WIND INDUCED NOISE IN MICROPHONE CAVITY OF TWO WAY RADIO

NUR SYUHADAH BINTI HUSEIN

SCHOOL OF AEROSPACE ENGINEERING

UNIVERSITI SAINS MALAYSIA

2019

WIND INDUCED NOISE IN MICROPHONE CAVITY

OF TWO WAY RADIO

by

NUR SYUHADAH BINTI HUSEIN

Thesis submitted in fulfilment of the requirements for the Bachelor Degree of

Engineering (Honours) (Aerospace Engineering)

June 2019

ENDORSEMENT

I, Nur Syuhadah binti Husein hereby declare that all corrections and comments made by the supervisor and examiner have been taken consideration and rectified accordingly.

____

(Signature of Student)

Date :

____

(Signature of Supervisor)

Name :

Date :

____

(Signature of Examiner)

Name :

Date :

WIND-INDUCED NOISE IN MICROPHONE CAVITY OF TWO-WAY RADIO

ABSTRACT

This research investigated the relationship of occurrence of wind noise during the operation of using two-way radio for communication purpose. It is done by visualizing the wind flow and the formation of vortex shedding inside cavities. Noise measurement due to wind flow of a two-way radio and its bigger scale model (specimen) was referred to a study that was conducted inside an open-loop wind tunnel. A Computational Fluid Dynamics (CFD) analysis was performed based on the noise measurement wind speed. The effect of wind flow direction, wind velocity of 2 and 4 m/s, and Reynolds Number of 8694 and 17879 were presented in wind noise measurement for a two-way radio. The wind velocity that was used for CFD simulation is calculated by using Reynold Number formula. CFD was used to simulate wind flow over a model to observe the formation of vortex shedding inside cavities with different cavity shape, positions and wind velocity. CFD was used to simulate the velocity magnitude, static pressure and vorticity contour. The result shows that the wind velocity has increased the wind noise for the left-to-right direction of wind flow. The CFD simulation shows a stronger vortex shedding has been formed at the opening edge of the cavity. The velocity contour of cavity at higher wind velocity and higher position is smaller than the velocity contour at nearer cavity position. The result from this research is very significant for the improvement of a two-way radio design in reducing noise during communication.

KEBISINGAN ANGIN TERARUH DALAM RONGGA MIKROFON RADIO DUA HALA

ABSTRAK

Kajian ini menyiasat tentang kejadian bunyi angin semasa operasi menggunakan radio dua hala bagi tujuan komunikasi. Kajian dijalankan dengan menggambarkan alirang angina dan pembentukan vortex di dalam rongga. Pengukuran kebisingan yang disebabkan oleh aliran angin daripada radio dua hala dan model berskala besar (spesimen) dirujuk dari kajian yang telah dijalankan dalam terowong angin gelung terbuka. Analisis perkomputeran dinamik bendalir dilakukan berdasarkan kelajuan aliran angin pengukuran kebisingan. Kesan daripada arah aliran angin, kelajuan angin 2 dan 4 m/s, dan nombor Reynolds masing-masing 8694 dan 17879 telah dibentangkan dalam pengukuran kebisingan angin radio dua hala. Kelajuan aliran angina yang digunakan bagi tujuan simulasi Perkomputeran Dinamik Bendalir dikira dengan menggunakan formula nombor Reynolds. Perkomputeran Dinamik Bendalir telah digunakan untuk mensimulasikan magnitude halaju, tekanan static dan kontur pusaran. Hasil kajian menunjukkan bahawa halaju angin telah menyebabkan kebisingan angin meningkat bagi arah aliran angin dari kiri ke kanan. Simulasi perkomputeran dinamik bendalir menunjukkan vorteks yang lebih besar telah terbentuk pada pinggir pembukaan rongga. Magnitude halaju pada rongga dengan aliran angina deras dan kedudukan yang jauh lebih rendah dari magnitude halaju pada kedudukan yang dekat. Pembentukan vorteks telah meningkat di kedudukan rongga yang lebih jauh dan pada halaju angin yang lebih tinggi. Hasil dari kajian ini sangat penting kepada penambah baikan reka bentuk radio dua hala dalam mengurangkan bunyi bising semasa komunikasi.

ACKNOWLEDGEMENTS

First and foremost, I have to thank my research supervisor, Dr Norilmi Amilia Ismail. Without her assistance and dedicated involvement this paper would not have been accomplished. I truly appreciate the opportunity to work under her supervision.

I would also like to show gratitude to research partner, Mrs Wan Masrurah Hairudin for guiding me every step of the process of completing this thesis. I have gain experience that I cannot get anywhere else as this topic is not actually related to my scope of study. Thank you for your excellent guidance, caring and patience in order for me to complete this research.

My special thanks to the technicians of School of Aerospace Engineering for helping me to complete the experiment. They are Mr. Ismail, Mr Shukri, Mrs Rahayu and other technicians who I missed to mention their names.

I would like to thank my friends, Nor Raweyah, Sharfa Eliya, Nur Juliana, Sim Sing Mei, Anis Natasha, and the rest of my coursemate. They helped me a lot in terms of support for this research until the completion. Thank you for being a part of my support system. Without them, I will not finish my work on time.

Most importantly, none of this could have happened without my family. To my parents, my brothers and sisters, thank you for always supporting me with your kind words and believe in me for this past 4 years of struggle to complete this degree. Thank you so much for your patience and love towards me. Thank you for encouraging me with your best wishes. To those who indirectly contributed in this project, your gentleness means a lot to me. Thank you very much.

DECLARATION

This thesis is the result of my own investigation, except where otherwise stated and has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any other degree.

____

(Signature of Student)

Date :

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

CFD	:	Computational Fluid Dynamics
PBL	:	Planetary Boundary Layer

LIST OF SYMBOLS

Re	:	Reynolds Number
ρ	:	Density [kg/m^3]
μ	:	Dynamic Viscosity [kg/ms^{-1}]
v	:	Velocity [ms^{-1}]
l	:	Length [m]
V_W	:	Velocity of two-way radio
V_{cfd}	:	Velocity of CFD simulation
L_W	:	Length of working fluid of two way radio
L_{cfd}	:	Length of working fluid of CFD simulation
Re_W	:	Reynolds Number of the two-way radio
Re_{cfd}	:	Reynolds Number of CFD simulation.

CHAPTER ONE

INTRODUCTION

1.1 Background

Wind induced noise in microphone's cavity of a two way radio is known to disturb many outdoor measurements. The word noise itself means unwanted sound. Noise is unwanted signals that randomly combines with the original signal, which is very much related to the generalization of the static noise. Static noise is heard when listening to a weak radio transmission with significant electrical noise. Noise is generated by the unsteadiness of the wave like sound wave and electrical signal wave as said by Rabani in his study (Hashim, 2015). Frequencies of noise depend on the source of sound. Mitigation usually takes the form of applying windshields and windscreens. The mesh in windscreens will essentially divides a large gust of wind into numerous smaller gust of wind thus reducing the wind noise. But when the signal level is low and wind speed near the ground is high, even very efficient windshields may not be sufficient (Kendrick, von Hünerbein, & Cox, 2016)

Wind noise can also be generated by wind flowing through gaps within elements with turbulence flow. The audible noise generated from these aero-acoustic phenomena can be disturbing when the wind noise has a higher air speed than the desired audio. Hart and Wilson,2016 stated that wind-induced noise, as sensed by a screened microphone in the atmosphere, produces a highly varying noise floor. At low frequencies, this noise source may dominate the ambient noise spectrum and present a challenge for signal detection.

