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

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

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 WEI EE..

Matrix No. ..I08001549..

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

LIST OF FIGURES

LIST OF TABLES

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

Symbol	*Definition*
C_p	Power coefficient
P_{mech}	Mechanical power
ρ	Density
u_∞	Free wind Speed
λ	Tip speed ratio
R	Rotor Radius
Ω	Rotor angular speed
V_∞	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^{th} 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 boots 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

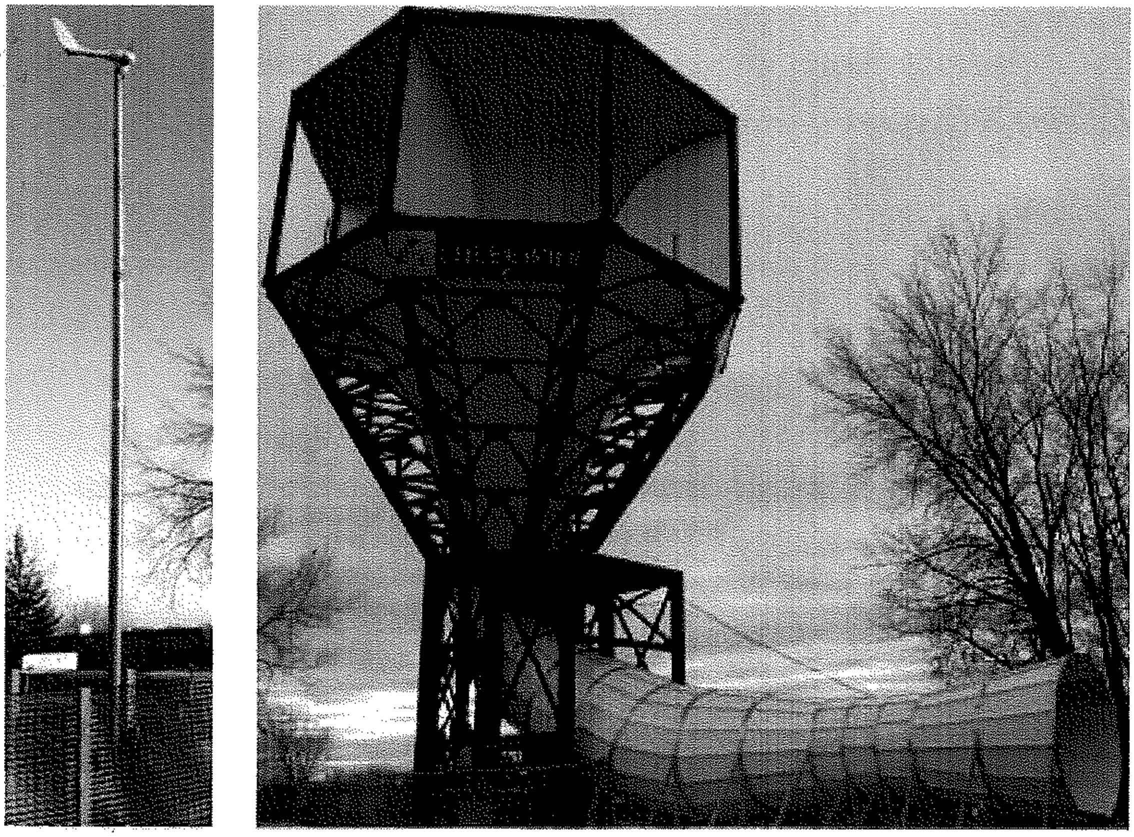

Figure 2.2.1 Typical turbine tower system (left) and Fielded INVELOX demo (right) under evaluation in Chaska, Minnesota (Allaei and Andreopoulos, 2014).

