RDU140345

OPTIMIZATION OF THE FLOW CHARACTERISTIC IN VARIOUS ARRAY ARRANGEMENTS OF MULTIPLE PICOHYDRO POWER TURBINES

(PENGOPTIMUMAN CIRI ALIRAN DALAM TURBIN HIDRO PIKO UNTUK PELBAGAI SUSUNAN)

IDRIS BIN MAT SAHAT
AT-TASNEEM BINTI MOHD AMIN
MUHAMMAD AMMAR BIN NIK MU'TASIM
WAN ANUAR BIN WAN HASSAN
UMMU KULTHUM BINTI JAMALUDIN
ROSMAZI BIN ROSLI

RESEARCH VOTE NO.:
RDU140345

Computational Analysis Group
Centre For Earth Resources Research & Management
&
Fakulti Kejuruteraan Mekanikal
Universiti Malaysia Pahang

2017

ACKNOWLEDGEMENT

The authors would like to express their gratitude to Universiti Malaysia Pahang and Centre For Earth Resources Research & Management (CERRM) for the awarded fund of RDU140345, for supporting these research activities. Special thanks also dedicated to fellow faculty members for their assistance and support throughout this project.

ABSTRACT

OPTIMIZATION OF THE FLOW CHARACTERISTIC IN VARIOUS ARRAY ARRANGEMENTS OF MULTIPLE PICOHYDRO POWER TURBINES
(*Keywords: Optimization,flow characteristics, pikohidro, turbine)*

Renewable energy is an energy which is generated from the natural resources such as wind, water, sunlight, rain and geothermal. Naturally, it can be replenished. This energy can reduce the pollutions that occur all around the world. It is also can be the best solution for the global issue such as climate changes that is happened rapidly nowadays. Likewise, it is practically suitable in the rural areas or the remote areas because of the difficulty in getting fuel which is also expensive. Micro hydropower (MHP) is one of renewable energy resources in the world. Hydro turbines are used to change water pressure from river flow into mechanical or kinetic energy, which can generate electricity. It can be classified by its capacity such as pico, mini, and small which the power are up to 100KW, 1MW and 25MW respectively. Accordingly, MHP is the most economical technology to generate power. This is because MHP use the water flow in a river to generate power, where it is a continuous process. Besides, the costs for manufacture and install MHP turbines are cheaper compared to the tidal and wind turbines. Most of the computational researches on arrays arrangement are done for tidal turbines, which is almost similar to MHP turbines. Computational Fluid Dynamics (CFD) models are used to analyze the wake effects in arrays of tidal turbines. The simulations of the turbines are done to obtain the wake characteristics of actuator disc and compare with experimental data. According to, Fluent software is used to simulate 3-D models of Tidal Energy Converter (TEC) turbines with three-row array. In this research, the TEC array arrangement and spacing simulated to identify the performance variation of electrical power output. Besides that, adaptive mesh method with Gerris solver is used to optimize multiple arrays of picohydro turbines in a channel. Reynolds-averaged Navier Stokes (RANS) equations are used with the turbines to stand for frozen rotor. Static blade method is similar with the frozen rotor method. The frozen rotor method is used in the computational domain to consider effect of velocity on it. The CFD software is used here to analyse the best 3D array arrangement of river picohydro turbines, which generate maximum power output. Meanwhile, the MATLAB and DasyLab simulations will validate the CFD results obtained from a single picohydro turbine at three different locations.

Key researchers **:**

IDRIS BIN MAT SAHAT
AT-TASNEEM BINTI MOHD AMIN
MUHAMMAD AMMAR BIN NIK MU'TASIM
WAN ANUAR BIN WAN HASSAN
UMMU KULTHUM BINTI JAMALUDIN
ROSMAZI BIN ROSLI
Tel. No. **: +609-4246223**
Vote No. **: RDU140345**

ABSTRAK

PENGOPTIMUMAN CIRI ALIRAN DALAM TURBIN HIDRO PIKO UNTUK PELBAGAI SUSUNAN

(Kata Kunci: pengoptimuman, cirri aliran dalaman, pikohidro, turbin)

Tenaga boleh diperbaharui adalah tenaga yang dijana daripada sumber semula jadi seperti angin, air, cahaya matahari, hujan dan panas bumi. Secara semula jadi, ia boleh diisi semula. Tenaga ini boleh mengurangkan pencemaran yang berlaku di seluruh dunia. Ia juga boleh menjadi penyelesaian terbaik untuk isu global seperti perubahan iklim yang berlaku dengan pesat pada masa kini. Begitu juga, ia boleh dikatakan sesuai di kawasan luar bandar atau kawasan pedalaman kerana kesukaran dalam mendapatkan bahan api yang juga mahal. kuasa hidro mikro (MHP) adalah salah satu daripada sumber tenaga boleh diperbaharui di dunia. Hydro turbin digunakan untuk menukar tekanan air dari aliran sungai kepada tenaga mekanikal atau kinetik, yang boleh menjana elektrik. Ia boleh diklasifikasikan oleh kapasitinya seperti pico, mini, dan kecil yang kuasa sehingga masing-masing 100KW, 1MW dan 25MW. Oleh itu, MHP adalah teknologi yang paling ekonomi untuk menjana kuasa. Ini kerana MHP menggunakan aliran air di dalam sungai untuk menjana kuasa, di mana ia adalah satu proses yang berterusan. Selain itu, kos bagi pembuatan dan pemasangan MHP turbin adalah lebih murah berbanding dengan turbin pasang surut dan angin. Kebanyakan kajian pengiraan pada susunan tatasusunan dilakukan untuk turbin pasang surut, yang hampir sama dengan turbin MHP. Dinamik bendalir pengiraan (CFD) model digunakan untuk menganalisis kesan bangun dalam tatasusunan turbin pasang surut. Simulasi turbin yang dilakukan untuk mendapatkan ciri-ciri berikutan cakera penggerak dan bandingkan dengan data eksperimen. Menurut, perisian Fasih digunakan untuk mensimulasikan model 3-D Tidal Energy Converter (TEC) turbin dengan pelbagai tiga-berturut-turut. Dalam kajian ini, susunan TEC lokasi dan jarak simulasi untuk mengenal pasti variasi prestasi output kuasa elektrik. Selain itu, kaedah mesh penyesuaian dengan Gerris penyelesai digunakan untuk mengoptimumkan pelbagai tatasusunan turbin picohydro dalam saluran. Reynolds-purata Navier Stokes (Rans) persamaan digunakan dengan turbin untuk berdiri untuk rotor beku. kaedah bilah statik adalah sama dengan kaedah pemutar beku. kaedah pemutar beku digunakan dalam domain pengiraan untuk mempertimbangkan kesan halaju di atasnya. Perisian CFD digunakan di sini untuk menganalisis pengaturan pelbagai 3D terbaik turbin picohydro sungai, yang menghasilkan output kuasa maksimum. Sementara itu, MATLAB dan DasyLab simulasi akan mengesahkan keputusan CFD diperolehi daripada turbin picohydro tunggal di tiga lokasi yang berbeza.

TABLE OF CONTENTS **Page**

CHAPTER 1

INTRODUCTION

1.1 Introduction

Renewable energy is an energy which is generated from the natural resources such as wind, water, sunlight, rain and geothermal. Naturally, it can be replenished. This energy can reduce the pollutions that occur all around the world. It is also can be the best solution for the global issue such as climate changes that is happened rapidly nowadays. Likewise, it is practically suitable in the rural areas or the remote areas because of the difficulty in getting fuel which is also expensive.

1.2. Problem Statement

Micro hydropower (MHP) is one of renewable energy resources in the world. Hydro turbines are used to change water pressure from river flow into mechanical or kinetic energy, which can generate electricity [1]. It can be classified by its capacity such as pico, mini, and small which the power are up to 100KW, 1MW and 25MW respectively [2]. According to [1], MHP is the most economical technology to generate power. This is because MHP use the water flow in a river to generate power, where it is a continuous process. Besides, the costs for manufacture and install MHP turbines are cheaper compared to the tidal and wind turbines.

Most of the computational researches on arrays arrangement are done for tidal turbines, which is almost similar to MHP turbines. Computational Fluid Dynamics (CFD) models are used to analyze the wake effects in arrays of tidal turbines [3]. The simulations of the turbines are done to obtain the wake characteristics of actuator disc and compare with experimental data. According to [4], Fluent software is used to simulate 3-D models of Tidal Energy Converter (TEC) turbines with three-row array. In this research, the TEC array arrangement and spacing simulated to identify the performance variation of electrical power output.

Besides that, adaptive mesh method with Gerris solver is used to optimize multiple arrays of picohydro turbines in a channel [5]. Reynolds-averaged Navier Stokes (RANS) equations are used with the turbines to stand for frozen rotor. Static blade method is similar with the frozen rotor method. The frozen rotor method is used in the computational domain to consider effect of velocity on it [6]. The CFD software is used here to analyse the best 3D array arrangement of river picohydro turbines, which generate maximum power output. Meanwhile, the MATLAB and DasyLab simulations will validate the CFD results obtained from a single picohydro turbine at three different locations.

Thus, this research investigates the possibility of finding the best multiple picohydro turbines array arrangement to generate the possible maximum power output. The pump which functioned as turbines is actually an aquarium pump with the diameter of 47.9mm and 54mm of length. The motivation in using small pump as turbine is driven by the rapid growth in planting the giant size of generators. The flow of the water will gradually can cause damage to the ecosystem especially in the river. In fact, the large scale of turbine will influence higher power generations and also higher pollution in the ecosystem. [7-8, 9]. Thus, the needs of the small scale of turbines which can generate the high value of power generation are mandatory to minimize the disturbance on the natural flow and also to preserve the ecology. However, the latest finding [10] shown that the single small size of turbine might not be enough to generate the desired power. As a result, the cumulative of the power generation from multiple small sizes of turbines are necessary for small electrical appliances.

1.3 Objectives

- To design numerical model of multiple picohydro turbines array arrangements
- To obtain the experimental flow characteristic and pattern from multiple picohydro turbines array arrangements in the free flow

1.4 Scopes

- To generate the power output of 80 Watts from multiple picohydro turbines
- To validate the experimental flow characteristics with numerical models of multiple picohydro turbines array arrangements in the free flow setting

CHAPTER 2

A STUDY ON THE EFFECT OF FLOW RATE ON THE POWER GENERATED BY A PICO HYDRO POWER TURBINE

World Applied Sciences Journal 30 (Innovation Challenges in Multidiciplinary Research & Practice): 420-423, 2014
ISSN 1818-4952

DOI: 10.5829/idosi.wasj.2014.30.icmrp.40

A Study on the Effect of Flow Rate on the Power Generated by a Pico hydro Power Turbine

At-Tasneem M. A.[1], Wan Mohamad Azam and Ummu Jamaludin
Faculty of Mechanical Engineering, University Malaysia Pahang,
26600 Pekan, Pahang, MALAYSIA
[1]tasneem@ump.edu.my

Abstract - The experimental investigation was conducted to explore the effect of the flow rate on the power produced by a picohydro power turbine. The pelton turbine is originally an aquarium pump that required 12V and 1.05A of direct current. The pump is functioned as a pelton turbine. The potential energy created by the stream of the water is converted into mechanical rotation of the fans before it generate electricity. Tthe constant magnetic field that is produced by the stator is caused by the impulsion on the electrons in the metal inside of the turbine. A range of velocities were tested on the turbine. The results reveal that the power produced by the turbine is increased as the increment on the velocity of the water. The modified pump can produce 6 watt of power with the velocity of 3 m/s of the water. Thus, the result present in this paper may facilitate the development of the multiple picohydro power turbines which is designed to minimize the abuse of the ecosystem as the development of hydropower generator is usually planted across the habitat of river ecology.

Keywords-picohydro power, renewable energy, pump as turbine.

