# ANALYSIS OF PROPELLERS FOR MICRO HYDRO POWER FOR DOMESTIC AND COMERCIAL LOADS

ABID ABBAS BIN MOHAMED ARIFIN

Thesis submitted to the Faculty of Mechanical Engineering in
partial fulfillment of the requirement for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2013

# EXAMINER'S DECLARATION

I have examined the final copy of this report and in my opinion, it is fully adequate in terms of language standard, and report formatting requirement for the award of the degree of Bachelor of Mechanical Engineering.

Signature :

Name of Examiner : AP. Dr. Tuan Mohd Yussof Shah Tuan Ya

Position : Associate Professor

Date : 26 June 2013

# SUPERVISOR'S DECLARATION

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

Signature :

Name of Supervisor : Mr. Muhammad Ammar Nik Mu'tasim

Position : Senior Lecturer

Date : 26 June 2013

## STUDENT'S DECLARATION

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

Signature :

Name : Abid Abbas Mohamed Arifin

ID Number : MA09086

Date : 26 June 2013

بِسْمِ ٱللَّهِ ٱلرَّحْمَٰنِ ٱلرَّحِيمِ

In the Name of Allah, the Most Gracious and the Most Merciful

Specially dedicated to

My beloved family and those who have guided and inspired me

Throughout my journey of learning

## ACKNOWLEDGEMENTS

In the name of Allah, the most Benevolent, the most Merciful. First of all, I wish to record immeasurable and thankfulness to the One and The Almighty Creator, the Lord and Sustainer of the universe, and the Mankind in particular. It is only through His mercy and help that this work could be completed, and it is ardently desired that this little effort be accepted by Him to be of some service to the cause of humanity.

I am grateful and I would like to express my sincere gratitude to my supervisor, Muhammad Ammar bin Nik Mu'tasim for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, and he belief that a Degree of Bachelor program is only a start of a life-long learning experience. I appreciate his consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naive mistakes, and his commitment to my future career.

My sincere thanks go to all my focus group members and the members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I am also grateful to family members for their sacrifice, patience, and understanding that were inevitable to make this work possible. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. Special thanks should be given to my friends. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

## ABSTRACT

The reaction turbine technology for low head application in the micro hydro power range has been vastly neglected despite niche available in scattered region of valley flows as well as in waste water canals and other energy recovery schemes, where the available head does not exceed two meters. The main objective for this study is to develop the design of propeller turbine by using Computational Fluid Dynamic (CFD) for the micro hydro power range with a particular focus on ease of manufacture. Computational Fluid Dynamic (CFD) has become the most effective tool for predicting detailed information relating the flow in turbine space to enable the selection of the best design. In present work, three dimensional turbulent real flow analysis in propeller have been carried out with variety of geometrical optimization on a propeller runner, whose blades have been designed using the free vortex theory. All the designs have been analyzes with same initial and boundary condition by using Ansys CFX computational fluid dynamic (CFD) software. This paper also presents the methodology to analyze the effect of each optimization stages. This method shows the relative changes to shaft power and water flow at constant boundary condition. The average values of flow parameters like velocities and flow angles at the inlet and outlet of runner, guide vane of turbine are computed to derive flow characteristics. It was found that the performance of the runner were got effected in the changes of geometries of the propeller. The results show the modification has increase the efficiency of propeller to catches the energy from the water from 27.2% to 55.4%.

# ABSTRAK

Teknologi turbin tindak balas untuk aplikasi turus rendah di kuasa mikro hydro telah jauh diabaikan walaupun khusus terdapat di kawasan yang bertaburan di lembah aliran serta di terusan air sisa dan skim pemulihan tenaga yang lain, yang mana turus yang ada tidak melebihi dua meter. Objektif utama kajian ini adalah untuk membangunkan reka bentuk kipas turbin dengan menggunakan Pengiraan Bendalir Dinamik (CFD) untuk ruang skop kuasa hidro mikro dengan memberi tumpuan khusus kepada kemudahan dalam pembuatan. Pengiraan Bendalir Dinamik (CFD) telah menjadi alat yang paling berkesan untuk meramalkan maklumat terperinci mengenai aliran di ruang turbin untuk membolehkan pemilihan reka bentuk yang terbaik. Dalam kerja-kerja ini, tiga dimensi analisis aliran bergelora sebenar dalam kipas turbin telah dijalankan dengan pelbagai penambahbaikan geometri kipas turbin yang mana bilahnya telah direka dengan menggunakan teori vortek bebas. Semua reka bentuk yang telah dianalisis dengan keadaan awal dan sempadan yang sama dengan menggunakan perisian ANSYS CFX pengiraan bendalir dinamik (CFD). Kertas ini juga memaparkan kaedah untuk menganalisis kesan setiap peringkat penambahbaikan. Kaedah ini menunjukkan perubahan relatif kepada kuasa acid an aliran air pada keadaan sempadan yang sama. Nilai purata parameter aliran seperti halaju dan sudut aliran di bahagian masuk dan keluar daripada bilah turbin, panduan ram turbin dikira untuk mendapatkan ciri-ciri aliran. Berdasarkan analisis yang telah dijalankan mendapati bahawa perubahan geometri turbin telah memberikan kesan kepada prestasi turbin. Keputusan menunjkkan pengubahsuaian telah meningkatkan kecekapan kipas turbin untuk menangkap tenaga daripada air dari 22.7% kepada 55.4%.

# TABLE OF CONTENTS

**CHAPTER 5: CONCLUSION AND RECOMMENDATION**

## LIST OF TABLES

## LIST OF FIGURES

# LIST OF ABBREVIATIONS

| | |
|---|---|
| $N$ | Rotational speed (rpm) |
| $V$ | Absolute velocity |
| $U$ | Tangential velocity |
| $W$ | Relative velocity |
| $V_\theta$ | Tangential component of Absolute velocity |
| $V_f$ | Flow Velocity (Radial component of Absolute velocity) |
| $\alpha_1$ | Guide vane angle |
| $\beta_1$ | Inlet vane angle |
| $\beta_2$ | Outlet vane angle |
| $P_{IN}$ | Work generate by Francis turbine |
| $H_G$ | Gross head |
| $H_L$ | Head losses (due to friction in pipes) |
| $H$ | Effective head / Net head |

# CHAPTER 1

# INTRODUCTION

## 1.1 MICRO HYDRO POWER

The increasing consumption of conventional fuels coupled with environmental pollutions has led to the development of renewable energy sources which are naturally replenished. Nowadays, one of the main and most widely utilized sources of renewable energy is hydro power that generates power by using the potential energy of water. As a renewable energy, it is extremely site specific, economically feasible, the technology being used was proven from large to small scale depend on the demands.

In the modern world, the development of rural areas it is widely recognized that Micro Hydro Power (MHP) may play an important role in the power sector development with applying of specific technology which technically and economically feasible and socially acceptable. This is because the cost to supplying of electricity through the grid extension is very expensive base on the distances and not economically feasible for the remote or rural villages of most developing countries. In generating the electricity, Micro Hydro Power (MPH) is the best solution presents many advantage over the commonly used diesel generators.

The main part of micro hydro generator to ensure the generating of electricity from the potential energy of water is propeller. The design of propeller will give the effects in generating electricity base on the some properties of that should be considered.

**Table 1.1**: Classification of hydropower by size

| Types | Explanations |
|---|---|
| Large- hydro | More than 100 MW and usually feeding into a large electricity grid |
| Medium-hydro | 15 - 100 MW - usually feeding a grid |
| Small-hydro | 1 - 15 MW - usually feeding into a grid |
| Mini-hydro | Above 100 kW, but below 1 MW; either stand alone schemes or more often feeding into the grid |
| Micro-hydro | From 5kW up to 100 kW; usually provided power for a small community or rural industry in remote areas away from the grid |
| Pico-hydro | From a few hundred watts up to 5kW |

Source: Herzog et. al., (2000)

## 1.2 HISTORY

The use of hydropower as a source of mechanical energy dates back more than 2,000 years to the earliest waterwheels. Such wheels in one form or another were the primary source of power for many centuries. French engineers started making improvements in waterwheels in the mid 18th century and continued to lead the field until the mid 19th century. A French military engineer, Claude Burdin (1790-1873), first used the term "water turbine" from the Latin turbo, that which spins. (Although water wheels fit this definition, they are not now classed as turbines by most of those working in the hydropower field). The first commercially successful new breed of hydraulic turbine was a radial-outflow type. The water entered at the center of the turbine and flowed outward through the turbine runners (blades). The turbine was developed by a student of Burdin, Benoit Fournegron (1802-1867). In 1836, a patent was awarded to Samuel B. Howd of Geneva, New York for a radial "inflow" turbine.

The idea was perfected by James B. Francis and Uriah A. Boyden at Lowell, Massachusetts in 1847. In its developed form, the radial inflow hydraulic turbine, now known as the Francis turbine gave excellent efficiencies and was highly regarded.

Another class of turbine used the concept of a vertical wheel driven by a jet of water applied at one point in its circumference. The approach led ultimately to the Pelton wheel, which uses a jet or jets of water impinging on an array of specially shaped buckets closely spaced around the rim of a wheel. The Pelton wheel was developed at the end of the 19th century by a group of California engineers, among them Lester A. Pelton (1829-1908).

The generation of electric power from flowing water has been a source of energy in the United States for a century. The first electricity from hydropower was produced in 1882 by a 12.5-kilowatt (kW) plant in Appleton, Wisconsin. Since then, the number of hydroelectric power generating facilities in the U.S. has grown to more than 1,300, and total capacity now surpasses 76,000 megawatts (MW).

Early hydroelectric power plants were small, and the power they produced went to nearby users. But by the early 1900s, design and engineering advances had opened the way for larger facilities and greater transmission distances. Improvements in dam construction equipment and techniques made much larger dams possible, while the use of alternating current generators, transformers, and the development of suspension-type insulators led to long-distance, high-voltage power transmission systems.

