DESIGN, ANALYSIS AND FABRICATE SMALL CROSS FLOW TURBINE FOR SUNGAI PAHANG

MOHAMAD NORMAZUAN BIN MUSTAFA

Report submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

2012

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality ofthe award of the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature :

Name of Supervisor : PROF. DR. ROSLI BIN ABU BAKAR

Position : LECTURER

Date :

SECOND REVIEWER DECLARATION

I certify the thesis "Design, Analysis, And Fabricate Small Cross Flow Turbine For Sungai Pahang" is written byMohamadNormazuan Bin Mustafa. I have examined the final copy of this thesis and in my opinion; it is fully adequate in term of scope and quality for the award of the degree of Bachelor of Engineering. I here with recommend that it be accepted in fulfillment of the requirement for the degree of Bachelor of Mechanical Engineering with Automotive.

Signature :
Name of Supervisor : MOHAMAD ZAIRI BIN BAHAROM
Position : LECTURER
Date :

STUDENT'S DECLARATION

I hereby declare that the work in this report is my own except for quotations and summaries which have been duly acknowledged. The report has not been accepted for any degree and is not concurrently submitted for award of another degree.

Signature :

Name : MOHAMAD NORMAZUAN BIN MUSTAFA

ID Number : MH08039

Date :

ACKNOWLEDGEMENTS

I would like to express my deepest appreciation and sincere gratitude to my supervisor, Dr. Sugeng Ariyono for my first supervisor on my PSM1, then for Prof. Dr. Rosli Bin Abu Bakar supervisor for PSM2, for his wisdom, invaluable guidance and professionalism from the beginning to the end in making this research possible. He has been an excellent mentor and has provided continuous encouragement and constant support throughout my project. It should be recognized that the success of this thesis was through her cooperation and assistance from the initial to the final level which enabled me to develop an understanding of the subject.

I also would like to extend my heartiest thanks to my colleagues who have rendered assistance and support in one way or another to make this study possible. I am grateful for their support and invaluable help.

Special thanks to my beloved parents Amanan Binti Mat, Mustafa Bin Darus and siblings whose endless support and understanding have been profound throughout the difficult times of this project. Without your love and support I am sure that I would not have been able to achieve so much. Lastly, it is a pleasure to thank those who supported me in any respect during the completion of the project. Without the generous help of these individuals, this research would not have been possible.

ABSTRACT

This thesis is about doing analysis and investigations on river flow rate in Malaysian River which focusing at Sungai Pahang, which is form the investigation the small cross flow are design by consider the data and the characteristic that are collected. Through this analysis and investigation could obtain the potential of running a Water Turbine System (WTS). Demand for electricity around the world increase day by day, and plant that are used nowadays many of that are not renewable energy such as coal power plant, fossil-fuel derived electricity, where the source and supply of this raw material source are decrease day by day and the price are increase, because of this global crisis water turbine system are come out. This innovation are come out to overcome this crisis due to the hydropower that are known as renewable power energy. This project are dedicated to people that leave deep inland can reach and used a electricity especially for this project are dedicated and special for Sungai Pahang river.The main problem statement is on location limitation to obtain peak flow rate of river to running water turbine systems which is depending on depth and width of the river and also from the velocity of sectional area of river. Other than that, choosing most suitable water turbine system is also one of main criteria to be taken in this research. Water Turbine are one of clean renewable energy sources which holding high percentage of energy resource provider among renewable energy. This thesis will carry on with modeling a WTS that can be recommended to construct in Sungai Pahang. The analysis on flow rate of Pahang River are done by calculating potential Power (Watt) that could produced based on volume flow rate data that obtain from Department of Irrigation and Drainage of Malaysia. Since water turbine could provide electricity power without affecting pollution to the environment. Water turbines systems also are cheapest resource to be obtain. Final modeling of water turbine system is completed by Computer Aided Design,CAD software and will be analys bu ising Computetional Fluid Dynamic, CFD software.

ABSTRAK

Tesis ini adalah menajalan analisis dan penyiasatan ke atas kadar aliran sungai di Malaysia, yang mana memberikan tumpuan khusus pada Sungai Pahang, dimana hasil penyiasatan akan di gunakan dalam mereka bentuk turbine berskala kecil yang sesuai berpandukan maklumat dan ciri-ciri Sungai Pahang. Melalui analisis dan penyiasatan ini, membolehkan perolehan maklumat mengenai potensi penggunaan Sistem Turbin Air (WTS). Sedia maklum permintaan untuk tenaga elektrik di seluruh dunia dunia hari demi hari, dan loji yang digunakan sekarang adalah dari jenis tenaga tidak boleh diperbaharui seperti loji janakuasa arang batu dan bahan bakar fosil hi, di mana sumber dan bekalan sumber bahan mentahnya adalah semakin berkurangan hari demi hari dan ini menyebabkan harga peningkatan harganya, oleh sistem air turbin diperkanalkan bagi menangani krisis global yang melular ini. Inovasi ini adalah untuk mengatasi krisis ini kerana kuasa hidro adalah tenaga kuasa yang boleh diperbaharui. Projek ini didedikasikan untuk komuniti yang tinggal di pedalaman, melalui hasil projek ini capaian dalam penggunaan elektrik dapat dikurangkn jurangnya, dan projek ini adalah khusus bagi Sungai Pahang. Masalah utama adalah had lokasi untuk mendapatkan kadar aliran tertinggi sungai untuk menjalankan sistem ini dimana ianya turut berkait rapat dengan kelebaran sungai dan kedalaman sungai itu sendiri. Selain daripada itu, memilih sistem turbin yang paling sesuai juga merupakan salah satu kriteria utama yang perlu diambil dalam kajian ini. Sistem turbin air adalah salah satu sumber tenaga yang bersih boleh diperbaharui yang mana merupakan sistem utama di dalam pembekalan sumber bersih tenaga elektrik. Tesis ini dijalankn dengan memodelkan Sistem Turbin Air (WTS) untuk disyorkan untuk digunakan di Sungai Pahang. Analisis ke atas kadar aliran Sungai Pahang adalah dilakukan dengan mengira potensi Kuasa (Watt) yang boleh dihasilkan berdasarkan data kadar aliran isipadu yang diperolehi dari Jabatan Pengairan dan Saliran Malaysia. Dengan penggunaan sistem turbin air, ianya dapat menghasilkn kuasa elektrik tanpa menjejaskan alam sekitar. Sistem turbin air juga adalah sumber yang paling murah. Pemodelan akhir sistem turbin ini adalah menggunakan perisian Computer Aided Design, CAD dan akan analisis aliran adalah dengan mengunakan perisian Computetional Fluid Dynamic, CFD.

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURE

LIST OF ABBREVIATIONS

POM	Polyoxymethylene
WTS	Water Turbine System
DC	Direct Current
FRP	Fiberglass Reinforced Plastic
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
MIG	Metal Inert Gas
ARC	Shielded Metal Arc Welding
L	Length
W	Width
T	Temperature
K	Kelvin
Re	Reynold Number
kg	kilogram
Pa	Pascal
MPa	Megapascal
2-D	Two Dimension
V	Velocity
V_1	Velocity inlet
V_2	Velocity outlet
SI units	System International units
N	Newton
mm	Millimeter

ρ	Density
UNESCO	United Nations Educational, Scientific and Cultural Organization
$P\omega$	Extractable power of a water jet
Q	Volumetric flow rate
H	Head of the water
g	Acceleration due to gravity
A	Cross sectional area
A_1	Cross sectional area inlet
A_2	Cross sectional area outlet
Ps	Shaft power
Pe	Electrical power
F	Tangential force
ω	Angular speed
μ	dynamic viscosity
L	Pitch radius
n_m	Mechanical efficiency
U	Voltage across the load
I	load current
L_e	Length of nozzle
Dh	hydraulic diameter

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Nowadays, electricity is one of an important thing in daily life, the demand increase day by day due to the increasing the capacity of human being and also of the transportation demand, where such as electric motorcycle, car and train. Hydro power in conservation of water flow energy to another form of energy, electricity which is common power used in nowadays modern era world.

The electricity generate by the rotating of turbine blade cause of river flow into electrical current by means of an electrical generator where the power are stored in the cell. Water turbine is used in big scale like dam, where involve such an edifice that are changed the landscape of nature which is not eco-friendly. For this purpose small individual turbines are promoted to Sungai Pahang which is beneficial to all that are involved and attach to this river. Hydro power energy also plentiful, renewable, reliable source of energy, clean, easy to handle, easy to do maintenances and reduces toxic atmospheric and greenhouse gas emission if used widely where can replace fossil-fuel derived electricity

A small cross water flow turbine is a machine that is converts the kinetic energy to another form of energy, where in form of mechanical energy. This mechanical energy then converts to electricity energy which is called hydro generator, or hydro energy converter.

This turbines can defined in two types, depend on the axis of the turbine and there are Vertical and Horizontal axis cross water flow turbine.

This small scale cross flow turbine only can produce a small amount of energy, if compared to conventional type. But for this type it more eco-friendly which is no need such a massive construction that are known will affect the eco system of the area that are involve in the construction and also need such a long period of time to set it up. This small cross flow turbine may produce at kilowatt, kW, or lower electricity power. But for this project the aim is for the small community in inland and also for the bridge light purpose. Within affect a very minimum effect to eco system.

1.2 PROBLEM STATEMENT

For this century a renewable energy not just an optional but a need, because of a source of the conventional energy decrease day by day and the global price raised every day. The world stock and need are greater than supply and also increasing of the mankind day by day, (UNESCO). By using a small cross flow turbine is and a perfect fit because of fulfill all the criteria such as clean, green and renewable. This project target area is along Sungai Pahang, known as backbone off peninsular Malaysia.

This project is also hopefully be one and a popular alternative in Malaysia due to generate an electricity, especially due to the target area for inland community and also facilities such as bridge that are built to across the river that are located far inland along Sungai Pahang. This project that are cheaper and easy to conduct and maintenance where these criteria also include to the green way for the renewable energy.

This project are choose because of more weather friendly, compared to other alternative such as solar panel, this is because Pahang are located in the East of Malaysia that has a rainy season, for solar panel, during rainy season the sunlight are so minimum because of the long period raining time, and solar panel just can be at maximum output at day, differ for small water turbine it can keep produce an electricity a day and night. Compared to the wind turbine for target area such as a long Sungai Pahang, the open space are limited and it full of tree where it is obstacle for the maximum wind flow.

1.3 ROJECT OBJECTIVES

The objectives of this project is:

i. To fabricate the prototype cross flow water turbine system.
ii. Increase flow rate of river flow

1.4 SCOPES OF PROJECT

The scopes of this project are:

i. Modeling and simulating the prototype suitable software.
ii. Validate the design by performing test of flow rate water speed by using CFD software.

1.5 PROJECT METHODOLOGY

This methodology must be detail and closely dovetailing the flow chart of the overall project. After the objective has been set, it comes the way to setup the experiment from the early to the end of this project. The execution of this project has to follow the flow that is already set, to ensure that the project will be done smoothly and without any discrepancies. The overall of the project is shown in Figure 1.1

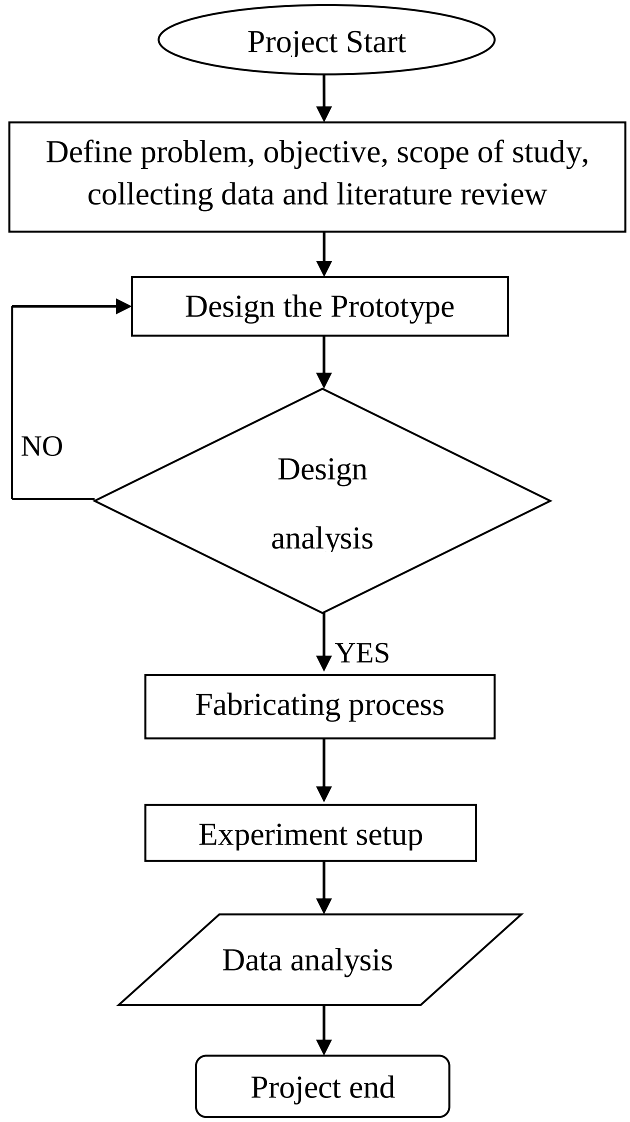


Figure 1.1: Methodology of the project.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

In 1892, John B. McCormick invented and a water wheel, before his design and a mixed-flow style turbine in 1899. Water turbine basically made of steel that are attach to the chassis that are hold the turbine in order to generate electricity. The reason John B. McCormick to develop of this invention is to "Going Green" before the term even existed, dedicating his life to inventing more efficient ways to create power for industry, he started with basic improvements to the water wheel, the simplest type of water turbine, and one of the oldest forms of power and irrigation, dating back to the ancient Egyptians. The water wheel utilizes falling or flowing water to turn a large wheel centered on a shaft. This shaft then spins and, in McCormick's time, would power a set of gears in a mill. The water is caught by either blades or buckets and nothing but the natural force (potential energy) of water is used to create power.