I. INTRODUCTION

Renewable energy is an energy which is generated from the natural resources such as wind, water, sunlight, rain and geothermal. Naturally, it can be replenished. This energy can reduce the pollutions that occur all around the world. It is also can be the best solution for the global issue such as climate changes that is happened rapidly nowadays. Likewise, it is practically suitable in the rural areas or the remote areas because of the difficulty in getting fuel which is also expensive.

There are several forms of renewable energy such as wind energy, hydropower energy, biomass energy, ocean energy and geothermal energy [1]. Wind energy is generated from the flow of the wind. The generated energy is affected by three factors: the earth’s terrain, bodies of water, and the vegetative cover by the mountain and trees and buildings. Electricity is generated from the airflow of the modern wind turbine. Geothermal energy is originated from the heat inside of the earth. The current commercial utilization, the types of resources is hydrothermal reservoirs, hot dry rock, geopressed brides and magma are yet to be developed. Biomass is originated from the carbonaceous material or residue from animals and plants. The composition of the carbon dioxide in atmosphere remains constant because the same amount of carbon release from biomass process is used in photosynthesis. There are three types of conversion in ocean energy which are tidal energy which caused by combined attraction of sun and moon on the waters revolving globe, wave energy which caused by motion of the water up and down and ocean thermal energy which came from the temperature difference between the surface and deep water.

II. HYDROPOWER TURBINE

Hydropower is generated from the water. It can be classified by its capacity such as pico, mini, and small which the power are up to 100KW, 1MW and 25MW respectively [2]. Based on prior studies of hydropower, the water mill is the oldest methods used by human to produce mechanical energy instead of electrical energy [3] because the water turbine was found only in 19th century [2].

The main part of the hydropower system is turbine. It converts energy from the dropping water into rotating shaft and generates the electricity. The selection of the turbine is crucial in generating the power according to the head of the water as shown in Table I.

TABLE 1
IMPULSE AND REACTION TURBINE [4]

Turbine type	Head Classification High (>50m)	Medium (10-50m)	Low (<10m)
Impulse	Pelton Turgo Multi-jet Pelton	Crossflow Turgo Multi-jet pelton	Crossflow
Reaction		Francis (spiral case)	Francis (Open-flume) Propeller Kaplan

The impulse turbine is operated in the air. The water pressure remains at atmospheric pressure before and after the runner driven by a jet of water making contact with the runner blades. The reaction turbine is immersed in the water and it is fully operate in the water.

Pelton turbine is commonly used in the picohydro power system according to it efficiency [4]. It can be operated at low flow rate (Fig. I [5]) and can generate the electricity easily. However, it cannot be operated in the free flow of water. In order to create a high speed-jet in the Pelton turbine, the water has to be driven from a nozzle [6].

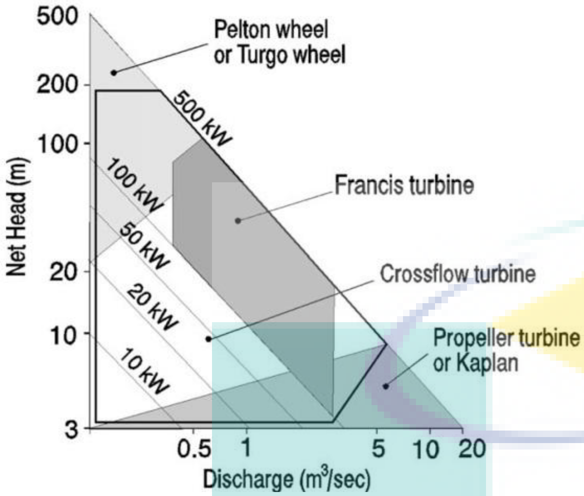


Fig I Head flow range for small hydro turbines [5]

In the recent finding by [7], the overshot turbine is the most suitable for the low discharge flow rate of water (Fig 2).

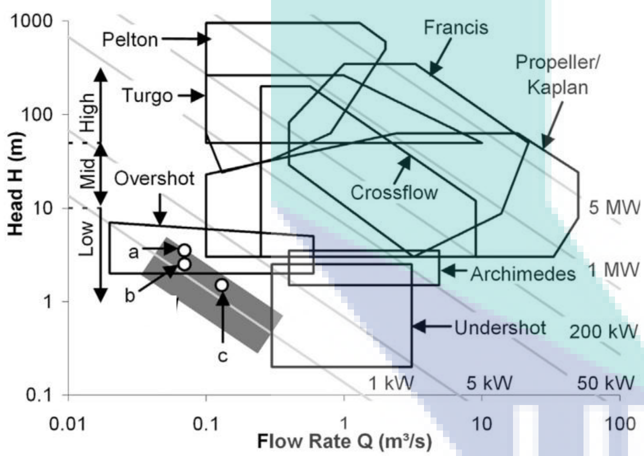


Fig. II Typical turbine application range chart [7]

Therefore, this paper is to investigate the possibility of modifying a pump to be functioned as a turbine [8]. The pump is actually an aquarium pump with the diameter of 47.9mm and 54mm of length.

The motivation in using small pump as turbine is driven by the rapid growth in planting the giant size of generators. The flow of the water will gradually can cause damage to the ecosystem especially in the river. In fact, the large scale of turbine will influence higher power generations and also higher pollution in the ecosystem. [6-7, 9].

Thus, the needs of the small scale of turbines which can generate the high value of power generation are mandatory to minimize the disturbance on the natural flow and also to preserve the ecology. The design of multiple small sizes of hydropower generators at the rural river and buoyance on the top of the water were stimulated from the tidal turbine in the ocean [10, 11]. It is shown that the single small size of turbine might not be enough to generate the desired power. As a result, the cumulative of the power generation from multiple small sizes of turbines are necessary for small electrical appliances.

III. PUMP AS TURBINE

The ZC-A40 aquarium pump is used in this experiment (Fig III). This submersible pump required 5V until 12V, 1.05 A of maximum direct current and 13 Watt of power. It is able to pond up until 550 liter per hour with 3.8 max head. The aquarium pump, the stator and the circuit board has been manufactured and sealed with epoxy resin [12] so that it is waterproof. Therefore, there will be no electric spark or current leakage which may abuse the ecosystem.

Fig. III ZC-A40 pump

The experiment on the effect of the water flow rate on the power generated from a single pump (Fig III) is conducted in the controlled environment. The head of the inlet water is 1 meter in the close channel of 3.124cm^2 of the area. For analysis, the pump is connected in data acquisition DAQ NI9239 (Fig IV). This DAQ is used to convert the analog input signal produced by turbine to digital signal in the computer processor. A range of water flow rate passing through the pump versus the voltage generated is recorded. The electrical power generated from the pump as a turbine is discussed.

IV. RESULT AND DISCUSSION

The voltage generated from the pump based on the water flow rate, Q is plotted in Fig V.

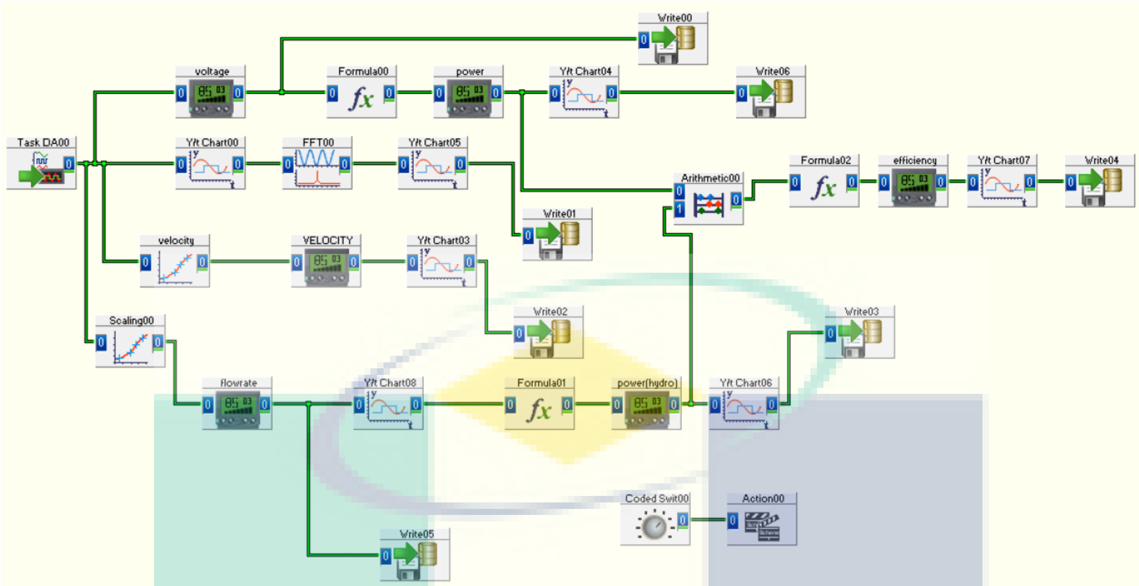


Fig IV DAQ Worksheet

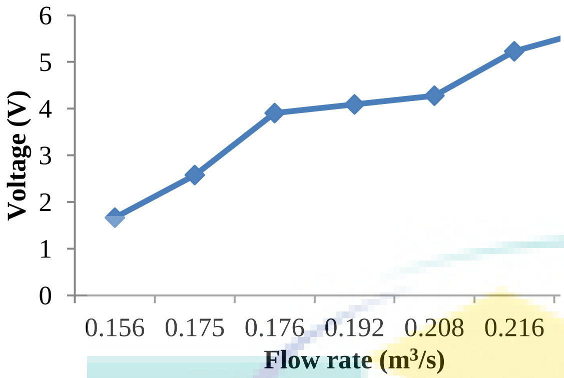


Fig V Water flow rate vs Voltage

The electrical power, P generated from the pump (functioned as turbine) is calculated as

$$P = VI \text{ or } V^2/R \quad (1.1)$$

where V = voltage across the component (V)
I = current flow (A)
R = resistance (Ω).

Based on eq 1.1 and the specification of the pump, the calculated resistance of the pump is 11.08 Ω. As a result, the relationship between the electrical power, P generated by the pump and flow rate, Q is shown in Fig V. The pump can produce 3Watt of electrical power with 0.2 m^3/s of water in controlled environment.

V. FUTURE WORK

Although the electrical power generated by pump is as low as 3Watt for 0.2 m^3/s of flow rate, the finding elucidates that the pump has a potential to become a turbine. Therefore, the pump can be modified as a turbine by removing the water inlet

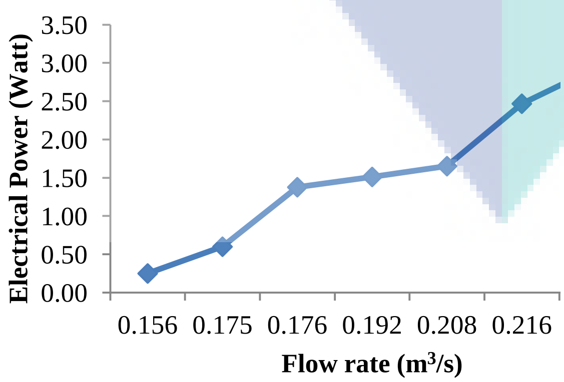


Fig VI Water flow rate vs Electrical Power

and outlet of the water and connect it with a propeller as in Fig.VI. It can be shown that the propeller turbine is the best choice for the low head (<10m) of discharge [4]. Also, the higher the flow of the water, the higher the power can be generated from this turbine [5].

This prototype turbine will be tested at Sungai Pahang, Malaysia where the recorded average flow rate of the water is 596.53m^3/s by Jabatan Pengaliran dan Saliran Air, Kuantan Malaysia. Based on results and Fig. II, this turbine is capable to generate as maximum power as possible. The possible maximum power generated by a single modified pump as propeller turbine is 10.8V with the efficiency (η) of the propeller turbine 0.90 [6] and the maximum voltage supplied by the pump is 12V.

The idea of developing multiple picohydro turbines is to minimize the sedimentation and ecosystem disturbance.in the future, it can be a device to monitor the sediment transport around the turbines is compulsory. In fact, the idea of floating the turbines instead of submerging it underwater is to minimize the disruption will also be considered.

