MODELLING AND ANALYSIS ON THE EFFECT OF WRAP-AROUND INLET CHANNEL TO VORTEX FLOW FOR MICRO HYDRO APPLICATION

RONALDO ULISI

BK18110241

FACULTY OF ENGINEERING

UNIVERSITI MALAYSIA SABAH

2022

HK08 FYP 2 FINAL THESIS DRAFT	
Supervisor	Ir. Dr. Mohd Azlan Bin Ismail
Examiner 1	Dr. Nazrein Adrian Bin Amaludin
Examiner 2	Pn. Fadzlita Binti Mohd Tamiri

MODELLING AND ANALYSIS ON THE EFFECT OF WRAP-AROUND INLET CHANNEL TO VORTEX FLOW FOR MICRO HYDRO APPLICATION

RONALDO ULISI

THESIS SUBMITTED IN PARTIAL FULFILMENT OF THE REQUIREMENT FOR THE DEGREE OF BACHELOR OF MECHANICAL ENGINEERING

FACULTY OF ENGINEERING

UNIVERSITI MALAYSIA SABAH

2022

DECLARATION

I declare that this thesis was submitted only to University Malaysia Sabah as the requirement for degree of Bachelor of Mechanical Engineering. I also certify that the values of results in this thesis is entirely taken from my simulation and experimental result. Additionally, I declare that the work designated is entirely my own, except for my quotation and summaries of sources which have been duly acknowledged.

26 MAY 2022

RONALDO ULISI

(BK18110241)

CERTIFIED BY

IR. DR. MOHD AZLAN BIN ISMAIL

SUPERVISOR

CERTIFICATE

NAME: **RONALDO ULISI**

NO MATRIX: **BK18110241**

TITLE: **MODELLING AND ANALYSIS ON THE EFFECT OF WRAP-AROUND INLET CHANNEL TO VORTEX FLOW FOR MICRO HYDRO APPLICATION**

BACHELOR: **BACHELOR WITH HONOR MECHANICAL ENGINEERING**

VIVA DATE: **25^{TH} JULY 2022**

CERTIFIED BY

SUPERVISOR **SIGN**

IR.DR. MOHD AZLAN BIN ISMAIL ---------------------------

ACKNOWLEDGEMENT

First out of all, I would like to express my special thanks of gratitude to my beloved supervisor, Ir.Dr Azlan Bin Ismail, for his continues support and guidance along with final year project. I am incredibly grateful that I had the chance to work on this fantastic project because I have learned a lot from it.

Secondly, I would also like to thank my parents and friends who helped me a lot in finalizing this project within the limited time frame. Thank you.

Ronaldo Ulisi

26 May 2022

ABSTRACT

The Gravitational Water Vortex Turbine is a new and underdeveloped micro hydro system. Thus, this field still not fully explored and many design and parameters need to study. It brings the efficiency of the vortex turbine still very low which is below than 40%. There are a lot of factors that able to influence the efficiency of vortex turbine performance such as the design of basin and inlet configuration of basin. This study is focused on Computational Fluid Dynamics (CFD) analysis of water vortex formation and hydrodynamic flow characteristic. For more specific, this study focusses to compare and analyse the vortex formation and its hydrodynamic characteristic with respect to the performance. There are two different configurations were developed which are no wrap-around inlet and wrap-around inlet. Aside that, three models of wrap-around inlet with radius curvature of 0.27 m, 0.28 m and 0.30 m were developed for the optimization purpose. These models were designed using Solidwork and simulated using Ansys CFX. The experimental work also done for validation purpose. The percentage difference between the experimental and simulation is 7.94%, which is modest enough to demonstrate that the simulation setting usable. In the result, it is focused on the water streamline, vortex profile, water velocity and optimization. Based on the findings, the wrap-around inlet computed higher and symmetric vortex formation, unform and high magnitude of velocity and the streamline also distributed well compared to no-wrap around inlet model. While for the optimize model, wrap-around with 0.27 m radius of curvature was the most efficient among the tested models. With the same reasons, it can create a high velocity with symmetrical vortex.

