

# **A PARAMETRIC STUDY OF LOW WIND SPEED ENHANCEMENT SYSTEM**

By

CHEW WEIEE  
I08001549

FINAL PROJECT REPORT

Submitted to  
the Faculty of Science, Technology, Engineering & Mathematics  
in Partial Fulfillment of the Requirements  
for the Degree of

BACHELOR OF ENGINEERING (HONS)  
in  
MECHANICAL ENGINEERING

At

INTI INTERNATIONAL UNIVERSITY  
Persiaran Perdana BBN, Putra Nilai, 71800 Nilai  
Negeri Sembilan, Malaysia

APRIL 2015

© Copyright 2015  
by  
Chew Weiee, 2015  
All Rights Reserved

# APPROVAL

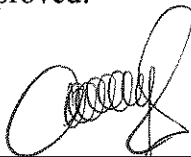
## A PARAMETRIC STUDY OF LOW WIND SPEED ENHANCEMENT SYSTEM

by

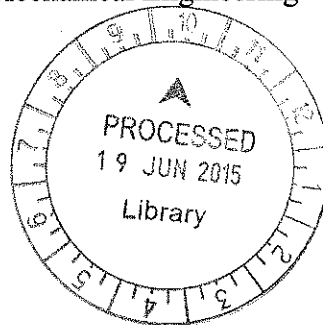
Chew Weiee

A project dissertation submitted to the  
Faculty of Science, Technology, Engineering & Mathematics  
INTI INTERNATIONAL UNIVERSITY  
in partial fulfilment of the requirement for the  
Bachelor of Engineering (Hons) in  
Mechanical Engineering

Approved:



DR. ABDULWEHAB ADEM IBRAHIM  
Project Supervisor



TJ  
INS  
CME  
2015


INTI INTERNATIONAL UNIVERSITY  
NILAI, NEGERI SEMBILAN

April 2015

## DECLARATION

I, the undersigned, hereby declare that this report is my own independent work except as specified in the references and acknowledgements. I have not committed plagiarism in the accomplishment of this work, nor have I falsified and/or invented the data in my work. I am aware of the University regulations on Plagiarism. I accept the academic penalties that may be imposed for any violation.

Signature

 .....

Name

..CHEW WELEE.....

Matrix No.

..10800154-9.....

Date

..21/05/2015.....

## ABSTRACT

A research study on a low wind speed enhancement system in Malaysia. The system is used to harness wind energy from all direction to generate electricity through a wind turbine. The objective of this research is to study the low wind speed enhancement system with different geometry parameter and optimize the performance of the design. There are 2 software that used in this research which is Autodesk Inventor and ANSYS Fluent. Besides that, there are total 4 system design and 4 research on the chapter 4 results and discussion. A series of parametric simulation has been done in this research to optimize the performance of the system. The wind speed at the nozzle has increase from the first design 2m/s to the final design 5.7m/s which is 96.1% in different.

## ACKNOWLEDGEMENTS

First of all, I would like to take this opportunity to thanks to all of the people that help me to complete my work successfully.

A special thanks to my supervisor Dr. Abdulwehab for helping, trusting, guiding and assisting me throughout this project. I really appreciate!

Also, I will like to extend my gratitude to all my friend who have been giving me a hand when I needed help.

## DEDICATION

*This thesis is dedicated to my beloved parent and family.*

# TABLE OF CONTENTS

DECLARATION.....	i
ABSTRACT .....	ii
ACKNOWLEDGEMENTS.....	iii
DEDICATION.....	iv
LIST OF FIGURES .....	viii
LIST OF TABLES.....	xiii
LIST OF ABBREVIATIONS.....	xv
NOMENCLATURE .....	xvi
CHAPTER 1 INTRODUCTION .....	1
1.1. Background.....	1
1.2. Problem Statement.....	2
1.3. Objectives of the Research .....	3
1.4. Scope of the Research.....	3
1.5. Report Organization.....	4
CHAPTER 2 LITERATURE REVIEW .....	5
2.1. Introduction.....	5
2.2. Design .....	5
2.3. Wind Turbine.....	8
2.4. Computational Fluid Dynamic (CFD) Simulation .....	11
2.5. Bernoulli's Principle.....	16
CHAPTER 3 METHODOLOGY .....	18
3.1. Introduction.....	18
3.2. Flow of Research .....	18
3.3. Geometry Modelling.....	20
3.3.1. Autodesk Inventor .....	20
3.3.2. Parametric Geometry .....	22
3.4. Simulation.....	24

3.4.1. ANSYS .....	24
3.4.2. Geometry Design Modeler .....	25
3.4.3. Meshing .....	26
3.4.4. Simulation Setup.....	27
3.4.5. Parameter Set .....	32
3.5. Mathematical Model.....	34
3.5.1. Bernoulli`s Equation.....	34
3.5.2. Response Surface.....	34
CHAPTER 4 RESULTS AND DISCUSSION.....	36
4.1. Introduction.....	36
4.2. Low Wind Speed Enhancement System Design 1 .....	36
4.2.1. Design 1 .....	36
4.2.2. Design 1A .....	40
4.2.3. Design 1B .....	43
4.2.4. Design 1C .....	46
4.3. Low Wind Speed Enhancement System Design 2 .....	48
4.4. Upper Nozzle Research .....	57
4.5. New Upper Nozzle Research.....	66
4.6. Nozzle Throat Fillet Research .....	78
4.7. Low Wind Speed Enhancement System Design 3 .....	86
4.8. Diffuser Research .....	95
4.8.1. Without Diffuser.....	95
4.8.2. With Diffuser.....	97
4.9. Low Wind Speed Enhancement System Final Design .....	99
CHAPTER 5 CONCLUSION AND FUTURE WORK.....	103
5.1. Conclusion.....	103
5.2. Recommendations.....	104