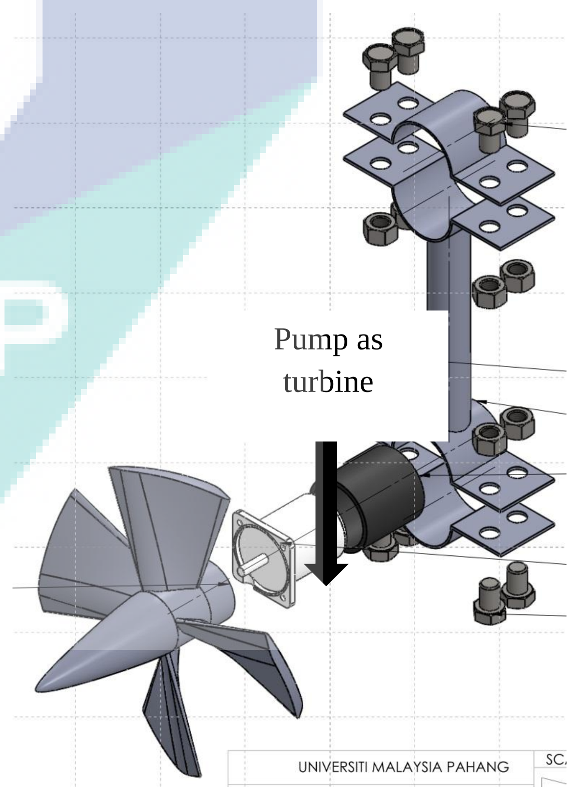


Fig VII Illustration of pump as turbine

REFERENCES

[1] Zhixiao Ma, Bing Xue, Yong Geng, Wanxia Ren, Tsuyoshi Fujita, Zilong Zhang, Jose A. Puppim de Oliveira, David A. Jacques and Fengming Xi, "Co-benefit analysis on climate change and environmental effect of wind-power: A case study from Xianjiang, China," *Renewable Energy*, vol 57, pp 35-42. 2013.

[2] Ahmed M.A. Haidar, Mohd F.M. Senan, Abdulhakim Noman, Taha Radman, "Utilization of pico hydro generation in domestic and commercial loads," *Renewable and Sustainable Energy Review*, vol 16, pp 518-524. 2011.

[3] R. Bakis, "The current status and future opportunities of hydroelectricity," *Energy Sources, PartB: Economics, Planning, and Policy*, vol 2:3, pp 259-266. 2007.

[4] Oliver Paish, "Small hydro power: technology and current status," *Renewable and Sustainable Energy Reviews*, vol 6, pp 537-556. 2002.

[5] Paish Oliver, "Micro-hydro power: Status and Prospects," *Journal of Power and Energy, Professional Engineering Publishing. 2002.*

[6] Johnson V, "Fluid flow in pico hydro power plant", 2007.

[7] S. J. Williamson, B. H.Stark, J. D. Booker, "Low head pico hydro turbine selection using a multi-criteria analysis," *Renewable Energy*, vol 61, pp 43-50. 2013.

[8] Armando Carravetta, Giuseppe Del Giudice, Oreste Fecarotta and Helena M. Ramos, "Pump as Turbine (PAT) Design in Water Distribution Network by System Effectiveness" Water, vol 5, pp 1211-1225, 2013.

[9] C. Beth Robertson, Mathew T. Langford, David Z. Zhu, and Alf Leake, " Reservoir Thermal Structure and its effect on Hydropower Operation Induced Fish Entrainment," in *Proc. of World Environmental and Water Resources Congress 2011: Bearing Knowledge for Sustainability,"* 2011.

[10] T. Divett, R. Vennell, and C. Stevens, "Optimization of multiple turbine arrays in a channel with tidally reversing flow by numerical modeling with adaptive mesh," *Philosophical Transactions of The Royal Society A (Mathematical Physical & Engineering Sciences,* vol 371, no 1985. 2013.

[11] Herman Jacobus Vernaak, Kanzumba Kusakana, Sandile Philip Koko, "Status of micro-hydrokinetic river technology in rural application: A review of literature," *Renewable and Sustainable Reviews*, vol 29, pp 625-633. 2013.

[12]R. Omranipour, S. U. Haq, " Evaluation of Grading System of Large Motors AC Stator Windings," in *Proc. of IEEE International Symposium on Electrical Insulation*, pp 158-161, 2008.

CHAPTER 3

NUMERICAL SIMULATION OF MULTIPLE ARRAYS ARRANGEMENT OF MICRO HYDRO POWER TURBINES

World Academy of Science, Engineering and Technology
International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering
Vol:8 No:5, 2014

World Academy of Science, Engineering and Technology
International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering Vol:8 No:5, 2014

Numerical Simulation of Multiple Arrays Arrangement of Micro Hydro Power Turbines

M. A. At-Tasneem, N. T. Rao, T. M. Y. S. Tuan Ya, M. S. Idris, M. Ammar

Abstract**—**River flow over micro hydro power (MHP) turbines of multiple arrays arrangement is simulated with computational fluid dynamics (CFD) software to obtain the flow characteristics. In this paper, CFD software is used to simulate the water flow over MHP turbines as they are placed in a river. Multiple arrays arrangement of MHP turbines lead to generate large amount of power. In this study, a river model is created and simulated in CFD software to obtain the water flow characteristic. The process then continued by simulating different types of arrays arrangement in the river model. A MHP turbine model consists of a turbine outer body and static propeller blade in it. Five types of arrangements are used which are parallel, series, triangular, square and rhombus with different spacing sizes. The velocity profiles on each MHP turbines are identified at the mouth of each turbine bodies. This study is required to obtain the arrangement with increasing spacing sizes that can produce highest power density through the water flow variation.

Keywords**—**Micro hydro power, CFD, arrays arrangement, spacing sizes, velocity profile, power.

III. INTRODUCTION

HYDRO turbines are used to change water pressure from river flow into mechanical or kinetic energy, which can generate electricity [1]. According to [2], MHP technology is needed to generate electricity to residential areas. Besides, MHP is one of renewable energy resources in the world. According to [1], small hydro or MHP is the most economical technology to generate power. This is because MHP use the water flow in a river to generate power, where it is a continuous process. Besides, the costs for manufacture and install MHP turbines are cheaper compared to the tidal and wind turbines.

Most of the computational researches on arrays arrangement are done for tidal turbines, which is almost similar to MHP turbines. Computational Fluid Dynamics (CFD) models are used to analyze the wake effects in arrays of tidal turbines [3]. The simulations of the turbines are done to obtain the wake characteristics of actuator disc and compare with experimental data. According to [4], fluent software is used to simulate 3-D models of Tidal Energy Converter (TEC) turbines with three-row array. In this research, the TEC array arrangement and spacing simulated to identify the performance variation.

Besides that, adaptive mesh method with Gerris solver is used to optimize multiple arrays of tidal turbines in a channel [5]. Reynolds-averaged Navier Stokes (RANS) equations are used with the turbines to stand for frozen rotor. Static blade method is similar with the frozen rotor method. The frozen rotor method is used in the computational domain to consider effect of velocity on it [6]. The CFD software is used here to consider the best arrangement of ocean current turbines, which generate maximum power.

Moreover, Large-eddy Scale (LES) is type of CFD, where the larger turbulent scales are resolved [7]. This method is carried out by creating a framework to generate inflow tidal turbulence data. With the framework, the wake characteristics and power generated have been considered. According to [8], Horizontal-axis Tidal Current Turbines (HATTs) are simulated with RANS CFD method to characterize the performance.

The objectives of this study are to develop a numerical model of river domain over a MHP turbine and obtain the flow characteristics of river flow when subjected with multiple arrays of MHP turbines. First, a numerical model of a river flow has been created to obtain flow characteristics without obstruction in it. Next, MHP turbine bodies with static blades are placed in the river domain with variation of arrays arrangements. Finally, the velocity profiles of each array are determined to identify the changes in the velocities.

IV. KINETIC ENERGY EQUATION

Kinetic energy (*KE*) is an energy formed as a result from the motion of a medium of fluid or air. The *KE* equation represents the relationship between velocity, mass and kinetic energy. The general equation of *KE* is:

$$KE = \frac{m.v^2}{2} \tag{1}$$

where *KE* is kinetic energy (J): *m* is mass (kg) and *v* is velocity of a fluid flow (m/s).

At-Tasneem M.A is with the Faculty of Mechanical Engineering, University Malaysia Pahang, 26600 Pekan, Pahang, Malaysia (corresponding author; phone: 09-4246211; e-mail: tasneem@ump.edu.my).
Rao N.T was with the Faculty of Mechanical Engineering, University Malaysia Pahang, 26600 Pekan, Pahang, Malaysia (e-mail: thiwaan@hotmail.com)
Tuan Ya T.M.Y.S. is with the Department of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Petronas, Bandar Seri Iskandar, 31750 Tronoh, Perak, Malaysia (e-mail: tyusoff.ty@petronas.com.my).
Idris M.S. and Ammar M. are with the Faculty of Mechanical Engineering, University Malaysia Pahang, 26600 Pekan, Pahang,Malaysia (e-mail: idris@ump.edu.my, ammar@ump.edu.my).

Meanwhile, for *KE* per unit mass, the formula is presented as following:

$$\frac{KE}{m} = \frac{v^2}{2} \tag{2}$$

The *KE* can be converted into mechanical energy. The kinetic energy from the water flow can be converted into kinetic energy of rotating shaft (*KE*) by rotating the turbine blade. The formula below proves that velocity is directly proportional to the angular velocity as the kinetic energy is converted into mechanical energy:

$$v = r\omega \tag{3}$$

where r is the radius of turbine shaft andω is an angular velocity

Substitute (3) into (1)

$$KE = \frac{m.r^2.\omega^2}{2} \tag{4}$$

$$KE = \frac{\omega^2}{2} \tag{5}$$

From (5), it is proven that the angular velocity of the rotating turbine blade increases when the kinetic energy increases. Furthermore, the power of the rotating turbine blade calculated with the following equation:

$$P = \omega M \tag{6}$$

whereω is the angular velocity and M is the momentum of torque.

These equations prove that the angular velocity of the turbine blade shaft increases when the velocity of the fluid flow increases. This is because the kinetic energy of the fluid flow is equals to the rotational kinetic energy. Hence, the more kinetic energy is converted into rotational kinetic energy in a turbine, the more power will be generated.

V. Turbine Body and Blade Design

The dimension of the existing MHP turbine blade obtained in Table I is 312mm diameter. The turbine body is designed in computer aided design software as shown in Figs. 1 and 2:

- Diameter (hollow part): 350 mm
- Outer diameter: 430 mm
- Thickness: 40 mm
- Length: 400 mm

A blade with diameter of 312 mm and thickness of 70 mm is designed, and assembled with the turbine body.

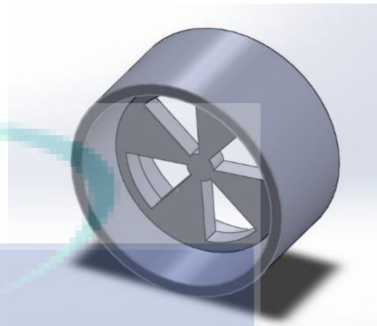


Fig. 1 The designed turbine body and static blade

Fig. 2 The blade must be smaller than the turbine body

VI. Model Description

Computational Domain

The river domain or the water flow domain is made up of box enclosure. The MHP turbine models are placed in the domain with chosen array arrangements and spacing lengths.

The height of the enclosure box is 4m from the base (Y-axis). The width is 5.5m (X-axis) and the length is 10m (Z-axis). However, this dimension is different according to the spacing lengths between the turbines. There are 4 MHP turbines for each array arrangements.

Meshing

The model mesh size used is fine for every array (see Table I). However, the total number of model elements is different according to the number of turbines and spacing lengths between the turbines.