ABSTRAK

PEMODELAN DAN ANALISIS TERHADAP KESAN SALURAN MASUK BALUT KE SEKITAR KEPADA ALIRAN VORTEX UNTUK APLIKASI MIKRO HIDRO

Turbin Vortex Air Gravitational ialah sistem hidro mikro yang baharu dan kurang dibangunkan. Oleh itu, bidang ini masih belum diterokai sepenuhnya dan banyak reka bentuk serta parameter yang perlu dikaji. Ia menjadikan kecekapan turbin vorteks masih sangat rendah iaitu di bawah 40%. Terdapat banyak faktor yang boleh mempengaruhi kecekapan prestasi turbin vorteks seperti reka bentuk lembangan dan konfigurasi salur masuk lembangan. Kajian ini tertumpu kepada analisis Computational Fluid Dynamics (CFD) pembentukan pusaran air dan ciri aliran hidrodinamik. Untuk lebih spesifik, kajian ini memfokuskan untuk membandingkan dan menganalisis pembentukan vorteks dan ciri hidrodinamiknya berkenaan dengan prestasi.Terdapat dua konfigurasi berbeza telah dibangunkan iaitu tiada salur masuk lilit dan salur masuk lilit. Selain itu, tiga model salur masuk lilit dengan kelengkungan jejari 0.27 m, 0.28 m dan 0.30 m telah dibangunkan untuk tujuan pengoptimuman. Model ini direka bentuk menggunakan Solidwork dan disimulasikan menggunakan Ansys CFX. Kerja eksperimen juga dilakukan untuk tujuan pengesahan. Perbezaan peratusan antara eksperimen dan simulasi ialah 7.94%, yang cukup kecil untuk menunjukkan bahawa tetapan simulasi boleh digunakan. Hasilnya, ia tertumpu pada aliran air, profil vorteks, halaju air dan pengoptimuman. Berdasarkan penemuan, salur masuk lilitan mengira pembentukan vorteks yang lebih tinggi dan simetri, tidak berbentuk dan magnitud tinggi halaju dan garis arus juga teragih dengan baik berbanding model salur masuk tanpa lilitan. Manakala untuk model pengoptimuman, lilitan dengan jejari kelengkungan 0.27 m adalah yang paling cekap antara model yang diuji. Dengan alasan yang sama, ia boleh mencipta halaju tinggi dengan pusaran simetri.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1 Introduction

Nowadays, global warming and pollution become more serious issues and being a concern of every county. Global warming has become increasingly dangerous as a result of increased carbon dioxide emissions and the depletion of fossil resources. Every year, an estimated seven million people die because of air pollution around the world. According to World Health Organization (WHO) data, nearly all the world's population (99%) breathes air that exceeds WHO guideline limits and contains high levels of pollutants, with low- and middle-income nations bearing the brunt of the burden. The World Health Organization (WHO) is assisting countries in combating air pollution. (WHO, 2021). Thus, the usage of environmentally friendly electric power is in demand.

Water energy is important for a sustainable future because it is a clean, inexpensive, and environmentally friendly form of power generation, but developing such an energy system to generate energy from water is usually a huge challenge. Water may be used to generate energy in a variety of ways, including hydrostatic and hydrokinetic processes. The hydrostatic technique is a traditional method of generating electricity that involves storing water in reservoirs to produce a pressure head and collecting the potential energy of water using appropriate turbomachinery (S. Dhakal *et al.*, 2015a). In hydrokinetic approach, the kinetic energy from moving water is directly converted into electricity by relatively small scales turbines without impoundment and with almost no head (S. Dhakal *et al.*, 2015a)

The hydropower system concept is promising renewable energy technology. Micro hydro power systems have a capacity of up to 100 kW to generate electricity. Electricity in rural, remote, and hilly locations is insufficient, with poor and unstable energy supplies. Micro hydropower can offer energy in rural areas where grid extension is too expensive, and consumers have low incomes. In general, hydropower system does not emit any pollutants into the atmosphere.

The gravitational water vortex turbine (GWVT) is a growing micro-hydro power plant that requires less expertise, has a low head, and requires less setup area. The gravitational water vortex turbine is a low-head turbine that can operate in the 0.7–2 m range(Nishi *et al.*, 2020a). The GWVT's performance has yet to be thoroughly evaluated using turbine performance curves. The recent research offers numerous performance parameters such as rotational speed, torque, brake power, and mechanical efficiency of single-stage GWVT under various flow and design conditions using mathematical expressions and experimental.

