CFD SIMULATION OF MULTIPLE ARRAY ARRANGEMENT OF MICRO HYDRO POWER TURBINES

THIWAAN RAO S/O NARASIMMA NAIDU

Thesis submitted in partial fulfilment of the requirements for the awards of Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2013

ABSTRACT

Micro Hydro Power (MHP) turbines were used to generate energy with a water flow. This energy is useful to the mankind. Computational Fluid Dynamics (CFD) software was used to simulate the water flow over MHP turbines as they are placed in a river domain. Multiple arrays arrangement of MHP turbines lead to generate large amount of energy. It is rare to find CFD simulation of MHP turbines. In this study, a river model was created and simulated in CFD software to obtain the water flow characteristic over MHP turbine bodies. 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 different array arrangements were used which are parallel, series, triangular, square and rhombus with different spacing sizes. The velocity profiles on each MHP turbines were identified at the mouth of turbine body. In this study, the triangular, square and rhombus arrangements were significant to generate power because their average velocities increase as the spacing sizes increase but parallel and series arrangements were not significant.

ABSTRAK

Turbin Penjana Micro Hydro (MHP) telah digunakan untuk menjana tenaga dengan aliran air. Tenaga ini amat berguna kepada manusia. Perisian Computational Fluid Dynamics (CFD) telah digunakan untuk mensimulasikan aliran air keatas turbin MHP semasa diletakkan di dalam domain sungai. Barisan susunan pelbagai turbin MHP akan membawa kepada penjanaan tenaga dalam kuantiti yang maksimum. Dalam kajian ini, model sungai telah dibentuk dan disimulasi dalam perisian CFD untuk mendapatkan ciri-ciri aliran air keatas badan turbin MHP. Proses ini kemudian diteruskan dengan simulasi pelbagai jenis susunan turbin yang berbeza dalam model sungai tersebut. Satu model turbin MHP terdiri daripada badan luar turbin dan bilah kipas statik di dalamnya. Lima jenis susunan turbin yang berbeza telah digunakan iaitu susuan selari, bersiri, segitiga, empat persegi dan rombus dengan saiz jarak yang berbeza antara turbin-turbin tersebut. Profil halaju pada setiap turbin MHP telah dikenal pasti di hujung badan turbin. Dalam kajian ini, susunan segitiga, empat persegi dan rombus dipilih untuk menjana tenaga kerana purata halaju meningkat apabila saiz jarak meningkat tetapi susunan selari dan bersiri tidak ketara untuk menjana tenaga.

TABLE OF CONTENTS

SUPERVISOR'S DECLARATION		ii
STUDENT'S DECLARATION		iii
DEDICATION		iv
ACKNOWLEDGEMENT		v
ABSTRACT		vi
ABSTRAK		vii
TABLE OF CONTENT		viii
LIST OF TABLES		xi
LIST OF FIGURES		xii
LIST OF SYMBOLS		xviii
LIST OF ABBREVIATIONS		xix
CHAPTER 1	**INTRODUCTION**	**1-4**
1.1	Project Background	1
1.2	Problem Statement	2
1.3	Objectives	3
1.4	Project Scope	3
1.5	Expected Results	4
CHAPTER 2	**LITERATURE REVIEW**	**5-15**
2.1	Introduction	5
2.1	Computational Fluid Dynamics (CFD)	6
2.3	Governing Equations	8
	2.3.1 Navier-Stokes Equation	8
	2.3.2 Kinetic Energy Equations	8
	2.3.3 Continuity Equations	9
	2.3.4 Momentum Equation	10
	2.3.5 Energy Equation	12