Boundary Conditions

For this study, the river flow is standardized to steady fluid flow. The fluid is consists of air at the top and water below since it is an open-channel flow

Inlet: Velocity with upstream height
Outlet: Constant static pressure
Surface: Zero pressure
Bottom: No slip wall
Sidewall: No slip wall

The inlet velocity is 2.5m/s with upstream and downstream heights of 1m. The upstream height is the height of water level of the river at the inlet. The downstream height is the height of the river water level at the outlet.

TABLE 1
MESH SIZES FOR DIFFERENT ARRAY ARRANGEMENT

Array arrangement	Configuration	Number of grid elements	Number of nodes
	0%	161,483	29,221
	50%	530,409	104,345
Parallel	100%	671,429	129,351
	150%	677,227	130,189
	200%	675,108	129,849
	0%	2.453,271	510,762
	50%	582,214	113,557
Series	100%	647,487	124,616
	150%	639,472	122,783
	200%	650,892	124,732
	0%	417,182	78,079
	50%	459,336	84,839
Triangular	100%	549,988	98,542
	150%	483,492	89,133
	200%	671,775	129,595
	0%	117,052	21,557
	50%	543,204	106,804
Square	100%	646,506	124,906
	150%	656,557	126,304
	200%	663,951	127,222
Rhombus	0%	602,708	117,588
	50%	664,226	127,868
	100%	683,481	131,211
	150%	664,979	127,921
	200%	671,396	129,098

VII. MODEL CONFIGURATION

In each array arrangement, four MHP turbines are placed in the river domain. There are five types of array arrangements are used which are:

1. Parallel arrangement (Fig. 3)
2. Series arrangement (Fig. 4)
3. Triangular arrangement (Fig. 5)
4. Square arrangement (Fig. 6)
5. Rhombus arrangement (Fig. 7)

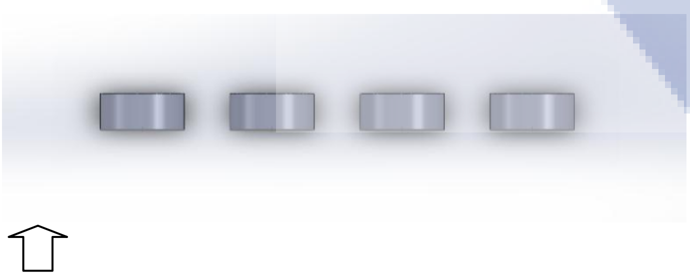


Fig. 3 Parallel arrangement

Fig. 4 Series arrangement

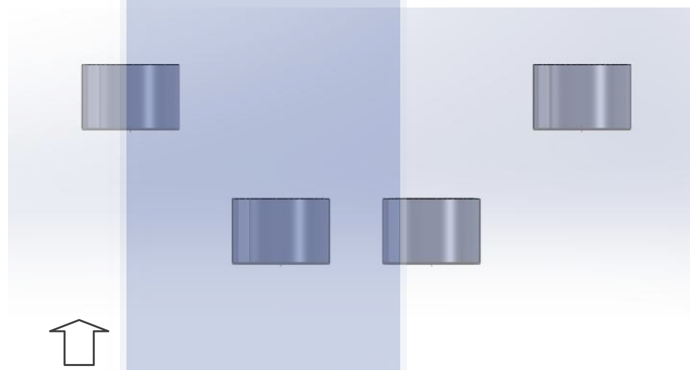


Fig. 5 Triangular arrangement

Fig. 6 Square arrangement

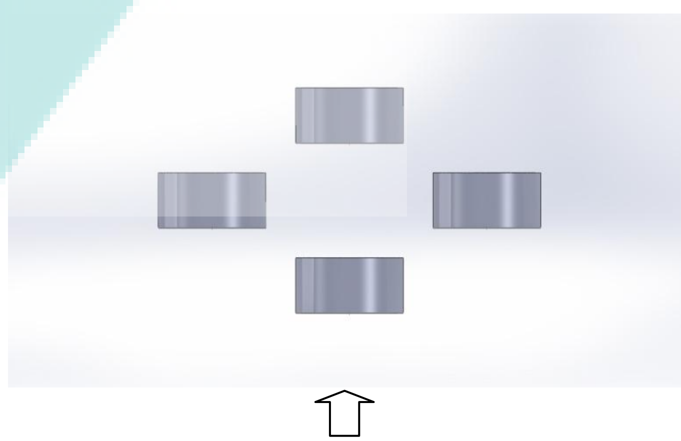


Fig. 7 Rhombus arrangement

The spacing sizes are calculated with respect to the turbine body diameter.

1. 0% diameter: 0mm spacing
2. 50% diameter: 215mm spacing
3. 100% of diameter: 430mm spacing
4. 150% of diameter: 645mm spacing
5. 200% of diameter: 860mm spacing

The first spacing size is 0mm between all the turbines, which the turbines, which the turbines are attached to each other side by side in each arrangement. The arrangements and spacing are done in Solid works 2012 software. The simulation is carried out for all the arrangements with calculated spacing sizes.

VIII. Results and Discussion

The CFD simulations and analysis were conducted to study the river water flow through MHP turbines of different types of array arrangements and specified spacing distances between the turbines. These simulations were conducted for two stages. A comparison data is conducted to present the differences of velocities in each turbine in every arrangement. These simulations were carried out with 1 velocity, and 5 different array arrangements with 5 different spacing sizes for each arrangement.

For velocity profile plot, velocity values at the entrance of the turbine were extracted from the line drawn across in front of the turbines. The profiles were plotted for all array arrangements. The velocity values at the turbine mouth must be closer the mean value of the river velocity, which is 2.5m/s. The average velocities for each array arrangement are calculated to determine the best one to generate maximum power.

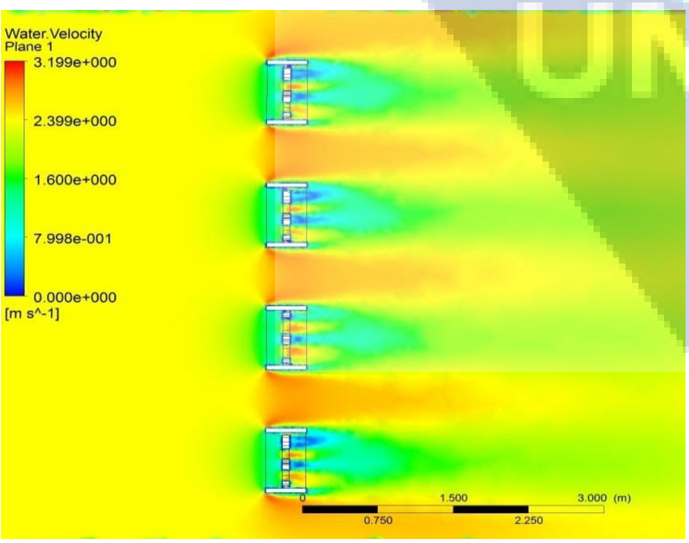


Fig. 8 The effect of velocity flow in MHP turbine with parallel arrangement at 645mm spacing size

Fig. 8 shows the color changes represent the variations of velocities of river flow throughout the MHP turbine body. The velocity decreases as the flow pass through the MHP turbine. However, the velocity changes are consistent in every spacing size for parallel arrangement.

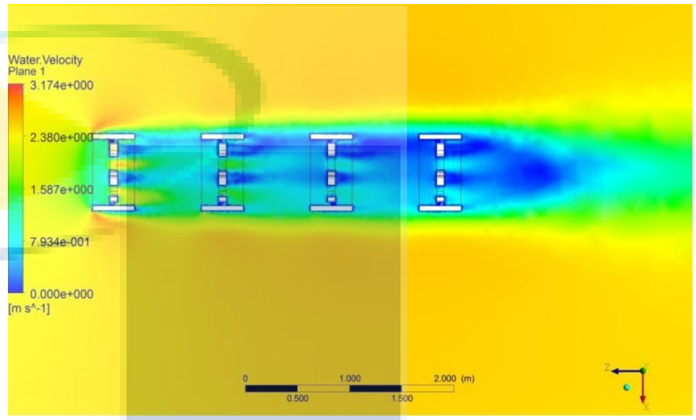


Fig. 9 The effect of velocity flow in MHP turbine with series arrangement at 645mm spacing size

Fig. 9 shows the effect on variations of the velocities in river flow throughout the MHP turbine body. The large drop of velocity occurs as the water pass through the MHP turbines. The water flow almost becomes stagnant. The same flow condition occurs in the other spacing sizes.

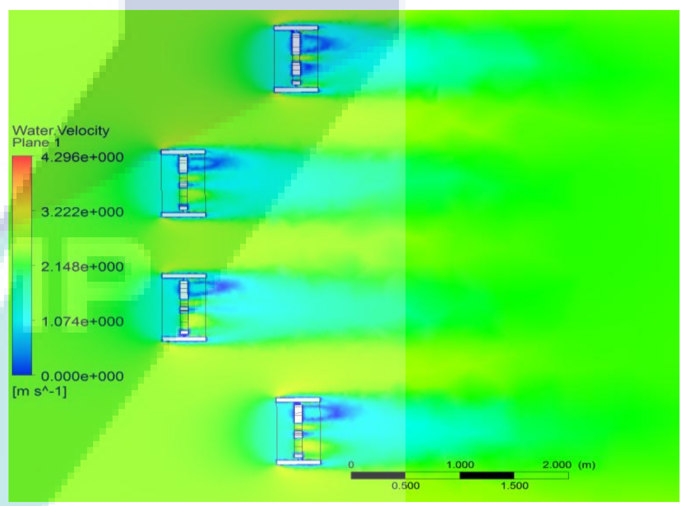


Fig. 10 The effect of velocity flow in MHP turbine with triangular arrangement at 645mm spacing size

Fig. 10 shows the effect of velocities in river flow throughout the MHP turbine body. The flow shows the decrement in velocity as it pass through the MHP turbines. Besides, as the spacing sizes increase, the average velocities increase except for 0mm spacing size, which shows the highest average velocity.

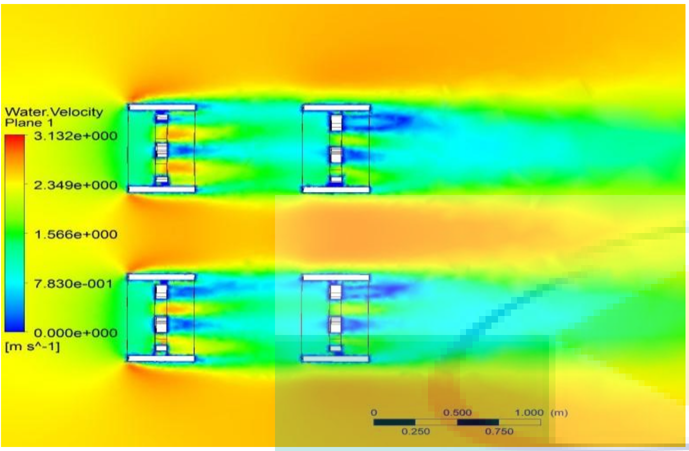


Fig. 11 The effect of velocity flow in MHP turbine with square arrangement at 645mm spacing size arrangement

Fig. 11 shows velocity contours in river flow throughout square arrangements of MHP turbine body. The upstream flow has been truncated by the downstream turbine and the velocity continuously dropped as the flow pass through the MHP turbines. Besides, as the spacing sizes increase, the average velocities increase.