The wind induced noise generally consists of two forms of pressure fluctuations: the turbulence provided by the local wind which depends on the atmospheric conditions and terrain properties (van den Berg, 2006). The sensed noise is affected by several factors in the atmospheric convective boundary layer: the mean flow at the height of the microphone, turbulent eddy interaction with the windscreen, and pressure fluctuations within the turbulent flow (Carl R. Hart, 2018).

In this research, six different types of cavity model are studied. Models are analysed under different effect such as wind speed and ratio per length in terms of the shape and position of the cavity. The pressure-fluctuation (Pa), velocity profile, and vorticity are visualize and examine to reduce the vortex shedding flow and noise level produced.

1.2 Problem Statement

A two-way radio is a radio that can transmit and receive a signal from other similar radio that are using the same frequency for mobile connection. This two-way radio is used in various condition such as windy condition that will affect the transmission of signal to the receiver. This problem is related to the noise created by the wind that flows around the two-way radio and causes wind-induced effect. Communication interference due to noise is difficult to be totally avoided in a two-way radio during communication from transmitter to receiver. These conditions serve the needs to study the relationship between the effect of wind speed and cavity design to the occurrence of noise by comparing the two dimensional simulation of the cavity models.

1.3 Objectives of Research

The objectives of this research are:

1.3.1 To design a two-dimensional cavity model with different geometries and position.

1.3.2 To investigate the effect of velocity, pressure and vorticity over the position of the microphone cavity of a two-way radio.

1.3.3 To analyse the effect of wind speed to the occurrence of noise in the cavity of a two-way radio.

1.4 Thesis Outline

Chapter one consist of the introduction, problem statement and objective of the research.

Chapter two present the literature review regarding the tittle of wind-induced noise, cavity, and noise reduction.

Chapter three explains the methodology of the project. Methodology consists of main steps such as designing two-dimensional cavity models, analysis of model using ANSYS CFD software and simulation of wind flow around the cavity.

Chapter four comprises the result and discussion of the simulation. Discussions on models with different geometry design and position will be analyse.

Chapter five concludes the research and suggest the potential impact that can be utilize for future advancement of two-way radio microphone.

CHAPTER TWO

LITERATURE REVIEW

This chapter reviews literatures related to wind-induced noise, wind-induced noise in microphone, types of cavity, two-way radio, vortex shedding and cavity.

2.1 Wind-induced Noise

Wind is a natural phenomenon that happens on a daily basis. It is caused by the spatial differences in atmospheric pressure. A study by Panofsky in 1985 (Panofsky, 1985), the first layer of wind from the ground is called the “friction layer” or the “planetary boundary layer” (PBL). Panofsky in his study also stated that the planetary boundary layer (PBL) is that part of the atmosphere that is affected by the characteristics of the ground by rapid vertical exchange processes, mostly by turbulence (Panofsky, 1985). Turbulence happens when in windy situation which depends on the terrain properties and the atmospheric pressure. There are two types of turbulence which are mechanical turbulence and convective turbulence. There are numerous sources of noise for example, noise from other source and wind flow that is the main cause of noise in this research. Wind that flows without any disturbance do not produce any noise. As stated before, noise is generated when wind flow interacts with disturbance and the phenomenon is called as wind induced noise. Wind induced noise is a major problem in designing a product especially for product which has high exposure to wind flow. One of the products is two-way radio or it is also known as the walkie-talkie.

2.2 Wind-induced noise in Microphones

A study by Morgan et al. showed that atmospheric turbulence is a significant contributor to outdoor wind-induced noise (Morgan & Raspet, 1992). Wind-induced noise inside a microphone studied by Zhang et al. to reduce the background noise in microphones. They have improved the existing design methods to understand the process of wind noise measurements. The study showed that the effects of wind-induced noise in a measuring microphone constantly increased with high wind speeds (Zhang & Kanapathipillai, 2008). In the study of Leclercq et al., a single-shielded microphone showed that the wind-induced noise increased with the flow velocity. Alamshah et al. also stated that the turbulence flow passing over a microphone surface causes wind-induced noise (Alamshah, Zander, & Lenchine, 2013). One of the method intended to reduce wind noise in microphone is by using windshield. The use of this windshield is expected to preserve the original signal and reduce unwanted the flow around the cavity.

2.3 Two-way Radio

Study on two-way radio has been conducted on the circulation of flow in the microphone cavity. The two-way radio has a cavity that receives flow, hits the diaphragm and the microphone, and then converts the flow to a signal. This two-way radio is available in mobile, stationary base and portable configurations. The cross section of the two-way radio microphone cavity is shown in Figure 2.2. In the case of two-way radio, the problem face during the operation is the generation of the environmental source that is the sound of the wind. The noise can be identified patently in situation of high rate of wind blow such as the hangar and the seaside. Wind noise will affect the quality of the transmitted signal.

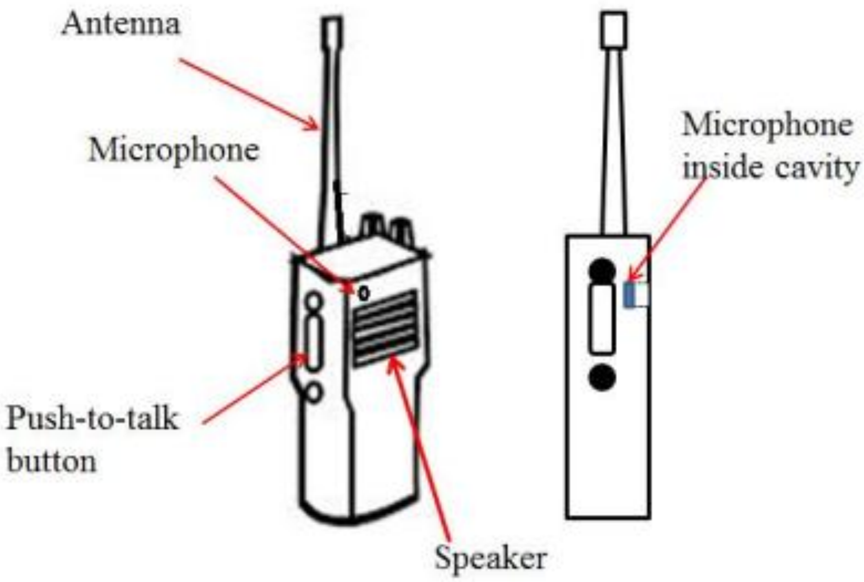

Figure 2.1: Basic configuration of a two-way radio (Fisol, 2016)

This type of communication have been available in the year 1907. In 1912, this two ways communication was installed in commercial and military ships to allow communication with the ground system. Two way radios are widely used for verbal communication in the work site, usually the open environment (Hashim, 2015). Typical obstacles, such as background and wind noise will be encountered by those who works in an open environment. Policeman, soldiers and fire fighters are few of the main user of the radio.