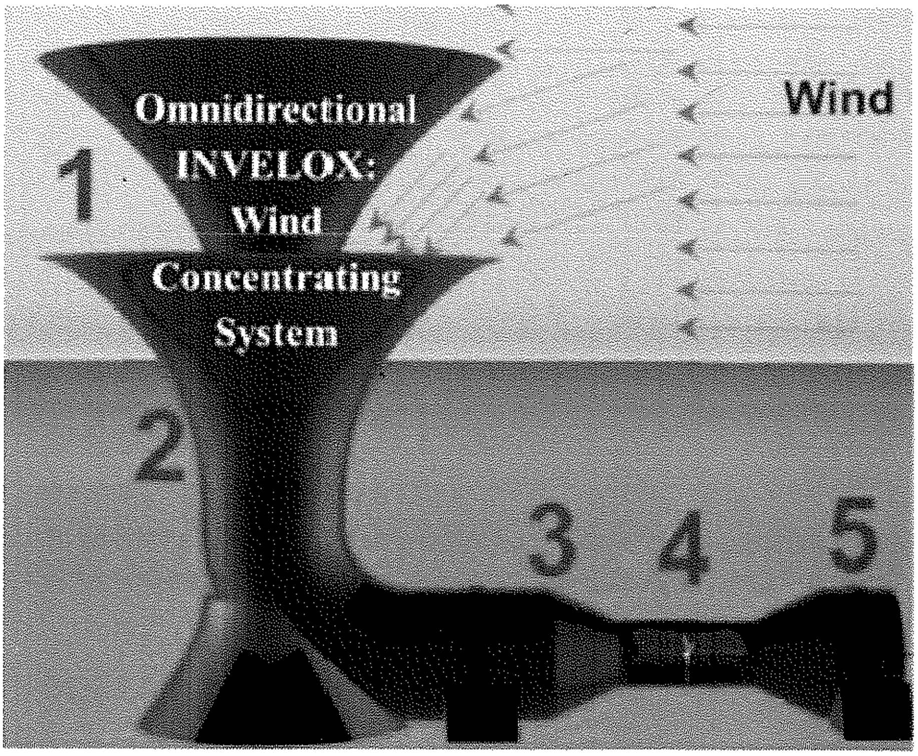

Figure 2.2.2 Schematic of the INVELOX wind delievery system with its key components (Allaei and Andreopoulos, 2014).

(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).

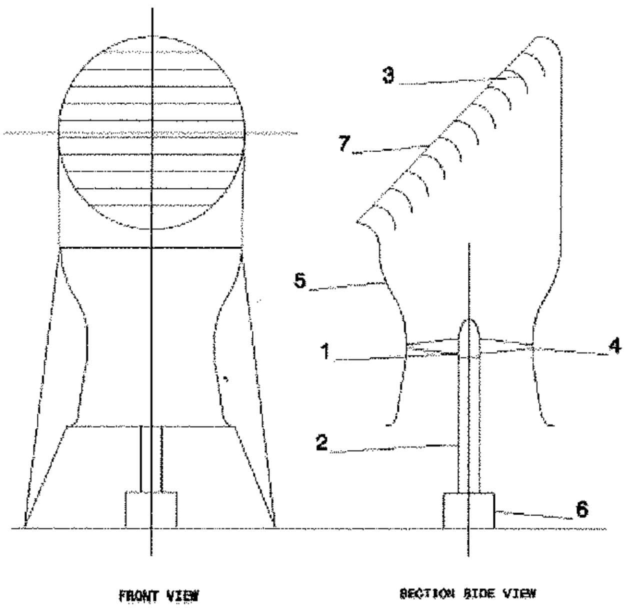

Figure 2.2.3 Schematic of a novel wind turbine (Ahmed, 2013).

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 make it operate like a horizontal axis wins 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 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 results in improved low wind speed start up and overall efficiency. The protective screen (7) can also use 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).

