • The highest negative pressure is occurred at 10 ° angle • From the analysis, the roof with slope angle of 35° was founded as the optimum roof slope as it experienced the least wind load distribution.

CHAPTER THREE

METHODOLOGY

3.1 Introduction

This chapter described all of the task involved in the Computational Fluid Dynamic work from pre-processing to post processing. Ansys Fluent version 14 is used in the analysis as this version has enough capability to produce best output for these studies. Ansys Fluent is an engineering software that can be used to make a modelling, simulation and testing of engineering objects. The Tecplot software was also used in this study to interpreted the result outcome from Ansys Fluent. Figure 3.1 shows the flow chart of the work involved in these studies and the further explanation on each work is presented in following section.

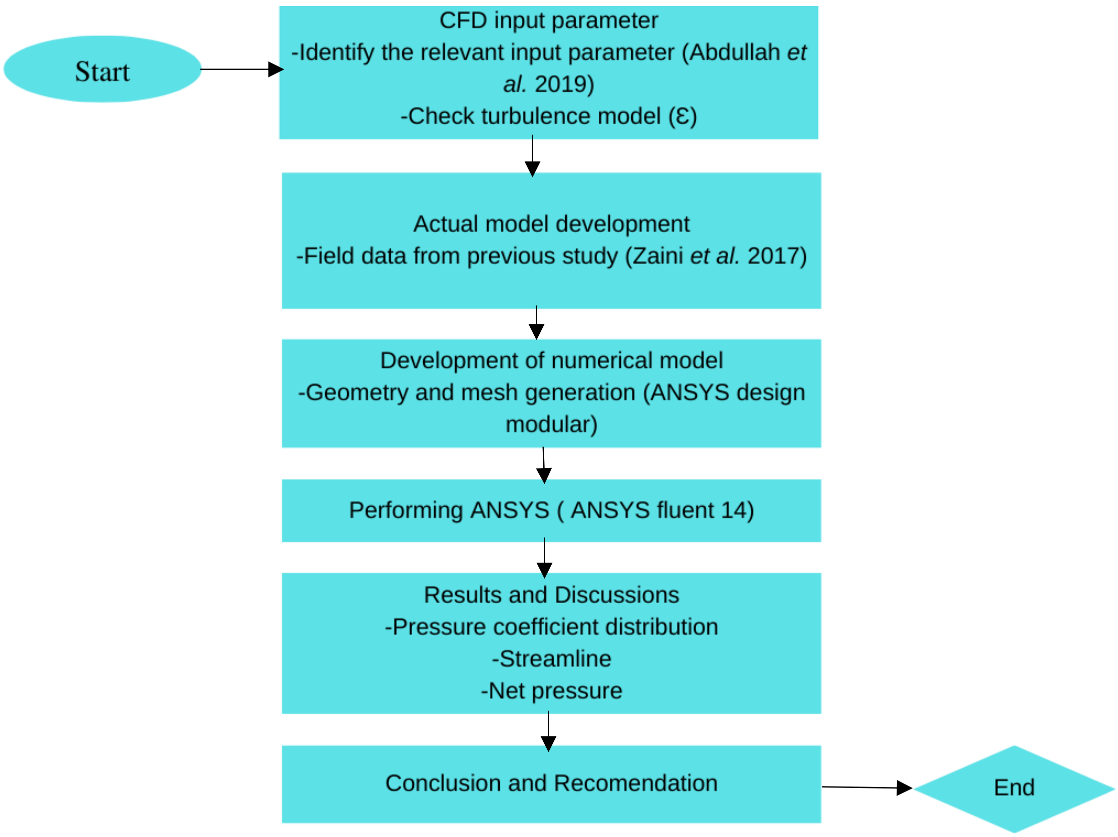


Figure 3.1 Flow chart of Study

3.2 CFD Input Parameter

To analyze the model using computational method, the input parameter must be obtained. In this study, the input parameter refers to the parameter that will be used to analyze the pressure coefficient and streamline of low-rise building. The input data is taken from previous study of CFD simulation (Abdullah *et al.*, 2019). The input data obtained has gone through the validation exercise. The validation exercise of the input data by Abdullah *et al.* (2019) has smallest error in comparison with Tominaga WTT.

The input data obtained in the study also suitable to be used for the flat terrain circumstances (Abdullah *et al.*, 2019). This input data can be used for other CFD studies involving low rise structure. Thus, the input from Jun Abdullah *et al.* (2019) is suitable to be used in this study to evaluate the pressure coefficient, Cp and stream line for low rise building with butterfly shaped roof. Thus, referring to the validation work done by Abdullah *et al.* (2019), the generated input data can be used in this study.