2.4	Simulation Methods	14
2.5	Boundary Conditions	15
	2.5.1 Velocity of Pahang River	15
CHAPTER 3	**METHODOLOGY**	**16-26**
3.1	Introduction	16
3.2	Flow Chart	16
3.3	Gantt Chart	16
3.4	Dimension of Turbine Body and Blade	17
	3.4.1 Designing the Turbine Body and Blade	17
3.5	Model Description	20
	3.5.1 Overview	20
	3.5.2 Computational Domain	20
	3.5.3 Mesh	21
	3.5.4 Boundary Conditions	22
3.6	Model Configurations	23
	3.6.1 Spacing Sizes between all the Array Arrangements	25
	3.6.2 Calculations of the Spacing Sizes	26
CHAPTER 4	**RESULTS & DISCUSSION**	**27-60**
4.1	Introduction	27
4.2	Analysis of Velocity Profiles	28
	4.2.1 Parallel Array Arrangement	28
	4.2.2 Series Array Arrangement	34
	4.2.3 Triangular Array Arrangement	41
	4.2.4 Square Array Arrangement	47
	4.2.5 Rhombus Array Arrangement	53
4.3	Comparison between the significant arrays arrangements	60
CHAPTER 5	**CONCLUSION & RECOMMENDATIONS**	**61-62**
5.1	Introduction	61
5.2	Conclusions	61

5.3	Recommendations	62
REFERENCES		**63-64**
APPENDICES		**65-97**
A	Flow Chart and Gantt Chart	66
B	Different Array Arrangements with Different Spacing Sizes	70
C	Velocity Profiles for Each Arrangements	83

LIST OF TABLES

Table No.	Title	Page
2.1	Simulation Method by Authors	14
2.2	Data Obtained Regarding Velocity of Pahang River	15
3.1	Dimensions of Turbine Body	18
3.2	Mesh sizes for different array arrangement	21
3.3	Boundary Conditions	22
3.4	Expressions used for River Simulations	22
3.5	Spacing Sizes	25

LIST OF FIGURES

Figure No.	Title	Page
2.1	The different discipline contained within computational fluid dynamics	7
2.2	The three basic approaches to solve problem in fluid dynamics and heat transfer	7
2.3	The conversion of mass in an infinitesimal control volume of a fluid flow between stationary parallel plates	9
2.4	Surface forces acting on the infinitesimal control volume for the velocity component *u*. Deformed fluid element due to the action of the surface forces	10
2.5	Work done by surface forces on the fluid within the infinitesimal control volume	11
3.1	The designed turbine body	17
3.2	The designed turbine blade	18
3.3	The assembled turbine body and static blade	19
3.4	The blade must be smaller than the turbine body	19
3.5	Computational domain of parallel arrays with 0% diameter configuration	20
3.6	Example of parallel array arrangement with the water flow direction	23
3.7	Example of series array arrangement with the water flow direction	23
3.8	Example of triangular array arrangement with the water flow direction	24
3.9	Example of square array arrangement with the water flow direction	24
3.10	Example of rhombus array arrangement with the water flow direction	25

4.1	Velocity profile along the four turbines' mouths in parallel arrangement and 0% spacing size	28
4.2	Velocity profile along the four turbines' mouths in parallel arrangement and 50% spacing size	29
4.3	Velocity profile along the four turbines' mouths in parallel arrangement and 100% spacing size	30
4.4	Velocity profile along the four turbines' mouths in parallel arrangement and 150% spacing size	31
4.5	Velocity profile along the four turbines' mouths in parallel arrangement and 200% spacing size	32
4.6	The changes in average velocities when spacing sizes were increased for parallel array arrangement	33
4.7	Velocity profile along the four turbines' mouths in series arrangement and 0% spacing size	34
4.8	Velocity profile along the four turbines' mouths in series arrangement and 50% spacing size	36
4.9	Velocity profile along the four turbines' mouths in series arrangement and 100% spacing size	37
4.10	Velocity profile along the four turbines' mouths in series arrangement and 150% spacing size	38
4.11	Velocity profile along the four turbines' mouths in series arrangement and 200% spacing size	39
4.12	The changes in average velocities when spacing sizes were increased for series array arrangements	40
4.13	Velocity profile along the four turbines' mouths in triangular arrangement and 0% spacing size	41
4.14	Velocity profile along the four turbines' mouths in triangular arrangement and 50% spacing size	42
4.15	Velocity profile along the four turbines' mouths in triangular arrangement and 100% spacing size	43