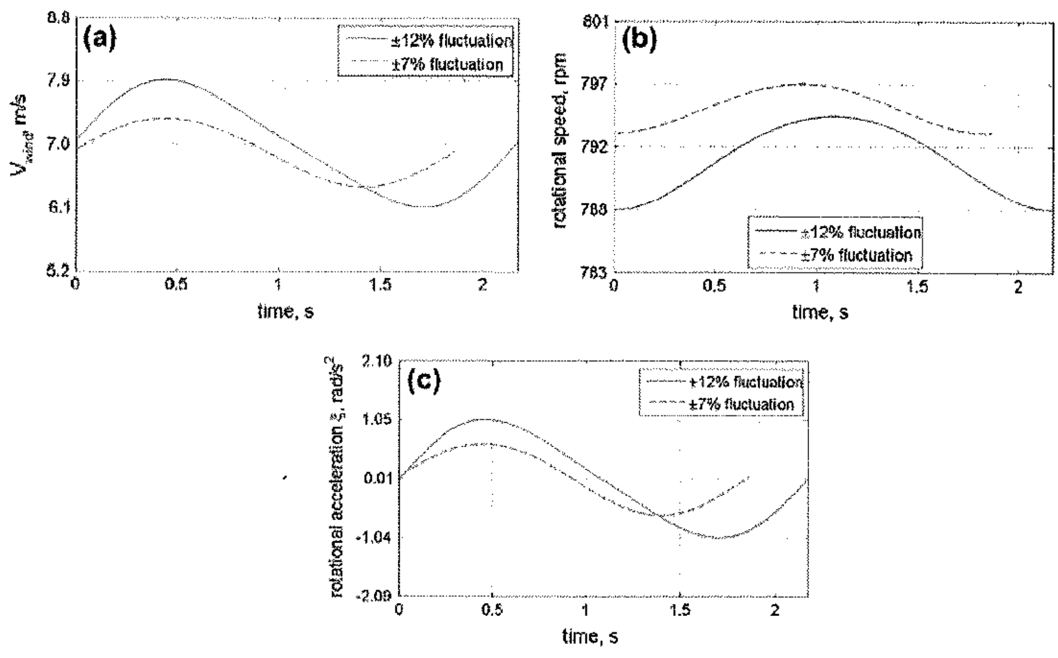

Figure 2.3.1 Unsteady Kinematics for different Vamp (a) wind velocity, (b) RPM and (c) acceleration (Danao, Eboibi and Howell, 2013).

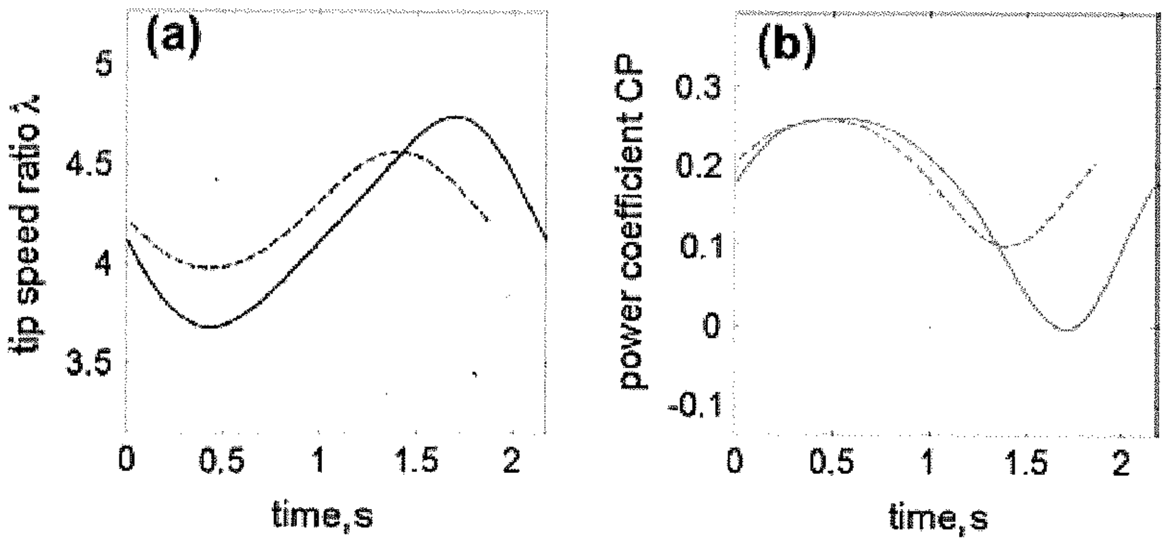

Figure 2.3.2 Unsteady Performance of the VAWT for the two V_{amp} cases: (a) λ and (b) CP (Danao, Eboibi and Howell, 2013).