The major components of GWVT are arranged in a separate channel outside the mainstream of a river or a canal. An upstream channel, a cylindrical basin with an opening at the bottom, a rotor with blades, and a vertical shaft are the main components of a GWVT. Under gravity, water flows upstream via a straight channel, then tangentially into a cylindrical basin to generate a vortex(S. Dhakal *et al.*, 2015b). Due to Coriolis force, the flow intake causes the vortex to form at the free surface. This vortex begins to increase as it approaches the bottom opening, and cause speeding up the rotation of the water. Using a rotor with blades, the vortex's flow energy can be converted to mechanical power. The generator is connected to the turbine and shaft motor converting the mechanical energy to electric energy.

Considering this background, this project aims to analyze and study about vortex turbine performance by conducting experiment dan simulation by using ANSYS. The experimental data will be used for verifying and validate the simulation result.

1.2 Problem Statement

Low-head hydroelectric technologies have recently attracted a lot of attention to expand distributed power systems into remote areas that are difficult to connect to the electrical grid, particularly in developing nations. Because these systems can operate with low head without the need for a large reservoir and installation area, gravitational vortex hydropower systems can be a renewable and suitable option for expanding electricity access and promoting development in these remote regions, which are also rich in hydric resources.

But the biggest problem of vortex turbine system is its efficiency. Based on the past experimental study, it found that the efficiency of the vortex turbine still very low which is below than 40% (Sritram & Suntivarakorn, 2017b). As for the Water Free Vortex Turbine, the highest generated energy was 14.5 W, the torque was 2.77 N-m, and the highest efficiency was 35.92% at rotational speed of 50 rpm.

There are a lot of factors that able to influence the efficiency of vortex turbine performance such as the design of basin and inlet configuration of basin. Thus, the purpose of this research is to investigate the influence of different configuration of inlet basin. The configuration of inlet vortex turbine that will investigate are tangential inlet of cylinder basin and wrap-around inlet of cylinder basin.

1.3 Objective

The main objective of this research is to study and investigate the factor that affecting the vortex turbine performance via CFD simulation. This objective can be further divided into three sub-objectives:

1. To develop CFD model of vortex turbine by utilizing Simulation Software and validating with experimental work.
2. To compare the vortex formation between tangential inlet of cylinder basin and wrap-around inlet of cylinder basin.

3. To analyze the influence of wrap-around inlet of cylinder basin to its hydrodynamic characteristic with respect to the performance.

1.4 Scope of Work

This research able to enhance the understanding of implementation Computational Fluid Dynamic simulation to define the factors that affecting the vortex turbine performance. The scope job related to this project are

1. Research and reading a lot of articles related to vortex turbine for better and clear understanding. The articles including the literature review of pass experimental study and CFD modeling of the vortex turbine. The general articles of vortex turbine give the fundamental knowledge and information about the factor affecting the performance of vortex turbine.
2. Define the factors affecting the vortex turbine performance and specify the parameter that will be chosen for further study. This project focus on the effect of different inlet configuration for cylinder basin.
3. Design different inlet configuration of vortex turbine basin. Software like SolidWork is the most suitable for designing purpose. By having good designs or plans, it can give a clear picture of the simulation process in the CFD analysis later.
4. This project is aiming experimental work for collecting related data such as vortex height produced to validate the simulation result. The pervious researcher's prototype will be used for collecting validation data. Thus, no fabrication of prototype needed. The previous prototype is Gravitational Water Vortex Turbine with cylindrical basin.
5. CFD simulation test is perform by using ANSYIS CFX. The efficiency of the vortex turbine can be tested by performing CFD simulation with different inlet configuration (tangential inlet and wrap-around inlet). The best inlet configuration of basin can be determined after collecting the simulation result.
6. The data collected from simulation and experimental will be optimized based on the validation done on the ANSYS software and experimental work.