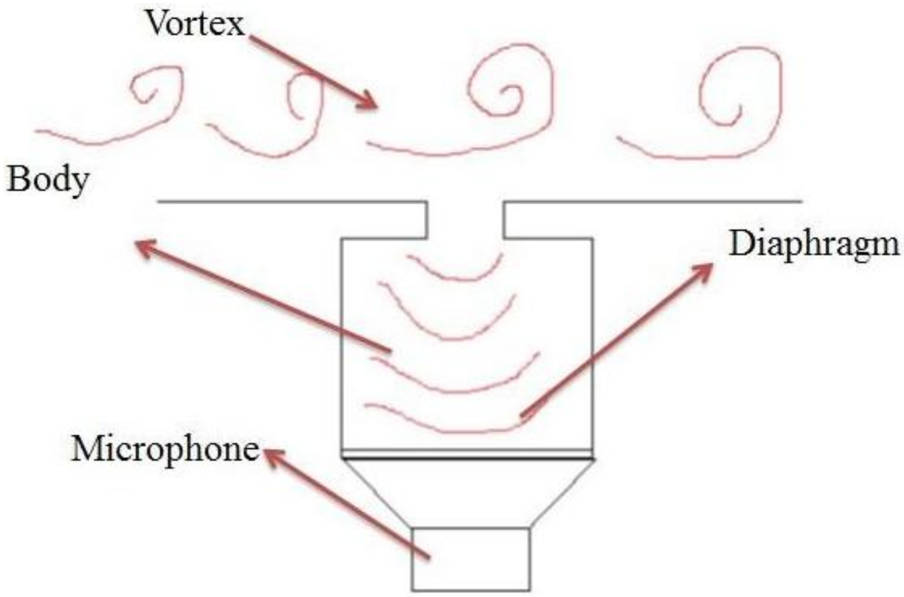

Figure 2.2: Cross section of a two-way radio microphone cavity

The noise measurement by Fisol et.al show the relation of flow speed to the level of noise. Higher flow speed of wind increases the noise level (Fisol, 2016). It can be compared with the situation of blowing air at the microphone. Investigation on how the vortices form inside the cavity and what is optimum design parameters for two-way radio to reduce the noise is conducted in this research.

2.4 Vortex Shedding

Vortex shedding is defined as an oscillating flow that develops when a fluid; water or air flows over a bluff body at a certain velocity that depends on the size and shape of the bluff. Vortex shedding is developed by the interaction between the wind flow and the surface of the ground-based object. In this research of two-way radio cavity, the vortices are developed at the leading edge and the trailing edge of the cavity body. The low pressure vortices on the downstream side of the object is created when the fluid flow past the object and moving to a lower pressure area of the object. Figure 2.2 visualize the vortex shedding that is generated at the cavity.

In 2002, a study was carried out by Kook, H. and Mongeau, L for wind induced noise over wind flow (Kook & Mongeau, 2002). The experiment focus on subsonic flows over Helmholtz resonators that causes pressure fluctuations inside the resonators. The frequency and relative amplitude of the cavity pressure fluctuations are predicted by the method for a range of flow velocities. Figure 2.3 shows the predicted vorticity formation under different frequency flow.

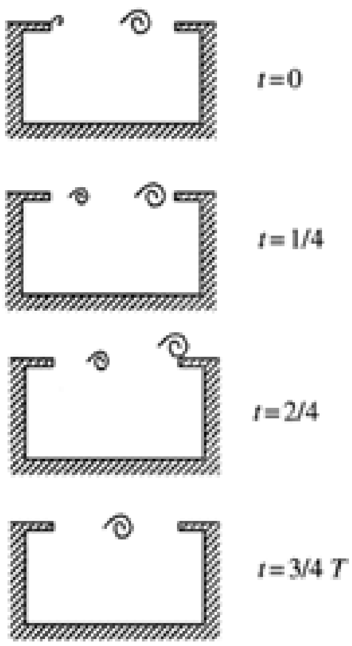

Figure2.3: A diagram of circulation strength fluctuation. (Kook & Mongeau, 2002)

The experimental design was setup using a rigid-walled cavity that is located at the middle of a low-speed wind tunnel as shown in Figure 2.4. The experimental data is reasonably to the model predictions as shown.

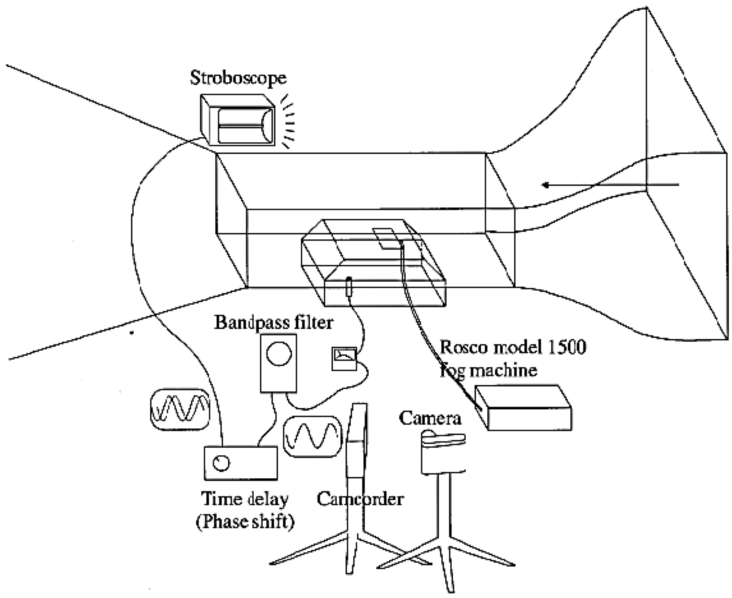

Figure 2.4: Experimental setup for flow visualization

2.5 Cavity

2.5.1 Types of Cavity

The occurrence and intensity of such oscillations are determined by the balance between the energy available from the free-stream flow and the energy dissipated through various loss mechanisms such as acoustic radiation, viscous losses, and convective mass exchange (Heller, Holmes, & Covert, 1971). A study on the rectangular shape cavity has been conducted in 1995 where experiment on unsteady, incompressible flow past a rectangular cavity is done. The characteristics of flow-induced pressure oscillations in open cavities depend strongly on parameters such as geometric, structural and aerodynamic. Research done by Rockwell et al. shows the effects of arrangements of cavities in groups and variations in cavity shape such as rectangular, circular, triangular, whistle-type, Helmholtz-type.