By the 1920s, emphasis had shifted to the development of large hydroelectric power projects, and as time went by, smaller developments those under 25 MW were more and more ignored. During the 1950s and 1960s, a combination of economic factors, the need to replace worn out turbine generator equipment and the availability of inexpensive fossil fuel and made it appear that a number of smaller hydropower facilities built early in the century had outlived their usefulness, and many of these were shut down and disposed of by their owners. Recently, however, the rapidly rising costs of fossil fuels and the high cost of meeting environmental standards for new thermal power plants have prompted a new look at hydropower's role in the national energy picture. And because almost all of the economically feasible and environmentally acceptable sites for large hydropower projects have already been developed, this new look at hydropower is focusing on smaller installations. (Mckinney *et al*, 1983)

## 1.3 PROBLEM STATEMENTS

The use and development of renewable energy sources can enhance diversity in energy supply markets, contributing to long-term supply of sustainable energy, helping to reduce local and global atmospheric emissions, and provide an attractive option in commercial terms to meet certain energy services, especially in developing countries and rural areas. One of the renewable energy sources is water. (Herzog *et al*, 2000). The propeller is an important component in Hydro Power that is responsible for capturing the most possible energy from the water. Regardless of the propeller type, efficiency is in the details. Each propeller type can be designed to meet vastly different requirements and for each of differences in specifications may affect the efficiency of power transfer. (Schumacher, 2000).

## 1.4 OBJECTIVES OF THE PROJECT

The main objective of this project is to conduct the computational fluid dynamic (CFD) analysis of propeller for Micro Hydro Power. The overall objectives are:

i. To design propeller base on the past research
ii. To analyze propeller using the software Computational Fluid Dynamic (CFD), ANSYS
iii. To compare the results of experiment and analysis results of the propellers design

## 1.5 SCOPE OF STUDY

This project concentrates on the performance of the axial blades base on the design. The scopes of study are as follows:

i. 3D modeling using SolidWorks base on concept design of blades
ii. Computational fluid dynamics analysis using ANSYS (CFX)
iii. Implement the initial condition and boundary condition
iv. Comparisons between simulation and experiment
v. Transition flow when propeller remain is static (U=0)

# CHAPTER 2

# LITERATURE REVIEW

## 2.1 INTRODUCTION

The purpose of this chapter is to provide a review of the propellers, past researches of the micro hydro power, performance characteristics of propeller, equation involve in this project and the simulation of propellers by using CFD. The review is fairly detailed so that the present research effort can be properly modified to add to the present body of literature as well as to justify the scope and direction of present research effort.

## 2.2 MICRO HYDRO POWER CHARACTERISTICS

Micro-hydro is hydropower that produces electrical output in range of 5 kilowatts to 100 kilowatts (kW), sufficient to power light bulbs, radios, televisions, refrigerators and food processors and so on. Hydropower systems of this size benefit over the larger systems in terms of cost and simplicity of design. Recent innovations in micro-hydro technology have made it an economic and versatile source of power even in some of the world's most resource-poor and inaccessible places. Standard AC electricity can be produced and distributed throughout a village to power electrical appliances, or it can charge large batteries for households. Recently, many researchers have been studied about the micro hydro scheme in order to help people in the remote area to get access with the electricity. Pico hydro has negligible environmental effect since large dams are not involved, and the scheme can be maintained and manage by the local village. In the third world country especially in the remote areas, another electrification sources such battery system and solar home system mostly only affordable for upper and middle class households only and the cost very expensive and inefficient to be use in the communities in the remote area. (Maher and Williams, 2003)

The suitable site for micro hydropower is usually the lowest cost option for off-grid rural electrification, and is environmentally sustainable. The technology has been developed for a wide range of site conditions, but the design, even for such small schemes, is usually site specific. In order to achieve low installation cost per unit power output, and hence low energy costs, it is necessary to select the components of the scheme to reduce cost and increase efficiency. For example, analysis of penstock diameter shows that design for less than 10% head loss is likely to give the optimum economic choice. Design guidelines have now been developed for most aspects of micro-hydro technology and will soon be made available for low-head turbines. There is now a need to build up technical and organizational capacity at a local level so that the benefits of this technology can be brought to rural populations. (Williams 2006)

Several researcher had found that micro hydro propeller (MHP) has many problem and cannot be used in certain topography area that has limited areas where the head meets this requirement and also the cost of the MHP system is to higher due to there's no local manufacture in certain country. Ramos (1999) on his paper also stated the same thing the Mariano, and have conclude that by using pumps as turbines seems to be a good alternative to dissipation of excess flow energy that, in normal conditions. Williams on his paper has stated even pump as turbines has many advantages over others micro hydro turbine too, but it is difficult to predict accurately the turbine performance. Even though from previous researches many researchers have found that the pump as turbine is the most efficient and suitable turbines for micro hydro scheme, but there are also agreed that for the low head, low flow and also for the low cost application, propeller turbine is the most suitable turbine that is meets these criteria.

The flows of the micro hydro system operation which normally consist some of components and each of the components have their own function. There are some components of micro hydro which is consists of water supply, reservoir, penstock, turbine, generator, electronic controller and distribution system.

## 2.3 PROPELLER FOR MICRO HYDRO POWER

In reaction turbines the available potential energy is progressively converted in the turbines rotors and the reaction of the accelerating water causes the turning of the runner. These are again divided into radial flow, mixed flow and axial flow machines. Radial flow machines are found suitable for moderate levels of potential energy and medium quantities of flow. The axial machines are suitable for low levels of potential energy and large flow rates. Reaction turbines are acted on by water, which changes pressure as it moves through the turbine and gives up its energy (Water Turbines 2010).

A propeller turbine is a runner that has shaped just like a boat propeller to turn the generator and it is usually has three to six blades. A variation of the propeller turbine is the Kaplan turbine in which the pitch of the propeller blades is adjustable. This type of turbine is often used in large hydroelectric plants. An advantage of propeller type of turbines is that they can be used in very low head conditions provided there is enough flow. (Water Turbines. 2010). The Figure 2.1 shows an example of propeller turbine for this research.

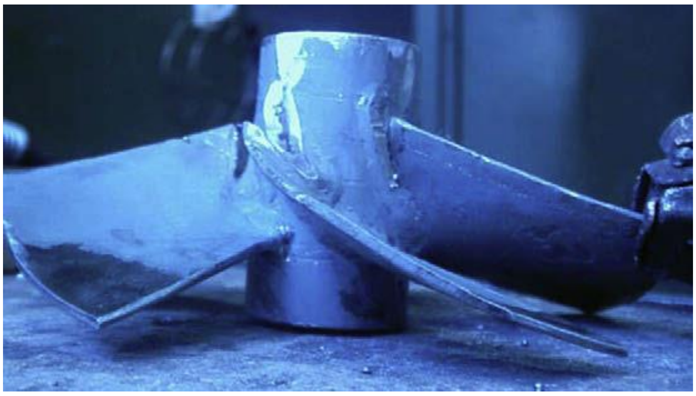


**Figure 2.1:** Propeller turbine

Source: Singh, P. & Nestmann, F. (2009)

### 2.3.1 Basic Nomenclatures of Propeller in Terms of Performance.

#### i. Size

Diameter is defined as the maximum radius of a blade multiplied by two. It is crucial geometric parameters in determining the amount of power that a propeller can absorb and deliver, thus determining the thrust available for propulsion. Usually the diameter is proportional to the efficiency of propeller, but in high speed vessels larger diameters equates high drag. For typical vessels a small increase in diameter translates into a dramatic increase in thrust and torque load on the engine shaft, thus the larger the diameter the slower the propeller will turn, limited by structural loading and engine rating.

#### ii. Pitch

Pitch is the theoretical distance travel in one revolution of propeller. For instance, the propeller move 10 inches in one revolution, thus the nominal pitch of the propeller is 10 inches. It is called nominal pitch because the actual pitch of propeller will be less than the nominal pitch. The difference between nominal and actual pitch is called slip. Slip is the difference between actual distance the vessel travels of the propeller blades through fluid in one revolution and theoretical distance base on formula. A properly matched propeller will actually move forward about 80 to 90 percent of the theoretical pitch. The ratio of pitch to diameter (P/D) is typically falls between 0.5 and 2.5 with an optimal value for most vessels closer to 0.8 to 1.8. Pitch effectively converts torque of the propeller shaft to thrust by deflecting or accelerating the water astern – simple Newton's Second Law.

#### iii. Hub

Solid centre disk that attached the propeller shaft and blade is called as hub. Ideally the hub should be as small in diameter as possible to obtain maximum thrust, however there is a tradeoff between size and strength. Too small a hub ultimately will not be strong enough. Usually, the hub shapes include radius, cylindrical, conical and barreled. The Propeller hub is split on a plane parallel to the plane of rotation of the propeller to allow for the installation of the blade.

### iv. Blade

Blades are the twisted fins or foils that protrude from the propeller hub. The shape of the blades and the speed at which they are driven dictates the torque a given propeller can deliver. Blade root is where the blade attached to the hub, and blade tip is the outermost edge of blade at a point furthest from the propeller shaft. High pressure side of blade is defined as blade face. This is the side that faces aft (backward) and pushes the vessels in forward motion. The back of the blade is the low pressure side or the suction face of the blade. This is the side that faces upstream or towards the front of the vessel.

### v. Number of blades

The number of blades will critically dictate the performance of a propeller. Basically greater speed, horsepower and load requirements of vessel need propeller with more blades. In design consideration, the number of blades is primarily determined by the need to avoid harmful resonant frequencies of the ship structure and machinery. It also found that both propeller efficiency and optimum propeller diameter increase as blade number decrease. With increased number of blades the surface area increases which reduces the slip in cruising speeds. Increased number of blades also gives a smoother behavior of the vessel with greater control in turns and in rough waters. Four and five blade propeller are also reducing vibrations and noise due to a better balance and are causing less wear on the transmission system. Adding blades however often decreases top speed with a couple of knots, but this is a trade-off in the propeller selection process. In this project have three propellers with different number of blades .

### 2.3.2 Material of Propellers

The materials from which propellers are made today can broadly be classed as members of the bronzes or stainless steels. The once popular of cast iron has now virtually disappeared, even for the production of spare propellers, in favor of materials with better mechanical and cavitation resistant properties. The propeller in this project is made from aluminium alloy.

The properties required in a propeller material will depend to a very large extent on the duty and service conditions of the vessel to which propeller are being fitted.

However, the most desirable set of properties which it should posses as follows:

i. High corrosion fatigue resistance in water
ii. High resistance to cavitation erosion
iii. Good resistance to general corrosion
iv. High strength to weight ratio
v. Good repair characteristics including weld ability and freedom from subsequent cracking
vi. Good casting characteristics

The material of this simulation is not considered because of this research only study about the condition of the flow.