1.5 Research Methodology

1.5.1 Research

Problem statement and background research

- The problem of the vortex turbine is determined from reading the related articles and journals.
- After getting some information/knowledge about the vortex turbine, the problem can be identified, and solution can be made through the project analysis.
- Information regarding the method of CFD modeling and simulation can be obtain from internet by reviewing articles and journals. The best journals that can be refer are the past experimental project dan pass research CFD modeling. It leads how to use CFD simulation feature, start from the pre-processing, post-processing, and outcome result. The CFD modeling and simulation research is important to know the correct method of setting the boundaries condition, the type of meshing and material selection. By having correct method, the outcome CFD result is more accurate and applicable.
- The past experimental research is important to give some general knowledge about the factors affecting the vortex turbine performance. Other than that, the internet sources information can provide basic knowledge of handling vortex turbine apparatus in the lab for collecting related data.

1.5.2 Analysis

Experimental

- The experimental analysis of vortex turbine performance with cylindrical basin type
  will be conducted in lab. The analysis will show the contribution of the basin design to the vortex performance in terms of power and torque generated.
- The result will be analyzed either the basin design should be improved or not.
- There is no fabrication work involve, where the previous prototype from previous researcher will be use.

CAD drawing

- The design of vortex turbine will be constructed by using CAD software such as SolidWork or Inventor.
- Analysis of basin parameters such as inlet configuration should be in consideration.
- Cad drawing can give a clear picture to run the simulation as a reference.

Design simulation and Analysis

- The basin design simulation is conducted by using ANSYS CFX software.
- All the design simulation must be correct process: pre-processing, post-processing, and result.
- The analysis of boundaries condition, type of meshing and materials selection also important in terms of running the CFD simulation. In other words, the analysis of ANSYS CFX features should be study properly.

CFD modeling

- Design will be conducted on simulation test to evaluate the performance of the vortex turbine system.

1.5.3 Testing

Validation

- The collected data from experiment in the lab is using for validating the simulation result by comparing the results.
- If the percentage error between experimental and simulation result is smaller, the simulation analysis can be further study and the result of simulation is applicable to be use.
- After the validation process, the design of basin can be various just using the design modelling simulation.

Optimization

- The vortex turbine parameters (inlet configuration) will be change accordingly.
- After testing/simulation done, the most optimize parameter will be define through the vortex performance such as output torque and output power.

1.5.4 Documentation

- The final finding of the project will be written in the report properly.