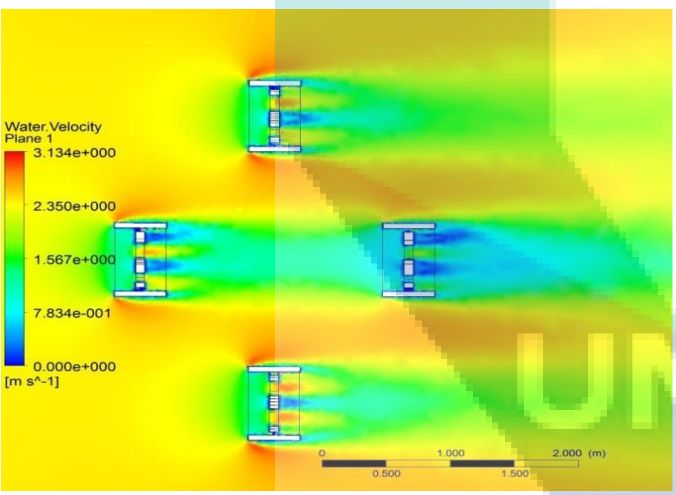


Fig. 12 The effect of velocity flow in MHP turbine with rhombus arrangement at 645mm spacing size arrangement

As shown in Fig. 12, the highest average velocity reached at 0mm spacing size which later it decreases as the flow pass through the rhombus arrangement of MHP turbines. As the spacing sizes increase, the average velocities of the flow increase gradually.

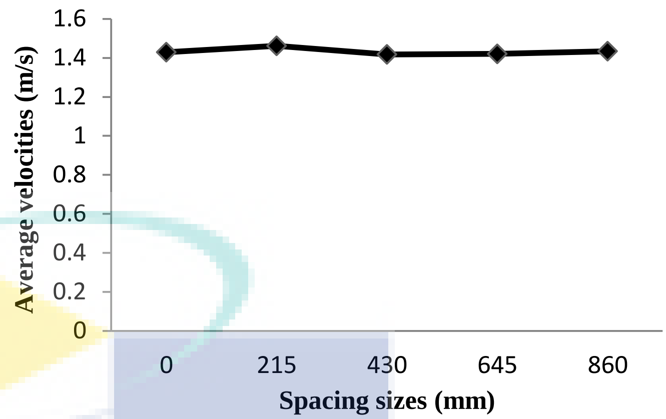


Fig. 13 The effect in average velocities when spacing sizes were increased for parallel array arrangement

Fig. 13 shows the effect of average velocities as the spacing sizes between the MHP turbines in parallel array arrangements were increased. The average values of the velocity at the mouth of the turbine bodies for all the parallel array arrangement were almost equal in every spacing size. Hence, the parallel arrangement is not significant enough to generate energy.

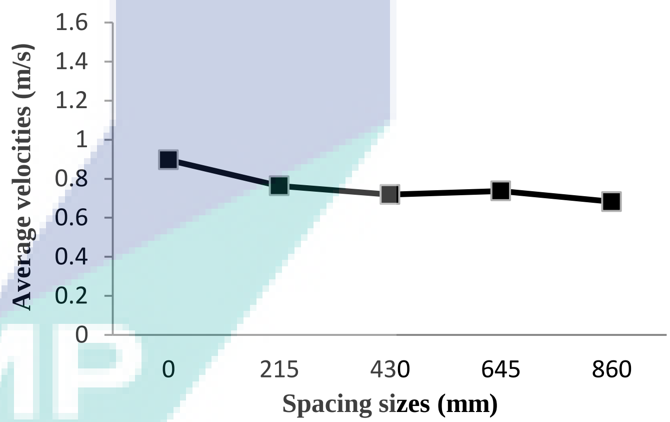


Fig. 14 The changes in average velocities when spacing sizes were increased for series array arrangement

Fig. 14 shows the effect of average velocities as the spacing sizes between the MHP turbines in series array arrangement were increased. The average velocities of the series array arrangement decrease when the spacing sizes increase. Besides, the values of the velocities almost reach 0 m/s as the water flow from Turbine 1 to Turbine 4. This shows that the water flow become stagnant inside the turbine bodies. Hence, this series arrangement is not significant enough to generate energy.

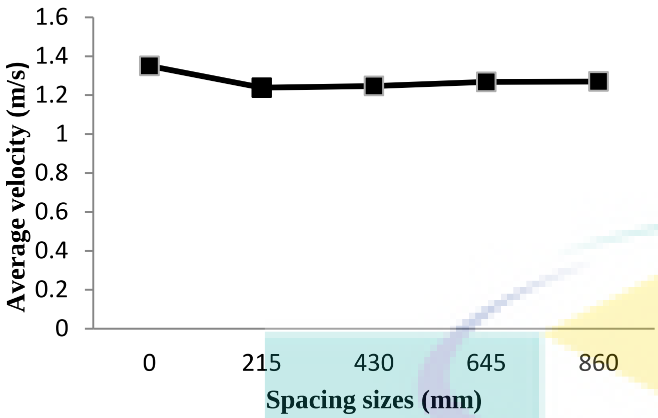


Fig. 15 The changes in average velocities when spacing sizes were increased for triangular array arrangement

Fig. 15 shows the effect of average velocities as the spacing sizes between the MHP turbines in triangular array arrangements were increased. For 0 mm spacing size, the average velocity was the highest compared to other spacing sizes. This proves that when MHP turbines are close to each other, the kinetic energy from the water flow can generate more energy. However, at 215mm, the average velocity decreases compared to 0mm. Then, the average velocities increases from 215mm until 860mm spacing sizes. Hence, this triangular arrangement is significant to generate energy.

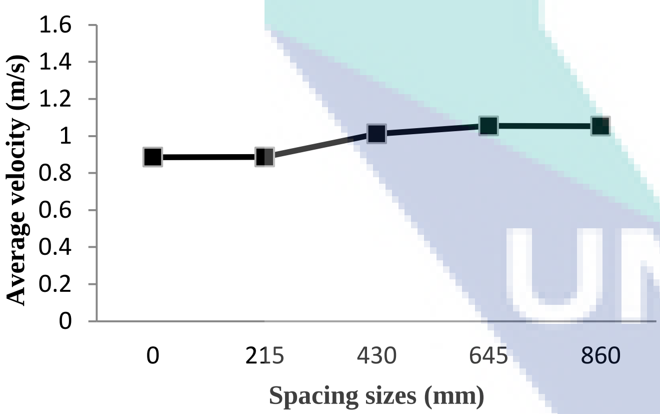


Fig. 16 The changes in average velocities when spacing sizes were increased for square array arrangement

Fig. 16 shows the effect of average velocities as the spacing sizes between the MHP turbines in square array arrangements were increased. The average velocities increase when the spacing sizes increase at the spacing sizes of 215 mm until 860mm. However, the average velocity does not have much change from 0mm until 215mm spacing sizes. Hence, this square arrangement is significant to generate energy because average velocity increases as the spacing sizes increased.

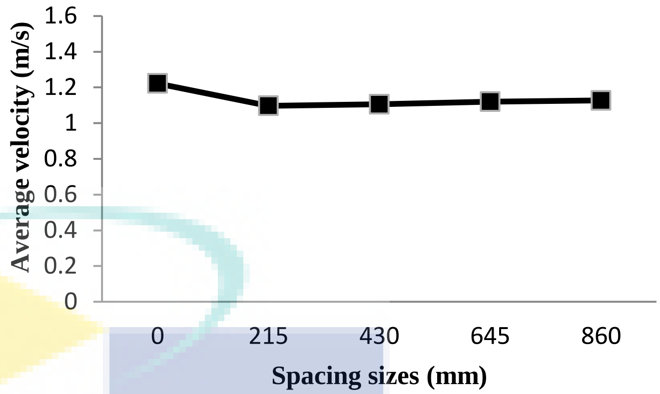


Fig. 17 The changes in average velocities when spacing sizes were increased for rhombus array arrangement

Fig. 17 shows the effect of average velocities as the spacing sizes between the MHP turbines in rhombus array arrangements were increased. For 0mm spacing size, the average velocity was the highest compared to other spacing sizes. This proves that when MHP turbines are close to each other, the kinetic energy from the water flow can generate more energy. However, the average velocity at 215mm was the lowest. From 215mm until 860mm spacing sizes, the average velocities of the rhombus array arrangement increases. Hence, this rhombus arrangement is significant to generate energy.

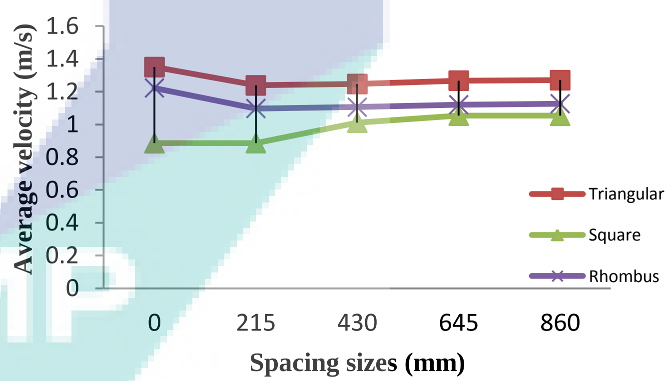


Fig. 18 The comparison of average velocities when spacing size increases between triangular, square and rhombus arrays arrangements

From Fig. 18, triangular, square and rhombus array arrangements of MHP turbines were compared since they were significant to generate energy from the river water flow according to the results above. As the spacing sizes increase, the average velocities at the mouth of the MHP turbines increases except for certain spacing sizes. At 0 mm spacing size for the entire array arrangements, the average velocity was the highest for triangular and rhombus arrays arrangements. The average velocities of triangular arrangement were the highest compared with rhombus and square arrangements. This shows that triangular arrangement is more significant to generate energy from kinetic energy of the water flow; however, the rhombus and square arrangements are still applicable to generate energy.

IX. Conclusions

From the study by using the array arrangements, it was proven that triangular, square and rhombus array arrangements as the significant arrangements to generate power, and the triangular arrangement was the best since the average velocity through the MHP turbines was the highest. The power was generated by extracting kinetic energy from multiple MHP from free-flowing stream of the water. However, the parallel and series array arrangements were not significant to generate power. The parallel array arrangement does not show significant changes in average velocities as the spacing sizes increases, meanwhile, the water flow series array arrangement almost become stagnant. Hence, the triangular, square and rhombus arrangements were significant to generate power but parallel and series arrangements were not significant.

However, since the application of MHP is in rivers the spacing between turbine bodies must be limited so as it would not be the source of obstruction to transports and other uses. Each river geographical properties must be analyzed for the most optimum spacing to harvest highest electrical energy and at the same time allowing various river activities to continue as normal.

References

[1] O. Paish, "Small hydro power: Technology and current status," *Renewable and Sustainable Energy Review,* vol. 6, pp. 537-556. 2002.

[2] P. Cunningham, and B. Atkinson, "Micro hydro power in the nineties," http://www.elements.nb.ca/theme/energy/micro/micro.htm. 1998.

[3] M. E. Harrison, W. M. J. Batten, L. E. Myersand, andA. S. Bahaj,"A comparison between CFD simulations and experiments for predicting the far wake of horizontal axis tidal turbines," in *Proc. 8th European Wave and Tidal Energy Conference*, Uppsala, Sweden, 2009.

[4] L. Bai,R. R. G. Spence, and G. Dudziak, "Investigation of the influence of array arrangement and spacing on tidal energy converter (TEC) performance using a 3-dimensional CFD model," in *Proc 8th European Wave and Tidal Energy Conference*, Uppsala, Sweden, 2009.

[5] T. Divett, R. Vennell, and C. Stevens, "Optimization of multiple turbine arrays in a channel with tidally reversing flow by numerical modeling with adaptive mesh,"*Philosophical Transactions of The Royal Society A (Mathematical Physical & Engineering Sciences,*vol 371, no 1985. 2013.

[6] S. H.Lee, S. H. Lee, K. Jang, J. Lee, and N. Hur, "A numerical study for the optimal arrangement of ocean current turbine generators in the ocean current power parks,"*Current Applied Physics*, vol 10, S136-S141. 2010.

[7] M. J. Churchfield, Y. Li, and P. J. Moriarty,"A large-eddy simulation study of wake propagation and power production in an array of tidal-current turbines," in *Proc. 9th European Wave and Tidal Energy Conference 2011*, Southampton, England, 2011.

[8] M. J. Lowson, Y. Li, and D. C. Sale, "Development and verification of a computational fluid dynamics model of a horizontal-axis tidal current turbine," in *Proc. 30th International Conference on Ocean, Offshore and Arctic Engineering*, Rotterdam, The Netehrlands, 2011.