John McCormick the physics of the water wheel took into consideration and created a version that was more suited for industry. In his design, the water wheel is enclosed in a metal casing that water is flown through. The blades of the water wheel have small flaps that are angled from the perpendicular that catch the water as it flows, and turn the wheel. The overall shape is similar to that of a bell as the diameter of the metal casing widens from top to bottom. The water enters through the thin end and the water wheel is in the thick end. The water flows from the thin end, to the thick end, turning the wheel in doing so. Design patented flow water turbine detail is shown in the Figure 2.1.

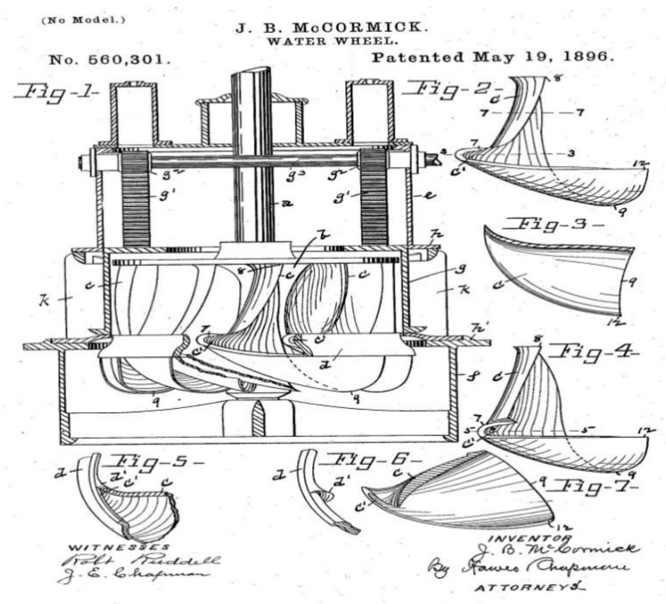


Figure 2.1: Patented flow water turbine

Source: John B. McCormick, 1896

The turbine style water wheel that McCormick invented in 1892 was just one of a few improvements he made to the water wheel before designing the mixed-flow style turbine in 1899. A turbine is a machine that uses the kinetic (moving) energy from a flowing gas or liquid such as steam or water to turn a series of blades or fans. This process converts kinetic energy to mechanical energy. In a power plant, a turbine is used to turn a conducting wire in a generator to produce power. A turbine is based on a principle made famous by physicist Isaac Newton. His third law of motion states that for every action there is an equal and opposite reaction. In a turbine, the action is the flowing fluid and the reaction is the turning blades.

The efficiency of an axial turbine is around 70%-75%. This is because there is space between the rotor blades for fluid to pass through. The only way for all of the air blowing at a windmill to be captured was if the windmill were a complete, solid circle. At the same time, if the windmill were a complete, solid circle there would be no way for the turbine to catch the air in a useful way, and because of this it is impossible to have an axial turbine with 100% efficiency.

2.2 THEORY OF WATER TURBINE

The total extractable power of a water jet $P\omega$ is shown in Eq.(2.1).

$$P\omega = \rho g h Q \tag{2.1}$$

where:

Q is the volumetric flow rate (m^3/s).

h is the available head of the water source (kg/m^3)

g is the acceleration due to gravity (9.81 m/s^2),

ρ is the density of the water (kg/m^3).

The speed of the water jet vows is found from the flow rate and the cross section area *A* of a circular nozzle with diameter *D*, the detail are shown in Eq.(2.2).

$$V\omega = Q/A = Q^4/\pi D^2 \tag{2.2}$$

where:

A is cross section area of nozzle (m^2)

D is the diameter of the nozzle (m)

The mechanical power *P*s available at the turbine shaft can be determined by measuring the torque t on the shaft at a corresponding angular speed ω. The torque is found by measuring the tangential force *F* on a brake lever with moment arm length *l*, while simultaneously measuring the rotational speed r of the shaft, as depicted in Figure 2.1. The shaft power is then *P*s is shown details in Eq.(2.3).

$$P_S = \omega t = 2\pi lF \tag{2.3}$$

where:

t is torque (N)

l is arm length (m)

$F$is the tangential force (N)

The angular speed ω is the tangential speed of the turbine v_t divided by the pitch radius L of the Pelton wheel, the detail calculation is shown in Eq.(2.4).

$$\omega = v_t/L \tag{2.4}$$

where:

L is pitch radius of the Pelton wheel (m)

v_t is the tangential speed of the turbine (m/s)

The mechanical efficiency, n_m of the turbine describes how effectively the available kinetic energy of the water jet is transformed into turbine motion and from Eq.(2.1) and Eq.(2.3) is shown details in Eq.(2.5).

$$n_m = P_S \,/\, P_w = 2\pi rlF/\rho ghQ \tag{2.5}$$

Cup and jet design are important parameters and it can be shown in Eq.(2.5), that maximum theoretical mechanical efficiency is achieved when the tangential speed of the turbine is roughly one-half the speed of the water coming from the jet or, the Eq.(2.6), is shown the detail.

$$v_t/\, v_w \approx 0.5. \tag{2.6}$$

where:

v_t is the tangential speed of the turbine (m/s)

v_w is the speed of the water (m/s)

When the turbine shaft is coupled to an electric generator which supplies electricity to a variable resistive load *R*, the electrical power *Pe* of the load is shown in Eq.(2.7).

$$Pe = IU \tag{2.7}$$

where:

I is the is the load current (amps)

U is the voltage across the load (V)

The rotating shaft will produce the torque at the shaft with length *l* of the shaft rotating rod, the detail are shown in the Figure 2.2 below.

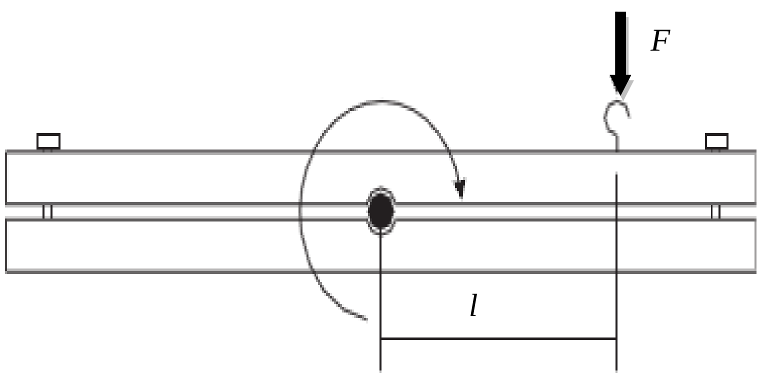


Figure 2.2: The measurement of the torque on the turbine shaft is performed using a brake lever with a moment arm length *l*.

Source: Renewable Energy 33 (2008)

The lever consists of two wooden splints which are clamped around the shaft by use of two screws. The tangential force F is measured directly using a spring scale. In which l is the load current and U the voltage across the load? The electrical efficiency n_c of the generator and the overall efficiency is shown by Eq.(2.8) and n_0 of hydroelectric generation are then shown in Eq.(2.9).

$$n_c = P_e/P_s = IU/2\pi lF \tag{2.8}$$

$$n_0 = P_e/P_w = IU/\rho ghQ \tag{2.9}$$

The Pelton turbine consists of a cylindrical hub onto which 12 vanes have been fastened using screws and right angle aluminum brackets. The hub and vanes were made of polyoxymethylene, POM, plastic for easy machine ability. The pitch radius of the turbine, which is the distance from the center of the shaft to the center of the impact point of the water jet, should be roughly 10 times the diameter of the jet so that adjacent vanes do not interfere with the water flow. The twin cups on each vane were shaped by milling two parallel 26 mm hemispherical grooves using an 8 mm ball-end cutter. The tips and back sides of the vanes were shaped by free-hand grinding with a belt sander. The shape and size of the hub and cups were selected according to available tools and materials.

Other geometries or manufacturing methods would, most certainly, work equally well. The turbine shaft is supported directly by the transparent acrylic turbine housing which allows the action of the water to be easily observed. No attempt was made to balance the finished turbine. The device is shown in Figure 2.3 with the upper section of the housing removed for better viewing. A hose-coupling connector from a garden supply store is used as a water nozzle. It is located on the lower corner of the housing, and has been attached using thick flexible rubber gaskets which allow the impact point of the jet to be adjusted so that it strikes the dividing ridge between cups on the lower side of the turbine. The nozzle diameter is about 6 mm and the pitch radius of the turbine is about 60 mm.

Figure 2.3: Details of the Pelton wheel showing the 12 vanes evenly spaced around the central hub.

Source: Renewable Energy 33 (2008)

The turbine shaft is fitted with a pulley wheel (foreground) which drives the Direct Current, DC, generator (not shown) using a rubber belt. The nozzle is shown in the lower-right corner. The upper part of the housing was removed for the photograph.

Outside the housing, the back of the nozzle is fitted with a T-connection which is used for the incoming water source and for the measurement of pressure, respectively. A pressure gauge can be fitted directly to the back of the nozzle or a connection to a mercury manometer can be made. In the system under discussion both methods have been used to determine the pressure but the manometer method appears more instructive to students.

Water is supplied to the nozzle by an electric pump and flexible tubing. After striking the vanes, water exits the turbine housing through a hole at the base and flows back to an enclosure in which the intake of the pump is located. The volume flow rate of water is measured and controlled using a magnetic volume flowmeter1 and hand-operated valve. A more simple low-cost method, however, is to use a graduated liquid measure and a stop watch to measure the volume flow.

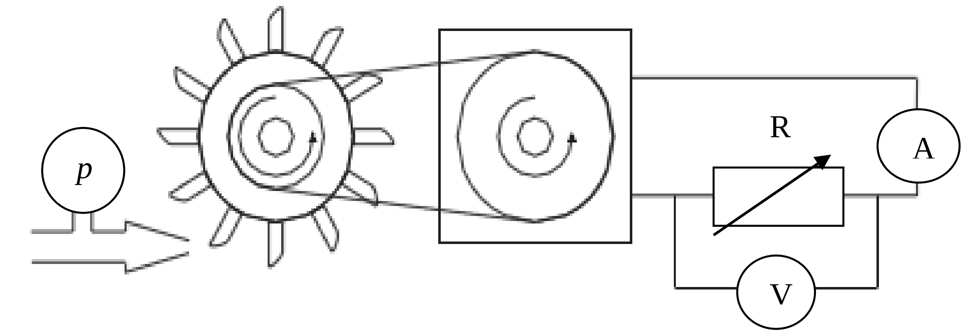


Figure 2.4: A schematic showing the experimental setup for the Pelton wheel water turbine experiment.

Source: Renewable Energy 33 (2008)

One end of the turbine shaft extends through the wall of the housing and is fitted with a pulley wheel, which drives a small Direct Current, DC, electric generator by use of a rubber belt. The generator itself is a salvaged motor from a Direct Current, DC, and cooling fan. In order to determine the rotational speed of the turbine, a frequency-adjustable stroboscopic light is used to illuminate the pulley wheel. While observing reference marks on the spinning pulley wheel, the strobe frequency is adjusted until a particular mark appears to be stationary.

When this is observed, the frequency of the strobe and the speed of rotation of the wheel are equal, assuming the experimenter has eliminated the possibilities of corresponding harmonic frequencies.

A brake lever is used to determine the torque on the turbine shaft. The lever is made from two wooden splints, which when fitted together and tightened with two screws; apply a frictional force on the turbine shaft. While the shaft is spinning, the force acting at right angles on the lever can be measured directly with a spring scale calibrated in Newton. For the turbine under discussion, measured forces have been in the range of 0.1–1.5N with a moment arm length of some 70 mm.

The load circuit of the DC generator consists of an adjustable 0-100 kΩ, decade resistor 2 Load current and Voltage have been measured using digital multi-meters. The experimental setup is shown in Figure 2.4.