2.5.2 Cavity Flow Oscillation

Unstable flow over cavities is grouped into three types of flow which are fluid-dynamic, fluid-resonant, and fluid-elastic categories. A study shows that the fluid interactions with the wall of a structure give rise to aerodynamic damping which varies with the fluid velocity, which contributes to the flow oscillation in cavity. Fluid-dynamic oscillations are attributable to instability of the cavity shear layer and are enhanced through a feedback mechanism. Fluid-resonant oscillations are governed by resonance conditions associated with compressibility or free surface wave phenomena. Fluid-elastic oscillations are primarily controlled by the elastic displacements of a solid boundary (Rockwell & Naudascher, 1978). This research focus on the fluid-dynamic oscillation.

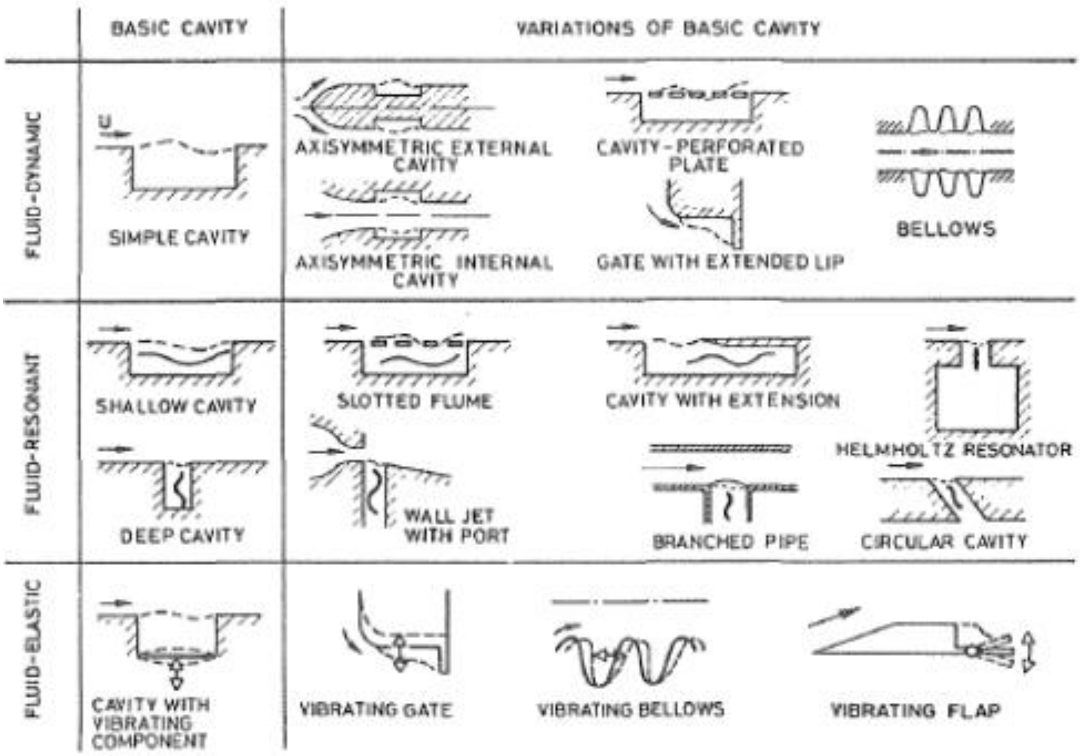

Figure 2.5: Representative fluid-dynamic, fluid-resonant, and fluid-elastic types of cavity oscillations (Rockwell & Naudascher, 1978)

Rockwell et al. also studies the experimental information and theoretical models for the frequencies and amplitudes of each of the categories of oscillations as mention above as well as a combination of them. The structural response to oscillatory energy may be so significant as to damage an aircraft structure. The study of these oscillations is to avoid, if possible, critical design configurations or, at least, to make reasonably accurate predictions of the magnitudes and spectral distributions.

2.5.3 Cavity Shape

Flow past an open cavity is known to give rise to self-sustained oscillations in a wide variety of configurations and these cavity oscillations are the origin of coherent and broadband sources of noise and, if the structure is sufficiently flexible, flow induced vibration as well (Lin & Rockwell, 2001). Studies on cavity flow with square and rectangular shape cavity have been done although the cavities might not be rectangular-shaped or square-shape in real life application. Erturk and Gokcol (Peyre et al., 2003) studied 2-D, steady and incompressible flow inside a triangular driven cavity numerically using a very fine grid mesh for high Reynolds numbers. Vortex formation and heat transfer in turbulent flow past a transverse cavity with a low aspect ratio and with inclined frontal and rear walls was studied experimentally by D’yachenko et al. (D’yachenko, Terekhov, & Yarygina, 2008). Flow past cavities have received great attention in the past decades in both experimental and numerical investigations due to its relevance to many practical engineering applications (Ozalp, Pinarbasi, & Sahin, 2010). A study by Ozalp et. al. (Ozalp et al., 2010) shows that apart from cavity shapes, Reynolds number changes has influence on the structure flow and turbulence quantities.

2.6 Summary

Wind induced noise clearly affected the transmitter's signal of the two way radio. The noise is generated when wind flow interacts with disturbance and the phenomenon is called as wind induced noise. Based on the previous study, there are few studies on the shape of the microphone cavity involving the rectangular and square shape. There are few ways that have been studied in order to reduce the wind noise such as using windshield or amplitude filter. The noise reduction can also occur by alternating the shape of the two-way radio microphone cavity.

CHAPTER THREE

METHODOLOGY

In this chapter, the design process of models with different geometry and position and simulation of two-way radio cavity model using ANSYS Software are presented and discussed.

3.1 Computational Fluid Dynamic (CFD)

3.1.1 Introduction

The design of the two-dimensional models; rectangular baseline (open cavity) model and slanted wall model was done by using ANSYS Software. The mesh of nodes and elements are generated on the drawn geometry by using mapped meshing, edge meshing and face meshing. Mapped meshing is a meshing process which is done by parting the sketch into a few sections and in this research the sketch was parted into seven sections. Both face meshing and edge meshing are generated for each section separately. Figure 3.21 shows the flow of CFD simulation that need to be run to obtain the result. In the Setup interface, FLUENT solver is selected and physical properties such as type of model, cell zone conditions, boundary conditions and reference properties are defined. The simulation will result in pressure contour, velocity contour and vorticity contour of the cavity model.