CHAPTER 4

Parametric Study of Multiple Configurations of Pico Hydrokinetic Turbines Using CFD

MATEC Web of Conferences 74, 00024 (2016)
DOI: 10.1051/matecconf/20167400024

Parametric Study of Multiple Configurations of Pico Hydrokinetic Turbines Using CFD

B. Yogaraja[1] and M. A. At-tasneem[1,2*]

[1]Faculty of Mechanical Engineering, Universiti Malaysia Pahang,
26600 Pekan, Pahang, Malaysia
*Email: tasneem@ump.edu.my
*Phone: +6094246211; Fax: +6094246222
[2]Fellow Researcher, Centre for Earth Resources Research & Management (CERRM),
Universiti Malaysia Pahang, Lebuhraya Tun Razak,
26300 Gambang, Kuantan, Pahang, Malaysia

ABSTRACT

This paper aims to study the river flow characteristics over pico hydrokinetic turbines with variation of arrangement using computational fluid dynamics (CFD) software. This study is required to obtain the optimum spacing and angle between the turbines which leads to higher turbine effective utilisation and performance in terms of power generated. In this study, a river model is created in CFD software to simulate the water flow over the turbines as they are placed in a river to obtain the water flow characteristic. Different types of array arrangements are simulated in the river model. Multiple turbines are used to accumulate more power. The turbine model which consists of eight turbines is arranged in series with different spacing, ranging from a size of diameter (1D) to four times diameter (4D) is simulated to identify the optimum spacing between the turbines. Then, the simulation is continued using a sufficient spacing of 0.5D with angles ranging of 10° to 60° from datum of original position to minimise the disruption of the aquatic environment. The velocity profiles of each turbine are obtained and analysed. The 4D spacing and 40° angle displayed higher average velocities compared to other arrangements. Thus, from this study, the 4D and 40° are deduced as the optimum spacing and angle, respectively.

Keywords: CFD; hydrokinetic turbine; configurations; velocity profile; wake effect.

INTRODUCTION

The term pico hydropower is a small-scale hydropower with power generation of less than 5 kW. It has made significant contributions to remote and off-grid settlements [1]. Installing and operating a run of river application, such as a pico hydrokinetic system is fully environment friendly as it does not alter the environment's natural course. This technology is favourable because of its constant supply of electrical energy and cost effectiveness in terms of low investment and maintenance costs [2]. The turbines can be installed in any flow with a velocity greater than 0.5 m/s [3]. Kinetic energy from the flowing water can be converted into mechanical energy by rotating the turbine blade which then generates electricity.

Most computational researches on array arrangements are conducted for tidal turbines, which is comparable to run of river turbines [4]. Computational fluid dynamics (CFD) models are used to study and analyse wake effects. The simulations of the turbines are done using the boundary conditions as employed by [4, 5] to obtain the wake characteristics and comparing it later with experimental data.

A wake is the region of disturbed flow or recirculating flow immediately behind a stationary or moving solid body, caused by the fluid flowing around the body. The flow in this trail is slowed down and quite turbulent when compared with the flow arriving in front of the turbine. In incompressible fluids such as water, it results in a wave. Similar to all wave forms, it then spreads outward from the source until the energy it possesses is overcome or lost, normally by friction or dispersion [6].

According to A.S Bahaj et. al. [7], near wake may take the form of vortices shed from the tip of the blades or from the support structure and will bound the slower moving flow from the stream. A discontinuity in the stream velocity profile is created by these vortices. Usually the near wake exists from 0D - 3D/4D, in which time the surrounding turbulence of the free flowing stream breaks down the bounding vortices.

Based on the study conducted by Wu et. al. [8], the formation and recovery of turbine wakes are known to be affected by the incoming flow characteristics, the operational states of a turbine and the turbine-generated turbulence characteristics. Both numerical and experimental studies of single turbine wakes in turbulent boundary layer flows has shown that the turbulence intensity level in the incoming flow can have a great effect on the wake recovery rate. A.S Bahaj et. al. [7] mentioned that the faster the free stream flows, the greater the convection of the wake. This means that the wake is carried further downstream in the same period of time. Therefore, the speed of the free stream is a key parameter in analysing the wake velocity.

Study by Churchfield et. al. [9] reveals that the way in which the turbulent inflow is simulated greatly affects the wake propagation and power production of the array flow. From the study, it shows that the low speed wakes of the first row of turbines have an effect upon the second row. Hence, the wakes of the second row of turbines exhibit low stream velocities. The second row of turbine wakes also appears to contain more turbulent motions and meander significantly.

The work of Bai et. al. [5] presents the relative effect of increasing the inter row spacing and row spacing between TEC turbines. The results display that the array with larger width spacing shows the fluid entering the central TEC to be less influenced by the wakes from the upstream TEC row. Thus, this variation in performance highlights that there is an opportunity to modify spacing to fully optimise array performance.

The objectives of this study are to create a river model using CFD software and to simulate the water flow over multiple hydrokinetic turbines over different array arrangements to obtain the water flow characteristics. The aim is to obtain the optimum spacing and angle between the turbines which leads to higher turbine effective utilisation and performance in terms of power generated.

MATHEMATICAL FORMULATIONS

Governing Equations

In this study, the flow field is assumed to be incompressible, steady, non-isothermal and two-dimensional (2D) flow. Therefore, the governing equations for the continuity, momentum and energy can be expressed as [10]:

Continuity equation for incompressible flow:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad (1)$$

$$\vec{\nabla}\cdot\vec{V}=0 \quad (2)$$

where u, v and w are velocity components in the *x*, *y* and *z* directions, respectively, and $\vec{\nabla}$ is divergence operator.

Momentum equation:

$$\rho\frac{D\vec{V}}{Dt}=\rho\left[\frac{\partial\vec{V}}{\partial t}+\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V}\right]=-\vec{\nabla}p+\rho\vec{g}+\mu\nabla^2\vec{V} \quad (3)$$

where ρ is density of the fluid (kg/m^3), V is velocity vector of the fluid (m/s), t is time (seconds), g is gravitational acceleration (m/s^2) and μ in the fluid viscosity (kg/m.s)

Conservation of energy equation:

$$\rho\frac{DE}{Dt}=\frac{\partial(u\sigma_{xx})}{\partial x}+\frac{\partial(v\sigma_{yy})}{\partial y}+\frac{\partial(w\sigma_{zz})}{\partial z}+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(v\tau_{zx})}{\partial z}+\frac{\partial(v\tau_{xy})}{\partial x}+\frac{\partial(v\tau_{zy})}{\partial z}$$
$$+\frac{\partial(w\tau_{xz})}{\partial x}+\frac{\partial(w\tau_{yz})}{\partial y}-\frac{\partial q_x}{\partial x}-\frac{\partial q_y}{\partial y}-\frac{\partial q_z}{\partial z} \quad (4)$$

where σ is stress and τ is shear stress.

Kinetic energy (KE) is the energy of motion. The kinetic energy of an object is the energy it possesses because of its motion. Kinetic energy equation is given as:

$$KE=\frac{1}{2}\,m\,v^2 \quad (5)$$

where *KE* represents kinetic energy (J), *m* is mass (kg) and *v* is the velocity (m/s).

The kinetic energy can be converted into mechanical energy. Kinetic energy from the flowing water can be converted into kinetic energy of rotating shaft by rotating the turbine blade. The formula below shows that velocity is directly proportional to the angular velocity:

$$v = r\,\omega \tag{6}$$

where r is the radius (m) and ω is the angular velocity (rad/s).

The kinetic energy of a rotating object is analogous to linear kinetic energy and can be expressed in terms of the moment of inertia and angular velocity. For a given fixed axis of rotation, the rotational kinetic energy can be expressed in the form:

$$KE = \frac{1}{2}\, I\,\omega^2 \tag{7}$$

where I is the moment of inertia or rotational inertia (kg.m^2).

If a constant torque rotates a body through an angle θ, the work done is:

$$W = T\,\theta \tag{8}$$

where W is the work done (J), T is the torque (J/rad) and θ is the angle (rad).

The power of the rotating turbine blade is given by:

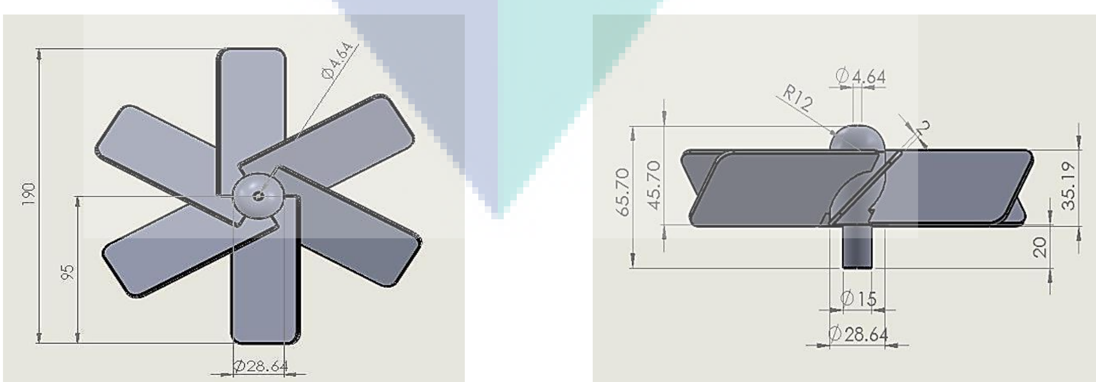
$$P = T\,\omega \tag{9}$$

where P is power in watts (W).

NUMERICAL ANALYSIS

Turbine and Blade Design

The dimensions of the turbine blade are shown in Figure 1. All values are in millimetres (mm).



(a) Front view (b) Side view

Figure 1. Dimensions of the blade

The blade is assembled to the turbine body as shown in Figure 2. For this study, eight identical turbines were used. The turbines were fitted through a shaft as shown in Figure 3.

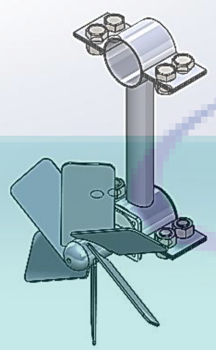


Figure 2. Turbine body

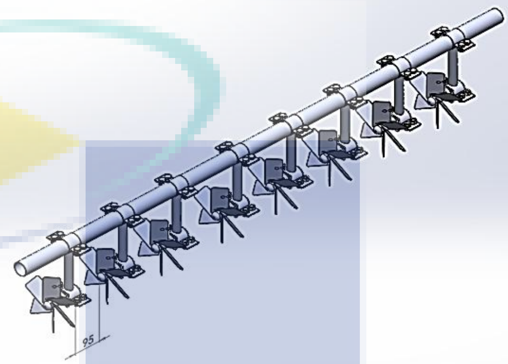


Figure 3. Full turbine arrangement

Computational Domain

The CFD software used is ANSYS 15.0 and the flow is studied using the Fluid Flow (CFX) analysis system. A river domain was set up using a box as an enclosure. The turbines were placed within the enclosure with the desired array arrangements of spacing and angle. Eight turbines are used for the simulation of different spacing while four turbines are used for the simulation of different angles.

Meshing

Fine mesh size is used on the model for every array arrangements. The number of element counts differs according to the spacing and angle between the turbines.

Boundary Conditions

For this study, the river flow was set as a steady flow fluid. As it is an open-channel flow, the fluid is composed of air (at the top) and water. However, the presence of air was neglected. The inlet velocity is 2.0 m/s with equal upstream and downstream heights. Hence, the gradient of the river is zero. The upstream height is the height of water level of the river at the inlet and the downstream height is the height of the river water level at the outlet. The boundary conditions used in the simulation are summarised in Table 1.

Table 1. Boundary conditions of river model

Boundaries	**Parameter**
Inlet	Velocity = 2.0 m/s
Outlet	Constant static

	pressure
Top surface	Zero pressure
Sidewall	No slip wall
Bottom	No slip wall

Model Configuration

There are ten different array arrangements used for this study. The simulations were conducted on eight turbines which are arranged in series, as shown in Figure 4 and tested with different spacing, x. Four tests were conducted, in which x is varied from 1D to 4D spacing, with an increment of 1D consecutively. D represents the diameter of the turbine blade, where D equals to 190 mm.