4.16	Velocity profile along the four turbines' mouths in triangular arrangement and 150% spacing size	44
4.17	Velocity profile along the four turbines' mouths in triangular arrangement and 200% spacing size	45
4.18	The changes in average velocities when spacing sizes were increased for triangular array arrangement	46
4.19	Velocity profile along the four turbines' mouths in square arrangement and 0% spacing size	47
4.20	Velocity profile along the four turbines' mouths in square arrangement and 500% spacing size	48
4.21	Velocity profile along the four turbines' mouths in square arrangement and 100% spacing size	49
4.22	Velocity profile along the four turbines' mouths in square arrangement and 150% spacing size	50
4.23	Velocity profile along the four turbines' mouths in square arrangement and 200% spacing size	51
4.24	The changes in average velocities when spacing sizes were increased for square array arrangement	52
4.25	Velocity profile along the four turbines' mouths in rhombus arrangement and 0% spacing size	53
4.26	Velocity profile along the four turbines' mouths in rhombus arrangement and 50% spacing size	55
4.27	Velocity profile along the four turbines' mouths in rhombus arrangement and 100% spacing size	56
4.28	Velocity profile along the four turbines' mouths in rhombus arrangement and 150% spacing size	57
4.29	Velocity profile along the four turbines' mouths in rhombus arrangement and 200% spacing size	58
4.30	The changes in average velocities when spacing sizes were increased for rhombus array arrangement	59

4.31	The comparison of average velocities when spacing size increases between triangular, square and rhombus arrays arrangements	60
B1	Parallel with 0% diameter spacing	71
B2	Parallel with 50% diameter spacing	71
B3	Parallel with 100% diameter spacing	72
B4	Parallel with 150% diameter spacing	72
B5	Parallel with 200% diameter spacing	73
B6	Series with 0% diameter spacing	73
B7	Series with 50% diameter spacing	74
B8	Series with 100% diameter spacing	74
B9	Series with150% diameter spacing	75
B10	Series with 200% diameter spacing	75
B11	Triangular with 0% diameter spacing	76
B12	Triangular with 50% diameter spacing	76
B13	Triangular with 100% diameter spacing	77
B14	Triangular with 150% diameter spacing	77
B15	Triangular with 200% diameter spacing	78
B16	Square with 0% diameter spacing	78
B17	Square with 50% diameter spacing	79
B18	Square with 100% diameter spacing	79
B19	Square with 150% diameter spacing	80
B20	Square with 200% diameter spacing	80
B21	Rhombus with 0% diameter spacing	81

B22	Rhombus with 50% diameter spacing	81
B23	Rhombus with 100% diameter spacing	82
B24	Rhombus with 150% diameter spacing	82
B25	Rhombus with 200% diameter spacing	83
C1	Velocity plane of parallel arrangement with 0% diameter spacing	85
C2	Velocity plane of parallel arrangement with 50% diameter spacing	85
C3	Velocity plane of parallel arrangement with 100% diameter spacing	86
C4	Velocity plane of parallel arrangement with 150% diameter spacing	86
C5	Velocity plane of parallel arrangement with 200% diameter spacing	87
C6	Velocity plane of series arrangement with 0% diameter spacing	88
C7	Velocity plane of series arrangement with 50% diameter spacing	88
C8	Velocity plane of series arrangement with 100% diameter spacing	89
C9	Velocity plane of series arrangement with 150% diameter spacing	89
C10	Velocity plane of series arrangement with 200% diameter spacing	90
C11	Velocity plane of triangular arrangement with 0% diameter spacing	91
C12	Velocity plane of triangular arrangement with 50% diameter spacing	91
C13	Velocity plane of triangular arrangement with 100% diameter spacing	92
C14	Velocity plane of triangular arrangement with 150% diameter spacing	92
C15	Velocity plane of triangular arrangement with 200% diameter spacing	93