## 2.4 THEORY OF FLOW

Since each potential site for small-scale hydropower scheme is unique turbine selection is based mostly on the water head and the available flow rate. As the scheme head reduces, the flow rate should be higher. It is important that steps are taken to find successful approaches to provide standardized equipment, engineering designs and implementation methods specifically for a particular location. The power produced by hydropower turbine can calculate using some of equations (Williams 2008).

Turbine selection is based mostly on the available water head, and less so on the available flow rate. In general, impulse turbines are used for high head sites, and reaction turbines are used for low head sites. There are many type of turbines use for micro hydro schemes which their characteristics are quite different and depend on the application.

### 2.4.1 Reaction Turbine Triangle Velocity

The fluid velocity at the turbine entry and exit can have three components in the tangential, axial and radial directions of the rotor. This means that the fluid momentum can have three components at the entry and exit. This also means that the force exerted on the runner can have three components. Out of these the tangential force only can cause the rotation of the runner and produce work. The axial component produces a thrust in the axial direction, which is taken by suitable thrust bearings. The radial component produces a bending of the shaft which is taken by the journal bearings.

Thus it is necessary to consider the tangential component for the determination of work done and power produced. The work done or power produced by the tangential force equals the product of the mass flow, tangential force and the tangential velocity. As the tangential velocity varies with the radius, the work done also will be vary with the radius. The moment of momentum theorem is used for this purpose. It states that the torque on the rotor equals the rate of change of moment of momentum of the fluid as it passes through the runner.

Let $u_1$ be the tangential velocity at entry and $u_2$ be the tangential velocity at exit. Let $V_{u1}$ be the tangential component of the absolute velocity of the fluid at inlet and let $V_{u2}$ be the tangential component of the absolute velocity of the fluid at exit.

Let $r_1$ and $r_2$ be the radii at inlet and exit.

The tangential momentum of the fluid at inlet = $\dot{m}V_{u1}$ (2.1)

The tangential momentum of the fluid at exit $= \dot{m}V_{u2}$ (2.2)

The moment of momentum at inlet $= \dot{m}V_{u1}r_1$ (2.3)

The moment of momentum at exit $= \dot{m}V_{u1}r_2$ (2.4)

Torque, τ $= \dot{m}\ (V_{u1}r_1 - V_{u1}r_2)$ (2.5)

Depending on the direction of $V_{u2}$ with reference to $V_{u1}$, the – sign will become + ve sign.

Power = ωτ and $\omega = \frac{2\pi\ \mathrm{N}}{60}$ (2.6)

Whereas N is in rpm.

Power = $\dot{m}\ \frac{2\pi\ \mathrm{N}}{60}\ (V_{w1}r_1 - V_{w1}r_2)$ (2.7)

But $\frac{2\pi\ \mathrm{N}}{60}\ r_1 = u_1$ and $\frac{2\pi\ \mathrm{N}}{60}\ r_2 = u_2$ (2.8)

Power = $\dot{m}\ (V_{u1}u_1 - V_{u2}u_2)$ (Euler's Turbine Equation) (2.9)

In the reaction turbine, there are three types of runners have been classified base on the inlet tip angle that are slow runner, medium runner and fast runner. For the slow runner the inlet angle was more than 90 degree, whereas for the fast runner was classified for less than 90 degree of inlet tip angle. For the reaction turbine which classified as medium runner when the inlet tip angle was equal to 90 degree.

The power produced can be expressed as due to three effects. These are the dynamic, centrifugal and acceleration effects. Consider the general velocity triangles at inlet and exit of fast runner turbine, shown in Figure 2.2.

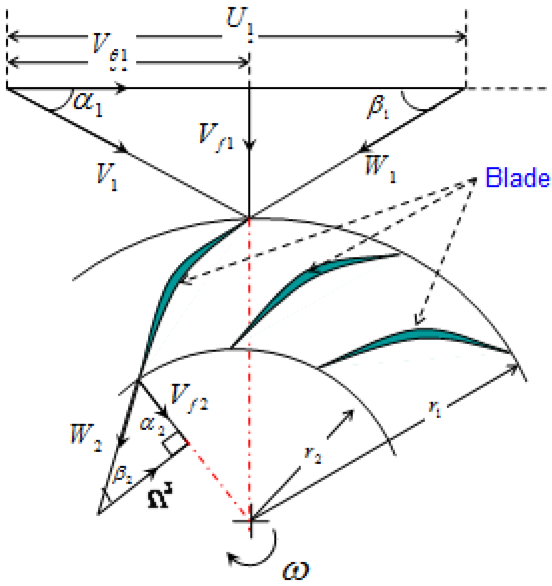



**Figure 2.2:** Forward curve (fast runner) triangle velocity

In the equations 2.10 to 2.11 describes the shaft power relation with flow velocity.

$$P_{SHAFT} = \dot{m}(V_{\theta 1}U_1 - V_{\theta 2}U_2) \tag{2.10}$$

$$P_{SHAFT} = \rho Q(V_{\theta 1}U_1 - V_{\theta 2}U_2) \tag{2.11}$$

To describe to the change of tangential velocity outlet, let the value of $\alpha_2 = 90°$

$$V_2 = V_{f2}$$

$$\alpha_2 = 90° \quad \rightarrow \quad V_{\theta 2} = 0$$

$$\text{(Mandatory Assumption)}$$

The equation of effective head or net head will become

$$H_E = \frac{V_{\theta 1}U_1}{g} \tag{2.12}$$

So, new equation for shaft power is:

$$P_{SHAFT} = \frac{V_{\theta 1}U_1 - V_{\theta 2}U_2}{g} = \dot{m}gH_E \tag{2.13}$$

Base on the figure of velocity triangle, the equation 2.14 to 2.16 describe the relation between vane angle and flow velocity.

$$\tan\alpha_1 = \frac{V_{f1}}{V_{\theta 1}} \tag{2.14}$$

$$\tan\beta_1 = \frac{V_{f1}}{U_1 - V_{\theta 1}} \tag{2.15}$$

$$\tan\beta_2 = \frac{V_{f2}}{U_2} \tag{2.16}$$

Mass flow rate: $\dot{m} = \rho Q$ (2.17)

From the relation of equation above, the equation of hydraulic efficiency is became

$$\eta_H = \frac{P_{\mathrm{RunnerBlade}}}{P_{\mathrm{From\ Water}}} = \frac{P_{\mathrm{SHAFT}}}{\rho g Q H} = \frac{\rho g Q H_E}{\rho g Q H} = \frac{V_{\theta 1}U_1}{gH} \tag{2.18}$$

The velocity triangles associated with the optimization of the blade configuration. It also shows the parameters related to each other from the direction of the blade profiles, such as the angle variation, as indicated by the vectors. A blade model configuration (BMC) was developed to estimate the best blade orientation, which to lead to best efficiency operating conditions. (Ramos et. al, 2010)

### 2.4.2 Free vortex theory

Propeller turbine is classified in the category of incompressible axial flow turbines, thus the free vortex law is suitable to be use for the analysis. Punit has used the free vortex theory for the design his propeller runners. The origins of free vortex law come essentially from the law of conservation of angular momentum. The primary conditions like irrotational flow and constant axial velocity need to be satisfied for this law. Equation 2.19 represents the final form of the free vortex law. (Punit, 2010)

$$C_u . r = \text{Constant} \tag{2.19}$$

The free vortex law calls for maintaining the product of tangential flow velocity and the radius vector constant all along the inlet region and the exit region of the blade as given by equation 2.20 (Punit, 2010)

$$[\, C_u . r \,]_{\text{inlet}} = K_{\text{inlet}} \quad \text{and} \quad [\, C_u . r \,]_{\text{exit}} = K_{\text{exit}} \tag{2.20}$$

The constants of equation 2.19 are not same in magnitude. In general for an axial flow turbine the constant (K inlet) at the inlet depends on the hydraulic (Euler) head to be realized on the shaft. In order to maximize the energy transfer, the exit tangential velocity is taken as zero (i.e. $C_{u,exit}$. = 0) all along the exit blade profile and hence Kexit = 0. Further, the radius vector of the axial flow turbine increases continuously from the hub to the tip, which causes the $C_u$ component to decrease. This causes fluid to enter each radial section with a different swirl angle, a. Moreover, since every radial section has a different the tangential blade velocity (u), the blade angle (or relative flow angle, b) should also change from the hub to tip (refer to velocity triangles in figure 2.3 and 2.4). The same holds true for the exit blade section despite ($C_{u,exit}$. = 0) (Punit, 2010)

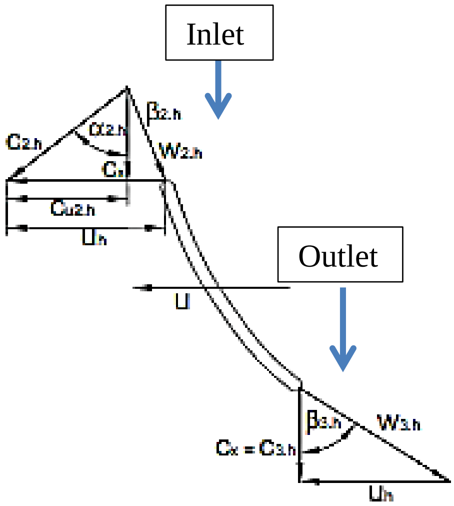



**Figure 2.3:** Inlet and exit velocity triangles at the runner hub.

Source: Punit, (2010)

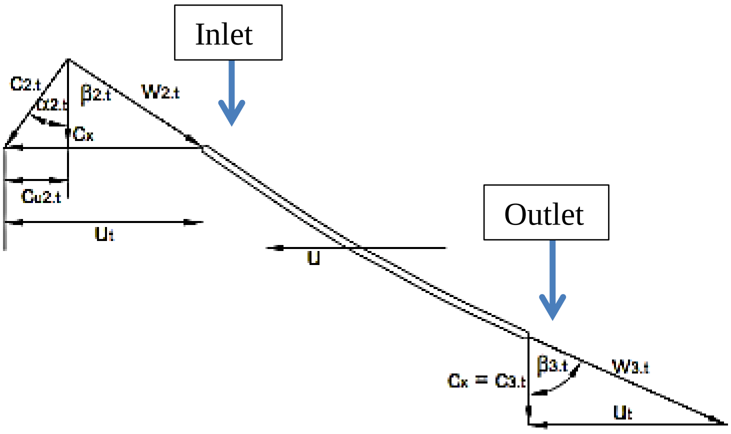



**Figure 2.4:** Inlet and exit velocity triangles at the runner tip

Source:Punit, (2010)

## 2.5 PROPELLER PERFORMANCE CHARACTERISTICS

The performance of propellers are mostly depends on the thrust produced. Thrust is a force that driven the propeller forward. Actually the details on how a propeller generates thrust are very complex. A lot of consideration must be made including the wake field, temperature, water density and others. However it still can be elaborated using simplified momentum theory. Simple momentum theory state that the thrust of a propeller depends on the volume of fluid accelerated per time unit, fluid acceleration and density of medium.