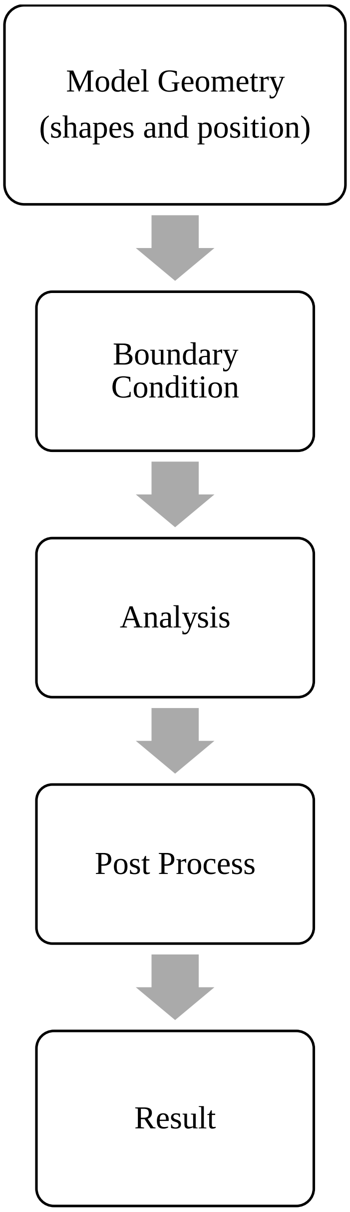

Figure 3.21: Flow of CFD simulation

3.1.2 Geometry

A two-dimensional geometry was selected instead of three-dimensional to ease the research to focus only on the formation of vortices inside the cavity. The rectangular shape open cavity is the actual model to represent the microphone cavity of the two-way radio. Whilst a few other different geometry cavity model such as curved opening cavity and slanted cavity model as well as the different position of cavity model are chosen as the hybrid model to use as the research model to determine which model that can be used in order to reduce the wind noise in a two-way radio microphone cavity. The geometry of the actual cavity model is shown in Figure 3.22 below.

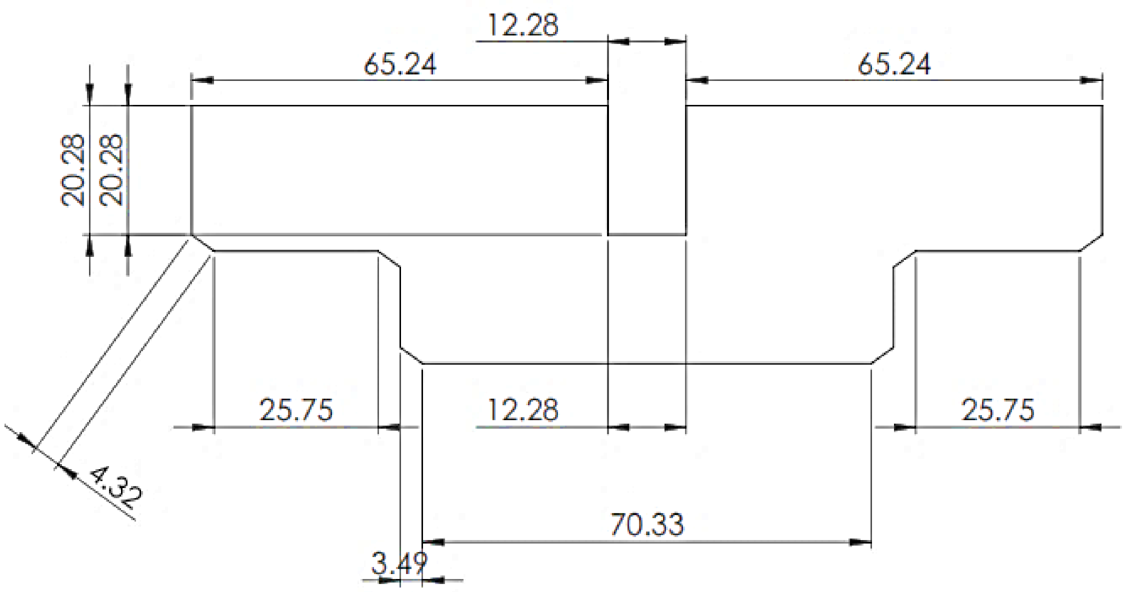

Figure 3.22: Geometry of Model 1 (Rectangle wall cavity)

Figure 3.22 is drawn according to the fabricated model of two-way radio cavity that is used as a part of this research. Geometry for Figure 3.22 is drawn based on Figure 3.24. Based on the fabricated model, the geometry of the model was altered to study the effect of wind velocity and vortex shedding to the shape and position of the cavity. Both figures below show the actual size of the cavity model. For simulation purpose, only the cavity part is redraw and analyse to focus only on the shape of the cavity wall.