C16	Velocity plane of square arrangement with 0% diameter spacing	94
C17	Velocity plane of square arrangement with 50% diameter spacing	94
C18	Velocity plane of square arrangement with 100% diameter spacing	95
C19	Velocity plane of square arrangement with 150% diameter spacing	95
C20	Velocity plane of square arrangement with 200% diameter spacing	96
C21	Velocity plane of rhombus arrangement with 0% diameter spacing	97
C22	Velocity plane of rhombus arrangement with 50% diameter spacing	97
C23	Velocity plane of rhombus arrangement with 100% diameter spacing	98
C24	Velocity plane of rhombus arrangement with 150% diameter spacing	98
C25	Velocity plane of rhombus arrangement with 200% diameter spacing	99

LIST OF SYMBOLS

Ø	Diameter
”	Inch
’	Feet
%	Percentage
±	Plus-minus sign
°	Degree sign
=	Equal to
≈	Almost equal to
(+)	Positive
+	Plus sign
(-)	Negative
-	Minus sign
×	Multiplication sign
÷	Division sign

LIST OF ABBREVIATIONS

MHP	Micro Hydro Power
CFD	Computational Fluid Dynamics
TEC	Tidal Energy Conversion
RANS	Reynolds-averaged Navier Stokes
LES	Large-eddy Scale
HATT	Horizontal-axis Tidal Current Turbine
KE	Kinetic Energy

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

Hydro turbines are generally used to generate electricity from the water flow of a river or an ocean. For example, the hydroelectric dams in Malaysia are used to generate electricity at upstream location of a river. The same concept can be done at downstream location by placing hydro turbines in the river. It is another method of extracting energy from kinetic energy of the water flow. Micro hydro turbines are used in large quantity to generate maximum energy

Computational Fluid Dynamics (CFD) is a subtopic of Fluid Mechanics subject, which involves simulations with numerical methods and algorithms by using the CFD software in computers. The liquids and gases are defined by boundary conditions. This boundary condition is created to form a numerical model. The numerical model can be used for calculation of the parameters, which are set up during defining the boundary condition. After calculation, the simulation can be performed together with data presentation in many different methods. Overall, CFD is used to analyze and simulate the flow in a domain. With CFD, the water flow of river at downstream through arrays of MHP turbines simulated.

1.2 PROBLEM STATEMENT

In terms of arrangement of the micro hydro power turbines, the best arrangement of micro hydro power turbines in a river is needed to generator maximum amount of power. These MHP turbines can be used to supply electricity to rural areas.

"Hydropower on a small scale, or micro-hydro, is one of the most cost-efficient energy technologies to be considered for rural electrification in less developed countries."(Paish, 2002)

*"Micro hydro power is almost always more cost-effective than any other form of renewable power."(*Paul and Barbara, 1994)

Most researches are conducted on the flow over array arrangements of tidal turbines.

"This work investigates the performance variation across a tidal energy converter (TEC) located centrally in an array (measure as a pressure differential)."(Bai et al., 2009)

"This study presents preliminary results of on-going work using Gerris, an adaptive mesh flow slower, to investigate the flow through 4 different arrays of 15 turbines each, over a tidal cycle."(Divett et al., 2010)

This research on MHP turbines was conducted based on the flow condition Pahang River downstream.

1.3 OBJECTIVES

The main objective of this project is to obtain the flow characteristics of a fiver flow when subjected with multiple arrays of micro hydro power (MHP) turbines. Before the main objective is achieved, another two objectives must be achieved first, which are to setup a free surface numerical model of a river and to setup a numerical model of a river flow on a MHP turbine.

1.4 PROJECT SCOPE

The first scope of the project was to create a numerical model of a river to simulate the water flow. First, the geometry of turbine body is designed with Solidworks 2012 software. The unit for the river model and the turbine body is in meter (m). The meshing, setting up boundary condition, calculation, simulation and data presentation can be done with ANSYS 14, which is limited to the ANSYS CFX program. ANSYS CFX is used to simulate a flow in a domain in which the river is a flow domain for water and the turbine bodies as the solid domain in the river. The parameters are velocity, upstream and downstream heights, water density, and reference density have been set in the setup to run the simulation to obtain the flow characteristics over the turbine bodies.