The performance characteristics of a propeller can be divided into two that are open water and behind-hull properties. Open water characteristics relate to the description of the forces and moments acting on the propeller when operating in a uniform fluid stream. Behind-hull properties are performance of propeller in a mixed wake field behind a body, thus more complex and closer to actual performance. Many types of turbines have been implemented in the MHP schemes such as Kaplan, Francis, Propeller, Pelton and Cross flow. The application of each turbines has own characteristic depending on the input of head and water flow rate.

### 2.5.1 Forces acting on propellers

The forces that act on propellers can be categorized into pressure forces and friction forces. The local angle of attack determines the operating condition of an element of the propeller blade, resulting in a certain pressure distribution around the airfoil. In normal (positive thrust) conditions the pressure is decreased at the upper side of the airfoil (the upstream side of the propeller disc) and increased at the lower side of the airfoil (the downstream side of the propeller disc). Integration of the pressure distribution along the airfoil leads to a resulting section pressure force. Lift and drag components can be derived as well as thrust and torque-force components. Integration of the contribution of all elements along the blade will lead to the total thrust and drag of the blade.

The friction force is the force due to viscous effects in the boundary layer on the propeller blade. The friction force acts in the direction of the local flow on the surface of the blade. Because of centrifugal and Coriolis forces the development of the boundary layer on a propeller blade can be highly three dimensional. An accurate calculation of the friction force will require detailed information of the boundary layer development. Calculation of the boundary layer is not customary in propeller analysis methods. Usually the empirically obtained friction force is thought as acting in the direction of the local flow just outside the boundary layer.

The energy came from water can be divide into two types, there are potential energy and kinetic energy. Potential energy from the dammed water is the energy extracted from the water depends on the volume and difference in height between the source and the water outlet. This height difference is called the head. The amount of potential energy in water is proportional to the head. The higher the level of water from the release valve, the higher potential energy will be produced. Kinetic energy from the waterwheel is the energy extracted from the water depends on the velocity of the water flow in the river. The higher velocity of the water river, the higher kinetic energy will be produced. (Maher and Smith, 2001)

## 2.6 DESIGN CRITERIA FOR PROPELLER TURBINE

The variation of number of blades and pitch angles has marginal effect on the energy efficiency, when operating in water. It is possible to improve efficiency for impellers operating in highly shear-thinning viscous non-Newtonian fluids to achieve increased velocities at a given power input and tank diameter, via optimizing impeller geometrical parameters. The smaller the Reynolds number, the greater is the potential percentage improvement in the flow energy efficiency. (Jie Woo, 2005)

Low head hydro sites (2 to 10m) have an even larger potential for providing electricity in rural areas of developing countries but the harnessing of this potential is severely hampered by the lack of an appropriate turbine design. Fixed geometry propeller turbines are one of the most cost-effective turbine options for low head micro hydro power. (Simpson and Williams, 2006) A sensible improvement of the turbine performances has been obtained using an optimal design technique to redesign the runner. The adopted optimization strategy is well adapted for this type of small hydro turbine. The parameterization method used fits with a good accuracy the initial proposed geometry. (Kueny and Lestriez, 2004)

It was found that an incorrect matching between the turbine rotor design and the available flow rate at the site significantly affected the turbine operation and in order to provide an acceptable performance it was possible to adjust just the runner design and operating speed of the turbine.( Simpson & Willians, 2006)

The runner is designed to spin in the casing at a speed compatible with the generator, nominally 1500 rev./min. The hub size can be varied by the designer taking into account several issues. The circumferential flow at the hub must be less than the velocity ur at the hub in order to obtain a nonnegative angle of attack along the runner leading edge. Too large a hub-to-tip ratio, while addressing the first issue, results in a high axial flow velocity in the annulus, creating excess friction losses and abrupt expansion losses as the hub truncates just downstream of the blades. Too high a hub-to-tip ratio also creates short-span, highly loaded blades with insufficient area to resist cavitation. The Figure 2.5 illustrates the blade and runner hub arrangement. ( Alexander at. el., 2009)

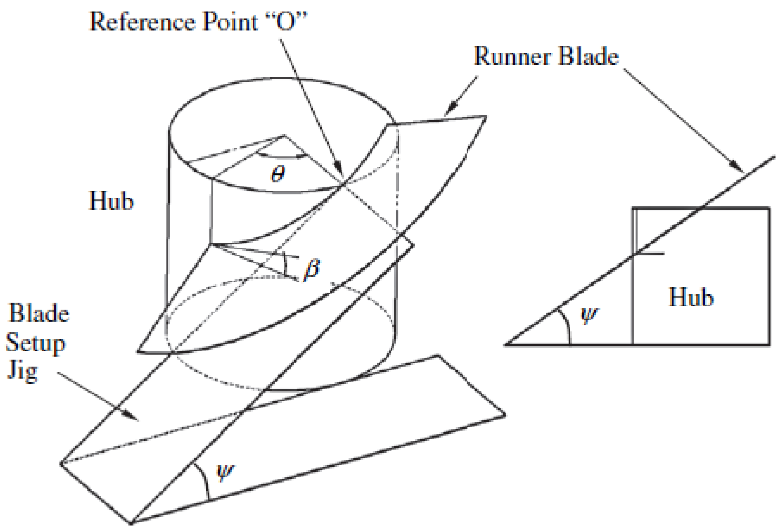



**Figure 2.5:** The set-up arrangement for attaching the blade to the runner hub

Source: Alexander at. el., (2009)

## 2.7 A SIGNIFICANT RESEARCH OF CFD ON PROPELLER FOR MICRO HYDRO POWER

Computational Fluid Dynamic are needed as a tools to predict the results from the calibration base on experimental tests and it give better understanding of the phenomenon associated with the flow behavior in turbines for different flow condition. The behavior of such micro hydro converters can differ depending on the flow conditions and on the runner configuration, particularly, on the inlet and outlet flow and on the shape of the blades. The design of new low-head devices cannot be solely based on a large turbine scaling-down methodology. To overcome this limitation, CFD analysis can be used to provide design and hydrodynamic behavior information, which can strongly support the experimental tools. CFD analysis is important to help in the design of the best configuration for different system's characteristics and flow scenarios. (Ramos et. al., 2010)

To predict the best results that lead to a best efficiency point (BEP), it is important to analyze different slopes for example of angle variation in designing the impeller blades of the tubular propeller to avoiding disturbances in the flow and causing additional losses that might constrain the efficiency. Another parameter is to adopt a minimal thickness in its development. For the flow rates considered in micro turbines, the maximum thickness of 1 mm was taken owing to limitations of the computational fluid dynamics (CFD) mesh generation that will be created for model simulations. (Helena et. al., 2013)

Computational Fluid Dynamics (CFD) has become a useful and complementary tool to the designer and can be used for simulating, designing and analyzing complex three-dimensional flows in turbo machinery. With the emergence and improvement of the commercial CFD software industry, computational modeling is being used much more by engineers with applications spanning a very wide area of engineering. To date, there appears to be a limited amount of research into the use of CFD for analyzing the design of small propeller turbines suitable for decentralized applications. (Simpson & Williams, 2006)

# CHAPTER 3

# METHODOLOGY

## 3.1 INTRODUCTION

In this chapter will explain about how to set up the analysis of propellers. This chapter also will include the design concepts, simulation, boundary condition, meshing component and lastly to compare the results of simulation and experiment.

The methodology consists of design concept which is a component of designing. The design concept requires lifetime understanding and market consideration. The final stage of designing is the Computer Aided Design (CAD). CAD is then converted to IGS file to generate the geometry or design in Computational Fluid Dynamic (CFD) before proceed to simulation process by using ANSYS.

The simulation process requires meshing the grid and refining the design. After the meshing is successful generated, it will converted into CFX software for boundaries condition and initial condition setup. The next process is run the solver to produces the solution for the task given. The final step, the results was defined to get the analysis result that was needed. Figure 3.1 illustrates the flow chart for this project.