5.3. Future work.....	104
REFERENCES .....	105
APPENDIX A Design Drawings .....	107
APPENDIX B Gantt Chart .....	108

## LIST OF FIGURES

Figure 2.2.1 Typical turbine tower system (left) and Fielded INVELOX demo (right) under evaluation in Chaska, Minnesota (Allaei and Andreopoulos, 2014). .....	5
Figure 2.2.2 Schematic of the INVELOX wind delivery system with its key components (Allaei and Andreopoulos, 2014). .....	6
Figure 2.2.3 Schematic of a novel wind turbine (Ahmed, 2013). .....	7
Figure 2.3.1 Unsteady Kinematics for different Vamp (a) wind velocity, (b) RPM and (c) acceleration (Danao, Eboibi and Howell, 2013). .....	8
Figure 2.3.2 Unsteady Performance of the VAWT for the two Vamp cases: (a) $\lambda$ and (b) CP (Danao, Eboibi and Howell, 2013). .....	9
Figure 2.3.3 unsteady performance of the VAWT at different Vamp (Danao, Eboibi and Howell, 2013). .....	9
Figure 2.4.1 Velocity profile in cutaway slice on the plane of symmetry using ANSYS model (Allaei and Andreopoulos, 2014). .....	11
Figure 2.4.2 Velocity profile in cutaway slice on the plane of symmetry using COMSOL model (Allaei and Andreopoulos, 2014). .....	11
Figure 2.4.3 Schematic of the numerical model, CD = Constriction diameter, CH = Cowling height, COA = Cowling outlet angle, VG = Vane gap (Ahmed, 2013). .....	13
Figure 2.4.4 Flow field with cowling outlet at angle of 5° and 50° (Ahmed, 2013). ..	13
Figure 2.4.5 Increasing in wind speed with changes in cowling angle (Ahmed, 2013). .....	14
Figure 2.4.6 Flow fields for different turning vanes gaps (a) 5mm (b) 20mm (Ahmed, 2013). .....	15
Figure 2.4.7 Increasing in wind speed with changes in turning vane gap (Ahmed, 2013). .....	15
Figure 2.5.1 Flow in a Venturi (Health.learninginfo.org, 2015). .....	17
Figure 3.2.1 Flow of research in vertical process .....	19
Figure 3.3.1 Autodesk Inventor Project File .....	21
Figure 3.3.2 Autodesk Inventor New File Selection .....	22
Figure 3.3.3 Set Parametric Geometry in Autodesk Inventor. ....	23
Figure 3.4.1 Sample of an ANSYS simulation flow .....	24
Figure 3.4.2 Geometry Design Modeler .....	25
Figure 3.4.3 Meshing in Fluent.....	26

Figure 3.4.4 Simulation Setup in Fluent.....	27
Figure 3.4.5 Graph Residuals versus Iterations .....	31
Figure 3.4.6 Simulation Results Viewer in Fluent (CFD-Post).....	32
Figure 3.4.7 Response Surface Design Exploration in ANSYS.....	33
Figure 4.2.1 Low Wind Speed Enhancement System Design 1 3D View .....	37
Figure 4.2.2 Low Wind Speed Enhancement System Design 1 With Dimension .....	37
Figure 4.2.3 Simulation on Low Wind Speed Enhancement System Design 1 (Contour).....	39
Figure 4.2.4 Simulation on Low Wind Speed Enhancement System Design 1 (Streamline) .....	40
Figure 4.2.5 Simulation on Low Wind Speed Enhancement System Design 1A (Contour).....	42
Figure 4.2.6 Simulation on Low Wind Speed Enhancement System Design 1A (Streamline) .....	42
Figure 4.2.7 Low Wind Speed Enhancement System Design 1B in 3D View.....	44
Figure 4.2.8 Simulation on Low Wind Speed Enhancement System Design 1B (Contour).....	45
Figure 4.2.9 Simulation on Low Wind Speed Enhancement System Design 1B (Streamline) .....	45
Figure 4.2.10 Simulation on Low Wind Speed Enhancement System Design 1C (Contour).....	47
Figure 4.2.11 Simulation on Low Wind Speed Enhancement System Design 1C (Streamline) .....	47
Figure 4.3.1 Low Wind Speed Enhancement System Design 2 3D View .....	48
Figure 4.3.2 Low Wind Speed Enhancement System Design 2 with Dimension. ....	49
Figure 4.3.3 Simulation on Low Wind Speed Enhancement System Design 2 (Contour).....	50
Figure 4.3.4 Simulation on Low Wind Speed Enhancement System Design 2 (Streamline) .....	50
Figure 4.3.5 Graph of Inlet Height versus Wind Speed .....	51
Figure 4.3.6 Graph of Inlet Width versus Wind Speed .....	52
Figure 4.3.7 Graph of Middle Width versus Wind Speed .....	53
Figure 4.3.8 Upper Nozzle Height versus Wind Speed.....	54
Figure 4.3.9 Graph of Lower Nozzle Height versus Wind Speed.....	55

Figure 4.3.10 Graph of Lower Nozzle Angle versus Wind Speed .....	56
Figure 4.4.1 Upper Nozzle Angle 11 degree .....	58
Figure 4.4.2 Simulation on Upper Nozzle Angle 13 degree (Contour).....	58
Figure 4.4.3 Simulation on Upper Nozzle Angle 13 degree (Streamline).....	59
Figure 4.4.4 Simulation on Upper Nozzle Angle 15 degree (Contour).....	60
Figure 4.4.5 Simulation on Upper Nozzle Angle 15 degree (Streamline).....	60
Figure 4.4.6 Simulation on Upper Nozzle Angle 17 degree (Contour).....	61
Figure 4.4.7 Simulation on Upper Nozzle Angle 17 degree (Streamline).....	62
Figure 4.4.8 Simulation on Upper Nozzle Angle 19 degree (Contour).....	63
Figure 4.4.9 Simulation on Upper Nozzle Angle 19 degree (Streamline).....	63
Figure 4.4.10 Simulation on Upper Nozzle Angle 21 degree (Contour).....	64
Figure 4.4.11 Simulation on Upper Nozzle Angle 21 degree (Streamline).....	65
Figure 4.4.12 Graph of Nozzle Angle versus Wind Speed.....	66
Figure 4.5.1 Simulation on Upper Nozzle Angle 11 degree with Fillet Radius 300 mm (Contour).....	68
Figure 4.5.2 Simulation on Upper Nozzle Angle 11 degree with Fillet Radius 300 mm (Streamline) .....	68
Figure 4.5.3 Simulation on Upper Nozzle Angle 13 degree with Fillet Radius 300 mm (Contour).....	69
Figure 4.5.4 Simulation on Upper Nozzle Angle 13 degree with Fillet Radius 300 mm (Streamline) .....	70
Figure 4.5.5 Simulation on Upper Nozzle Angle 15 degree with Fillet Radius 300 mm (Contour).....	71
Figure 4.5.6 Simulation on Upper Nozzle Angle 15 degree with Fillet Radius 300 mm (Streamline) .....	71
Figure 4.5.7 Simulation on Upper Nozzle Angle 17 degree with Fillet Radius 300 mm (Contour).....	72
Figure 4.5.8 Simulation on Upper Nozzle Angle 17 degree with Fillet Radius 300 mm (Streamline) .....	73
Figure 4.5.9 Simulation on Upper Nozzle Angle 19 degree with Fillet Radius 300 mm (Contour).....	74
Figure 4.5.10 Simulation on Upper Nozzle Angle 19 degree with Fillet Radius 300 mm (Streamline) .....	74