Students have conducted experiments with the Pelton turbine using three different water volume flow rates, referred to as *Q1*, *Q2* and *Q3*. The water pressure p at the nozzle is measured for each *Q* value with the turbine shaft rotating freely. In Eq.(2.10) is shown the equation to calculate, if a mercury manometer is used, the power *Pw* from Eq.(2.1) is,

$$Pw = pQ = \rho_{Hg} g Dh Q \qquad (2.10)$$

where:

Q, *Q1*, *Q2*, *Q3* is the volumetric flow rate (m^3/s).

g is the acceleration due to gravity (9.81 m/s^2),

ρ_{Hg} is the density of mercury (13 550 kg/m^3)

Dh is height difference between the two mercury columns (m)

The relationship between the tangential force on the shaft and the speed of rotation is then determined using the brake lever for each *Q* value. An empirical expression for the force as a function of rotational speed is fitted to this data using appropriate software. This relationship is generally found to be of a linear nature in the range of variables described.

With the drive belt connected to the Direct Current, DC, generator and the brake lever removed from the turbine shaft, the current and voltage of the resistive load are measured, while simultaneously measuring the rotational speed of the turbine. This is repeated for several values of load resistance. The measurement yields the electrical power produced by the generator as a function of rotational speed. The expressions found for tangential force are required to calculate the electrical efficiency of the Direct Current, DC generator at rotational speeds not directly measured with the brake lever.

Using experimental data, the mechanical, electrical and overall efficiency of the laboratory-scale hydroelectric power plant and their accompanying experimental uncertainties are calculated for each Q value.

2.3 DESIGN PARAMETER OF THE NOZZLE

The designs for this nozzle that are will develop are follow the continuity equation, shown in Eq.(2.11) that states, in a tube, the following must be true:

$$\dot{m}_{in} = \dot{m}_{out} \qquad (2.11)$$

This equation can be rewritten as shown in Eq.(2.12).

$$\rho_1 A_1 V_1 = \rho_2 A_2 V_2 \qquad (2.12)$$

Or for incompressible flow, is shown in Eq.(2.13). (assume $\rho_1 = \rho_2$)

$$A_1 V_1 = A_2 V_2 \qquad (2.13)$$

where:

A_1 is the inlet cross sectional area (m^2),

A_2 is the outlet cross sectional area (m^2),

V_1, $V2$ is the speed of the water flow (m/s^1),

ρ_1, ρ_2 is the density of the river water (kg/m^3).

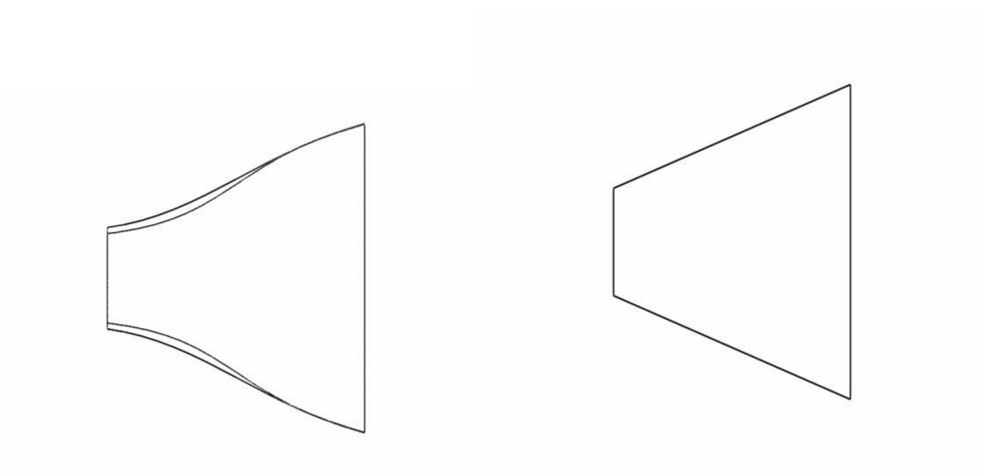
2.3.1 Developed Nozzle

With several of outlet shape, with same length and shape of nozzle developed nozzle are with several criteria into 4 groups, which are:

i. Inlet with constant (same) cross sectional area.
ii. Outlet with same cross sectional area size.
iii. Rectangular inlet with rectangular shape of outlet.
iv. Rectangular inlet with circular shape of outlet.

2.3.2 Inlet and outlet Nozzle

Inlet Nozzle consists of nozzle and cylinder. Figure 2.5 shows the shape of each nozzle mated with cylinder and the models of the Inlet Ducts are designated label as shown below, where for this project the outlet of the nozzle are selected at 0.0072 m². (with *L*=0.12 m and *W*=0.06 m).



(a) (b)

Figure 2.5: Inlet nozzle for (a) Shape A and (b) Shape B.

Source: Azliza, A. A. (2010)

The size of the inlet and outlet are defined by using ratio calculation, that are shown in Figure 2.6, where the inlet is six (6) time of the outlet area size.

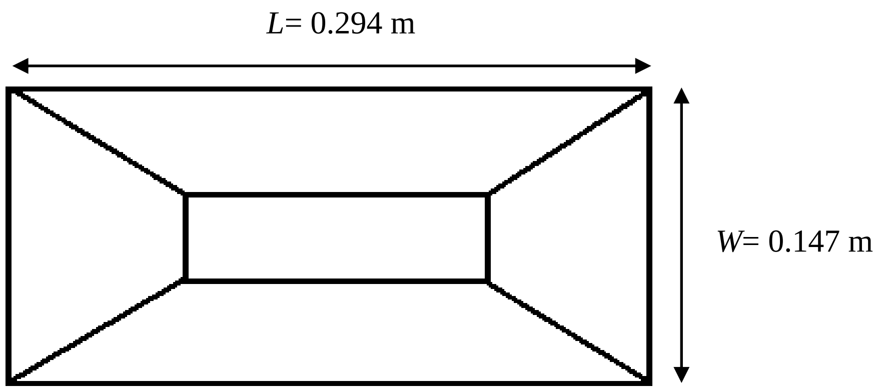


Figure 2.6: Nozzle inlet dimension

And the length L_e of the nozzle is defined by the calculation. With define as shown in Eq.(2.14), where defined the diameter of the rectangular first.

$$D_H = \frac{4LW}{2L+W} \tag{2.14}$$

where:

L is the length of the nozzle

W is the width of the nozzle

With diameter that is calculated are used to define the Reynold Number, ***Re.*** in other to define the characteristic of the flow rate either laminar or turbulent, the detail of the step are shown in the Eq.(2.15).

$$Re = \frac{\rho v D_H}{\mu} \tag{2.15}$$

where:

D_H is the hydraulic diameter of the pipe; its characteristic travelled length, L, (m).

Q is the volumetric flow rate (m^3/s).

v is the mean velocity of the object relative to the fluid (m/s).

μ is the dynamic viscosity of the fluid (Pa·s or N·s/m^2 or kg/(m·s)).

ρ is the density of the fluid (kg/m^3).

From the value Reynold Number, *Re,* that is calculated, the flow characteristic of the river can be defined according to the state value below.

laminar flow when $Re < 2300$

transient flow when $2300 < Re < 4000$

turbulent flow when $Re > 4000$

Nozzle Length, L_e in Eq.(2.16) are define by depend on the characteristic of the river flow.

$$Le = el/Dh \tag{2.16}$$

where:

Le is length to fully developed velocity profile (m)

Dh is the hydraulic diameter of the pipe; its characteristic travelled length, *L*, (m).

el : for laminar flow = 0.06 *Re* and

for turbulent flow = 4.4 $Re^{1/6}$

By calculation that is made to define the *Le* of the nozzle, now we can define the length of the nozzle, as shown in the Figure 2.7.

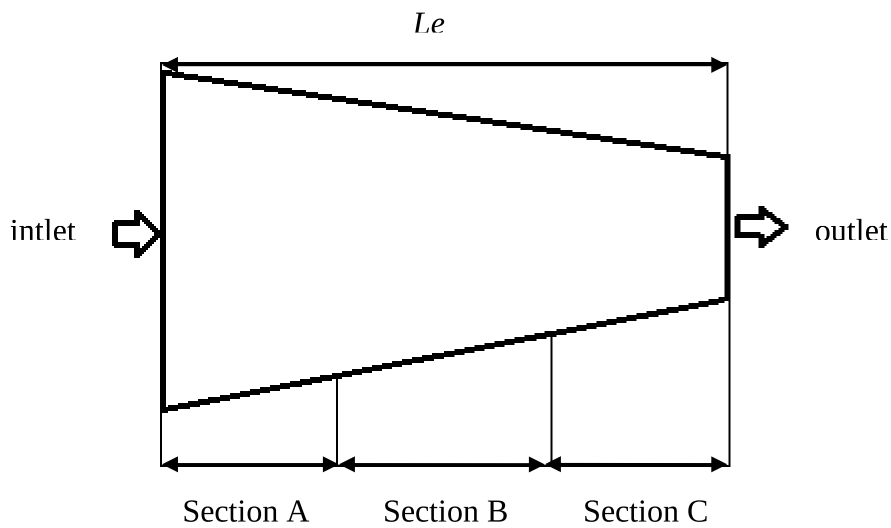


Figure2.7: The maximum length of the nozzle.

Velocities, *V* of the flow rate is defined from the figure are shown in Table 2.1, where the average of the flow rate from January until July yearly that are collected. With considering of the cross sectional area of the Sungai Pahang is approximately 1700 m².

Table 2.1: Flow rate from January until July yearly of Sungai Pahang

Month	Max (m^3/s)	Min (m^3/s)	Average (m^3/s)
Jan	404.3	196.1	260.8
Feb	432.2	169.7	215.8
Mac	583.2	233.8	334.4
Apr	583.2	233.8	334.4
May	734.4	221.9	496.3
June	735.2	230.2	496.3
July	1065.0	371.3	604.4
Average	708.09	252.24	421.44

Source: Makhzan, (2010)

So the value of the *V*is details shown in Eq. (2.17).

$$V = \frac{Q}{A} \tag{2.17}$$

where:

Q is average river flow rate (m^3/s)

A is cross sectional area of the Sungai Pahang is approximately (m^2)

2.4 COMPUTER AIDED ENGINEERING SOFTWARE IMPLEMENTATION

Allows the development of three dimensional (3-D) designs from which conventional two-dimensional orthographic views with automatic dimensioning can be produced. The integration of Computational Fluid Dynamics, CFD methods into a wide range of engineering disciplines is rising sharply, mainly due to the positive trends in computational power and affordability. (Joseph, 2008) Manufacturing tools path can be generated from the 3D model, and in some cases, part can be produced directly for a 3d database by using rapid prototyping and manufacturing methods, paperless manufacturing. The term Computer Aided Engineering, CAE generally applies to all computers related engineering applications. Some software is integrated with this software system that allows us to perform finite element analysis, computational fluid dynamic analysis and simulation.

One advantage of these methods, when used in the race car industry, is the large body of information provided by the solution. Contrary to wind tunnel tests, the data can be viewed, investigated, and analyzed over and over, after the experiment ends. Furthermore, such virtual solutions can be created before a vehicle is built and can provide information on aerodynamic loads on various components, flow visualization.

2.5 PRODUCT MATERIAL

2.5.1 Fiber glass

Fiberglass is a fiber reinforced polymer made of a plastic matrix reinforced by fine fibers of glass. Fiberglass is a lightweight, extremely strong, and robust material. Although strength properties are lower than carbon fiber and it is less stiff, the material is typically far less brittle, and the raw materials are much less expensive. Its bulk strength and weight properties are also very favorable when compared to metals, and it can be easily formed using molding processes.

Fiberglass is an immensely versatile material which combines its light weight with an inherent strength to provide a weather resistant finish, with a variety of surface textures. The 'glass' in fiberglass is the same basic substance used in windows and glassware. Molten glass is extruded through ultra-fine openings measured in microns, resulting in thread-like formations which can be woven together to form a rough cloth or patch. Different resins can then be added to this fiberglass material, allowing it to be formed and pressed into molds. Fiberglass is also highly resistant to environmental extremes. Fiberglass Reinforced Plastic, FRP, does not rust and is highly resistant to corrosion. In fact, the non-corrosive properties of fiberglass give it a much longer life expectancy than metal, wood, and non-reinforced plastics when used in highly corrosive application environments. Fiberglass is chemically inert. This means that it will not react chemically with other substances with which it may come into contact.

This can prevent potentially hazardous and explosive situations that arise with other metallic or petroleum based materials. Fiberglass also has superior and more desirable acoustic qualities than plastic or metal. Under similar conditions fiberglass and composites tend to vibrate less and remain quieter than sheet metals. This can reduce the overall operating volume of the machinery and help to achieve acceptable or required sound levels.