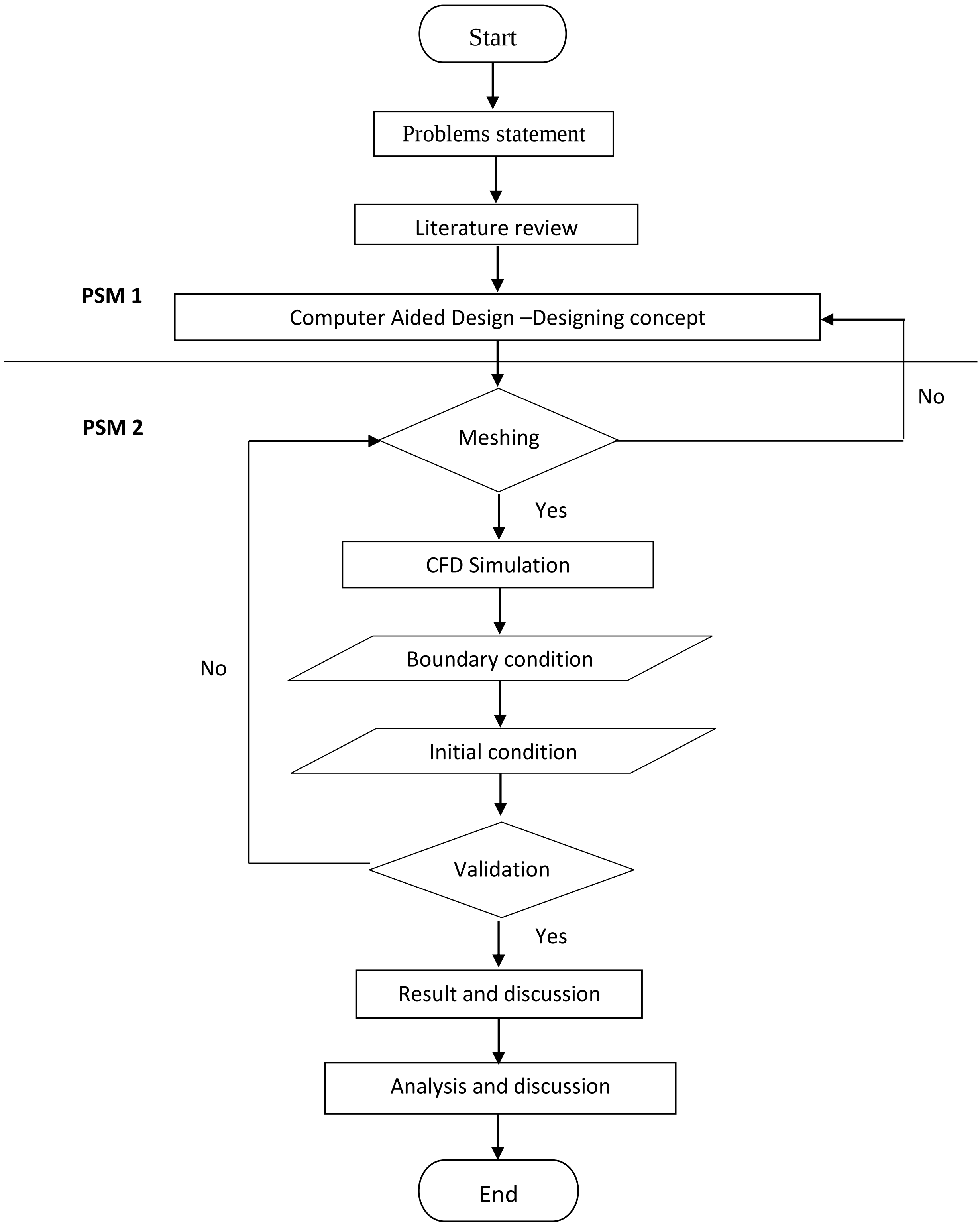



**Figure 3.1:** Flow chart

## 3.2 DESIGN CONCEPT

### 3.2.1 Sketching

In this design concept reached the point where the drafting initial design drawing were started. At the sketch design phase is very important to draw to scale. In fact, even there have many powerful CAD systems engineers usually begin their design exploration with a pencil and paper, turn to the computer only after they have established the basic concepts. The initial design should be a combination of propeller sections and form drawings or what ever best explains the design and how it works. The importance of drawing, informal sketching and both formal drafting, has been the subject of widely research. In the early years of propeller design, designers normally used analytical theory to make various engineering calculations that go into the design process and followed a lot of experimentation. But for this analysis, all the information has gathered form journal that related to this project that the design have been validate or simulation. After this step have finished, step will proceed to sketch in three-dimensional by using Computer-Aided Design (CAD).

### 3.2.2 Computer Aided Design (CAD)

The engineering design process is generally referred to the specification, conceptual design, embodiment and detailed design. In addition, the conceptual design stage is by far the most challenge in the design process. Moreover, the computer aided design systems mainly geared towards the later, more detailed stage. In practice, the computer aided design not just as passive three-dimensional drawing boards, but it have advanced ability to perceive a three-dimensional model depicted in a single freehand sketch presents the practical possibility of bringing numerous analysis tools.

For the software that was used to sketch 3D design of propeller in this project is SolidWorks. SolidWorks is a one of three-dimensional mechanical computer aided design program that runs on Microsoft Window and is being developed by Dassault Systèmes SolidWorks Corporation. The improvement design applications on some 3D program enhance collaboration, speed model creation and simplify the development process of the product. The new version of SolidWorks also have new drawing capabilities and powerful design tools, network rendering, cost estimation, sub-model simulation and wider sharing and also increased connectivity. Building on the success of the last 20 releases, the new offering covers the range of SolidWorks applications for three-dimensional design, simulation, technical communication, product data management, and sustainable design.

Figures 3.2 show the real runner at the left side, that used for experiment and at right side is a runner that was drawing by using SolidWorks. Figure 3.3 show the propellers geometry that has been taken from the journal. Figure 3.4 and Figure 3.5 shows the other turbine components.

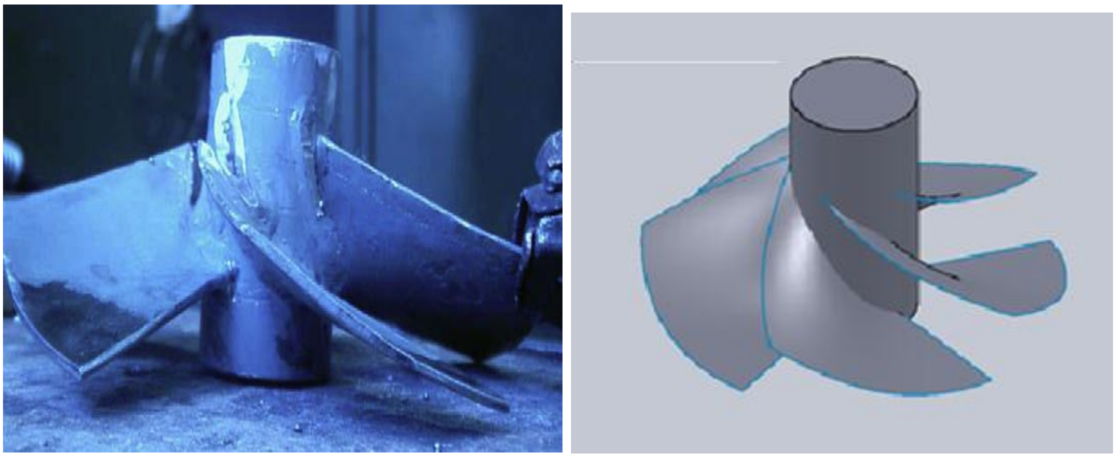


**Figure 3.2:** Runner

Source: Singh and Nestmann, (2009)

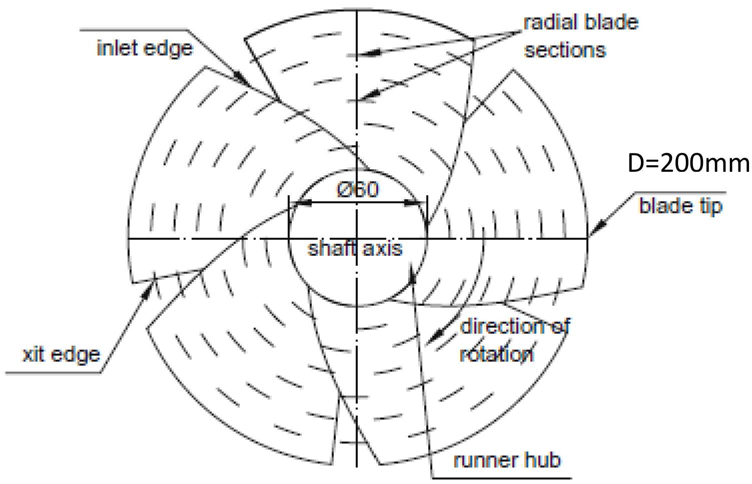



**Figure 3.3:** Runner geometry

Source: Singh and Nestmann, (2009)

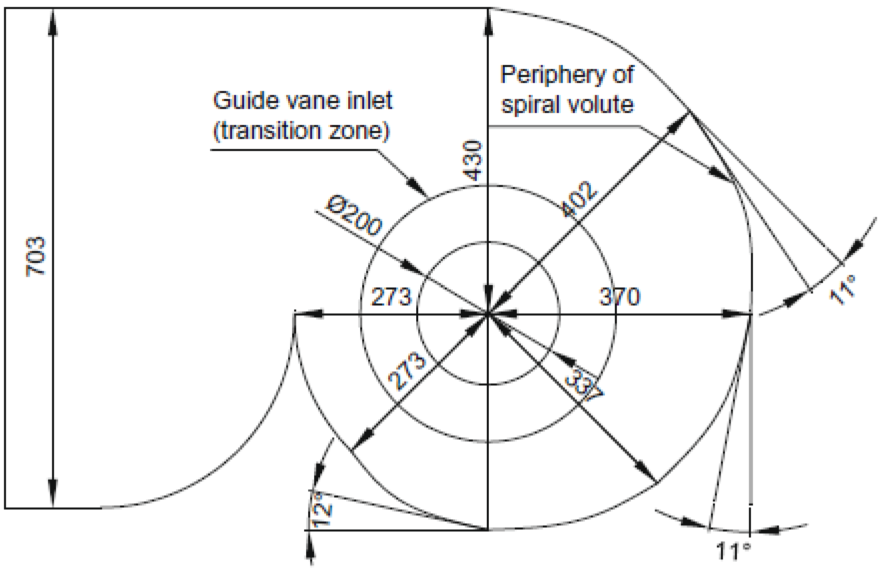



**Figure 3.4:** Open spiral volute [dimensions in mm]

Source: Singh and Nestmann, (2009)

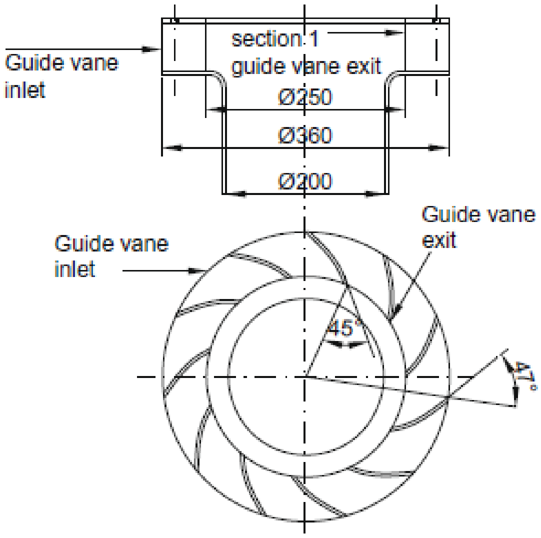



**Figure 3.5:** Radial guide vane

Source: Singh and Nestmann, (2009)

All the components have been sketched by using SolidWorks and save as IGS file to import the geometry in Ansys for simulation. This sketching will use to analyze it design base on some parameters that will applies in this project.