3.3 Modelling Work

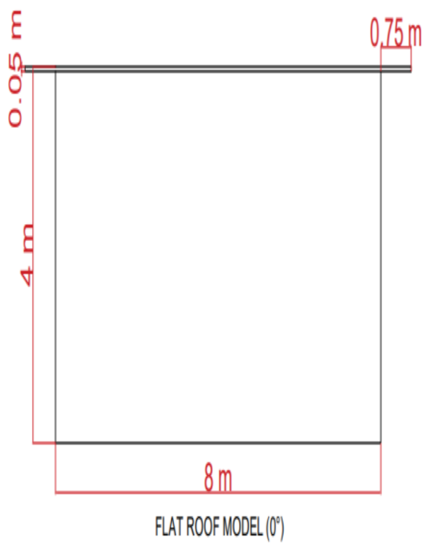
3.3.1 Actual Model Development

This section discussed the development of the actual model used in the analysis part. The butterfly shape roof models with the slope angle of 0 °, 10°, 15°, 20°, 25° and 30° is developed. The 0° model is made without the present of roof overhang while the 10 °, 15 °, 20 °, 25 ° and 30 ° is made by considering the roof overhang. The field data of the model is obtained from previous study by Zaini *et al.* (2017). The actual model is sketched in Ansys fluent software based on field data obtained. The actual model sketch is displayed in Figure 3.2. The line that connected the must be fully closed to avoid any error during model generation. The detail of the low-rise building model and its sketch presented in Table 3.1

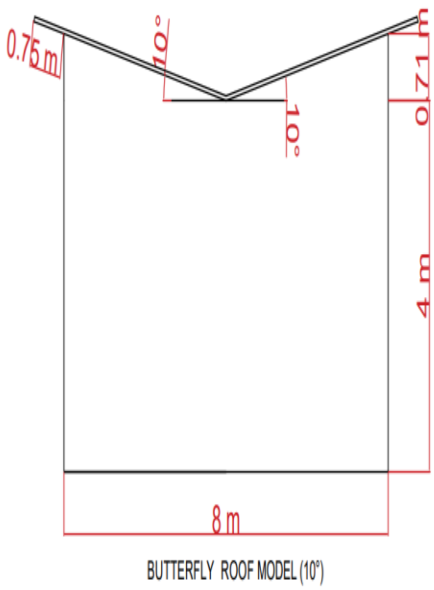
Table 3.1 Dimension of model component

Component	Dimension (m)
Height	4 m
Length	8 m
Overhang length	0.75 m
Overhang width	0.50 m
Roof height	0 m ,0.71 m,1.07 m 1.46 m, 1.87 m, 2.31 m (For roof angle 0°, 10°, 15°, 20°, 25°, 30 °, respectively)

Model details (2-Dimension)



(a)



(b)

Figure 3.2 Actual model sketch for (a) flat roof and (b) butterfly roof with 10° angle

3.3.2 Development of Numerical Model

3.3.2.1 Model Labelling

In these studies, 6 model were developed and each model have different slope angle. To make the model easy to read and presentable, the model is labelled based on it slope angle. The model is labeled base on the type of roof and number by sequence. The detailed information on the labelling of the model is displayed in the Table 3.2

Table 3.2 Model Labelling

Slope angle	Label
0°	Flat roof
10°	10° slope
15°	15° slope
20°	20° slope
25°	25° slope
30°	30° slope

3.3.2.2 Building Data

Figure 3.3 shows the cross section and marking of the parts on the low-rise building model. In attempt to make the model presentable and easy to understand, each component on the building is marked using alphabet. This model dimension is taken from the field data of previous study by Zaini *et al.* (2017). This dimension represents the average size of low rise building in Malaysia.

- Zone A (Windward wall of core house)
- Zone B (Windward overhang)
- Zone C (Decline roof)
- Zone D (Center of the roof)
- Zone E (Incline roof)

- Zone F (Leeward overhang)
- Zone G (Leeward wall of core house)