Fiberglass and composites are structurally stable. Fiberglass and composites exhibit the least amount of expansion and contraction with heat and stress compared to plastic, metal, or wood. This means that the body will hold their shape better under severe mechanical and environmental stresses

2.5.2 Stainless Steel

For the nozzle fabrication the stainless steel are used, this is because stainless steel does not corrode, rust or water contaminated with conventional steel, but although full name is not stain-proof, especially under low oxygen, high salinity, or environment of the bloodstream. It is also known as the crescent pending corrosion of steel or alloy type and grade are not detailed, particularly in the aviation industry. There are different grades and surface finishes of stainless steel alloys to suit the environment must endure. Stainless steel is used in which the behavior of steel and resistance to corrosion is required.

The nozzle will sink in the river, so the stainless steel are chosen because of his characteristic that are immune to corrosion, then stainless steel also easy to fabricate, can be welded by Metal Inert Gas, MIG, or Shielded Metal Arc Welding, ARC, weld that are much more cheaper compare to the plasma weld that are used to weld an aluminum.

The stainless steel not such a good conductor, so it suitable for used in the project because of this project are to generate of electricity, so by using the non-conducting material are such a good way to avoid the electricity lose. The material that is used is sheet stainless steel with 1 mm thickness.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter includes the step from the beginning until the end of the process of performance optimization of Cross Flow Small Water Turbine using experimental method. This chapter also explains how the experiment was conducted and methodology of this project can be classified as follows. The methodology experiment is classified in 3 different which are, analysis of existing model design of Small Cross Flow Water Turbine, development of different design with different inlet and outlet of duct and lastly is comparison of the analysis result between the designs that are proposed.

3.2 NOZZLE PARAMETRIC STUDY

As mentioned in Chapter 2, designs of nozzle and rotor that have been developed are based on Eq.(2.14).

a. Nozzle diameter D_H

$$D_H = \frac{4LW}{2L+W}$$

$=4\ (0.294\ \text{m})\ (0.1469\ \text{m})\ /\ 2\ (0.294\ \text{m} + 0.147\ \text{m})$

$=0.1959\ \text{m}$

b. Average flow rate *V*, is calculating by using Eq.(2.17).

$V = Q/A$

$= 421.44\ m^3/s\ /\ 1700\ m^2$

$= 0.25\ m/s$

c. Nozzle length

Reynold number was defining first, by using Eq.(2.15).

$$Re = \frac{\rho v D_H}{\mu}$$

where,

v=0.25 m/s

Q=421.44 m^3/s

ρ=998 kg/m^3

μ=1 kg/(m·s)

D_H=0.1959 m^2

*At T=288 K

$Re = (998\ kg/m^3)\ (0.25\ m/s)\ (0.1959\ m^2)\ /\ (1\ kg/m \cdot s).$

$= 48.877$

(Laminar condition where $Re \leq 2400$)

$el = 0.06Re$

$= 0.06(48.877)$

$= 1.466$

So, by using Eq.(2.16), *Le* is defined.

$Le \quad = el/D$

$= 1.466/0.1959$

$= 1.466$ m

So

$Le < 1.466$ m

0.3 m < 1.466 m

Where *Le* that is chosen is 0.3 m

d. Theoretical value of velocity at the outlet can be define by using Eq.(2.13).

$$A_1V_1 = A_2V_2$$

$$V_2 = (0.0276 \text{ m}^2)(0.25 \text{ m/s}) / (0.0072 \text{ m}^2)$$

$$= 1.5 \text{ m/s}$$

The nozzle design parameters are as shown in Table 3.1:

Table 3.1: Design parameters of the nozzle

Parameter/unit	**Value**
D_H(m)	0.1959
Le (m)	0.3
Ai (m^2)	0.0428
A/B (m^2)	0.0276
B/C (m^2)	0.0167
Ao(*v*)	0.0072
Q (m^3/s)	421.44
V (m/s)	0.25

Initially, there are various aspect need to be considered to fulfill the design requirements. The product must analysis the experimental method in order to produce or analysis the design there are proposed and also selected the best design for the Sungai Pahang, which is need meet a few condition such as the flow rate that are needed to be collected in order to produce a best scale of size oh small turbine water flow. In the Figure 3.2 shown the sectional point for the area of the nozzle at the same length, 0.1 m each.

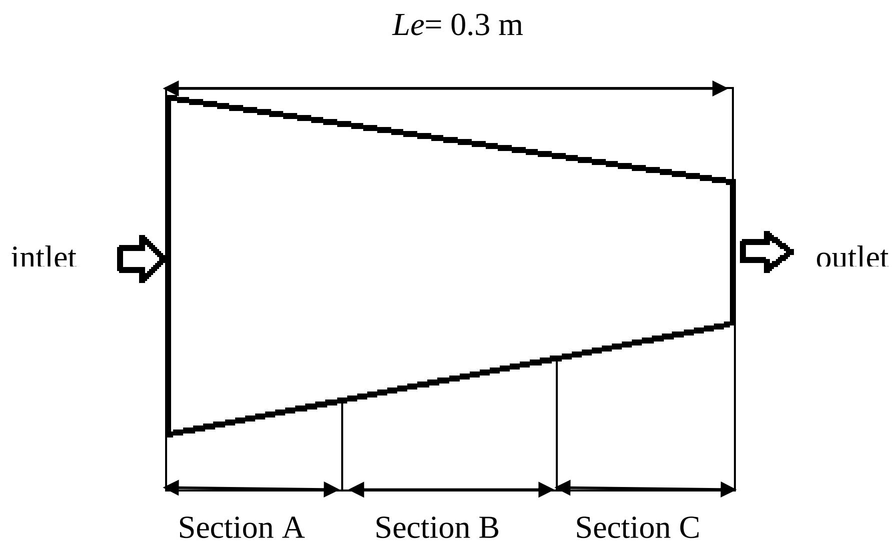


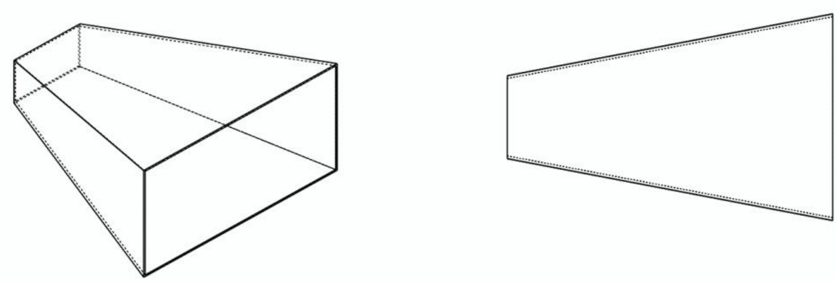
Figure 3.1: Shown the sectional point for the area, where the value of *Le* is 0.3 m

3.3 NOZZLE MODELING DESIGN

By using a design parameter that are calculated detail in Table 3.1, the four (4) different model are made by using by using Computational Aided Engineering, CAE, software that also take an considering design in Figure 2.5. The design is made with a same specification but with different outlet shape and design. Where the two (2) main shapes that are used for sectional area is rectangle and circular shape. The model is shown in the two dimensional, 2-D, and side view of the design.

3.3.1 Nozzle Model 1

As shown in figure below for Figure 3.3, the figure (a) is for 2-D view and (b) is for the side view of the model and the figure are shown is for design Model 1. For design in Model 1 the straight design are made with rectangular shape for input and output of the nozzle, where the rectangular shape are lofted between inlet and outlet straightly.

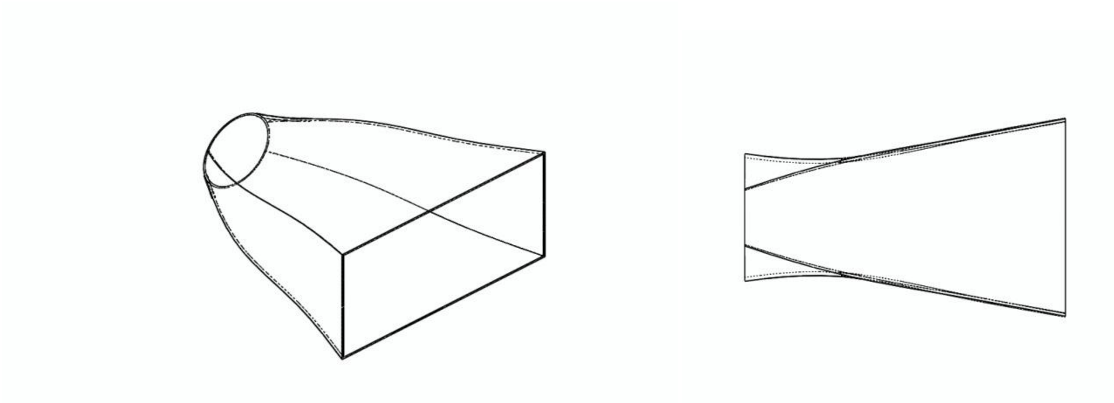


(a) (b)

Figure 3.2: Model 1, (a) for 2-D view and (b) is for side view

3.3.2 Nozzle Model 2

Figure 3.4, is for Model 2, where the outlet of the nozzle is circular shape, with curvature shape, where the design are made by lofted all the shape at each 0.1 m. Where lofted of rectangular inlet, rectangular at A/B cross section and circular shape of outlet.

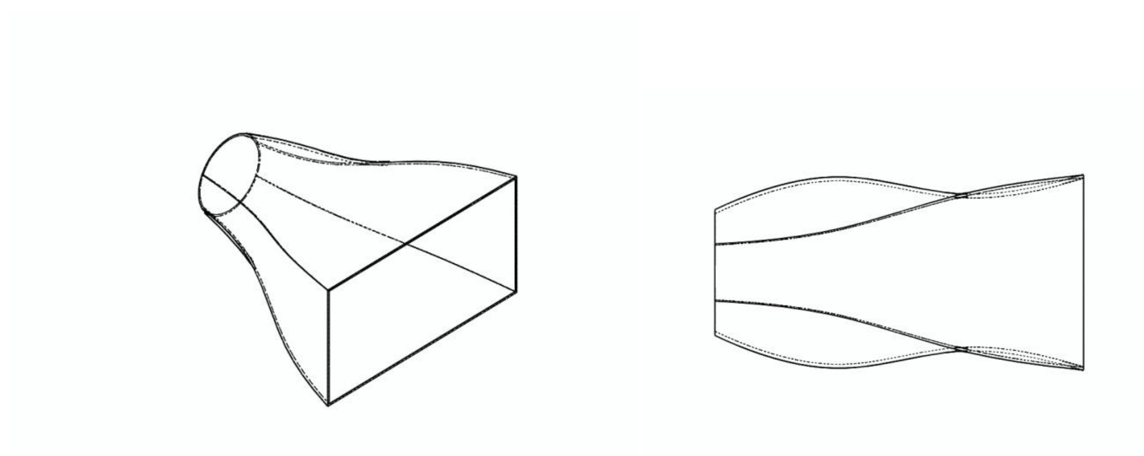


(a) (b)

Figure 3.3: Model 2, (a) for 2-D view and (b) is for side view

3.3.3 Nozzle Model 3

Figure 3.5, (a) is for 2-D view and (b) is for the side view of the model and the figure are shown is for Model 3, where the outlet and the sectional area of B/C of the nozzle is circular shape, where the design are made by lofted the rectangular shape of inlet connected with circular shape at B/C with same area and outlet with also same cross sectional area.

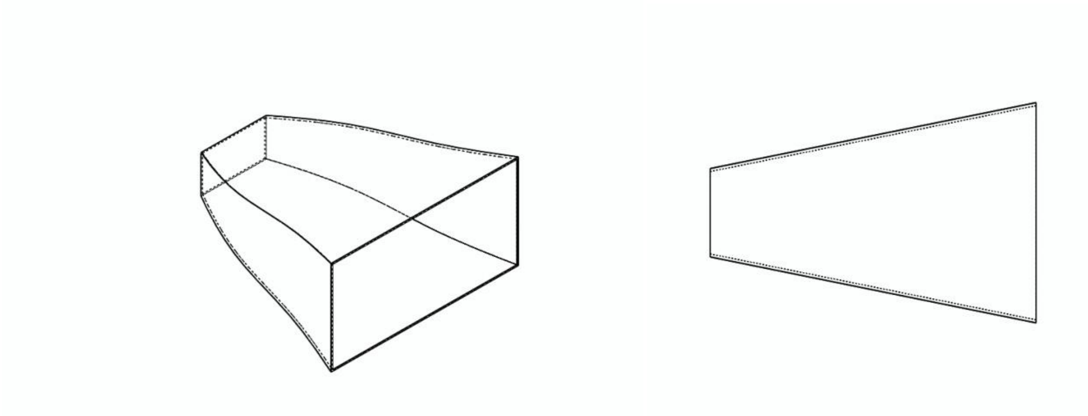


(a) (b)

Figure 3.4: Model 3, (a) for 2-D view and (b) is for side view

3.3.4 Nozzle Model 4

Figure 3.6, (a) is for 2-D view and (b) is for the side view of the model and the figure are shown is for Model 4, where the outlet of the nozzle is rectangular shape, where the design are made by lofted the rectangular shape of inlet connected with rectangular shape at A/B and B/C with same area and outlet with also same cross sectional area.