1.6 Project Flow Chart

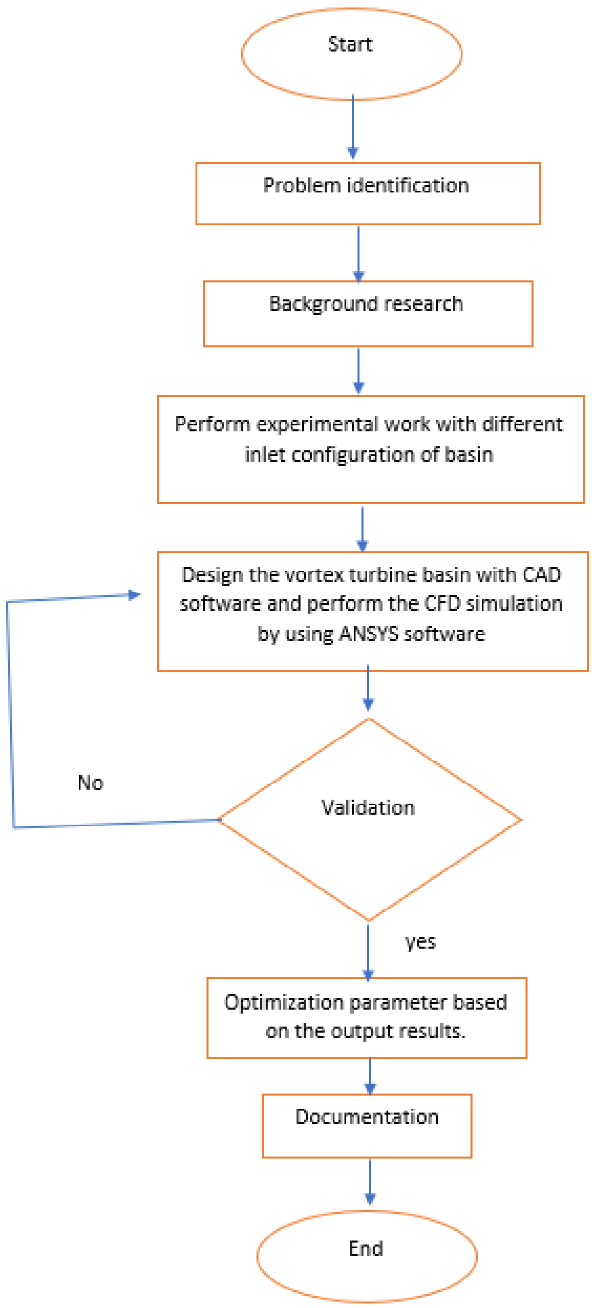

Figure 1.1: Flow Chart

1.7 Research Expected Outcomes

This project is focus on the analysis of the factor affecting of vortex turbine performance by implementing the CFD modeling simulation analysis. The parameters that highlighted and being manipulated variables are the configuration of basin (conical type and cylindrical type). Each of the basin will perform simulation with different height of basin. The expected outcomes of this project are:

- ✓ Understanding the flow process of CFD simulation application in solving the problem.
- ✓ Having a proper and acceptable validation of project analysis.
- ✓ Defining the best inlet configuration of cylindrical basin either tangential or wrap-around inlet.
- ✓ Improving the efficiency vortex turbine by generating high and strong vortex formation

1.8 Research Contributions

A gravitational water vortex power plant is a green technology that uses alternative or renewable energy sources to generate electricity. Water is tangentially fed into a circular basin to generate a free vortex, and energy is collected from the free vortex using a turbine in a vortex power plant. The discovery of vortex turbine research can help in a variety of ways, as seen below.

- ✓ The vortex turbine analysis can contribute to a better design of turbine that can perform in high efficiency.
- ✓ The modification and CFD simulation testing can give a clear knowledge to choose the optimize parameter that affecting the vortex turbine performance.

- ✓ The research also encouraging people to develop an efficient turbine. It is a new and not well-developed technology to harvest electricity from low pressure water energy sources.
- ✓ There are limited literatures available on the design, fabrication and physical geometry of the vortex turbine and generator. This research can contribute in terms of simulation results that will validate with experimental works.

1.9 Research Commercialization

Energy demand is increasing all the time, especially in developing countries like the Philippines and India. Hydropower, a renewable energy source, has become one of the most sought-after sources of energy due to its clean generation. Low-head hydropower plants are needed in areas where grid extension is impossible due to difficult geographical terrain and other factors. Due to its significant potential for commercialization, vortex turbine power generation has a bright future.

- ✓ Vortex turbine can operate in low head condition to generate electricity energy. Gravitational water vortex turbine is an ultra-low head turbine which can operate in as low head range of 0.7–2m with similar yield as conventional hydroelectric turbines used for production of renewable energy characterized with positive environmental yield.
- ✓ Vortex turbine just use free renewable energy which is gravitational water from river. Water energy being a clean, cheap and environment friendly source of power generation is of great importance for sustainable future.
- ✓ Can provide electricity energy to the people that live in the area which cannot see grid extension due to difficult geographical terrain.

CHAPTER 2

LITERATURE REVIEW

2.1 History Development of Hydropower Plant

Hydropower is the most common renewable energy source on the globe. Hydropower plants are typically used in peak regulation and frequency modulation in the power grid, and their flexibility and reliability are vital for the overall electrical power system's safety and stability (Yang *et al.*, 2021). Hydropower plants are taking on greater obligations for the power grid than ever before, as the economic society's demand for electricity resources grows (Yang *et al.*, 2021). As a result, ensuring the stability of hydropower plants is critical, particularly for hydro-dominant power systems.

As the hydropower plant become more significant, researcher start to do research regarding the characteristic and development of hydropower plant. In 1827, French specialist Benoit Fourneyron fostered a turbine fit for creating around 6 pull which is the most punctual variant of the Fourneyron response turbine (Britannica, 2017.) . By 1837, Fourneyron had delivered a turbine fit for 2,300 cycles each moment, 80% productivity, and 60 pull, with a wheel a foot in measurement and weighing just 40 pounds (18 kilograms).