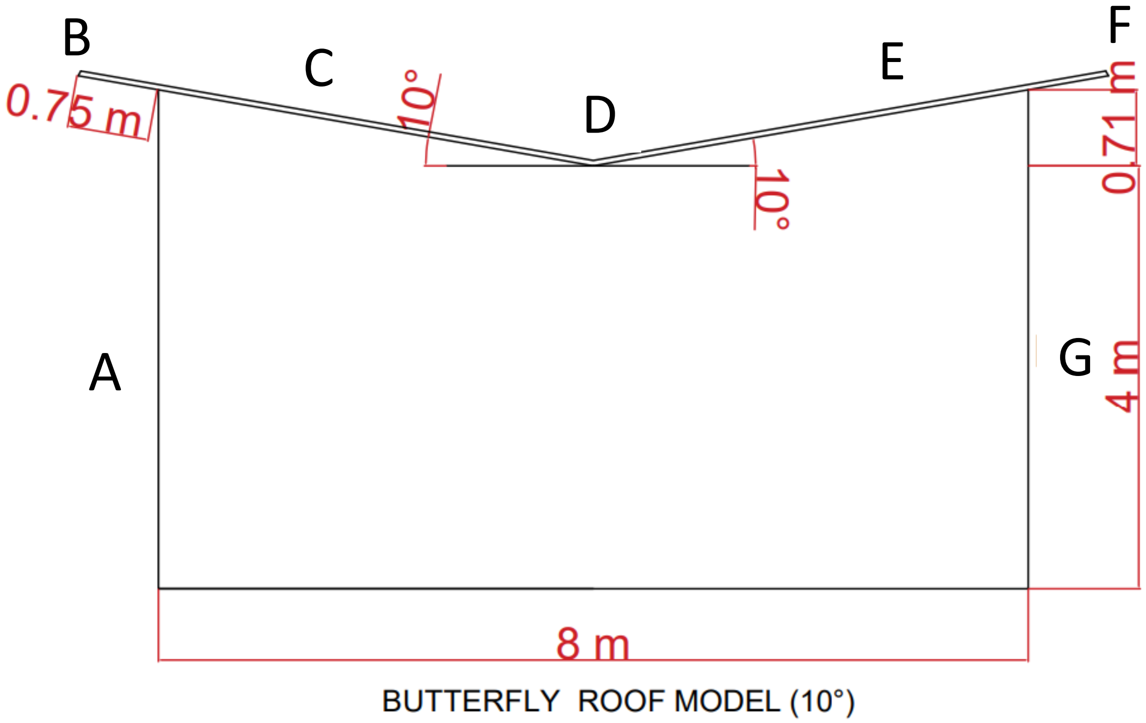


Figure 3.3: Marking of building model by parts

3.3.3 Domain and Boundary Condition Setup

This section briefly discussed the boundary condition used in this modelling analysis. The domain size used is 60m height and 60m width and the boundary condition and computational domain applied in this study is based on Franke, (2006). The line created must fully connected each other without left any gap. If the line was not connected each other or leaving the gap, the model could not be generated. The sketch and model developed in ANSYS software is displayed in Figure 3.4 bellow.

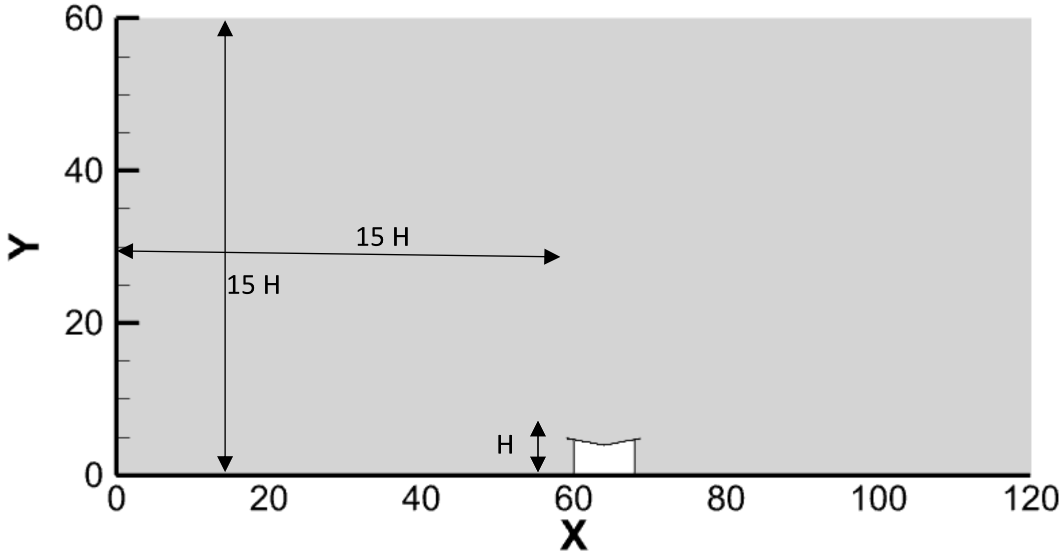


Figure 3.4 Boundary condition

The symmetry boundary condition must impose at the top section of the domain section. For the interior surface body, the user define function obtained from Abdullah *et al.* (2019) is interpreted. Then, the inlet and outlet condition are set at left and right section of the domain, respectively. Based on Deraman *et al.* (2019) the ground section of the domain is assigned as rough wall. The wall roughness height, K_s of the ground is 0.035m and roughness constant, C_s is 1.0 (Blocken *et al.*, 2007; Zhang, 2009).

The density, velocity and viscosity used in this analysis is 1.172kg/m3, 26.4m/s, and 1.7894 x10-5, respectively. For the surfaces, the domain operation is set as add material and the model operation (house) is set as add frozen. The size of the domain is fix for all type of model with the angle of the roof become the only variable parameter for this study. Then the model and domain need to be generated before the meshing work can be started.