(a) (b)

Figure 3.5: Model 4, (a) for 2-D view and (b) is side view

3.4 STUDYING THE FLOW

Computational Fluid Dynamics, CFD, became an important tool for studying the flow over complex configuration such as inlet and outlet of the main body it can be used as a preliminary design tool or to complement experimental methods. In providing flow visualization information and details such as the aerodynamic load on an access door, or expected pressure drop across a cooler.

This project, Computer Aided Design, CAD, are used to generate the model and to perform the analysis of fluid dynamic. Computational analysis cannot be considered 100% as the true result. It is impossible to get the exact condition for digital analysis. But the comparison for two different body shapes can be taken as reference of improvement in the real application.

> “Computer software is no substitute for the human thought process. You are the driver here; the computer is the vehicle to assist you on your journey to solution, numbers generated by computer is far from the truth if you entered incorrect input. ”. (J.Keith Nisbet, 2010).

The main point is that computational fluid dynamic approach helps a lot in preparing preliminary result without have to spend expensive investment. It can be tested and analyze single model over and over again. The step of the simulation that is set and run will show, where it set to 15 literation, at 0.25 m/s of velocity flow rate, will be refer to Appendix A.

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

This section presents the results of simulation Computational Fluid Dynamics, CFD, for the nozzle, the leading V_1= 0.25 m/s water flow in the nozzle, which is the outlet is consider as outflow characteristic. The comparison is made by four (4) models that are design.

4.2 CFD SIMULATION RESULT

The simulation results of ducts are mainly of flow characteristics but only velocity at the entrance of the nozzle (inlet) and end of the nozzle (outlet) is taken into consideration, which the values will be used to estimate the output power and its performance. This also to clear it out and to get the result of how many increasing of velocity that is achieved by the water flow that go true in to the nozzle.

4.2.1 Nozzle Model 1

The rough nozzle data are shown in the Figure 4.1 with included another 0.3 m outflow of the same length of the nozzle. Where we can simulate of the flow that are go true the nozzle by comparing the color of the flow, which is the outlet area face a high velocity of flow.

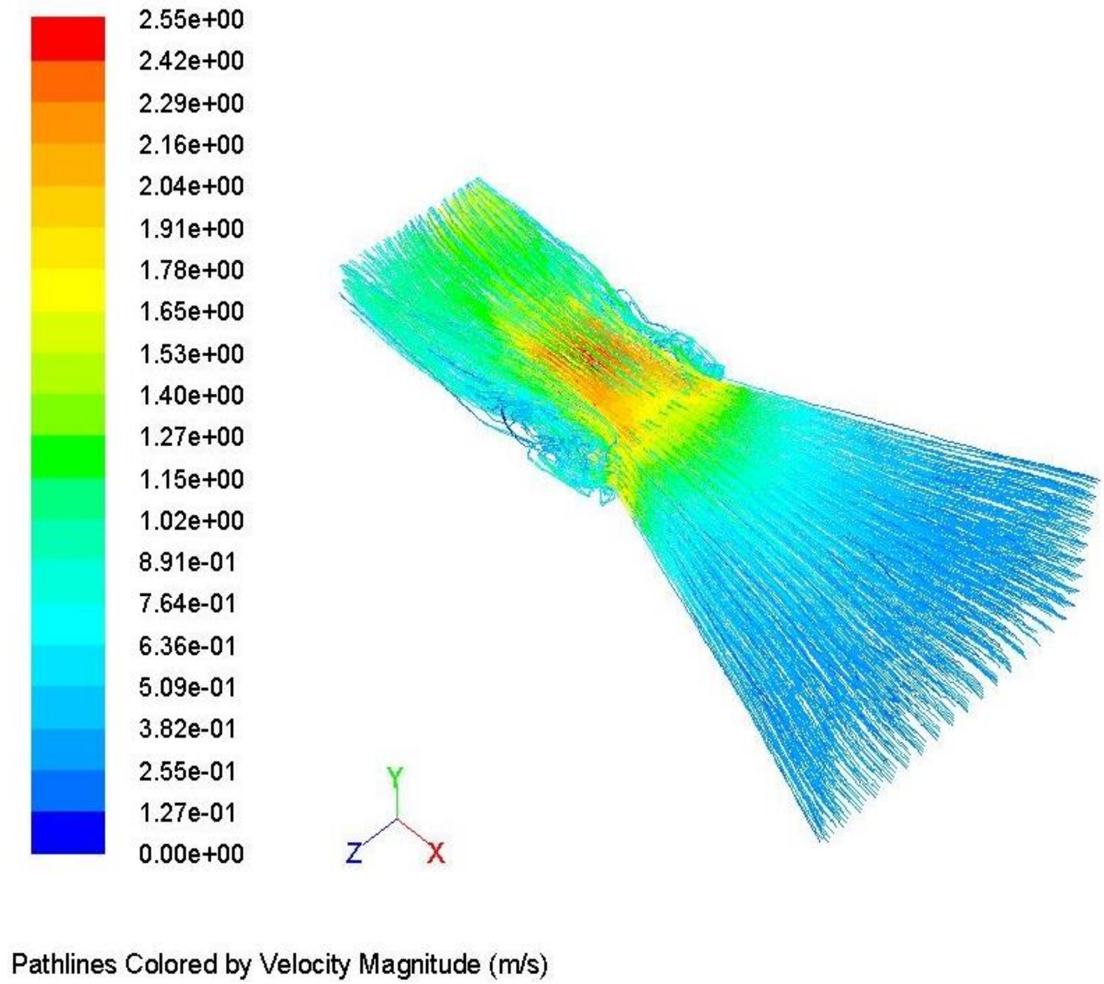


Figure 4.1: Rough data from analysis Model 1

Figure 4.2 velocity along the Model 1 which is 0.25 m/s of initial velocity enter the nozzle are included. The graph is plotted with value of the velocity that is increase versus the 21 position along the model that is tested.

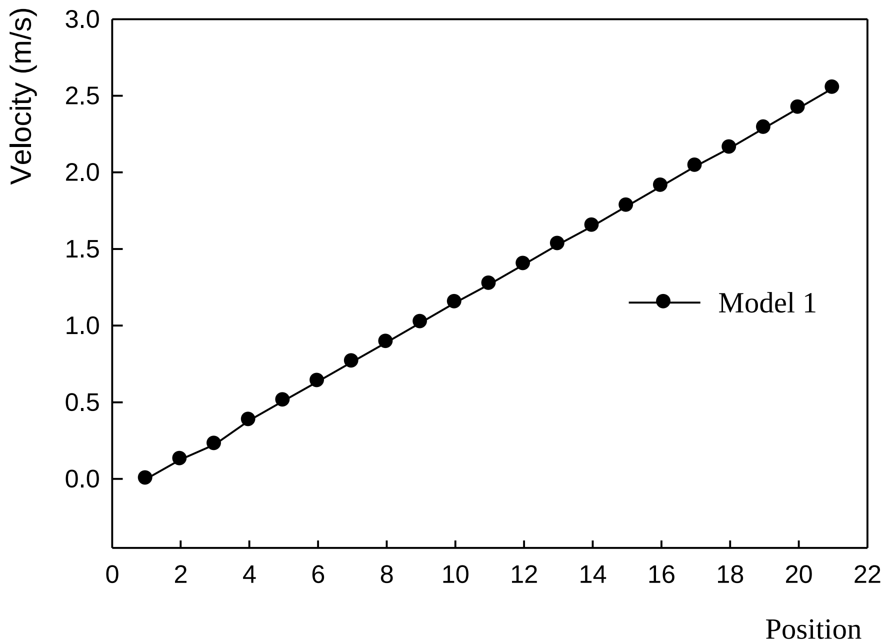


Figure 4.2: Velocity along the Model 1

From Figure 4.1 and Figure 4.2 the model with maximum value of the increasing velocity flow in the inlet until the outlet (outflow) is 2.55 m/s where are plotted in the graph and also data from the rough data in the Figure 4.1, by analysis the outflow at the output of the nozzle, the actual velocity at the output of Model 1 is 1.81692 m/s. In Figure 4.3 are shown the actual output velocity.

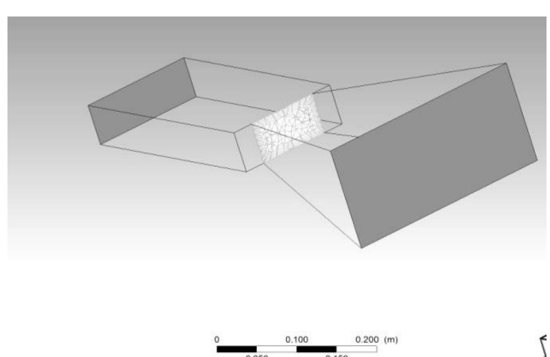


Figure 4.3: Actual velocity at the output of Model 1 is 1.81692 m/s

4.2.1. Nozzle Model 2

The rough nozzle data are shown in the Figure 4.4 with included another 0.3 m outflow of the same length of the nozzle. The simulation result of the flow that are go true the nozzle can be simulate by comparing the color of the flow, which is the outlet area face a high velocity of flow.

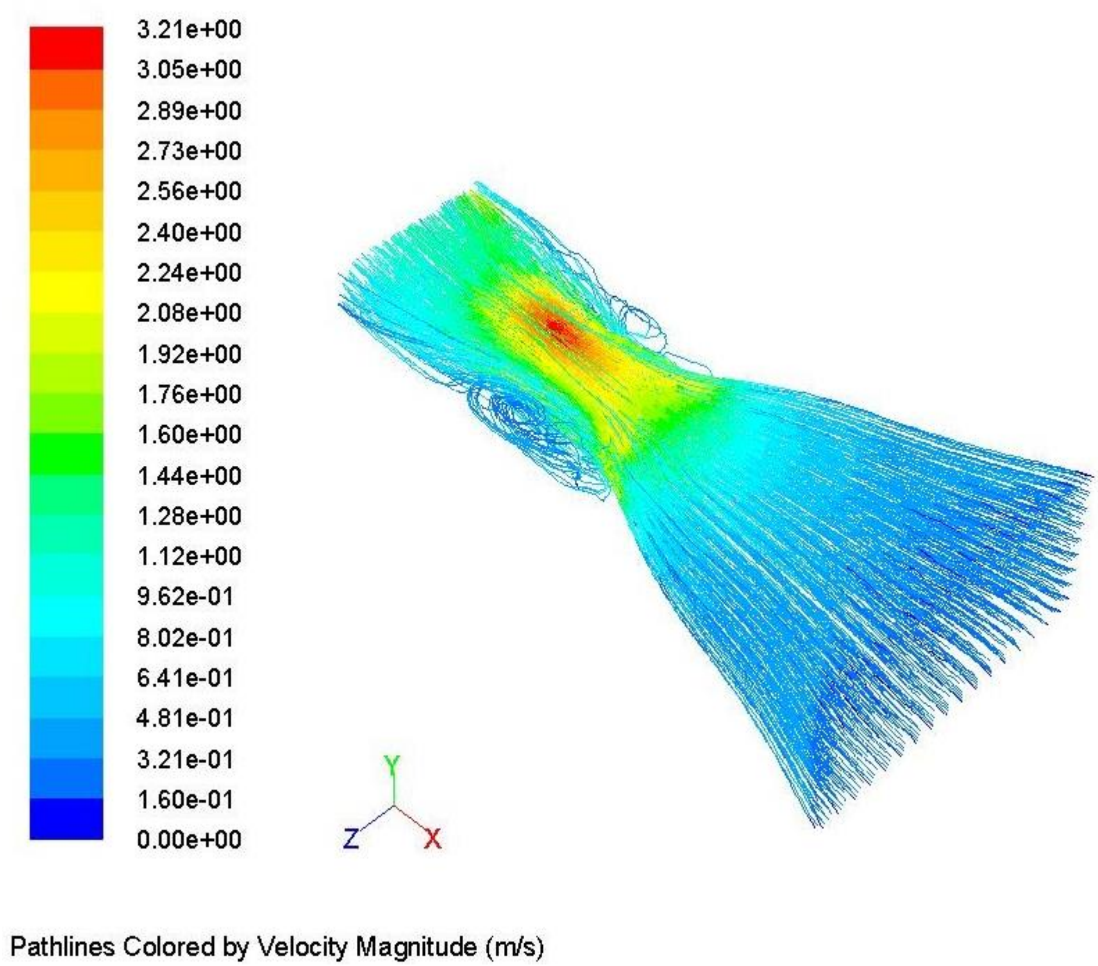


Figure 4.4: Rough data from analysis Model 2

Figure 4.5 velocity along the Model 1 which is 0.25 m/s of initial velocity enter the nozzle are included. Graph is plotted with value of the velocity that is increase versus the 21 position along the model that is tested.