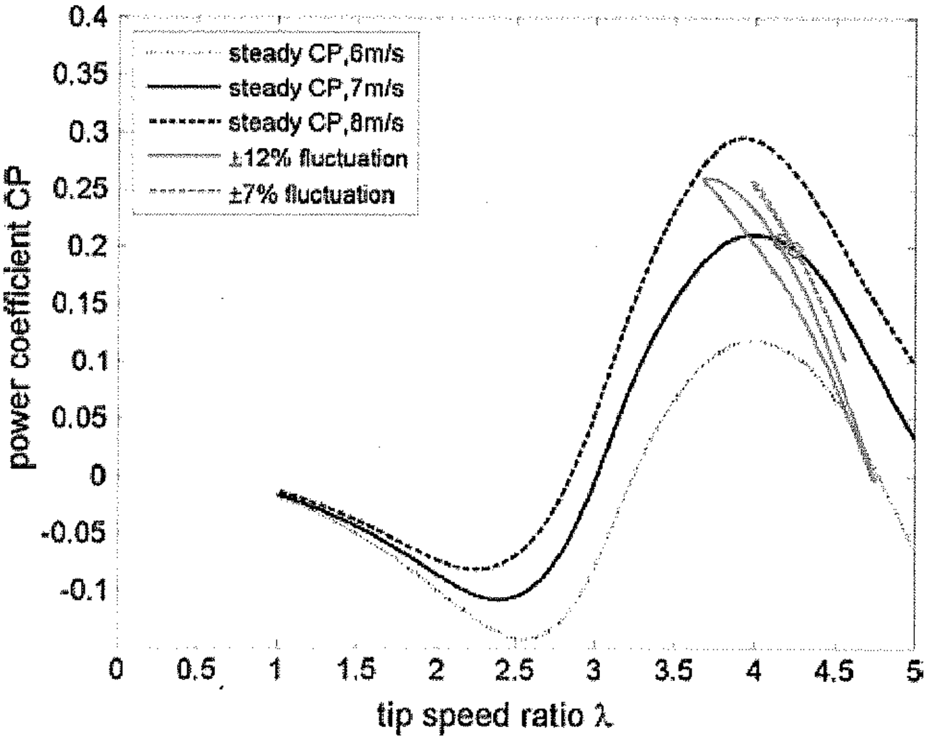

Figure 2.3.3 unsteady performance of the VAWT at different V_{amp} (Danao, Eboibi and Howell, 2013).

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

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

Where, λ = Tip speed ratio

R = Rotor Radius

Ω =Rotor angular speed

V_∞ =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):

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

Where, C_p = Power coefficient

P_{mech} = Mechanical power

ρ = Density (air)

u_∞ = Free wind Speed (Free stream wind speed)