## 3.3 MESHING COMPONENT

One of the most critical aspects of engineering simulation is mesh generation. In the other hand, meshing is an integral part of the computer-aided engineering (CAE) analysis process. The mesh affected the convergence, accuracy and speed of the solution. Moreover, the time it takes to create a mesh model is often an important portion of the time it takes to produce results from a CAE solution. Thus, to get a better solution, the automated the meshing tools was needed. From easy, automatic meshing to a very fine mesh, ANSYS provides the ultimate solution and powerful automation capabilities. Additionally, it also can update immediately to a parameter change, making the hand off from CAD to CAE seamless and aiding in upfront design.

For solid models, the ANSYS meshing solution provides a robust automatic tetrahedral meshing even the most complex geometries and it also provides the flexibility to produce a variety of meshes in the complexity of the hybrid pure hex very detailed. It also easily to use by put the right mesh in the right place and ensure that a simulation will accurately validate the physical model. Furthermore, it can generate pure hex meshes using one of several mesh methods, base on the type of models. It also has physics preferences for structural, fluid, explicit and electromagnetic simulations.

In fluid modeling, Meshing solutions from ANSYS for fluid models provide unstructured quad-surface meshing and tri-surface meshing driven by proximity, curvature, quality and smoothness, in combination with a pinch capability that automatically eliminate insignificant features. The combination of automated surface meshing, boundary layer technology (including automatic proximity handling) and an advancing front tet-mesh algorithm ensures high quality, push-button meshing for fluid flow analysis. Figure 3.6 illustrates the meshing of the components for simulation. For the default domain modified has 845 710 nodes and 4 696 486 elements. Whereas, the propeller it has 3 200 nodes and 2 355 elements.

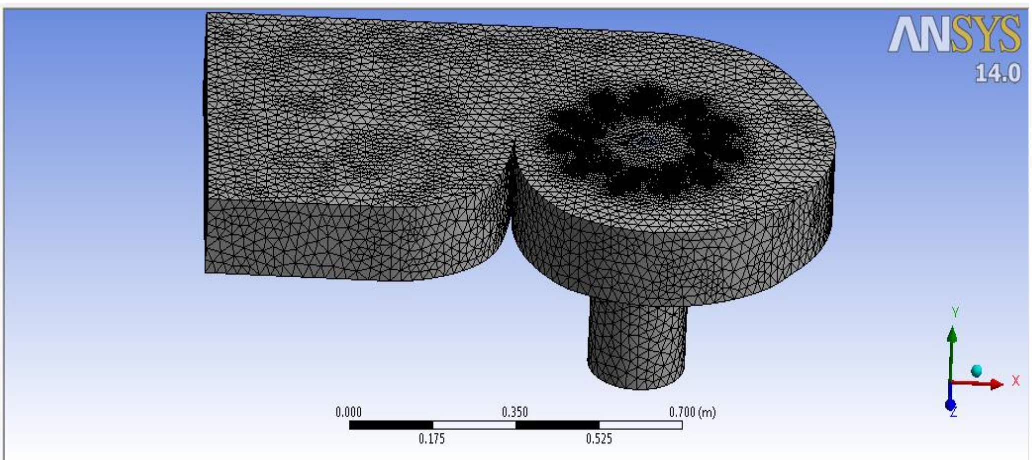



**Figure 3.6:** Meshing of the domain

## 3.4 SIMULATION

Simulation is defined as trying to predict the behavior of some aspect of the system by creating an approximation (mathematical) model of it and then done by physical modeling, by using a more general simulation package or writing a special-purpose computer program, probably still aimed at a particular kind of simulation. In this project the ANSYS CFX was used to analyze the propellers base on design. The general purpose CFD package that of this software is can solve complex and diverse three-dimensional fluid flow problems. It also uses the Navier–Stokes equations to describe the basic processes of mass transfer, momentum and heat. Moreover, it also combines several mathematical models that can be used in conjunction with the Navier–Stokes equations to describe other chemical or physical processes such as radiation, turbulence, combustion and so on.

This advance software also uses of some finite volume approach to convert the governing partial differential equations to become a system of discrete algebraic equations by discretizing the computational domain. It may produce a solution with the specified domain boundary conditions One of the most important features of CFX is that it uses a coupled solver, to solve the pressure as a single system, fluid flow and faster than the segregated solver up to a certain number of control volumes as it requires less iteration to get an equally converged solutions..

In the Ansys CFX 14 software there are some step must be done to get the analysis results. The sequences of the step in analysis are shown on Figure 3.7 and Figure 3.8.

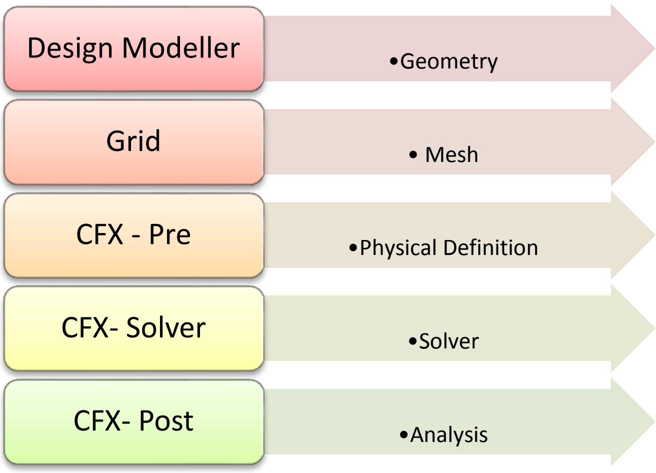



**Figure 3.7:** CFD process

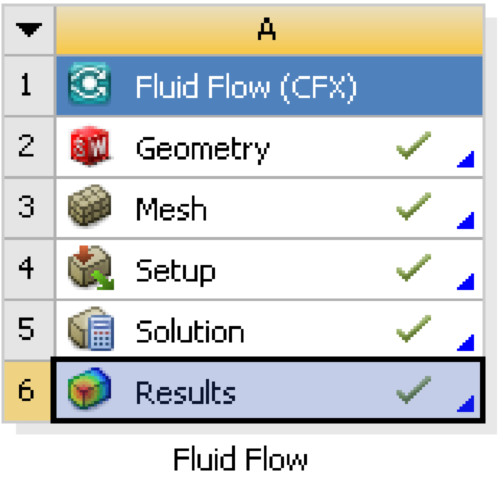



**Figure 3.8:** CFX Workbench

### 3.4.1 Initial Condition

Although distributed axial inflow velocity *V* set at the inlet and outside the boundaries of the calculation domain cylindrical surface. Set up at the outlet providing a specific value for the pressure condition. At the blade and hub surfaces No-slip condition is imposed. The axial inflow velocity *V* is also imposed as the initial condition of the whole flow field.

### 3.4.2 Boundary Condition

One of the simple things that should consider before set up the simulation is begin planning the boundary conditions of the set up. One of the popular meshes for simulating an airfoil in a stream is a Component Mesh, and that is what will be used. The calculation assumes uniform steady state incompressible fluid flow in the circumferential direction of the runner. The flow in the runner be explained in the rotating frame of reference, while the flow in the stationary components was expressed in the stationary frame of reference.

The undisturbed of free-surface elevation is assigned both at the inlet and outlet boundaries. The properties of variables-base on outflow and inflow boundary condition were applied on the propeller grid boundaries.

The computational were assumes in steady state incompressible continues fluid flow in open spiral volute. There are two domains part was created in the analysis. First domain is fluid flow as continues fluid and the flow in the stationary components was expressed in stationary frame of reference. The second domain was setup at the propeller that is solid domain and described in the rotating frame of reference. At the propeller blade in the rotating frame will be applied no-slip condition. The adiabatic will be assumed to the all solid surface.

### 3.4.2.1 Inlet Boundary Condition

For the inlet boundary condition was determined from the analytical calculations. From the inlet velocity profile, the inlet boundary condition was developed. The inlet boundary condition consists of two components that are the tangential (whirl) and radial velocity (velocity of flow) for the operating point. The value of velocity at inlet is constant for all simulation that is 2 meter per second.

### 3.4.2.2 Outlet Boundary Condition

The outlet of water flow is atmosphere space, and the boundary condition was set to an opening with zero Pascal of static pressure.

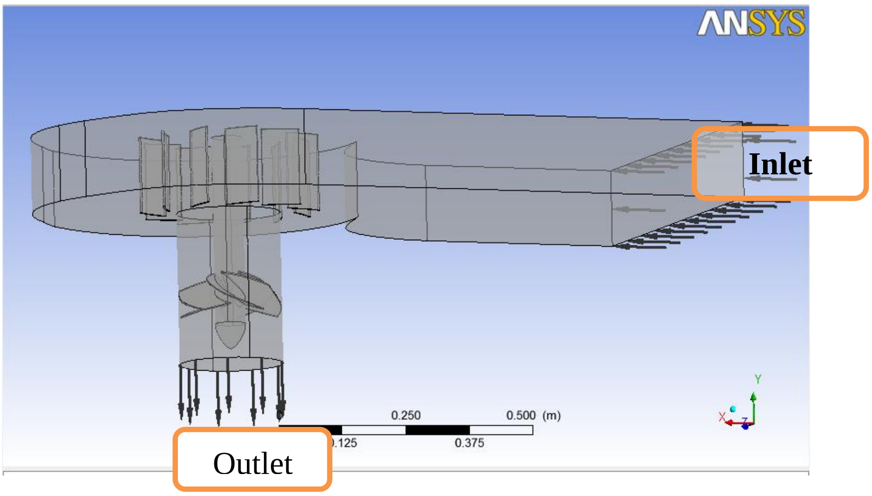



**Figure 3.9:** Setup of boundary condition in Ansys

## 3.5 COMPARISONS SIMULATION AND EXPERIMENT

Dimensionless characteristic parameters of experiments and CFD simulations were selected and compared. Comparing simulation and experiment a similar behavior of the fluid is evident, as well as the identification of the section in which the effect is more significant. The results from each method are compared to gain an understanding of the capabilities and limitations of each method. The simulations result will be analyzed to examine the improvements that have been made and the percentage of improvement that achieve for all type of the design of propeller turbine.

# CHAPTER 4

# RESULTS AND DISCUSSION

## 4.1 INTRODUCTION

The simulations of the reaction turbine have been carried out by using Ansys CFX version14 software. There are two major simulation have been done for the reaction turbine which are existing design and optimize design of propeller turbine.