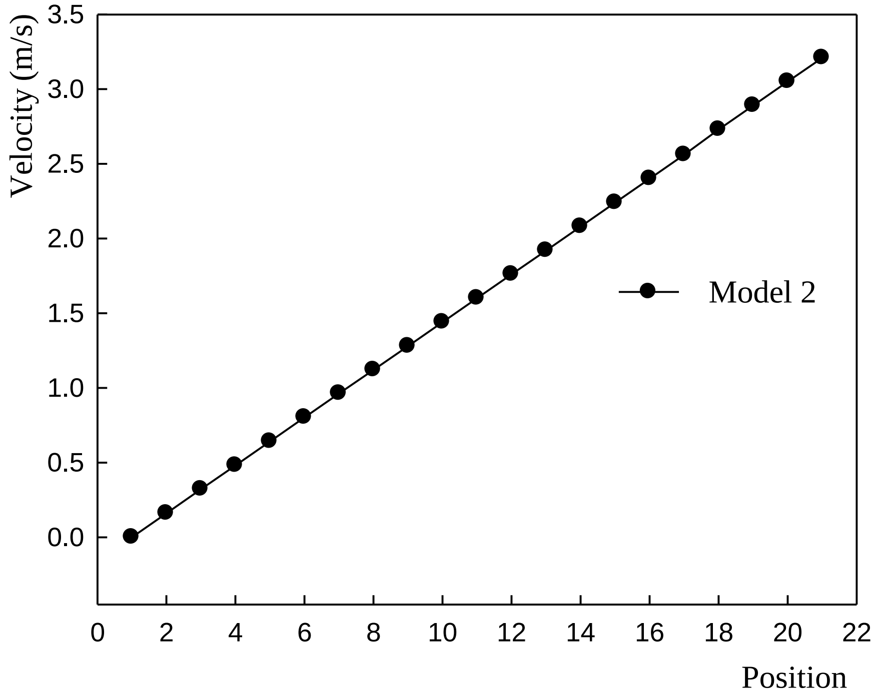


Figure 4.5: Velocity along the Model 2

From Figure 4.4 and Figure 4.5 the model with maximum value of the increasing velocity flow in the inlet until the outlet (outflow) is 3.21 m/s where are plotted in the graph and also data from the rough data in the Figure 4.4, by analysis the outflow at the output of the nozzle, the actual velocity at the output of Model 2 is 2.61925 m/s. In Figure 4.6 are shown the actual output velocity.

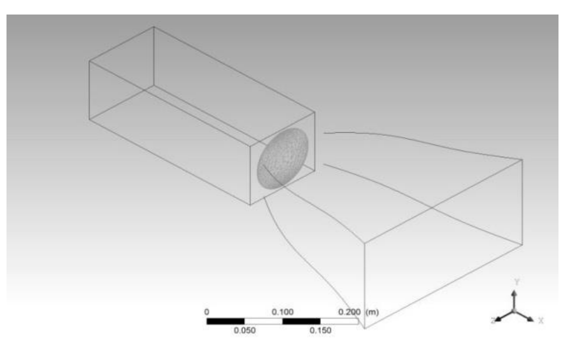


Figure 4.6: Actual velocity at the output of Model 2 is 2.61925 m/s

4.2.1 Nozzle Model 3

The rough nozzle data are shown in the Figure 4.7 with included another 0.3 m outflow of the same length of the nozzle. The simulation result of the flow that are go true the nozzle can be simulate by comparing the color of the flow, which is the outlet area face a high velocity of flow.

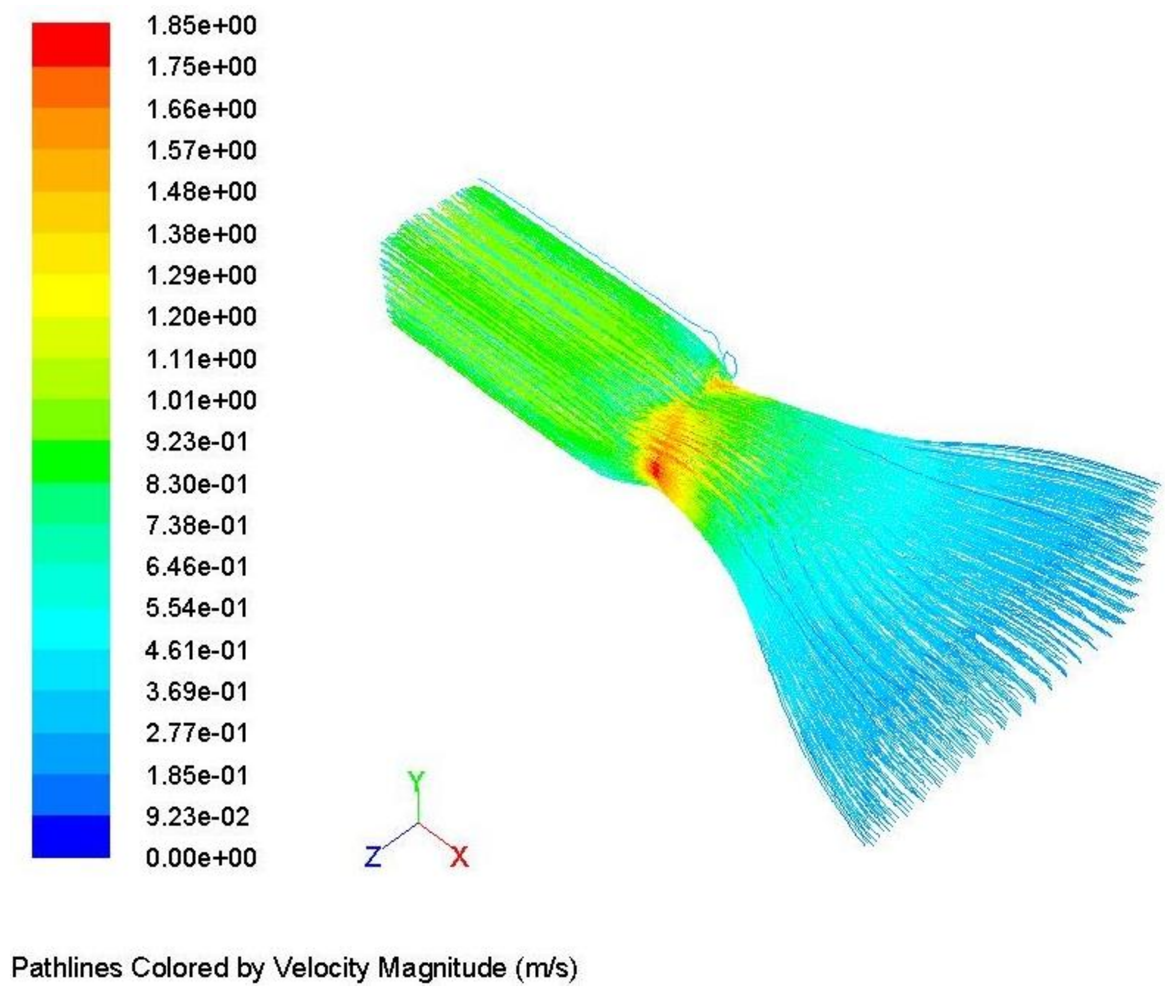


Figure 4.7: Rough data from analysis Model 3

Figure 4.8 velocity along the Model 1 which is 0.25 m/s of initial velocity enter the nozzle are included. Graph is plotted with value of the velocity that is increase versus the 21 position along the model that is tested.

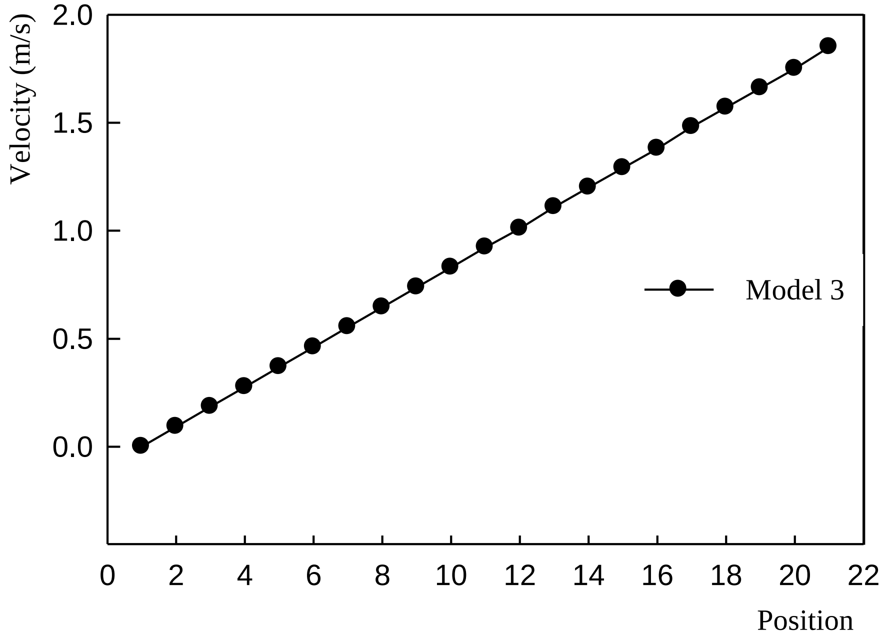


Figure 4.8: Velocity along the Model 3

From Figure 4.7 and Figure 4.8 the model with maximum value of the increasing velocity flow in the inlet until the outlet (outflow) is 1.85 m/s where are plotted in the graph and also data from the rough data in the Figure 4.7, by analysis the outflow at the output of the nozzle, the actual velocity at the output of Model 3 is 1.56995 m/s. In Figure 4.9 are shown the actual output velocity.

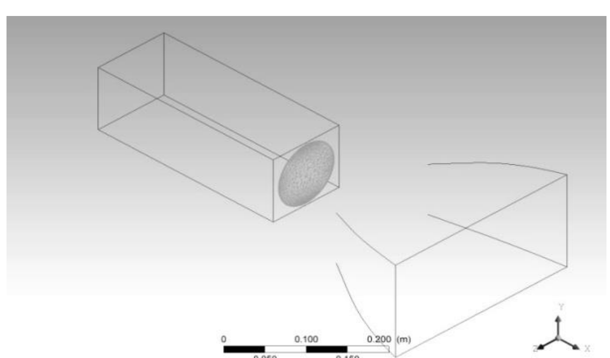


Figure 4.9: Actual velocity at the output of Model 3 is 1.56995 m/s

4.2.1 Nozzle Model 4

The rough nozzle data are shown in the Figure 4.10 with included another 0.3 m outflow of the same length of the nozzle. The simulation result of the flow that are go true the nozzle can be simulate by comparing the color of the flow, which is the outlet area face a high velocity of flow.

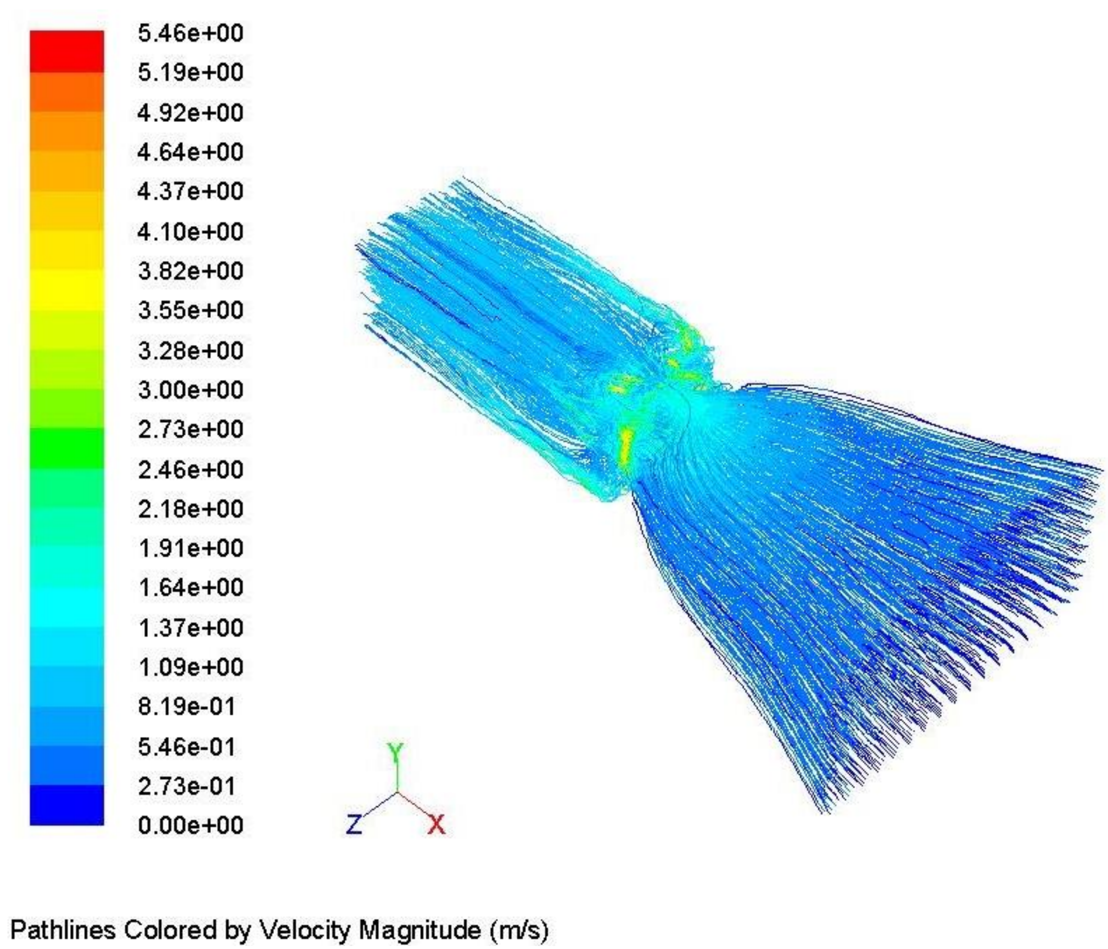


Figure 4.10: Rough data from analysis Model 4

Figure 4.11 velocity along the Model 1 which is 0.25 m/s of initial velocity enter the nozzle are included. Graph is plotted with value of the velocity that is increase versus the 21 position along the model that is tested.