The array arrangements of MHP are divided into five different arrangements, which are parallel, series, triangular, square and rhombus. Each array arrangement has different spacing sizes. These different array arrangements and spacing sizes are designed with the Solidworks software

The independent variable parameter used in the boundary condition is the velocity of the river flow. Meanwhile, the dependent variable is the velocity of the water reacts on the turbine blade. The temperature remains constant.

1.5 EXPECTED RESULTS

As the spacing sizes for each array arrangements increases, the average velocity of the water flow inside the turbine bodies will increase must decrease as it enters the turbine bodies and the pressure in the turbine bodies will be higher. The kinetic energy of the water flow will able to generate maximum energy. The array arrangements that obey this situation will be chosen as the most significant arrangements to generate energy and will be used for further researches.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Hydro turbines are used to change water pressure from river flow into mechanical or kinetic energy, which can generate electricity (Paish, 2002). According to Cunningham in 1994, Micro Hydro Power (MHP) technology is needed to generate electricity to residential areas. Besides, MHP was one of renewable energy resources in the world. According to Paish in 2002, small hydro or MHP was the most economical technology to generate power. This was because MHP turbines use the water flowing in a river to generate power, where it is a continuous process. Besides, the costs for manufacturing and installing the MHP turbines were cheaper compared to the tidal and wind turbines.

Most of the computational researches on arrays arrangement were done for tidal turbines, which was almost similar to MHP turbines. Computational fluid dynamics (CFD) models are used to analyze the wake effects in arrays of tidal turbines (Harrison, 2009). The simulations of the turbines were done to obtain the wake characteristics of actuator disc and compare with experimental data. According to Bai in 2009, Fluent software was used to simulate 3-D models of Tidal Energy Conversion (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 (Divett, 2010). Reynolds-averaged Navier Stokes (RANS) equations are used with the turbines to stand for frozen rotor. Static blade method was similar with the frozen rotor method. The frozen rotor method is used in the computational domain to consider the deliberate the rotating effect (Lee, 2010). The CFD software was 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 (Churchfield, 2012). This method was carried out by creating a framework to generate inflow tidal turbulence data. With the framework, the wake characteristics and power generated had been considered. According to Lawson in 2011, Horizontal-axis Tidal Current Turbines (HATTs) were simulated with RANS CFD method to characterize the performance.

2.2 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics is research software which is used in engineering fields to create simulation and analysis. This researches involve the flow characteristics of a fluid through a domain, heat transfer from solid to solid, solid to fluid or fluid to fluid, combustion process and the particle tracking of substances in a fluid. CFD is particularly dedicated to the former, fluids that were in motion, and how the fluid flow behavior influences processes that may include heat transfer and possibly chemical reactions in combusting flows (Jiyuan et al., 2008).

Moreover, the CFD software is used to create numerical models. The numerical models are consisting of the models of a real item which are design with the CAD software or the Design Modeler in ANSYS CFX. The CFD software creates simulation of fluid flow over or through the numerical model to get proof on capability of the items. Through the simulation and analysis, accurate data on a model can be obtained. The physical characteristics of the fluid motion can usually be described through fundamental mathematical equations, usually in partial differential form, which govern a process of interest and are often called governing equations in CFD (Jiyuan et al., 2008).