## 4.2 CFD SIMULATION

The simulation for the propeller turbine have been done using cfx fluid flow whereas the mesh that generated for this simulation are using automatic mesh method and using high mesh in order to get better result for the simulation. The simulations have been carried out by using single rotating domain motion. The setting for boundary condition, mesh, cfx pre, cfx solver manager and cfx post are same for all simulation. For the components of simulations that are open spiral volute and guide vane also same.

### 4.2.1 CFD simulation for inlet tip angle

In the simulation of the existing design, the variable parameters have been used to investigate the effects of designing components of micro hydro power to generating of power. Figure 4.1 illustrates the modification on the runner at the inlet tip angle. The conditions at the runner outlet tip angle remain unchanged. The outlet tip angle is $74^0$ as the old runner design.

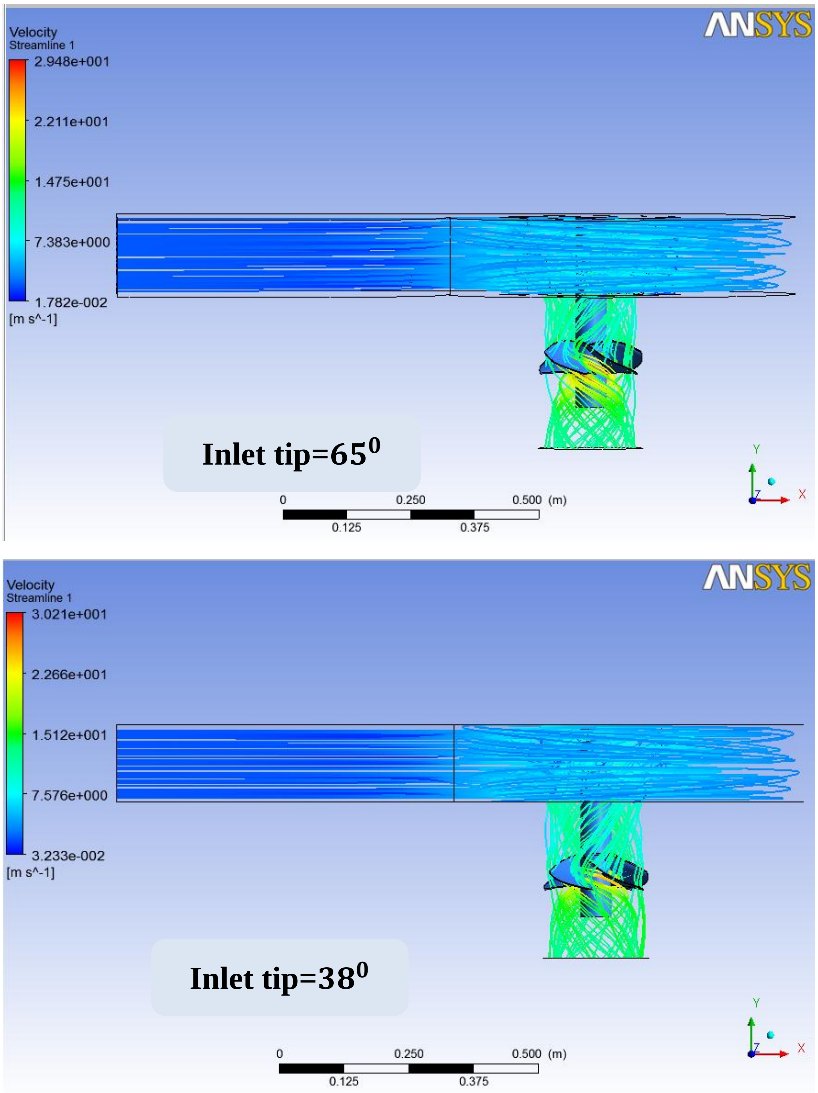



**Figure 4.1:** Propellers with constant exit tip angle $74^0$ and differences of inlet tip angle.

**Table 4.1:** Simulation result inlet tip angle modification with constant exit tip angle ($74^0$)

| Inlet Tip | Discharge $\left(\frac{m^3}{s}\right)$ | Discharge % change | Effective Head $(m)$ | Effective Head % change | Shaft Power $(kW)$ | Shaft Power % change | Efficiency (%) | Efficiency % change |
|---|---|---|---|---|---|---|---|---|
| $65^0$ | 0.4634 | +2.5 | 8.1474 | +63.7 | 37.0355 | +67.8 | 27.2 | +63.6 |
| $38^0$ | 0.4750 | | 13.3393 | | 62.1591 | | 44.5 | |

The results of simulation have calculated base on the formulas that have been stated on the chapters two. The performance characteristics have been compared in the Table 4.1. It can be clearly seen that the decrease of inlet tip angle opens up the inlet flow area creating a decreased contraction effect from inlet to exit. It can be observed that all the parameters of modified inlet stage have remarkably increased. The output power shaft has increased by 67.8% and effective head increased by 63.7%. While discharge increased by 2.5% resulting in a 63.6% rise in efficiency.

***Example of Calculation***

From Figure 4.1, the propellers with $74^0$ exit tip angle and $38^0$ inlet tip angle have $V_{outlet}$=30.21, $V_{inlet}$=7.576 and $V_{discharge}$=15.12. All the calculations were calculated base on equation 2.15 to equation 2.12 on the chapter two.

$$\tan\alpha_1 = \frac{V_{f1}}{V_{\theta 1}}$$

$$\tan 45° = \frac{7.5760}{V_{\theta 1}}$$

$$V_{\theta 1} = \frac{7.5760}{\tan 45°} = 7.5760 m/s$$

$$\tan\beta_1 = \frac{V_{f1}}{U_1 - V_{\theta 1}}$$

$$\tan 38° = \frac{7.5760}{U_1 - 7.5760}$$

$$U_1 = \frac{7.5760}{\tan 38°} + 7.5760 = 17.2728 m/s$$

$$\tan\beta_2 = \frac{V_{f2}}{U_2}$$

$$\tan 74° = \frac{30.21}{U_2}$$

$$U_2 = \frac{30.21}{\tan 74°} = 8.6626 m/s$$

$$Q = VA = \pi \times (0.1)^2 \times 15.12 = 0.4750 m/s$$

$$H_E = \frac{V_{\theta 1} U_1}{g}$$

$$H_E = \frac{7.5760 \times 17.2728}{9.81} = 13.3393 m$$

$$V_2 = V_{f2}$$

$$\alpha_2 = 90^\circ \quad \rightarrow \quad V_{\theta 2} = 0$$

$$\text{(Mandatory Assumption)}$$

$$P_{SHAFT} = \dot{m} g H_E = \rho Q g H_E$$

$$P_{SHAFT} = \rho Q g H_E = 1000 \times \pi \times (0.1)^2 \times 15.12 \times 9.81 \times 13.3393 = 62.1591 kW$$

Hydraulic efficiency: $\eta_H = \dfrac{P_{\text{RunnerBlade}}}{P_{\text{From Water}}} = \dfrac{P_{\text{SHAFT}}}{\rho g Q H} = \dfrac{\rho g Q H_E}{\rho g Q H} = \dfrac{V_{\theta 1} U_1}{g H}$

$$\eta_H = \frac{7.576 \times 17.2728}{9.81 \times 0.3} = 44.5\%$$

$$\% = \frac{New - Old}{Old} \times 100$$

$$Q,\% = \frac{0.4750 - 0.4634}{0.4634} \times 100 = 2.5\%$$

### 4.2.2 CFD simulation for exit tip angle

The second stage of modification of propellers with differences of exit tip angle is illustrated in Figure 4.2. The exit tip angle is changed from $74^0$ to $77^0$ and then changed to $85^0$.

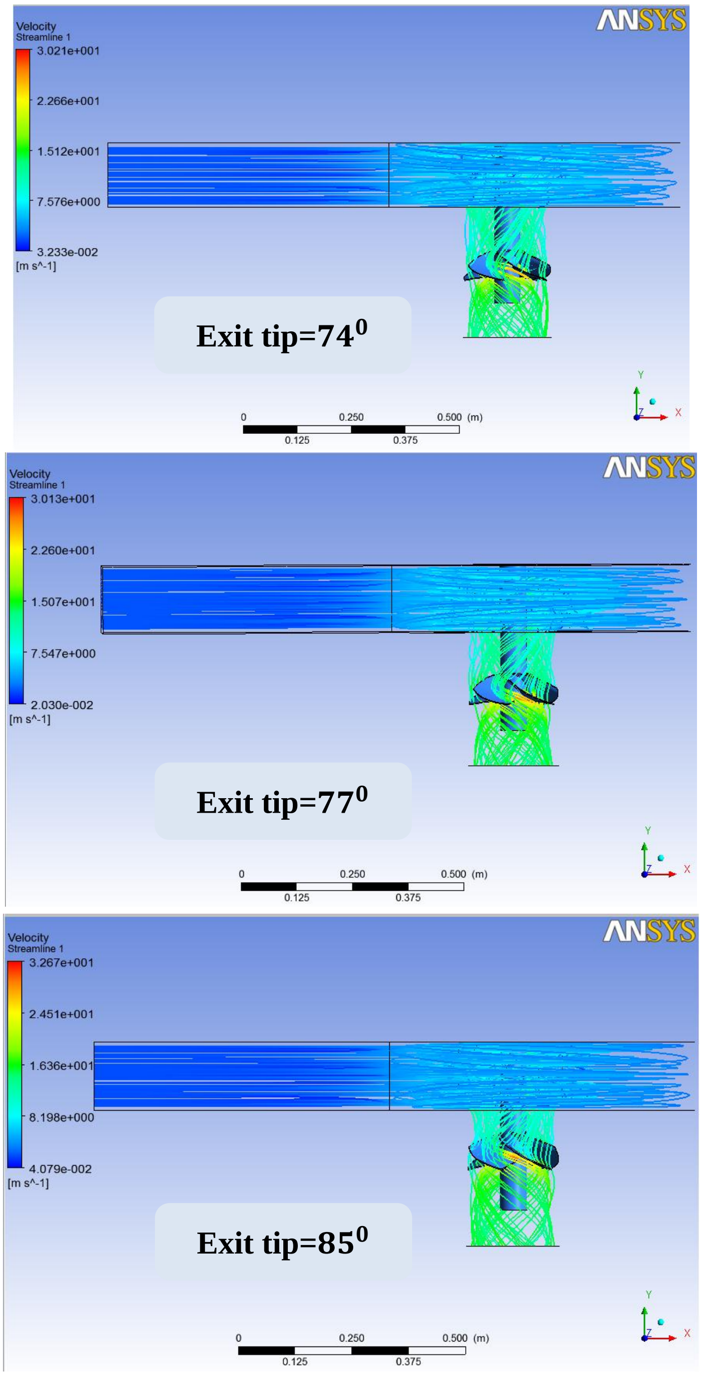



**Figure 4.2:** Propellers with constant inlet tip angle $38^0$ and differences of exit tip angle.

Table 4.2 shows the results of simulation and it clearly shows that propeller with $38^0$ inlet tip angle and $85^0$ exit tip angle is better than other two propellers at similar operating conditions. The result shows, it can produces 78.85 kW of shaft power with 15.62m effective head and 0.5146 meter cubic per second. This propeller has higher efficiency that is 52.1%.