2.4. Computational Fluid Dynamic (CFD) Simulation

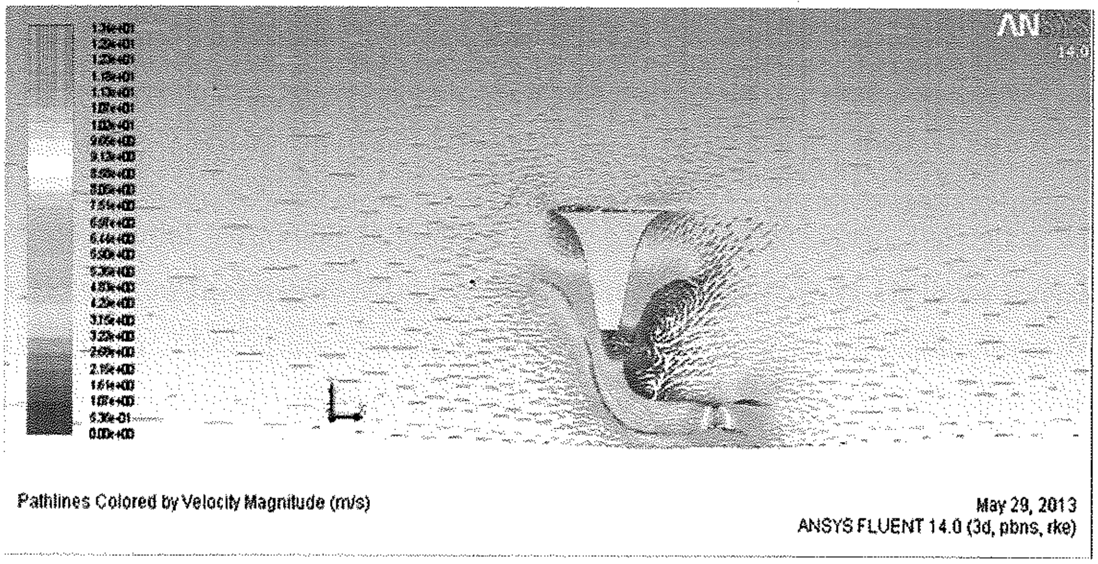

Figure 2.4.1 Velocity profile in cutaway slice on the plane of symmetry using ANSYS model (Allaei and Andreopoulos, 2014).

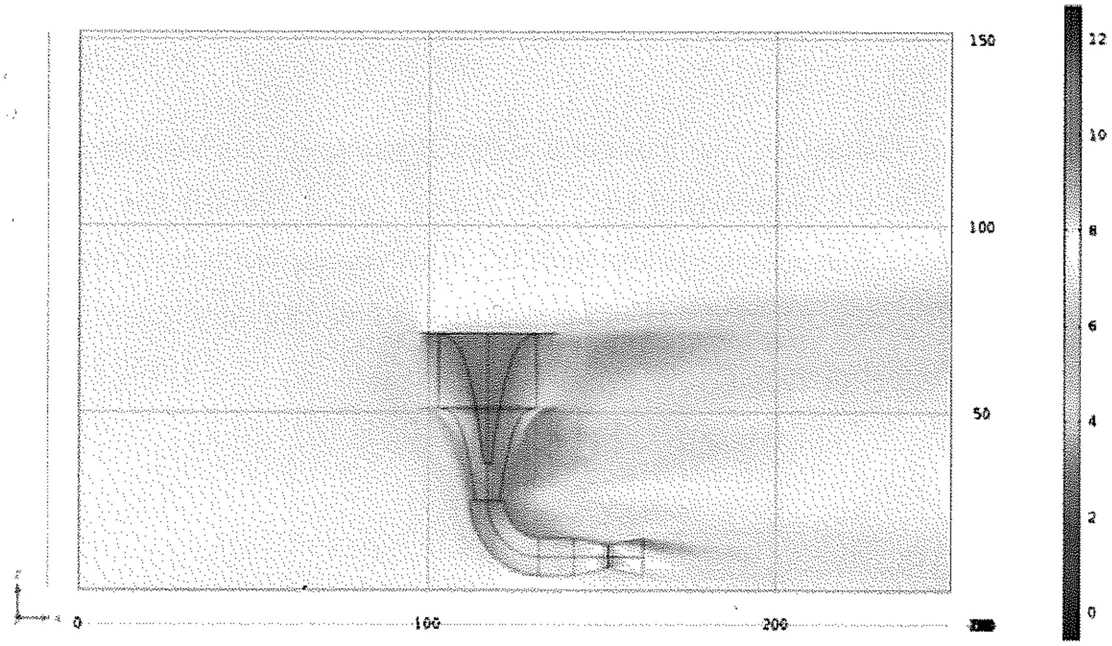

Figure 2.4.2 Velocity profile in cutaway slice on the plane of symmetry using COMSOL model (Allaei and Andreopoulos, 2014).

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).