Figure 4.5.11 Simulation on Upper Nozzle Angle 21 degree with Fillet Radius 300 mm (Contour) .....	75
Figure 4.5.12 Simulation on Upper Nozzle Angle 21 degree with Fillet Radius 300 mm (Streamline):.....	76
Figure 4.5.13 Graph of Nozzle Angle versus Wind Speed.....	77
Figure 4.6.1 Upper Nozzle Angle 11 degree .....	78
Figure 4.6.2 Simulation on Nozzle Angle 15 degree with Fillet Radius 300 mm (Contour).....	79
Figure 4.6.3 Simulation on Nozzle Angle 15 degree with Fillet Radius 300 mm (Streamline) .....	80
Figure 4.6.4 Simulation on Nozzle Angle 15 degree with Fillet Radius 500 mm (Contour).....	81
Figure 4.6.5 Simulation on Nozzle Angle 15 degree with Fillet Radius 500 mm (Streamline) .....	81
Figure 4.6.6 Simulation on Nozzle Angle 15 degree with Fillet Radius 700 mm (Contour).....	82
Figure 4.6.7 Simulation on Nozzle Angle 15 degree with Fillet Radius 700 mm (Streamline) .....	83
Figure 4.6.8 Simulation on Nozzle Angle 15 degree with Fillet Radius 900 mm (Contour).....	84
Figure 4.6.9 Simulation on Nozzle Angle 15 degree with Fillet Radius 900 mm (Streamline) .....	84
Figure 4.6.10 Graph of Fillet Radius versus Wind Speed. ....	85
Figure 4.7.1 Low Wind Speed Enhancement System Design 3 2D View with Dimension.....	87
Figure 4.7.2 Simulation on Low Wind Speed Enhancement System Design 3 (Contour).....	88
Figure 4.7.3 Simulation on Low Wind Speed Enhancement System Design 3 (Streamline) .....	88
Figure 4.7.4 Graph of Inlet Height versus Wind Speed .....	89
Figure 4.7.5 Graph of Inlet Width versus Wind Speed .....	90
Figure 4.7.6 Graph of Middle Width versus Wind Speed. ....	91
Figure 4.7.7 Graph of Middle Height versus Wind Speed. ....	92
Figure 4.7.8 Graph of Nozzle Throat Fillet versus Wind Speed. ....	93

Figure 4.7.9 Graph Upper Nozzle Angle versus Wind Speed.....	94
Figure 4.8.1 Low Wind Speed Enhancement System Design 3 without Diffuser .....	95
Figure 4.8.2 Simulation on Low Wind Speed Enhancement System Design 3 without Diffuser (Contour) .....	96
Figure 4.8.3 Simulation on Low Wind Speed Enhancement System Design 3 without Diffuser (Streamline).....	96
Figure 4.8.4 Low Wind Speed Enhancement System Design 3 without Diffuser .....	97
Figure 4.8.5 Simulation on Low Wind Speed Enhancement System Design 3 with Diffuser (Contour) .....	98
Figure 4.8.6 Simulation on Low Wind Speed Enhancement System Design 3 with Diffuser (Streamline).....	98
Figure 4.9.1 Low Wind Speed Enhancement System Final Design 3D View .....	99
Figure 4.9.2 Low Wind Speed Enhancement System Final Design 2D View with Dimension.....	100
Figure 4.9.3 Simulation on Low Wind Speed Enhancement System Final Design (Contour).....	101
Figure 4.9.4 Simulation on Low Wind Speed Enhancement System Final Design (Streamline) .....	101
Figure 4.9.5 Simulation on Low Wind Speed Enhancement System Final Design (Vector and Streamline).....	102

## LIST OF TABLES

Table 3.3.1 Variable Parameter Name and Equation .....	23
Table 3.4.1 Mesh Sizing .....	27
Table 3.4.2 Models settings for this research .....	28
Table 3.4.3 Material Selection.....	28
Table 3.4.4 Boundary Condition .....	29
Table 3.4.5 Solution Methods.....	29
Table 3.4.6 Solution Controls.....	29
Table 3.4.7 Residual Monitors.....	30
Table 4.2.1 Input Parameter for Design 1.....	38
Table 4.2.2 Geometry Dimension for Design 1.....	38
Table 4.2.3 Simulation Input for Design 1A .....	40
Table 4.2.4 Geometry Dimension for Design 1A.....	41
Table 4.2.5 Simulation Input for Design 1B.....	43
Table 4.2.6 Geometry Dimension for Design 1B.....	43
Table 4.2.7 Simulation Input for Design 1C.....	46
Table 4.2.8 Geometry Dimension for Design 1C.....	46
Table 4.3.1 Parametric Simulation on Inlet Height Results .....	51
Table 4.3.2 Parametric Simulation on Inlet Width Results .....	52
Table 4.3.3 Parametric Simulation on Middle Width Results .....	53
Table 4.3.4 Parametric Simulation on Upper Nozzle Height Results .....	54
Table 4.3.5 Parametric Simulation on Lower Nozzle Height Results.....	55
Table 4.3.6 Parametric Simulation on Lower Nozzle Angle Results.....	56
Table 4.4.1 Input Parameter.....	57
Table 4.4.2 Upper Nozzle Research Results .....	66
Table 4.5.1 Input Parameter.....	66
Table 4.5.2 New Upper Nozzle Research Results.....	77
Table 4.6.1 Input Parameter.....	78
Table 4.6.2 Parametric Simulation Results on Nozzle Throat.....	85
Table 4.7.1 Parametric Simulation Results on Inlet Height. ....	89
Table 4.7.2 Parametric Simulation Results on Inlet Width. ....	90
Table 4.7.3 Parametric Simulation Results on Middle Width.....	90
Table 4.7.4 Parametric Simulation Results on Middle Height. ....	91