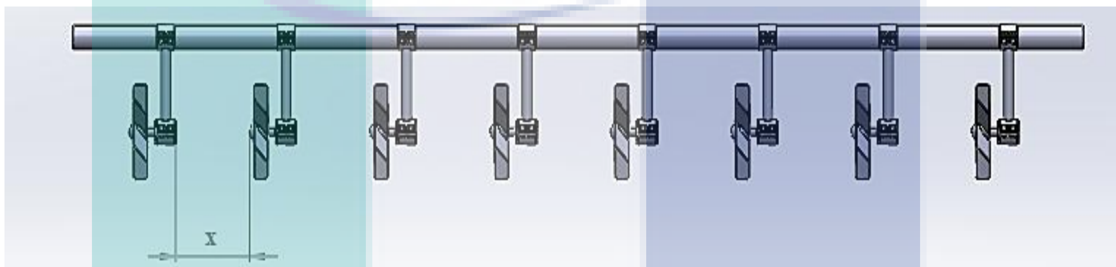


Figure 4. Turbines arranged in series

The simulation is continued by using four turbines with 0.5D spacing and tested with different angles, θ. Angles, as in shown in Figure 5 is introduced with the aim to minimise the aquatic environment. Six tests were conducted, where θ is varied from 10° to 60°, with an increment of 10° consecutively.

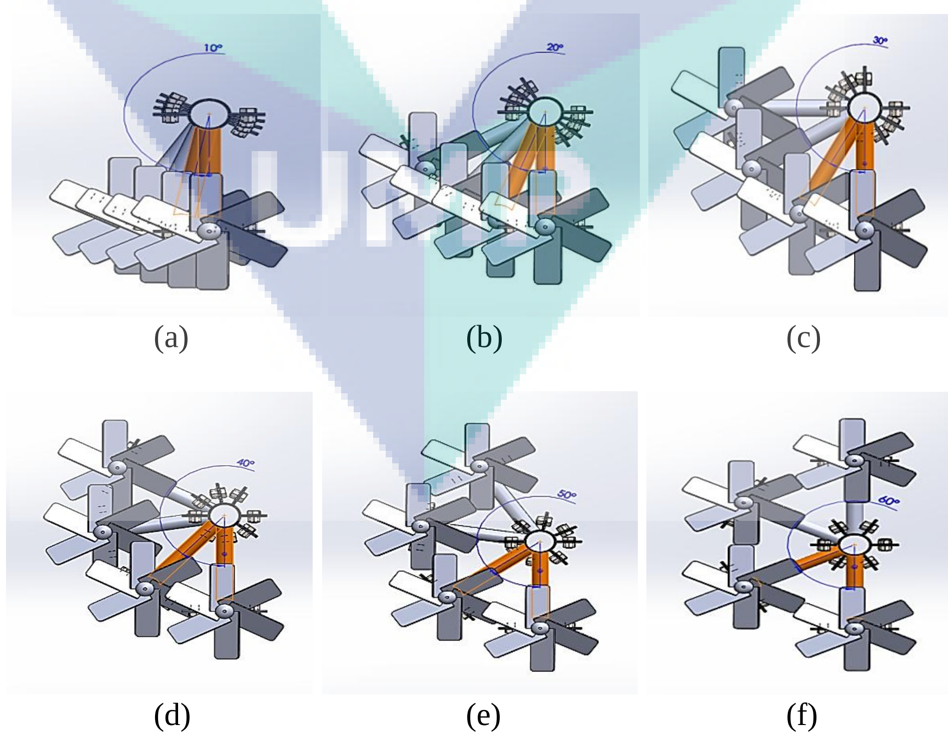


Figure 5. Turbines arranged under the influence of angles;
(a) 10°; (b) 20°; (c) 30°; (d) 40°; (e) 50°; (f) 60°

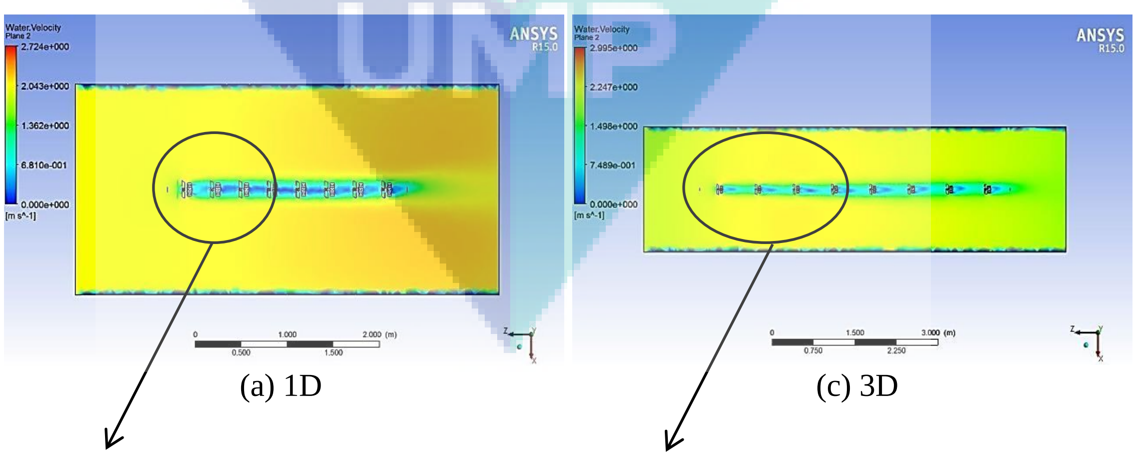
RESULTS AND DISCUSSION

The CFD simulations and analysis were conducted to study the river water flow through the multiple hydrokinetic turbines of different spacing and angles. A comparison data is conducted to present the differences of velocity of each turbine in every arrangement. These simulations were carried out with a constant mean value of river velocity of 2.0 m/s throughout ten different array arrangements with consists of four different spacing (1D, 2D, 3D and 4D) and six different angles (10°, 20°, 30°, 40°, 50° and 60°).

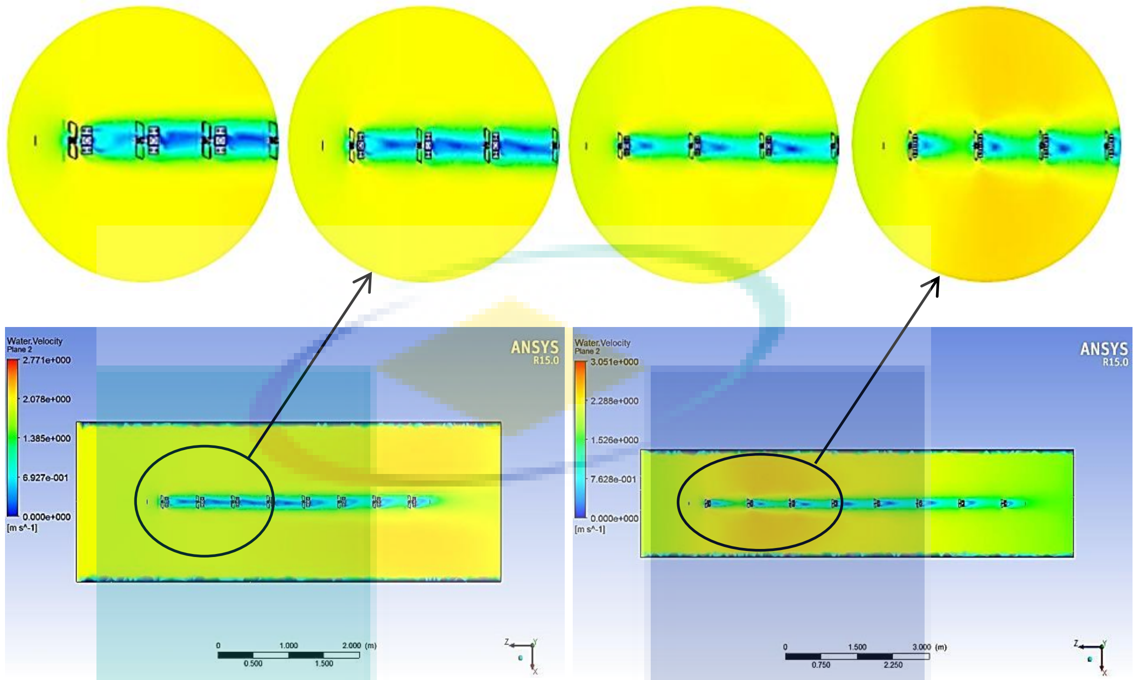
For the velocity plots, velocity values at the entrance of the turbine were extracted from the line drawn across the mouth of the turbines. The values for all array arrangements were plotted for analysis. The higher the value of velocity at the turbine mouth, the higher the performance of the turbine. This is due to the higher kinetic energy in the flow. The average velocity for each array arrangement is calculated to determine the best arrangement to generate maximum power.

Spacing

Figure 6 displays the changes in velocities of the river water as it flows throughout the turbines. It shows that the velocity of the flow decreases as it passes through the turbines. Figure 6(a) shows a large drop in velocity occurs as the water passes through the turbines of 1D spacing and the velocity remains low at the mouth of each turbine. However, as the spacing increases, the water velocity starts to recover slowly as it flows over a distance. Figure 6(b) shows a slight increase in velocity as well as in Figure 6(c), which exhibits a higher increase in velocity. Based on the four tests, the 4D spacing arrangement in Figure 6(d) shows the highest velocity at the mouth of each discrete turbine.