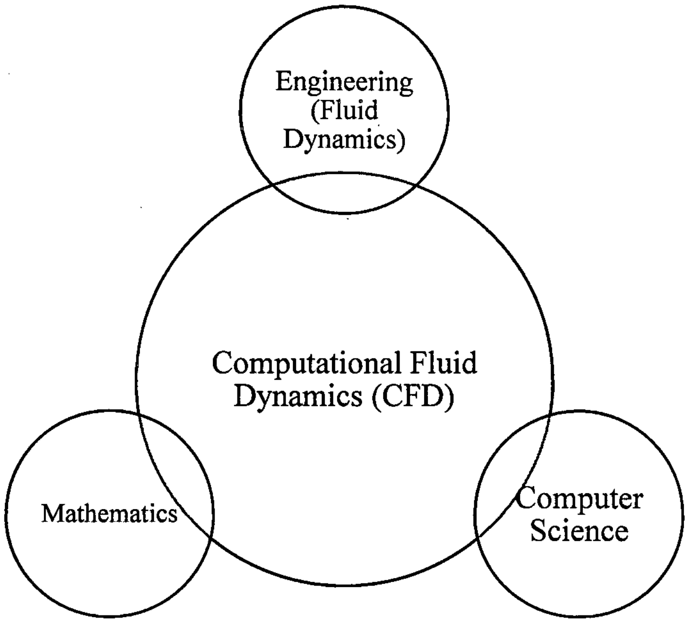

Figure 2.1: The different discipline contained within computational fluid dynamics.

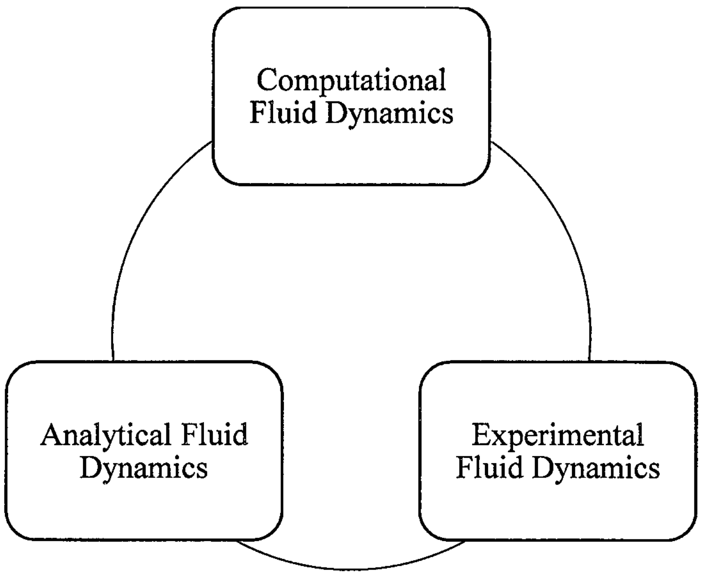

Figure 2.2: The three basic approaches to solve problem in fluid dynamics and heat transfer.

Source: Jiyuan et al., 2008.

2.3 GOVERNING EQUATIONS

2.3.1 Navier-Stokes Equation

This equation is used to calculate the fluid motion. The general Navier-Stokes equation is:

$$\rho\left(\frac{\partial v}{\partial t} + v.\nabla v\right) = -\nabla p + \nabla.T + f \qquad (2.1)$$

2.3.2 Kinetic Energy Equations

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:

$$KE = \frac{m.v^2}{2} \qquad (2.2)$$

Where, KE = kinetic energy (J)

m = mass (kg)

v = velocity of a fluid flow (m/s)

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

$$\frac{ke}{m} = \frac{v^2}{2} \qquad (2.3)$$

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_r) 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:

$$v = r\omega \qquad (2.4)$$

Where, *r* = radius of turbine shaft

ω = angular velocity

Substitute Eq. (2.4) into Eq. (2.2)

$$KE = \frac{m.r^2.\omega^2}{2} \tag{2.5}$$

$$KE = \frac{I\omega^2}{2} \tag{2.6}$$

From the Eq. (2.6), 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:

$$P = \omega M \tag{2.7}$$

Where, ω = angular velocity

M = 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.

2.3.3 Continuity Equation

This equation was involving the mass conversation in a control volume in a fixed space. The fluid flow is matter may neither be created nor destroyed (Jiyuan, 2008).