Table 4.7.5 Parametric Simulation Results on Nozzle Throat Fillet. ....	92
Table 4.7.6 Parametric Simulation Results on Upper Nozzle Angle. ....	94
Table 4.8.1 Wind Speed with and without diffuser .....	97



## LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamics
VAWT	Vertical Axis Wind Turbine
CP	Power Coefficient
RPM	Rotational Speed (Round Per Minute)
CD	Constriction Diameter
CH	Cowling Height
COA	Cowling Outlet Angle
VG	Vane Gap
DOE	Design of Experiment

## NOMENCLATURE

<i>Symbol</i>	<i>Definition</i>
$C_p$	Power coefficient
$P_{\text{mech}}$	Mechanical power
$\rho$	Density
$u_\infty$	Free wind Speed
$\lambda$	Tip speed ratio
$R$	Rotor Radius
$\Omega$	Rotor angular speed
$V_\infty$	Free stream wind speed

# CHAPTER 1

## INTRODUCTION

### 1.1. Background

Nowadays, the demand of electrical energy is getting higher. There are many type of energy source in this world to produce electricity such as fossil fuel, nuclear energy, wind energy, geothermal energy and others more. Hence, fossil fuel is one of the main energy sources of the global economy since 18<sup>th</sup> century (Oh, Pang and Chua, 2010). Besides that, fossil fuel also using in our common transport such as car, bus and airplane. However, it is a non-renewable energy source and which is also one of the major cause of environmental pollution problem.

Wind energy is one of the renewable energy and people have been using it more than 3000 years. In the past few centuries, wind energy was used in many applications such as moving boats using sail, power machinery in farms and others more. In early 1900s, wind energy making a big leap toward electric power generation industry. Besides that, due to the awareness of global warming and pollution, wind turbine technology has caught global attention and started to spread internationally in the past 3 decade. However, the wind energy technology is far beyond from reaching its full potential. Research and development on wind energy are accelerating during this past few years (Sopian, Othman and Wirsat, 1995).

In Malaysia, there are two main weather seasons in a year which is southwest monsoon (May to September) and northeast monsoon (November to March). However, the wind speed and direction are not constant in both monsoon. Southwest monsoon have lower wind speed compare to northeast monsoon which is 7m/s and 15m/s respectively. Thus, during April and September, the wind speed may exceeded 10m/s on Sabah and Sarawak due to typhoons striking neighboring countries such a Philippines (Sopian, Othman and Wirsat, 1995).

On the others hand, the mean wind speed at Malaysia is low as 1.8m/s. However, some of the city in this country see strong winds in certain month of a year such as Mersing,

Kuala Terengganu and Labuan. The wind speed of this area show significantly higher wind speed during the first and fourth quarter year.

The minimum average wind speed required in typical wind turbine is between 3 m/s – 5 m/s (Albani, Ibrahim and Yong, 2014). However most of the city in Malaysia does not meet the minimum average wind speed required. Besides that, the wind direction is not consistent. Therefore, wind energy cannot be harness effectively.

Low wind speed enhancement system is a wind collection system where it collects wind from all different direction and enhance the wind speed through a nozzle. The aim of this system is to increase the efficiency on wind energy electricity generation.

In this research, a low wind speed enhancement system will be design and simulate by using computer software which is Autodesk Inventor and ANSYS Fluent. Both of the software will be used to study air flow and optimize the performance of the system.

## **1.2. Problem Statement**

Global energy usage has increase dramatically in the past few decades due to modern civilization and increasing in human population. While non-renewable energy source such as fossil fuel is running out soon by time and causing air pollution to our environment. In order to reduce pollution and satisfy the high demand of energy in Malaysia, renewable energy is a new direction for clean and sustainable energy in our future.

Wind energy is one of the renewable energy which can be harness in Malaysia. However, there is problem to harness wind energy in Malaysia is due to the low wind speed and inconsistence-wind direction. The aim of this project is to design a low wind speed enhancement system which able to capture wind which coming from different direction. Simulation software will be used in this project to simulate the flow and wind speed with different geometry parameter.

### **1.3. Objectives of the Research**

This research will be focusing on optimize the performance of a low wind speed enhancement system with different geometry parameter of the design. The design will be simulated by using Computational Fluid Dynamic (CFD). The overall objectives of this research will be shown as below:

- To design a low wind speed enhancement system.
- To study the low wind speed enhancement system with different geometry parameter using CFD simulation software.
- To optimize the performance of the design.

### **1.4. Scope of the Research**

In this research, the information on the characteristic of wind in Malaysia is important. In most of the region or country may have different weather condition such as four seasons at Australia, cold weather at United Kingdom and hot weather at Saudi Arabia. The air humidity in Malaysia is high and it will cause metal to rust easily. Therefore, choosing the correct material for the low wind enhancement system are very critical in this research in order to make it more reliable and prevent tragedy happen such as the Titanic incident.

There are some countries which using the same funnel-based wind turbine system to harness wind energy for electricity generation such as Japan and United State. However, the wind speed and direction are not same with Malaysia. Therefore, the pattern and structural design has to be change according to the weather condition in Malaysia. The design of the system has been done by one of the senior who is Hiew C.K. Therefore, the system will be modify for improvement in this research.

In addition, the first priority in this research is to achieve the optimum performance for the low wind speed enhancement system. The dimension of the design will be varying in order get the optimum performance according to the wind characteristic in Malaysia. Design of experiment will used to simulate the low wind system enhancement system by using Computational Fluid Dynamic (CFD).

Besides that, literature review is part of this research. It is based on the related journal topic and research paper such as wind turbine system design, CFD simulation and surface respond. The valuable information and data, will be able to make this research more successful.

## **1.5. Report Organization**

Chapter 1 is the introduction part, it introduce the general information for the overall research. The background of wind energy and wind turbine will be written. In addition, the problem statement will be included in this chapter. Besides that, the objectives of the research are stated clearly.

Chapter 2 is about literature review. The past research, system, prototype, theory and Pros and cons of the wind turbine system will be discussed in this chapter.