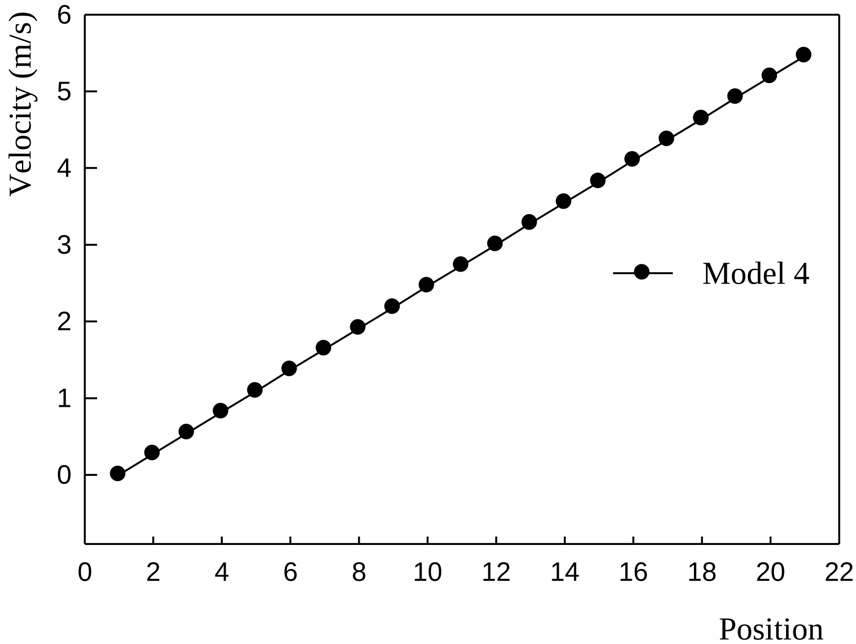


Figure 4.11: Velocity along the Model 4

From Figure 4.10 and Figure 4.11 the model with maximum value of the increasing velocity flow in the inlet until the outlet (outflow) is 5.46 m/s where are plotted in the graph and also data from the rough data in the Figure 4.10, by analysis the outflow at the output of the nozzle, the actual velocity at the output of Model 4 is 1.85099 m/s. In Figure 4.12 are shown the actual output velocity.

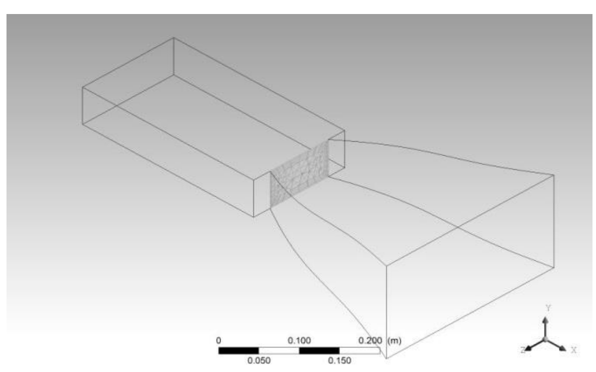


Figure 4.12: Actual velocity at the output of Model 4 is 1.85099 m/s

4.3 RESULT VALIDATION

From the four (4) model results of the nozzle that are go true the flow velocity simulation, we can compare all the model In order to choose the best design where all has same flow rate input at 0.25 m/s from inlet that are go true the 0.3 m long of the nozzle then for this simulation additional 0.3 m outflow body are attach in order to make the flow are clearly shown in the data rough collection with 25 literation. This also to clear it out and to get the result of how many increasing of velocity that is achieved by the water flow that go true in to the nozzle. Figure 4.13 are shown the plotted graph that are compared all four (4) model.

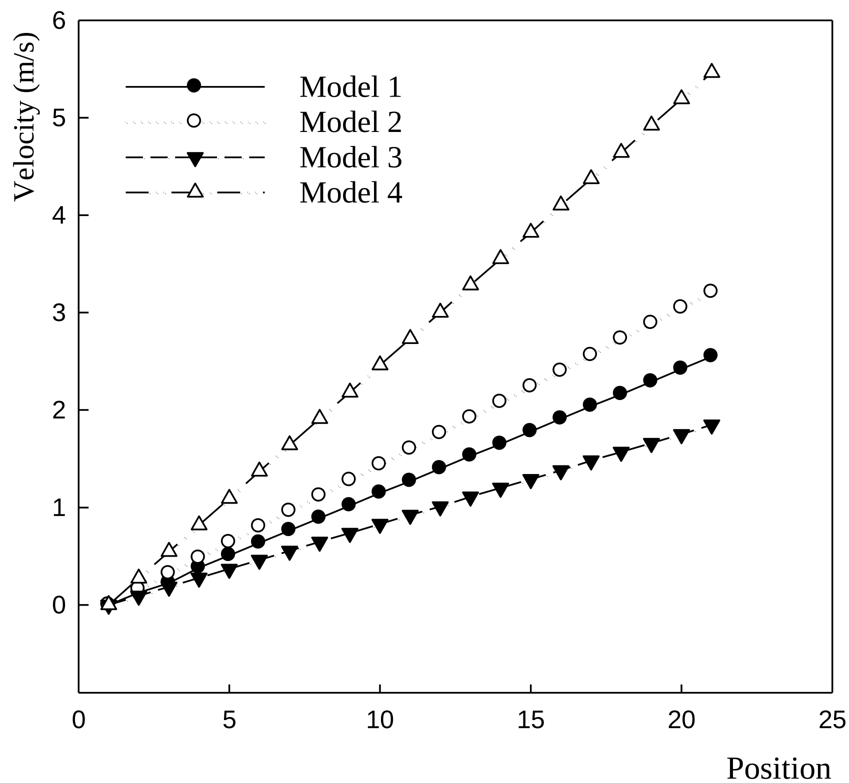


Figure 4.13: All four (4) Models plotted rough result

By referring the Figure 4.13, it shown that an the higher and average velocity flow that are go true the nozzle shown that the Model 4 is the most highest increasing, with the inlet are constant for all nozzle, at V_1= 0.25 m/s. In Table 4.1 are table of the result velocity at the output for each model, Table 4.2 is compared the Model 1, Model 2, Model 3, and Model 4 at the outlet reading velocity of the flow compared to the theoretical velocity output , V_2, from the calculation.

Table 4.1: Velocity at the Model outlet

Model	V_2 at output (m/s)
1	1.81692
2	2.61925
3	1.56995
4	1.85099

From the Eq. (2.13), so we know that theoretical value of the velocity outlet, $\dot{V}_2$is 1.5 m/s. from the Table 4.1 then the actual velocity outlet of the nozzle is compared to the actual velocity outlet, V_2 of each of the model nozzle. In Eq.(4.1), will show details about the equation of the percentage compared (%) of the each nozzle.

$$\text{Percentage compared (\%)} = \frac{V_2 - 1.5}{1.5} \times 100 \qquad (4.1)$$

where:

V_2 is the value of the outlet velocity of each nozzle (m/s)

Table 4.2: Percentage compared to the theoretical velocity output, V_2.

Model	*V2* (m/s)	Percentage compared (%)
1	1.81692	21.128
2	2.61925	74.617
3	1.56995	4.663
4	1.85099	23.399

Table 4.2 shown that Model 2 has the highest of the output velocity reading compared to the rest, and it also has highest the percentage (%) at 74.62%. This is because the curve curvature design of this model allows the water flow smoothly with minimum drag and surface friction among the water flow and the surface of the nozzle.

CHAPTER 5

CONCLUSION AND RECOMANDATION

5.1 CONCLUSION

The main goal of this project on the succeeded in develop a small cross flow turbine for Sungai Pahang is to overcome a few main problem in world now day, where we are stuck between a need energy and also in same time to maintain the environment landscape without doing any harm either to flora or fauna.

The following conclusion is directly contribute to the characteristic of Sungai Pahang where has a low current speed flow just only 0.25 m/s and flow rate at 421.44 m^3/s average yearly. This also considering the change of the season, such as raining season and the monsoon season, this is because Pahang are located in the east coast of Malaysia. Because of that the small cross flow of this project, one of the objective is to increase the current speed of the flow rate.

Nozzle design are made of the 4 main design where has a same length, *L*, the same area for each path, A, area inlet 0.0428 m^2, area between section A and B is 0.0276 m^2, area between section B and C is 0.0167 m^2 and lastly the outlet is 0.0072 m^2 and add on 0.3 m of the outflow at the outlet is another 0.3 m to make the flow can see clearly. From the model that undergo the simulation, it can be conclude that the best design for Sungai Pahang is design Model 2, because of its highest velocity of water at the output end of nozzle, which up to 2.61925 m/s, but the Model 1 are choose to be fabricate and used to take it value which up to 1.81692 m/s and this value is taken to be used in rotor blades simulations.

Models 1 are chosen because of the simple shape of the design, which is easy to fabricate, design and maintenances, because of this reason with simple design a fabrication work are much more easy, where it also effect the labor cost to fabricate this model, this design took a short time to fabricate and doesn’t need a mold compared to the best design, Model 4 where to fabricate the design need a mold because of the curvature shape of the model. And because of the nozzle made of the stainless steel 1 mm the production work are much easier for Model 1. And if the best designs Model 2 are choose, to fabricate the nozzle using a stainless steel will need a stamping machine which is take a lot of cost.

Rotor blade is designed by the another parties that are attach to this project, by using a velocity output of design Model 1 at 1.81692 m/s and this value is taken to be used in rotor blades simulations as.

5.2 RECOMMENDATION AND FUTURE WORK

There is much way to improve the project either in design, simulation and collecting the data. There are some recommendations that should be taken into consideration for future work n order to obtain the best design such as:

i. A detail study about flow rate of the Sungai Pahang for the whole year, which include a different speed at a different depth, place and area of the Sungai Pahang where its effect the performance of the rotor.

ii. Improvement of the nozzle design by attaching a settling chamber in front, by let this attach to the design the turbulent of the water flow can be less, so it will increase the performance of the nozzle where produce a better and higher velocity of the flow rate at the outlet.

iii. The best clearance of the space between nozzle outlet and the rotor blade need to be investigated in order avoid the loss during flow of the water deliver to the rotor blade.

iv. A restoration of the turbine in the river should be consider, by proposed the best method in order to minimize the maintenance job and also restoration cost.

The simulation results of ducts are mainly of flow characteristics but only velocity at the entrance of the nozzle (inlet) and end of the nozzle (outlet) is taken into consideration, which the values will be used to estimate the output power and its performance. This also to clear it out and to get the result of how many increasing of velocity that is achieved by the water flow that through in to the nozzle.

REFERENCES

Islam M., Fartaj A. and Ting D (2003).*Current Utilization and Future Prospects of Emerging Renewable Energy Applications in Canada.*2003

Fraenkel, P.L.19 March 2004. *Marine Current Turbines: an emerging technology*. Paper for Scottish Hydraulics Study Group Seminar in Glasgow

Akwensivie, F. (2004).*In The Wake of a Marine Current Turbine.* Master Thesis. University of Strathclyde, Glasgow

TengkuAb Rashid, T.M.A. (2000).*Preliminary Study of Ocean Energy Device.* Bachelor Degree. Universiti Teknologi Malaysia

Boyle G. *Renewable energy—power for a sustainable future*. New York: Oxford University Press Inc.; 1996.

White FM. *Fluid mechanics. 3rd ed*. New York: McGraw-Hill Inc.;1994.

El-Wakil MM. *Power plant technology—international edition*. New York: McGraw-Hill Inc.; 1984.

Fox, J. A. 1989. *Transient flow in pipes, open channels and sewers*. John Wiley and Sons.

D.Agar, M. Rasi. (2008). *Renewable Energy 33*

Azliza, A. A. (2010). *Development of a ducted horizontal axis marine current turbine rotor*. Master. Universiti Teknologi Malaysia.