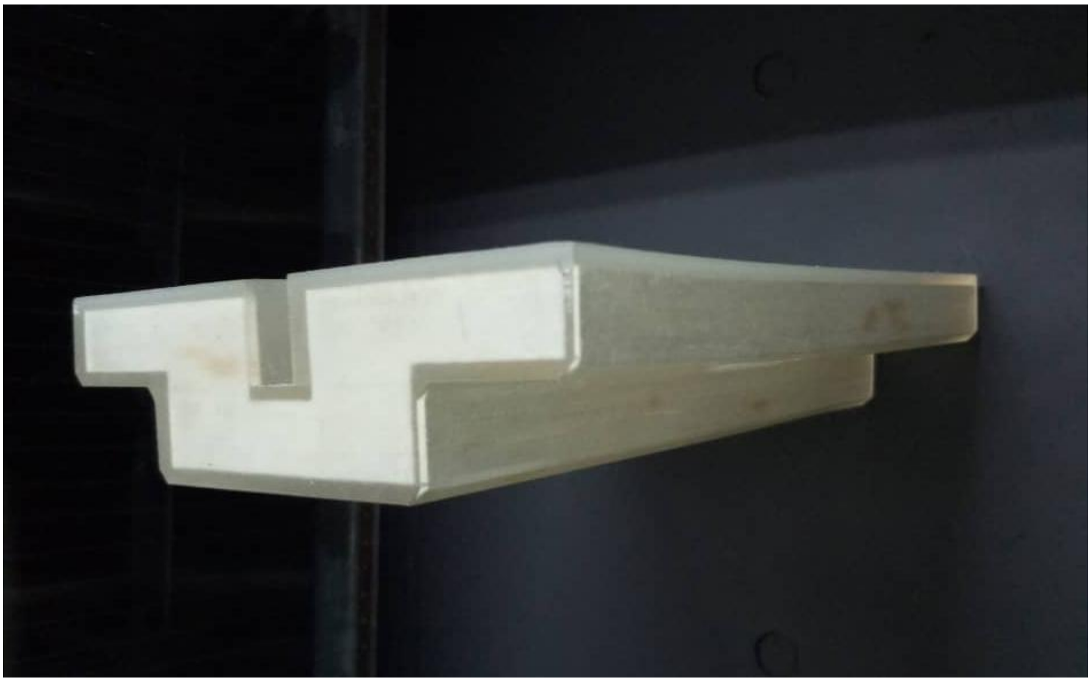

Figure 3.23: Isometric view of fabricated cavity model

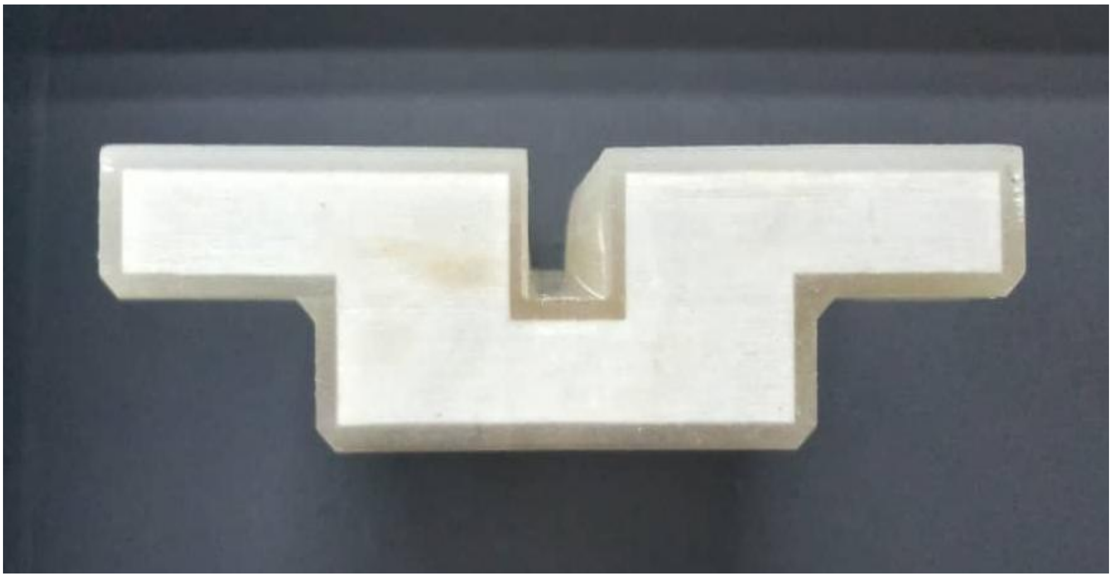

Figure 3.24: Front view of fabricated cavity model

The geometry are drawn using Design Modeler in ANSYS Fluent Software. The exact dimensions for rectangular cavity wall was measured from the fabricated model as shown in Figure 3.25.

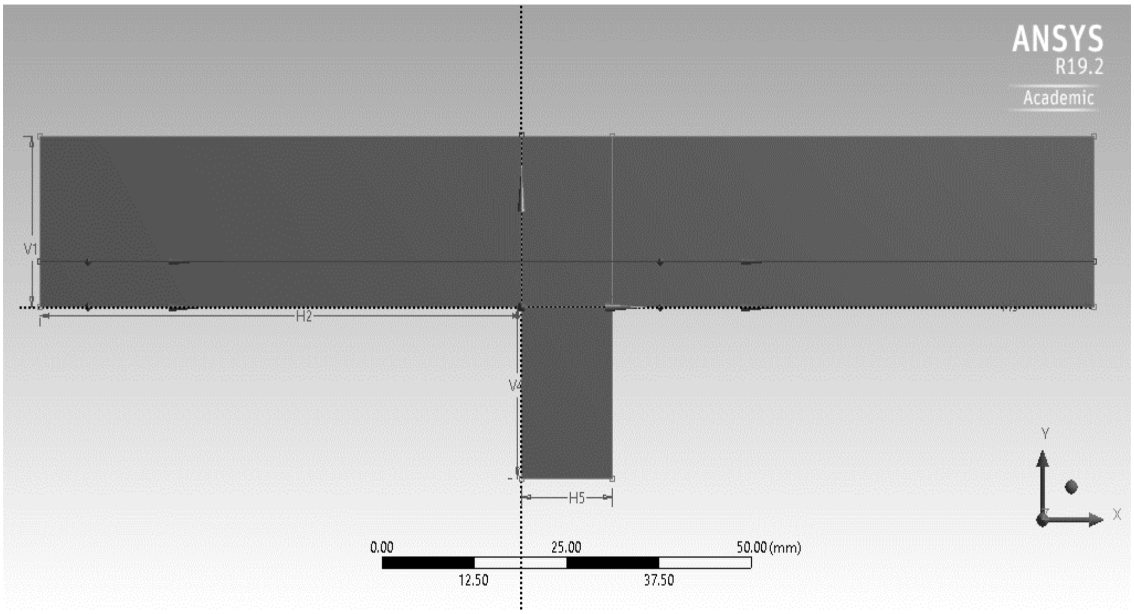

Figure 3.25: Model 1

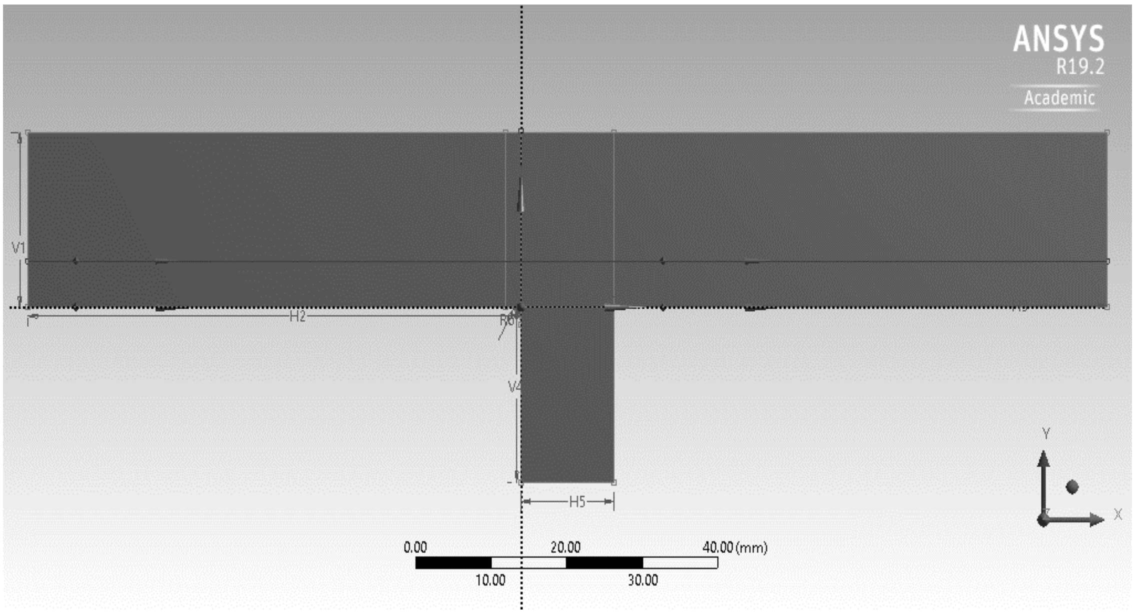

Figure 3.26: Model 2

Figure 3.26 shows the first shape modification based on Figure 3.25. Figure 3.26 consist of a smooth curve on the leading edge of the cavity which is the opening of the cavity. The radius size of the curve edge is 2cm.