Chapter 3 is about methodology, the way and method of doing this research will be discussed. In this chapter, software and mathematical model that used in this research such as Autodesk Inventor, ANSYS Fluent will be discuss and summarize.

Chapter 4 is the results and discussion. The results are based on the simulation software that using in this research which is ANSYS Fluent. Different geometry parameter has been simulated in this system using Computational Fluid Dynamic (CFD) by ANSYS Fluent. The results will used to compare with the previous design in order to improve the performance.

Chapter 5 is the conclusion. It concluded the overall research with everything that has been done during the past 8 months. The objective of this research will be discussed based on the results. In addition, some advice and future recommendation will be stated.

## CHAPTER 2

### LITERATURE REVIEW

#### 2.1. Introduction

In this chapter, the information and data which related to the topic of this research will be review. The study materials are taken from few different sources which are journals, research paper and internet. The purpose of doing literature review is to know what others researcher is doing with the similar or related topic. The information are very useful for example information that use to compare and support the result in this research. Besides that, some valuable ideal and method which taken from the literature review will make this research become more successful.

#### 2.2. Design

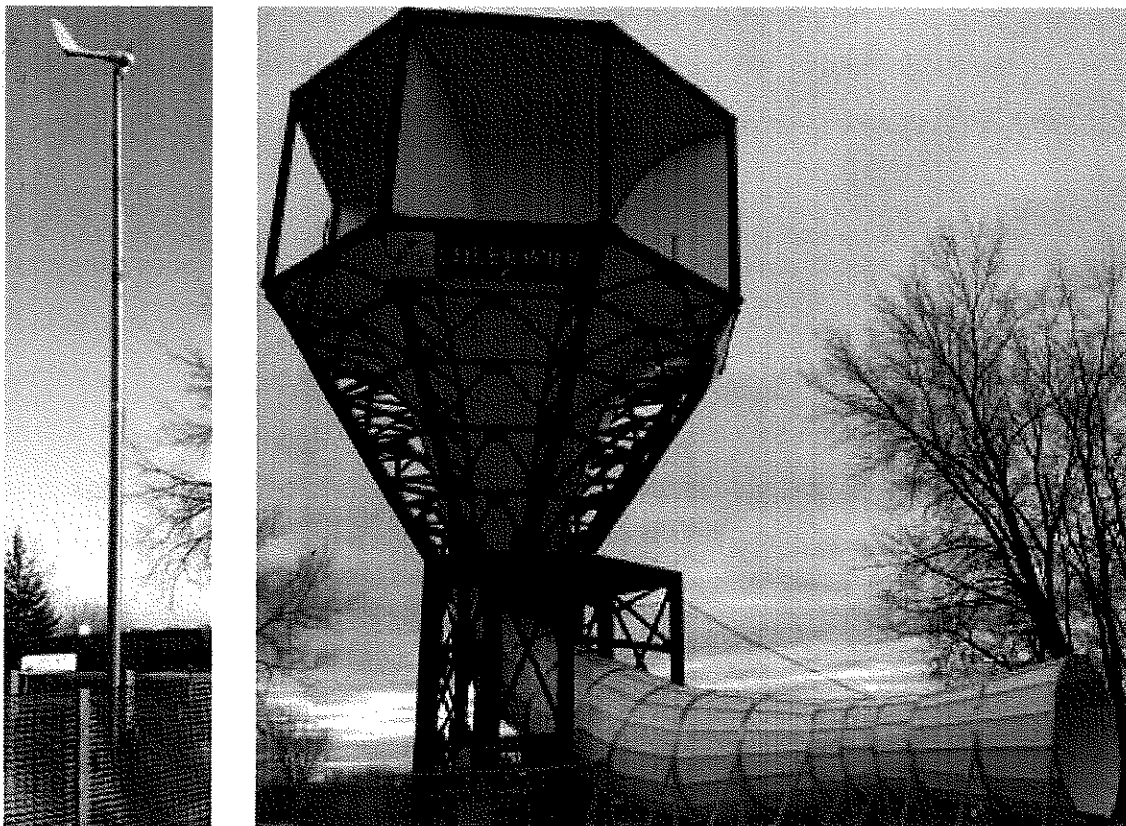


Figure 2.2.1 Typical turbine tower system (left) and Fielded INVELOX demo (right) under evaluation in Chaska, Minnesota (Allaei and Andreopoulos, 2014).

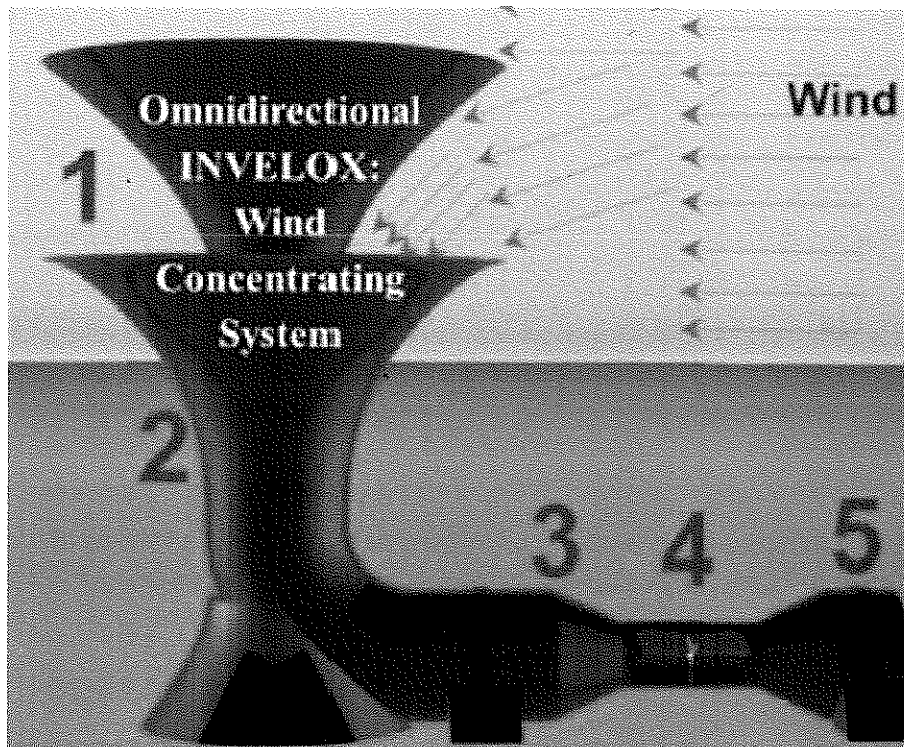


Figure 2.2.2 Schematic of the INVELOX wind delivery system with its key components (Allaei and Andreopoulos, 2014).