Appendix A

a. Defining the unit system. The SI system is applied to simulate the flow computation.

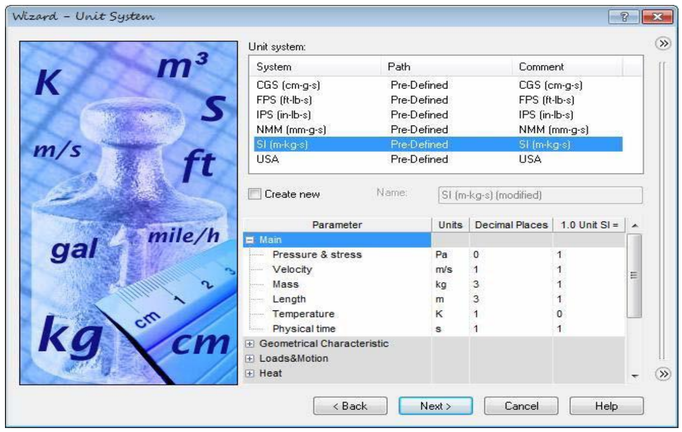


Figure 3.7: Defining the unit system in FloWorks.

b. Defining analysis type.

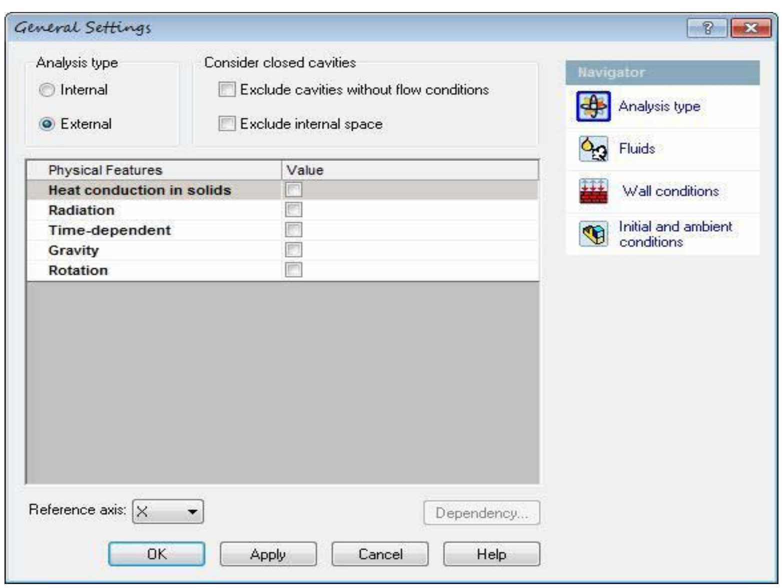


Figure 3.8: Defining analysis type in FloWorks.

c. Defining the type of the fluids. The fluid is chosen as water (liquids)

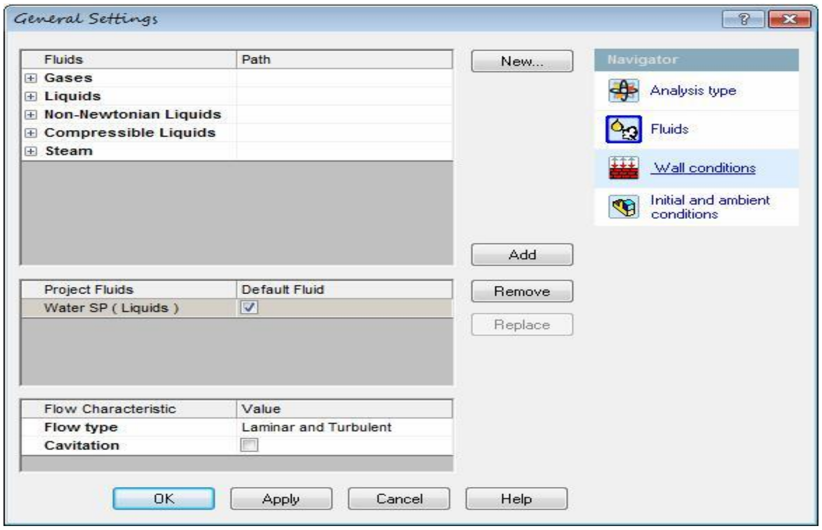


Figure 3.9: Defining type of the flow in FloWorks.

d. Defining the initial condition.

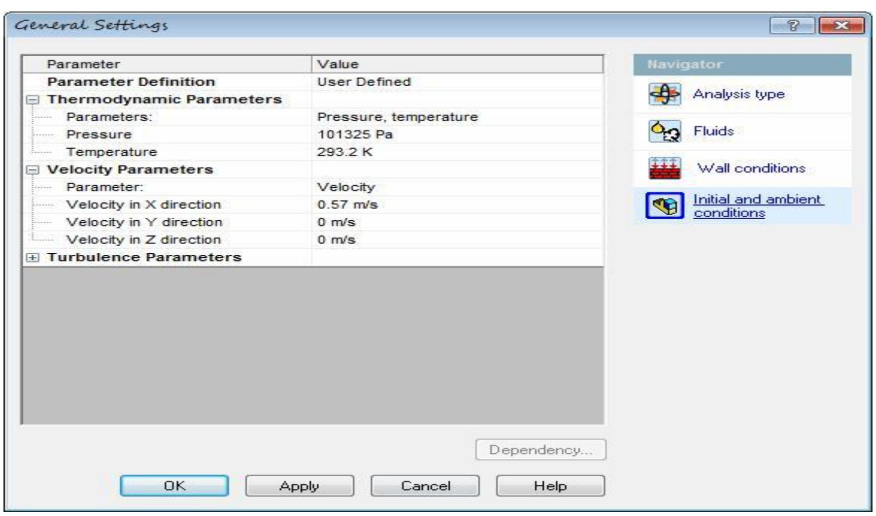


Figure 3.10: Defining the initial condition in FloWorks.

e. Defining the initial mesh.

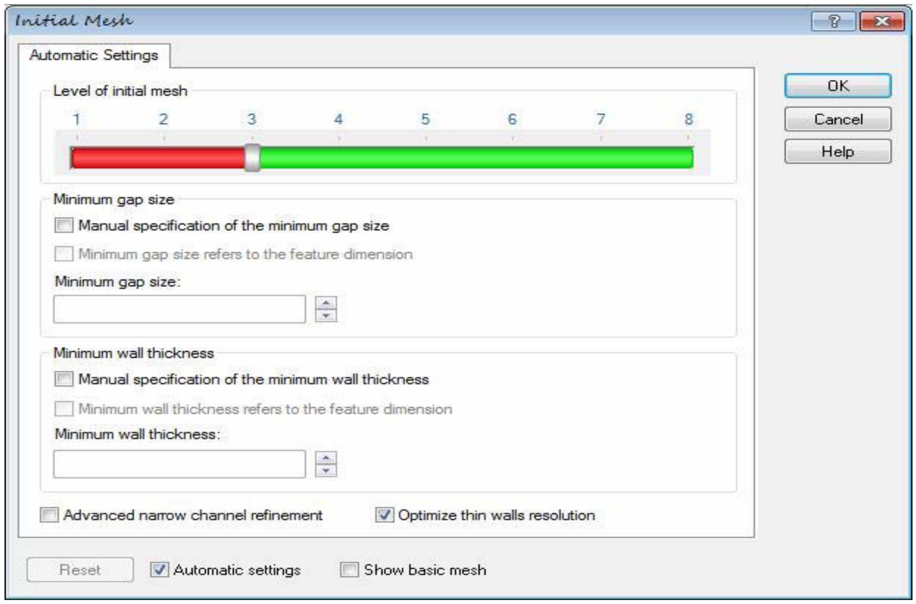
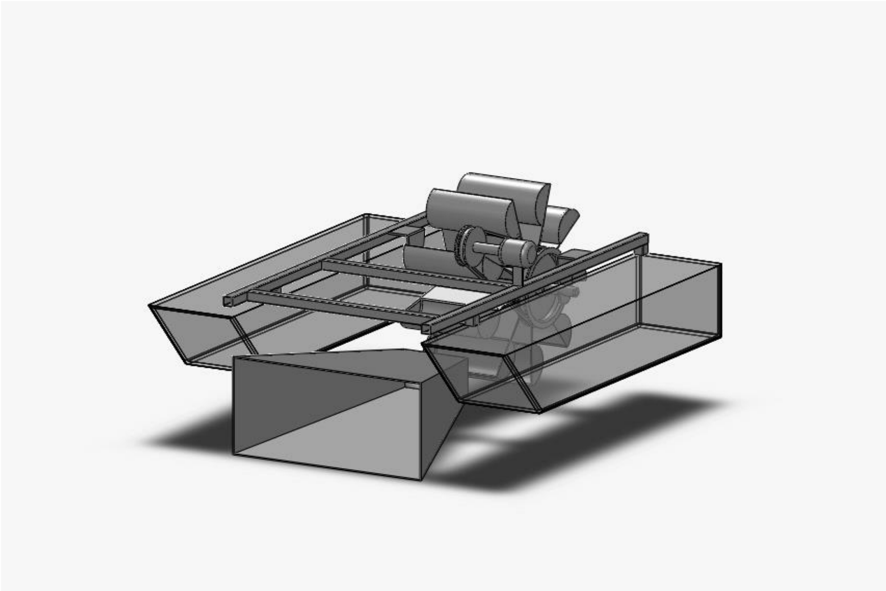


Figure 3.11: Defining initial mesh in FloWorks.

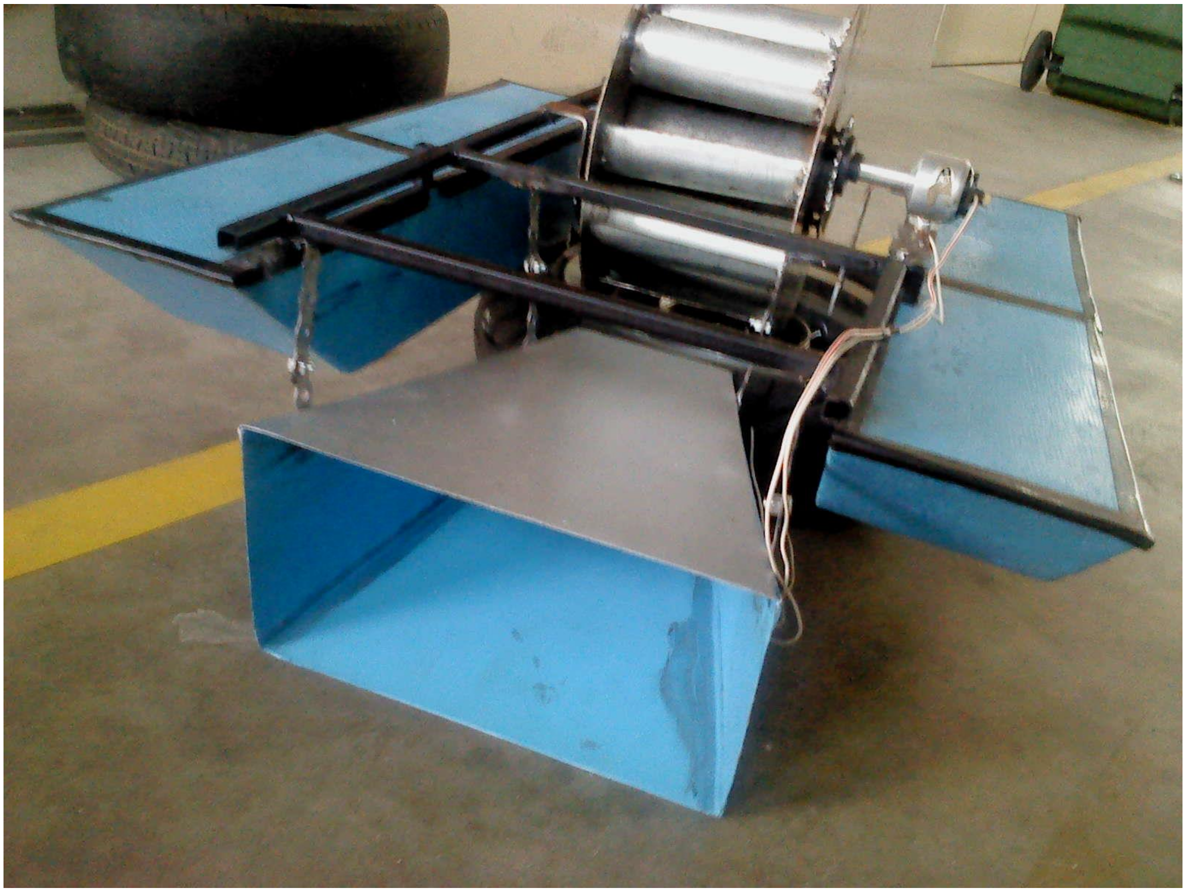
Appendix B

Full feature of the complete design and also a fabricating product.



Feature of the complete

Nozzle Model 1 Final Product



Fabricating product