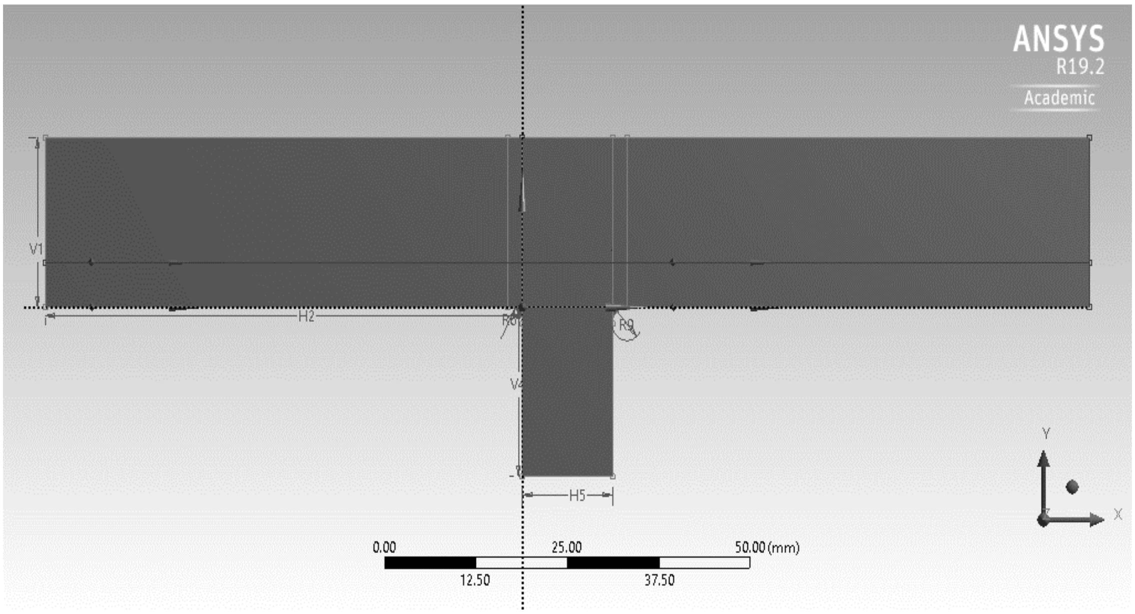

Figure 3.27: Model 3

Figure 3.27 shows model 3 which has smooth curve of 2 cm radius on both the leading edge and the trailing edge of the opening of the cavity.

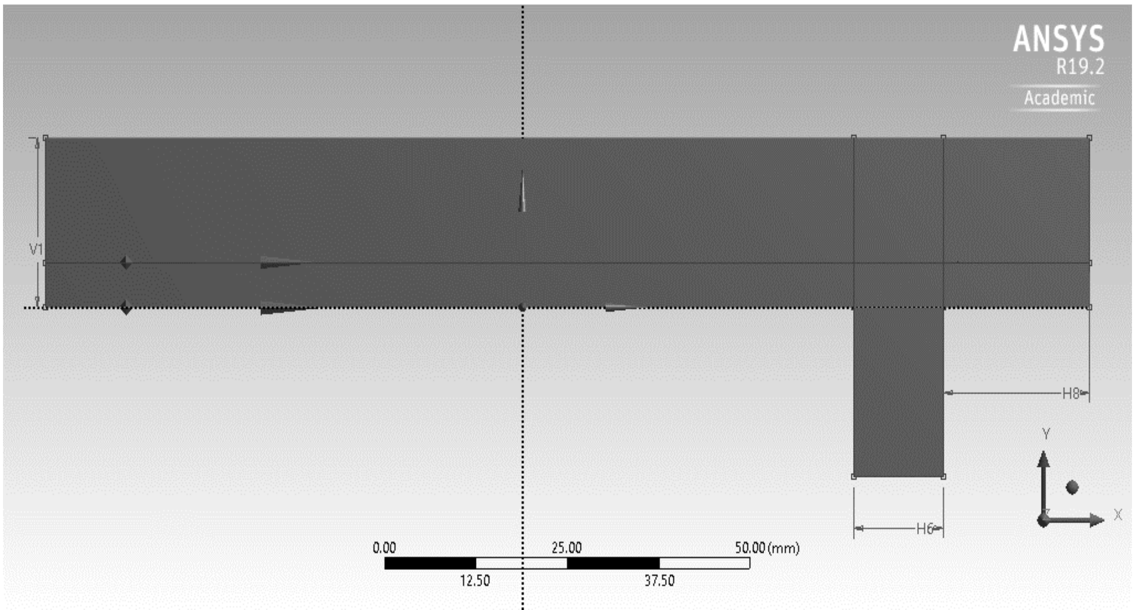

Figure 3.29: Model 4

Figure 3.28 above shows model 4 which has a different cavity position as compared to the cavity position in Figure 3.25. This position is studied to observe the occurrence of wind noise if the cavity is position further to the inlet, the wind source.

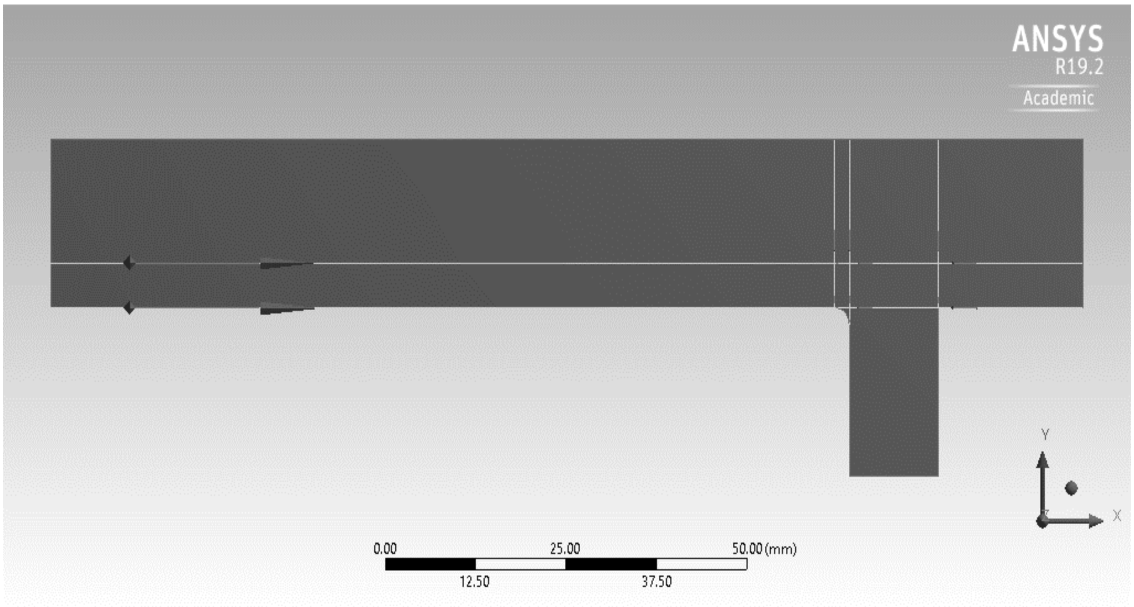

Figure 3.29: Model 5

Figure 3.29 shows Model 5 which have the same position as Model 4 but with the same shape modification as Model 2. A smooth curve with radius of 2 cm is added to the leading edge of the cavity in Model 5.