(1) intake, (2) channeling wind, (3) wind concentrator, (4) Venturi plus wind power conversion system, (5) diffuser returning wind to nature.

It is a new concept design of wind power harnessing system which significantly improved compare to traditional wind mill with the same wind conditions. Besides that, it produces significantly higher output and reduced structural cost. Typical wind mill is large and builds at the top of the tower. However, the cost of this tower are expansive, inefficient and hazardous to people and wildlife. The second advantage of INVELOX is that it captures wind in different direction instead of single direction with the active yaw control. Besides that, it accelerates the wind flow through a venturi section which is subsequently expanded and released at the middle of the diffuser outlet. Other than that, INVELOX solved all major problems that so far encounter by the wind industry such as wind turbine reliability, intermittency issues, and adverse environmental and radar impact (Allaei and Andreopoulos, 2014).



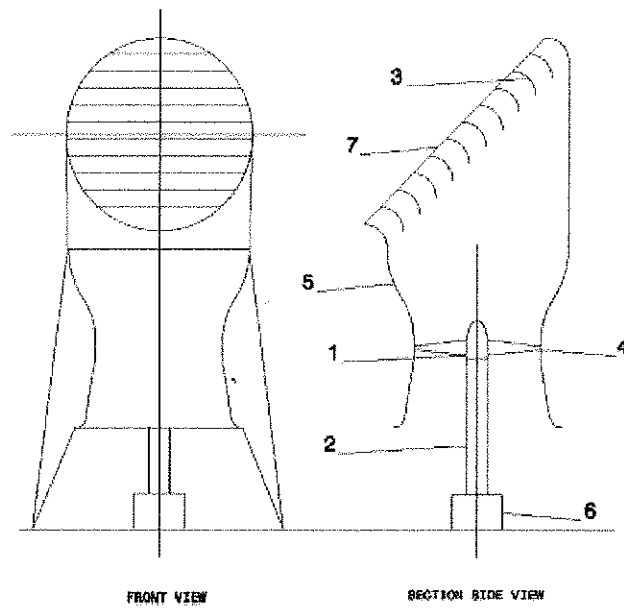


Figure 2.2.3 Schematic of a novel wind turbine (Ahmed, 2013).

The novel wind turbine shown in Figure 2.3 consists of several parts. The wind turbine (1) is mounted on the turbine support structure (2) with direction vanes (3) to divert the wind towards the turbine blades (4). The arrangement of this wind turbine makes it operate like a horizontal axis wind turbine. The turbine blades can be of fixed or either variable pitch. According to N.A. Ahmed (2013), an additional counter rotating blades added on the second row can be used to improve efficiency. The surrounding structure or the cowling (5) diverts the air flow to provide improved conversion of the wind energy through the turbine blade into either electrical or mechanical energy. Mechanical or electrical component can be mounted at the base of the structure (6). This would allow ease of maintenance and reduce weight in support structure for the turbine. The speed of the turbine can be controlled by rotating the surrounding structure (5), varying the turbine blade pitch (4) or varying the direction of the vanes (3). Flow modifying structure that attached to (5) can be varied to control speed of the turbine. The shape of the surrounding structure (5) can minimize the impact of sudden gusts on the turbine blades and facilitate the increase in air velocity on the turbine blade. This would result in improved low wind speed start up and overall efficiency. The protective screen (7) can also be used to protect the turbine from bird strike (Ahmed, 2013).

### 2.3. Wind Turbine

According to Danao, Eboibi and Howell (2013), they have conducted an experimental investigation on a wind tunnel scale vertical axis wind turbine with an unsteady wind conditions. During the experiment, wind speed was conducted at 7m/s (given a Reynolds number of around 50,000) with 7% and 12% fluctuations in wind velocity at a frequency of 0.5Hz. The boundary condition on the wind tunnel created an unsteady wind flow cause the rotational speed fluctuation in the vertical axis wind turbine (VAWT). The unsteady wind flow also create instantaneous power to the turbine rotor. The results show the unsteady power coefficient (CP) fluctuates according to the changes in wind velocity. Based on the results, the time average of the unsteady power coefficient (CP) with a 7% fluctuation in wind velocity was similar to the steady wind conditions. Besides that, the 12% fluctuations in wind speed resulted in a drop in the mean power coefficient (CP). It means unsteady wind with high fluctuation amplitudes will reduce the power coefficient (CP) and detrimental to the energy yields from the wind turbines (Danao, Eboibi and Howell, 2013).