(a) 1D (c) 3D

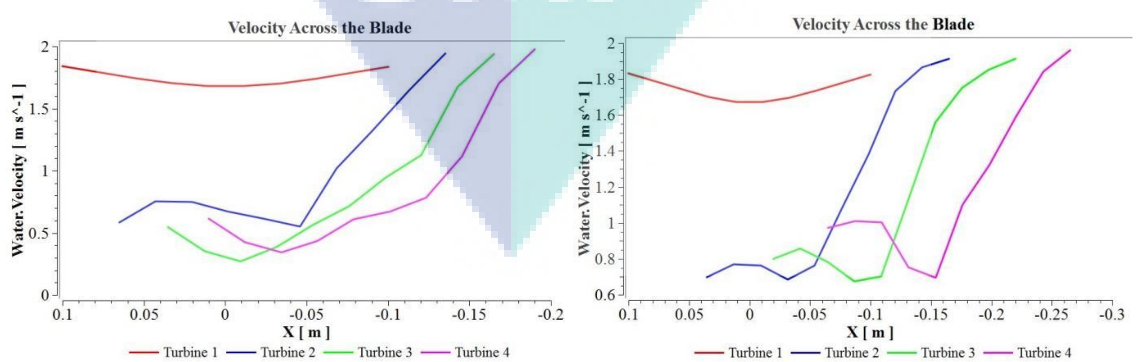


(b) 2D (d) 4D

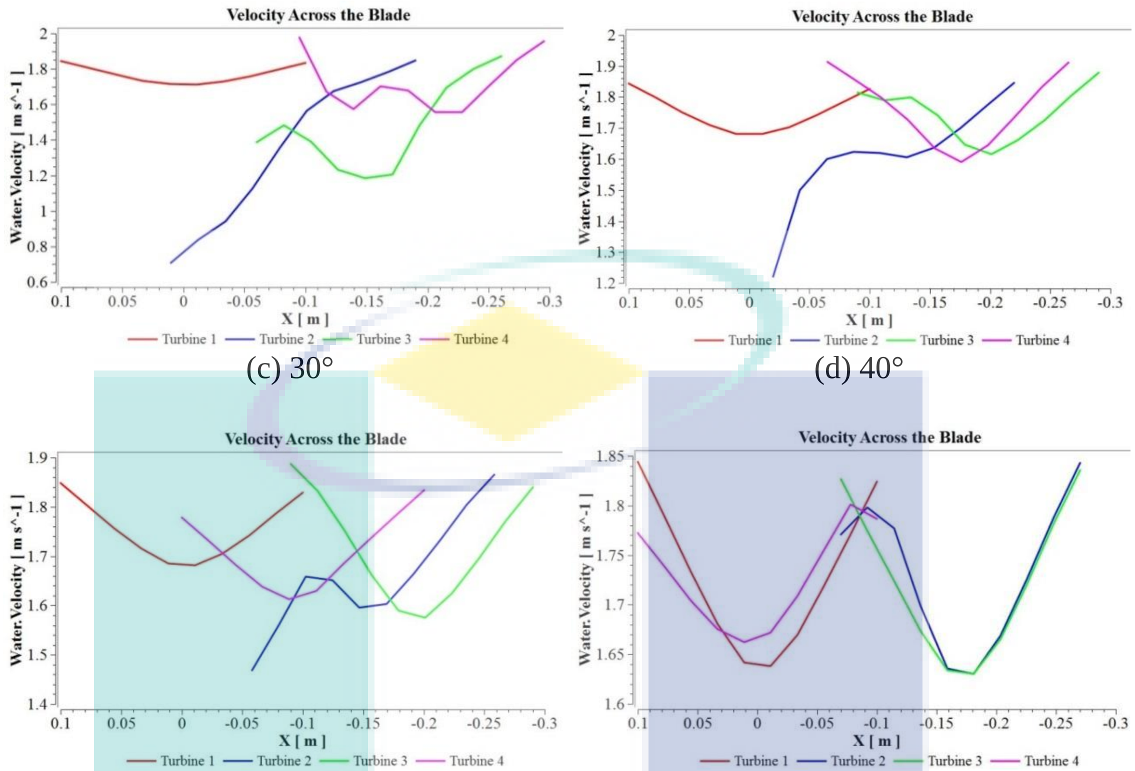
Figure 6. Velocity planes from simulation results of turbines with different spacing

Angle

Figure 7 displays the arrangement of turbines with different angles and its velocity profiles. Figure 7(a) shows a large drop in velocity occurs as the water passes through the turbines with 10° angle and the velocity at the mouth of each discrete turbine remains low. However, as the angle increases, the velocity at the mouth of each discrete turbine also increases. Based on the six tests, it can be concluded that when the angle increases, the position of the turbine moves away from the low velocity region that forms behind each turbine. Hence, Figure 7(f) with the 60° angle arrangement exhibits the highest velocity.



(a) 10° (b) 20°



(c) 30° (d) 40°

(e) 50° (f) 60°

Figure 7. Velocity plots from simulation results of turbines with different angles

Comparison Between Arrangements

Based on the tests conducted, the average velocity of each spacing and angle arrangement is obtained from the velocity plot and the results are tabulated in Table 2 and Table 3 respectively.

Table 2. Average velocity respective to their spacing

Spacing (mm)	Average velocity (m/s)
190 (1D)	1.022
380 (2D)	1.156
570 (3D)	1.292
760 (4D)	1.458

Table 3. Average velocity respective to their angle

Angle (°)	Average velocity (m/s)
10	1.318
20	1.417
30	1.588
40	1.699
50	1.722
60	1.736

Figure 13 shows the effect of spacing onto the average velocity at the mouth of each discrete turbine. In the series arrangement of the turbines, the average velocity increases as the spacing increases. Based on the graph, the 760 mm (4D) spacing arrangement shows the highest average velocity at the mouth of each discrete turbine. Hence, the larger the spacing between the turbines, the higher the velocity obtained. Therefore, larger spacing is needed to generate maximum power.

Figure 14 shows the effect of angle onto the average velocity at the mouth of each discrete turbine. The average velocity increases as the angle increases. Based on the graph, the 60° angle arrangement shows the highest average velocity at the mouth of each discrete turbine. However, it can be noticed from the graph that the angle of 40° is able to achieve higher velocity compared to smaller angles and the velocity at angle greater than 40° does not increase much and remains about the same. Therefore, angle greater than 40° is needed to generate maximum power.

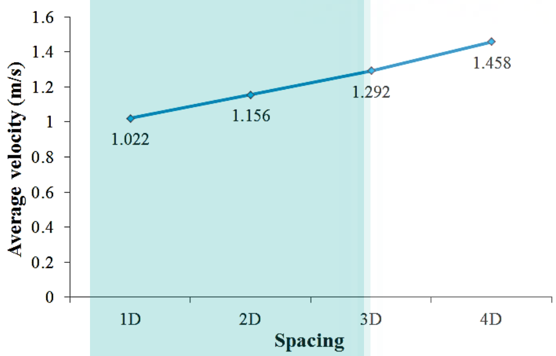


Figure 13. Graph of average velocity versus spacing

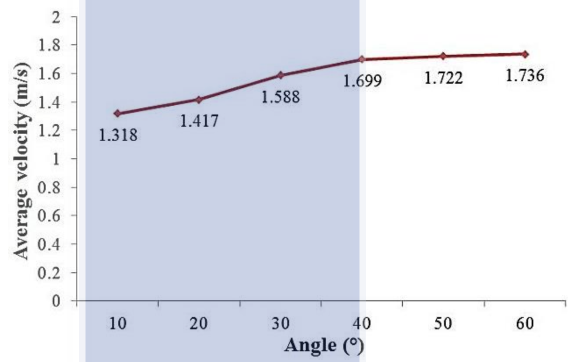


Figure 14. Graph of average velocity versus angle

CONCLUSIONS

In this study, the effect of velocity flow over different array arrangements could be studied and analysed. Based on the simulation results obtained, it is proven that for a series arrangement of turbines, the larger the spacing between the turbines, the higher the velocity obtained. High velocity is necessary to generate maximum power. Small spacing does not show signs of good performance due to its low average velocity throughout the turbines. Based on the tests conducted on the different spacing, the results show that the spacing of 4D exhibits the highest average velocity. Therefore, it is concluded as the optimum spacing between the turbines.

However, the application of hydrokinetic turbines in rivers might be restricted to the aquatic environment available at a particular area. Therefore, when taking that into consideration, angle spacing should be considered in the turbine arrangement to minimise the area involved for the installation of the turbines. From the tests conducted on different angles, 60° angle shows the highest velocity obtained. However, it is found that the angle of 40° is sufficient enough to be considered as the optimum angle where maximum power can be generated as it is capable in achieving higher velocity compared to smaller angles. Therefore, angles greater than 40° are applicable.

All discussions and results obtained are based on software simulation only. Due to the limitations of the study, the effect of the rotation of the turbine blades to the water flow is not considered. Hence, the results of this study should not be applied thoroughly without considering other possible factors when dealing with fluid flow.

For future application of hydrokinetic turbines, spacing and angle should both be accounted in order to achieve maximum power generation and to overcome environmental limitations as well as minimising the disruption of aquatic environment.

ACKNOWLEDGEMENTS

The authors would like to thank Centre for Earth Resources Research & Management (CERRM), Universiti Malaysia Pahang for providing financial assistance under project no. RDU140345.

REFERENCES

1. Kamaruzzaman, S. and A.R. Juhari, *Pico Hydro: Clean Power From Small Streams.* Proceedings of the 3rd WSEAS Int. Conf. on Renewable Energy Sources, 2009: p. 1-6.
2. Kapoor, R., *Pico Power: A Boon for Rural Electrification.* Advance in Electronic and Electric Engineering 2013. **3**: p. 865-872.
3. Kari, S., *Small-scale Water Current Turbines for River Applications.* 2010: p. 4.
4. At-tasneem, M.A., et al., *Numerical Simulation of Multiple Arrays Arrangement of Micro Hydro Power Turbines.* World Academy of Science, Engineering and Technology International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering, 2014. **8**: p. 966-972.
5. Bai, L., R.R.G. Spence, and G. Dudziak, *Investigation of the Influence of Array Arrangement and Spacing on Tidal Energy Converter (TEC) Performance using a 3-Dimensional CFD Model.* Proceedings of the 8th European Wave and Tidal Energy Conference, Uppsala, Sweden, 2009: p. 654-660.
6. Ximo, C.O.M., *Wake*. 2012: Ject Press.
7. Bahaj, A.S., et al., *Characterising the wake of horizontal axis marine current turbines.* Proceedings of the 7th European Wave and Tidal Energy Conference, Porto, Portugal, 2007.
8. Wu, Y.T. and P. Fernando, *Atmospheric Turbulence Effects on Wind-Turbine Wakes: An LES Study*. 2012: p. 1-23.
9. Churchfield, M.J., Y. Li, and P.J. Moriarty, *A Large-Eddy Simulation Study of Wake Propagation and Power Production in an Array of Tidal-Current Turbines*. 2011: p. 1-10.
10. Cengel, Y.A. and J.M. Cimbala, *Fluid Mechanics: Fundamentals and Applications*. 2013, New York: McGraw-Hill.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

Upon project completion, the result is very convincing and the objective is achieved. It is found that the microhydro power is successfully developed and tested, both experiment and throughout CFD simulation

5.2 Recommendation

It is recommended to further this study up to bigger scale as for smaller scale, the quantity of electricity is not enough to support even for a single home application.

REFERENCES

1. Zhixiao Ma, Bing Xue, Yong Geng, Wanxia Ren, Tsuyoshi Fujita, Zilong Zhang, Jose A. Puppim de Oliveira, David A. Jacques and Fengming Xi, "Co-benefit analysis on climate change and environmental effect of wind-power: A case study from Xianjiang, China," Renewable Energy, vol 57, pp 35-42. 2013.
2. Ahmed M.A. Haidar, Mohd F.M. Senan, Abdulhakim Noman, Taha Radman, "Utilization of pico hydro generation in domestic and commercial loads," Renewable and Sustainable Energy Review, vol 16, pp 518-524. 2011.
3. M. E. Harrison, W. M. J. Batten, L. E. Myersand, and A. S. Bahaj,"A comparison between CFD simulations and experiments for predicting the far wake of horizontal axis tidal turbines," in Proc. 8th European Wave and Tidal Energy Conference, Uppsala, Sweden, 2009.
4. L. Bai, R. R. G. Spence, and G. Dudziak, "Investigation of the influence of array arrangement and spacing on tidal energy converter (TEC) performance using a 3-dimensional CFD model," in Proc 8th European Wave and Tidal Energy Conference, Uppsala, Sweden, 2009.
5. T. Divett, R. Vennell, and C. Stevens, "Optimization of multiple turbine arrays in a channel with tidally reversing flow by numerical modeling with adaptive mesh," Philosophical Transactions of The Royal Society A (Mathematical Physical & Engineering Sciences, vol 371, no 1985. 2013.
6. S. H. Lee, S. H. Lee, K. Jang, J. Lee, and N. Hur, "A numerical study for the optimal arrangement of ocean current turbine generators in the ocean current power parks," Current Applied Physics, vol 10, S136-S141. 2010.
7. Johnson V, "Fluid flow in pico hydro power plant", 2007.
8. S. J. Williamson, B. H.Stark, J. D. Booker, "Low head pico hydro turbine selection using a multi-criteria analysis," Renewable Energy, vol 61, pp 43-50. 2013.
9. C. Beth Robertson, Mathew T. Langford, David Z. Zhu, and Alf Leake, "Reservoir Thermal Structure and its effect on Hydropower Operation Induced Fish Entrainment," in Proc. of World Environmental and Water Resources Congress 2011: Bearing Knowledge for Sustainability, 2011.
10. At-Tasneem M. A.1, Wan Mohamad Azam and Ummu Jamaludin, "A Study on the Effect of Flow Rate on the Power Generated by a Pico hydro Power Turbine" in Proc. of International Conference on Innovation Challenges in Multidisciplinary Research & Practice, 2013.

LIST OF OUTPUT

1. PATENT

NIL

2. PRODUCT

NIL

3. PAPER

– 1 Journal Paper

–2 Conference Paper

4. POST GRADURATE STUDENT

NIL