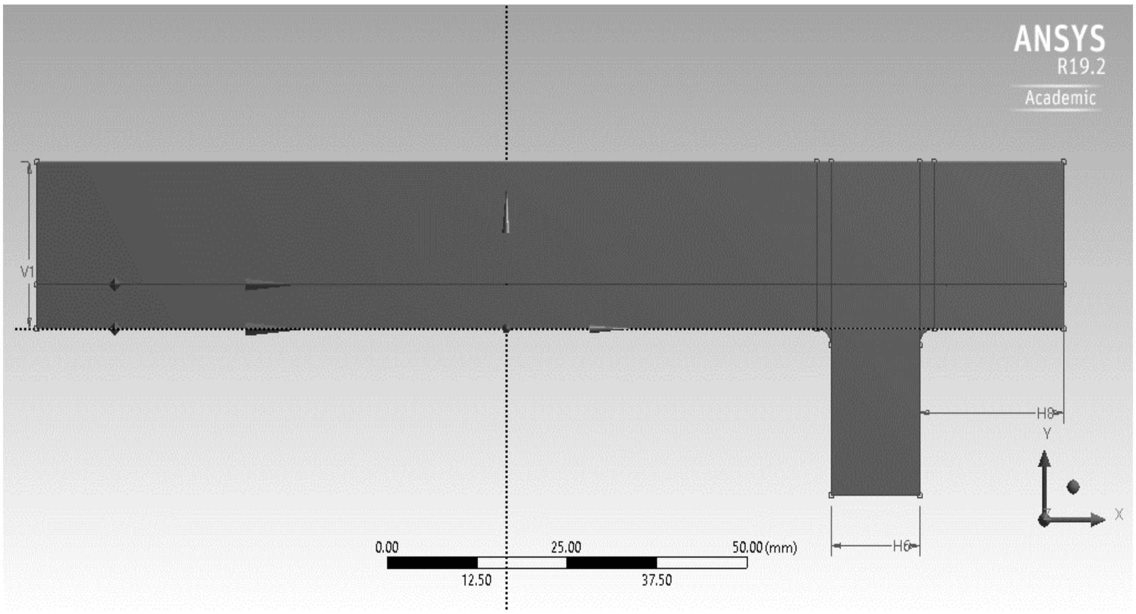

Figure 3.30: Model 6

Figure 3.30 above shows the same cavity position as model 4 but with the same shape modification as Model 3. Smooth curve with radius of 2 cm on both leading edge and trailing edge of the cavity in Model 6.

3.1.3 Mesh Independent Study

Mesh study was conducted to determine the optimum mesh to obtain desired solution. For this research, mapped meshing is used thus it requires a few additional steps compared to the normal edge meshing and face meshing. To create a finite element mesh for a geometry, mapped meshing is used. This mapped mesh can be created with normal rectangular regions or triangular regions. It could be used for an entire cavity model or a certain part of the model. In this research, only a part of the cavity model is used to focus on the cavity wall. Mapped meshing divides the model drawn into 7 to 9 parts. A new sketch must be drawn on the sketched model geometry to divide the model into desired parts. Next, the sketched lines have to be projected onto the surface of the sketched model to obtain the divided parts. Lines and surface from sketches are defined under the concept menu of Design Modeler. The surface body and line body are defined in the sketching section.

Next, in the meshing section of ANSYS Fluent software, each part is meshed accordingly. Both face meshing and edge meshing are used for each section. The wind flow have the highest effect on the cavity wall thus the number of divisions for edge meshing for the upper part of the model is smaller than the lower part which is the cavity wall. Number of divisions of edges near the cavity wall must be higher as it have high pressure as the wind touches the surface of the wall.

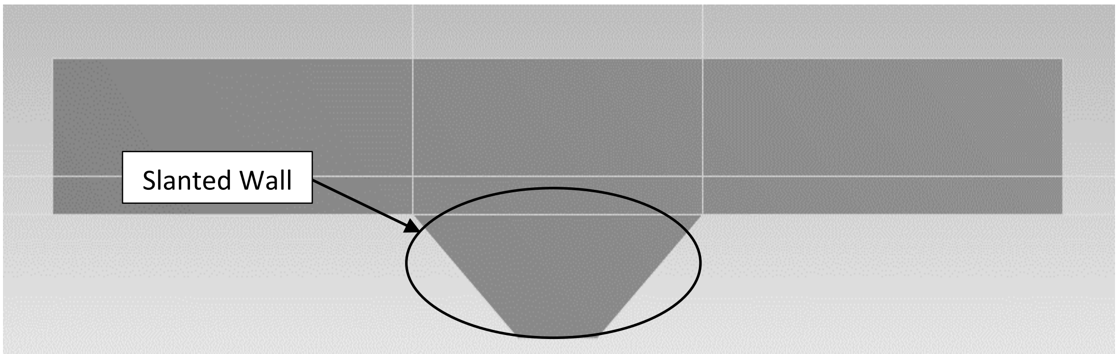

Figure 3.31: Sketched model for slanted wall

Edge meshing for each section is defined separately. Every rectangular section, the same side of the section is defined with the same number of division. For the cavity wall, trapezium shape section, the slanted wall both are defined with the same number of divisions. Same goes with the curve shape section, the wall are defined with the same number of division. Overall, there are two types of meshing which are face meshing and followed by edge meshing for each section. The selected type of mesh is Uniform Quadrilateral/Triangle meshing method for two-dimensional geometry.

Table 3.1: Table of Mesh Independent Study for Model 1

Mesh	Nodes	Elements	Time (s)
Mesh 1	8867	8539	16.87
Mesh 2	10946	10610	34.05
Mesh 3	52550	51852	56.54

Table 3.2: Table of Meshing Statistics of cavity models

Mesh / Model	Nodes	Elements
1	52550	51852
2	52876	52107
3	51381	50642
4	51092	50344
5	51373	50621
6	51650	50898

Table 3.2 shows the mesh analysis of each model. The target nodes and element for each model is 50,000 for both. The statistic depends on the shape of the model and the number of division for each section that was defines under edge meshing. The wind velocity for the above meshing analysis is 1.0 m/s.

3.1.4 Reynolds Number

Reynolds Numbers for this simulation are computed based on dynamic similarities between the two-way radio and the fabricated specimen. The Reynolds Number were calculated using the following equations:

$$Re_w = Re_{cfd} \tag{1}$$

$$\frac{\rho V_w L_w}{\mu} = \frac{\rho V_{cfd} L_{cfd}}{\mu} \tag{2}$$

$$V_{cfd} = \frac{V_w L_w}{L_{cfd}} \tag{3}$$

where V_{CFD} = velocity of CFD part, V_w = velocity of the two-way radio, L_{CFD} = length of working fluid for CFD and L_w = length of working fluid from the side view of the two-way radio.

Table 3.3: Reynolds Number Calculation Table

V_W (m/s)	L_W (m)	V_{cfd} (m/s)	L_{cfd} (m)	Re_W	Re_{cfd}
2	0.0635	0.876	0.145	8939	
4	0.0635	1.750	0.145	17879	
6	0.0635	2.628	0.145	26818	
8	0.0635	3.503	0.145	35757	
10	0.0635	4.380	0.145	44696	

Table 3.3 shows the Reynolds Number by taking the actual two-way radio wind velocities as reference. Wind velocities of the two-way radio are acquired from the