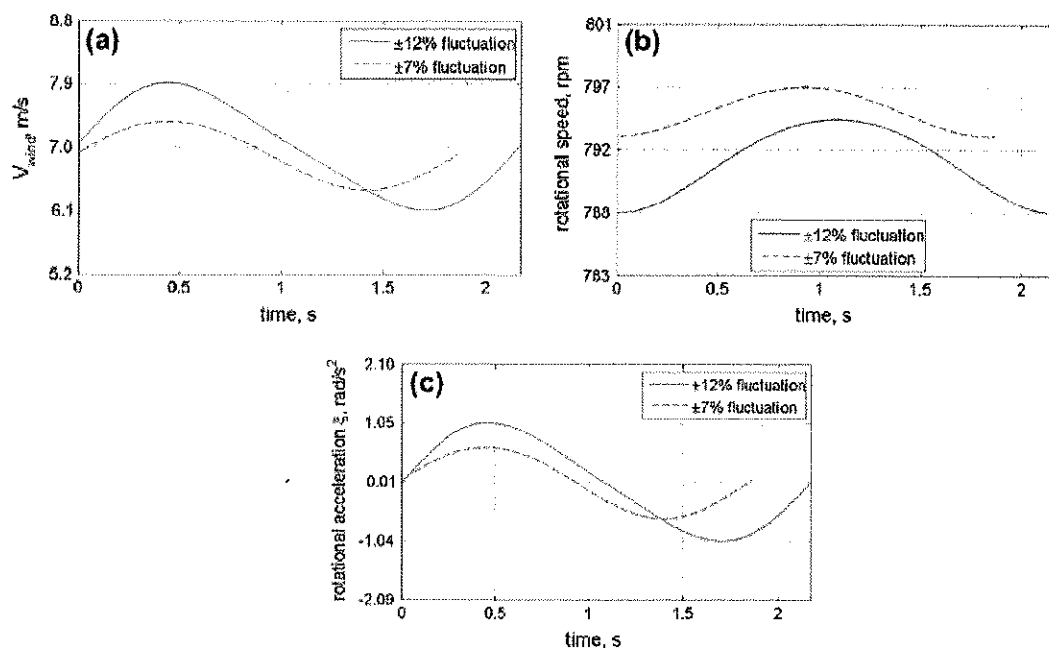


Figure 2.3.1 Unsteady Kinematics for different Vamp (a) wind velocity, (b) RPM and (c) acceleration (Danao, Eboibi and Howell, 2013).

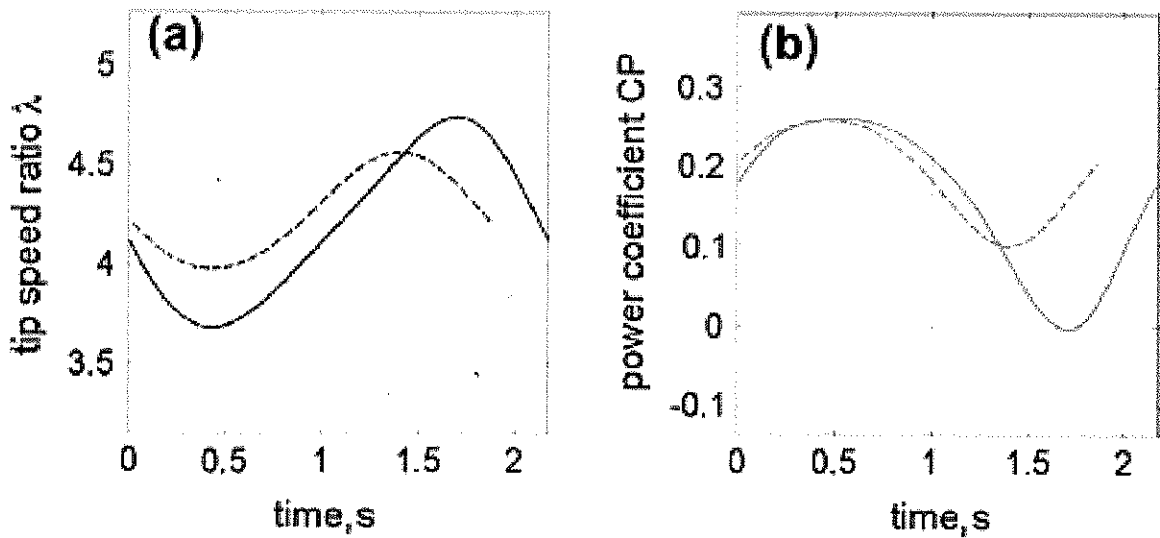


Figure 2.3.2 Unsteady Performance of the VAWT for the two  $V_{amp}$  cases: (a)  $\lambda$  and (b) CP (Danao, Eboibi and Howell, 2013).

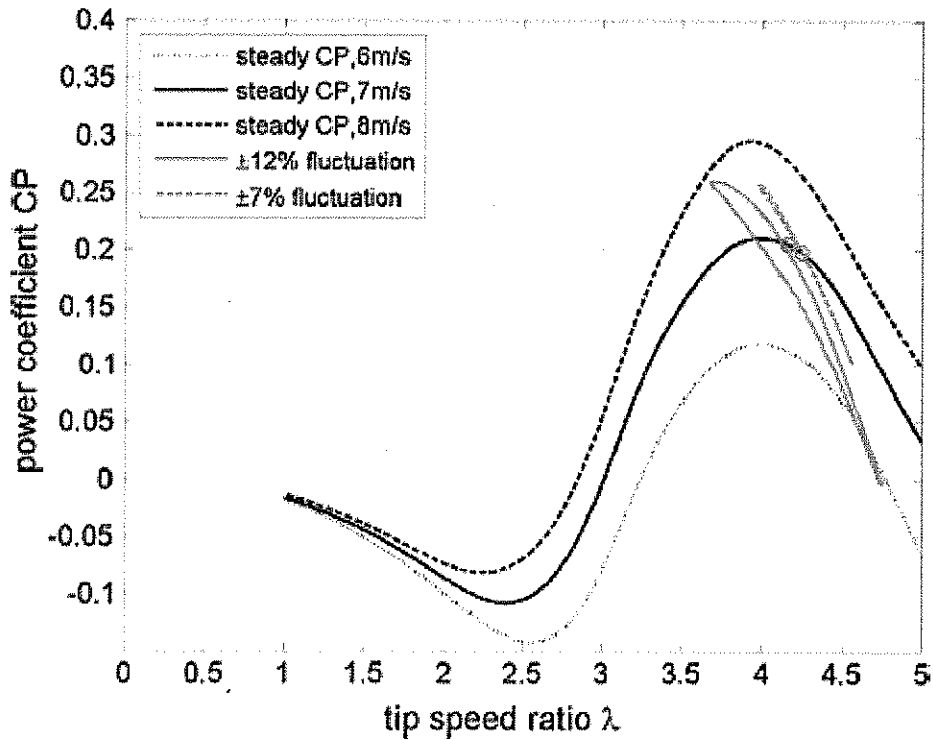


Figure 2.3.3 unsteady performance of the VAWT at different  $V_{amp}$  (Danao, Eboibi and Howell, 2013).

Tip speed ratio is a ratio between rotor speed and wind speed. The equation is shown as below:

$$\lambda = \frac{R\Omega}{V_{\infty}} \quad (1)$$

Where,  $\lambda$  = Tip speed ratio

$R$  = Rotor Radius

$\Omega$  = Rotor angular speed

$V_{\infty}$  = Free stream wind speed

The performance of a wind turbine is determined by three main parameter. The parameters are power coefficient, torque coefficient and overall efficiency. In addition, power coefficient can be defined as the amount of mechanical power that produced by the wind turbine against the total wind power. The form of equation of the power coefficient is shown as below (Anant Kishore and Priya, 2013):

$$C_p = \frac{P_{mech}}{(1/2)\rho\pi r^2 u_{\infty}^3} \quad (2)$$

Where,  $C_p$  = Power coefficient

$P_{mech}$  = Mechanical power

$\rho$  = Density (air)

$u_{\infty}$  = Free wind Speed (Free stream wind speed)

## 2.4. Computational Fluid Dynamic (CFD) Simulation

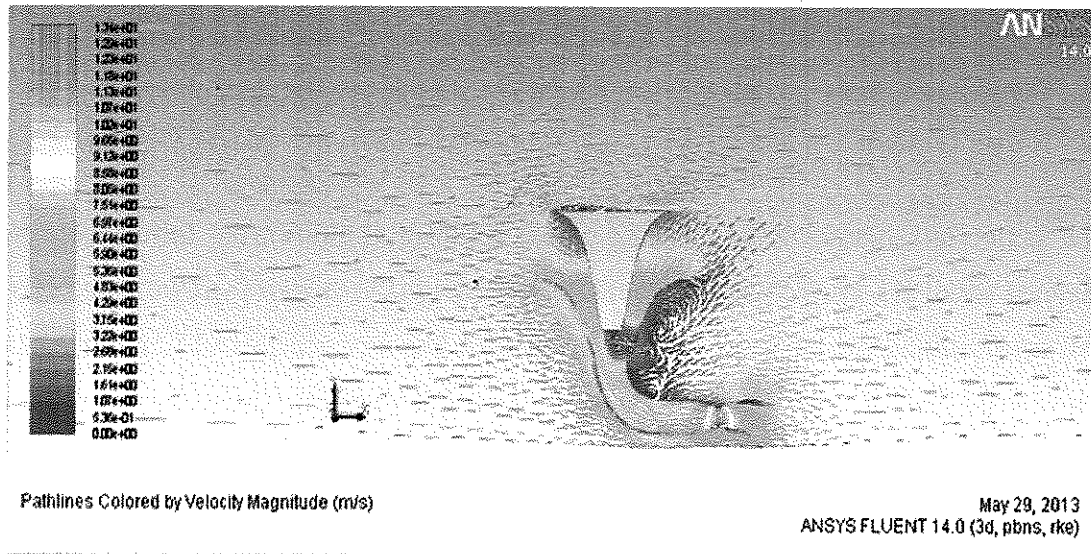


Figure 2.4.1 Velocity profile in cutaway slice on the plane of symmetry using ANSYS model (Allaei and Andreopoulos, 2014).

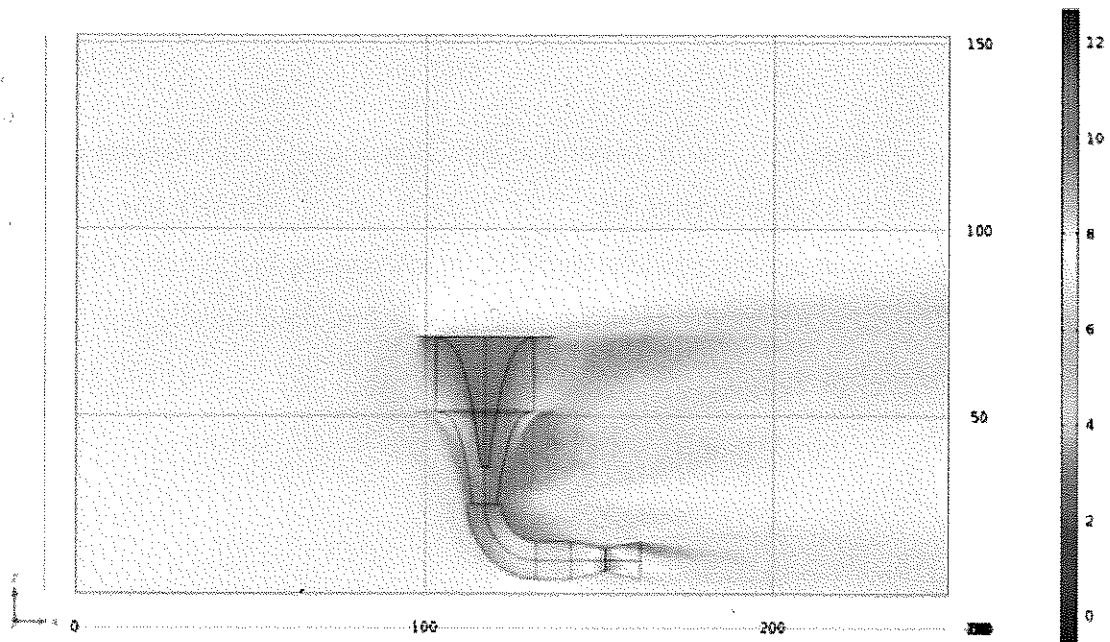


Figure 2.4.2 Velocity profile in cutaway slice on the plane of symmetry using COMSOL model (Allaei and Andreopoulos, 2014).

The simulation for the performance of wind delivery system is quite challenging due to the complexity of the wind delivery system. The objective of the simulation is to understand the air field inside the INVELOX where the actual wind turbine is located.

The simulation also involved case with different incoming wind direction and changes in the intake geometry. The result is present and it shows it is possible to capture, accelerate and concentrate the wind flow through the system. There is a significant improvement in the power output where the wind velocity increased (Allaei and Andreopoulos, 2014).

However, once the turbine is installed inside an INVELOX system, the output will reduce making the promise of superior performance no longer valid due to the increase in resistance. The free stream wind reduces its velocity when it reached the blades due to the induced velocity field by the vortex system. The reduction of the output could be up to half or to two-thirds (Allaei and Andreopoulos, 2014).

According to N.A. Ahmed (2013), the success of the concept proposed depends on ensuring that wind flow from the surrounding atmosphere can be guided to the turbine rotors with minimum losses. This including optimizes the geometry of the structure with the goal of achieving the maximum velocity at the place where rotor will be located (Ahmed, 2013).

Computational Fluid Dynamics (CFD) is a research tool is especially made for cost effective and flexible. It was used to study the interrelations between the various important geometrical parameters of the novel concept design such as cowling outlet angle turning, vane gap, and constriction diameter where the rotor will be located (Ahmed, 2013).