**Table 4.2:** Simulation results of exit tip angle modification with constant inlet tip angle ($38^0$)

| Parameter | Exit tip angle $\mathbf{74^0}$ | $\mathbf{77^0}$ | $\mathbf{85^0}$ |
|---|---|---|---|
| Q $(m^3/s)$ | 0.4750 | 0.4734 | 0.5146 |
| $H_E$ $(m)$ | 13.3393 | 13.2374 | 15.6200 |
| $P_{shaft}$ $(kW)$ | 62.1591 | 61.4801 | 78.8521 |
| $\eta$ (%) | 44.5 | 44.1 | 52.1 |

**4.2.3 CFD simulation for nose cone on the propeller**

For the improvement of the propeller, the nose cone was added at the end of hub propeller's. Figure 4.3 is shows nose cone at the hub of propeller. This nose cone will give some effects to the flow of fluid after through the blades. Figure 4.4 shows the result of simulation by using Ansys CFX.


**Figure 4.3:** Nose cone on propeller

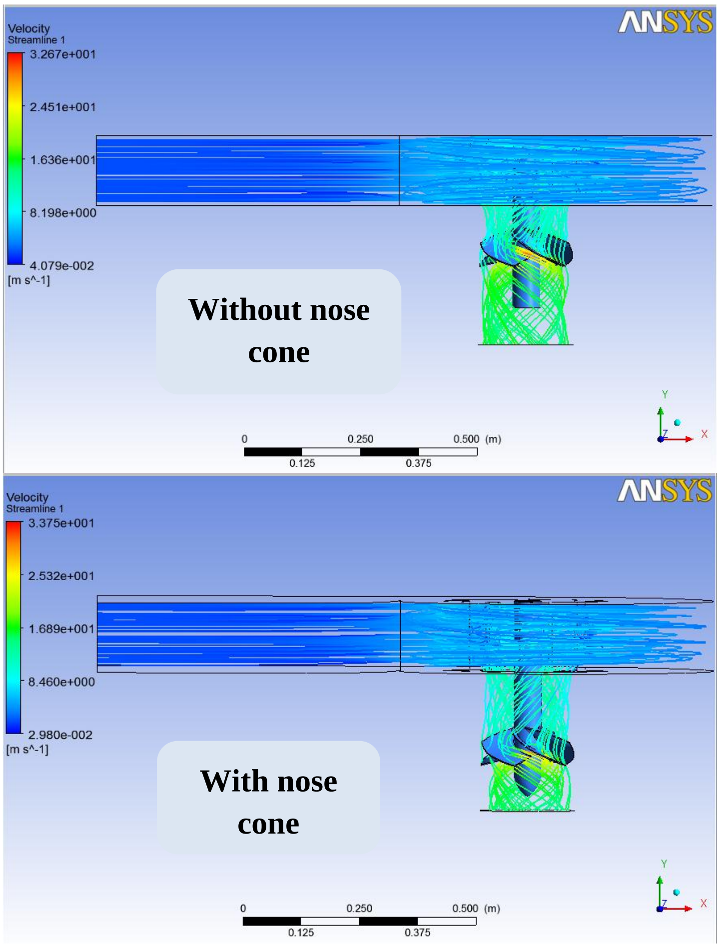



**Figure 4.4:** Differences the end of hub

**Table 4.3:** Results of propellers with difference structure at end of hub

| | $Discharge$ | | $Effective\ Head$ | | $Shaft\ Power$ | | $Efficiency$ | |
|---|---|---|---|---|---|---|---|---|
| | $\left(\frac{m^3}{s}\right)$ | $\%\ change$ | $(m)$ | $\%\ change$ | $(kW)$ | $\%\ change$ | $(\%)$ | $\%\ change$ |
| Without Nose Cone | 0.5146 | +3.1 | 15.6200 | +6.5 | 78.8521 | +9.8 | 52.1 | +6.3 |
| With Nose Cone | 0.5306 | | 16.6339 | | 86.5852 | | 55.4 | |

The performance characteristics have been compared in Table 4.3. It can be seen that the optimized model has a corresponding higher efficiency than the old turbine at similar operation conditions. From the percentage analysis, it can be observe that, the discharge has increased by 3.1%, but the shaft power has gone up by 9.8%, which means that in addition to the mass flow rate effects, the Euler shaft work has also increased, which internally refers to change in the velocity of water flow. It shows the nose cone is a significant component to increase the performance of propeller.

### 4.2.4 CFD simulation for surface area of blades

Figure 4.5 illustrates the modification high of blade or tip to tip length of blade. The same propellers have been modified on high of blade. Figure 4.6 illustrates the results of simulation that have been done.

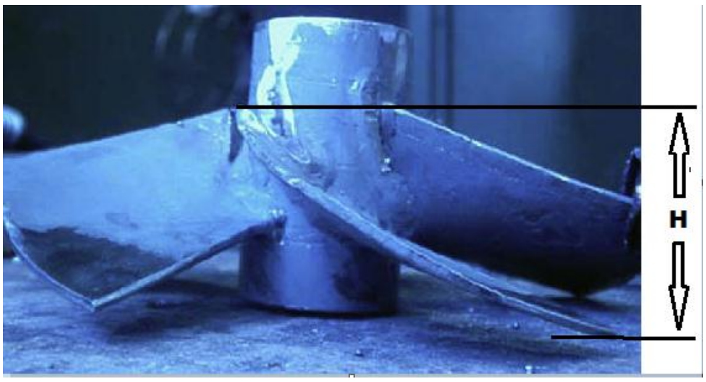



**Figure 4.5:** High of blade

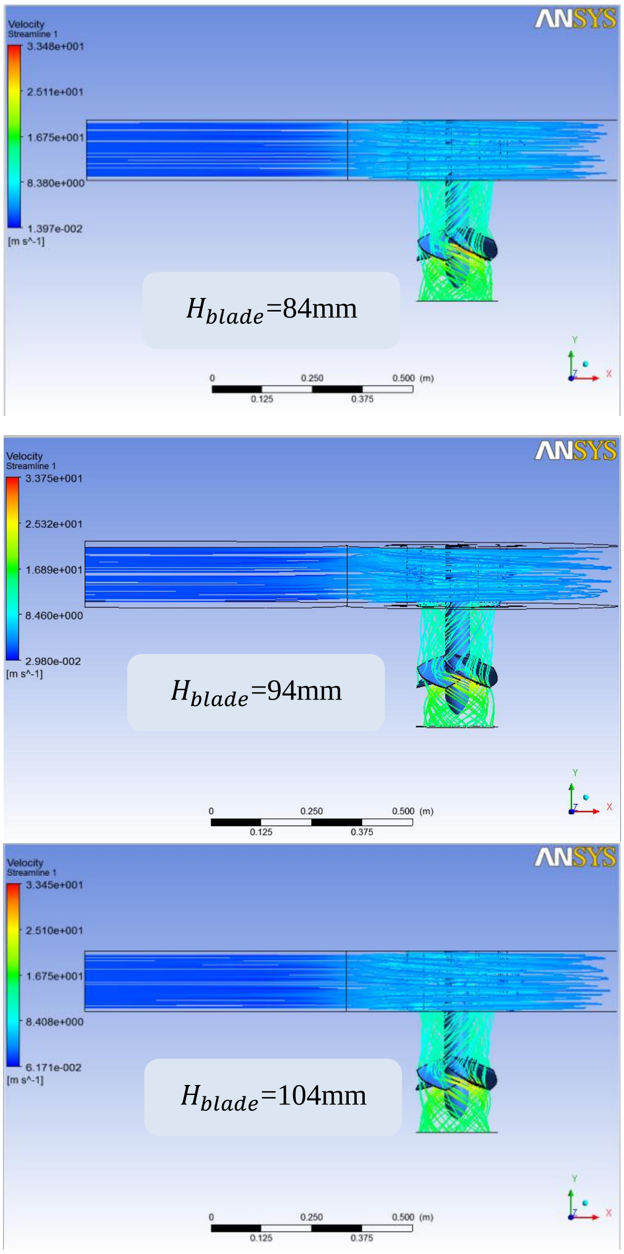



**Figure 4.6:** Simulation on propellers with differences high of blade

The comparison parameters of the simulation results are shows on Table 4.4.

**Table 4.4:** Tip angles of ($38^0$inlet $85^0$exit) propeller with differences of high of blades

| | $Discharge$ ($\frac{m^3}{s}$) | $\%\ change$ | $Effective\ Head$ $(m)$ | $\%\ change$ | $Shaft\ Power$ $(kW)$ | $\%\ change$ | $Efficiency$ $(\%)$ | $\%\ change$ |
|---|---|---|---|---|---|---|---|---|
| 84 mm | 0.5262 | +0.8 | 16.3208 | +1.9 | 84.2510 | +2.8 | 54.4 | +1.8 |
| 94 mm | 0.5306 | | 16.6339 | | 86.5852 | | 55.4 | |
| 104 mm | 0.5262 | +0.8 | 16.4301 | +1.2 | 84.8152 | +2.1 | 54.8 | +1.1 |

In this stage of modification, it shows the surface area of blade also has important role and must be suitable to the propeller. For this propeller, the suitable high is 94 mm because it has the highest efficiency that is 55.4 % and produces 86.6 kW of shaft power.

### 4.2.5 CFD simulation for number of blade

The final stage of modification on propeller is the number of blade. From the previous stage modification, the propeller that has 94mm high of blades has been modified on number of blade. In this stage will investigate the effect of blade number to the efficiency of propeller and the Figure 4.7 shows the results of simulation.

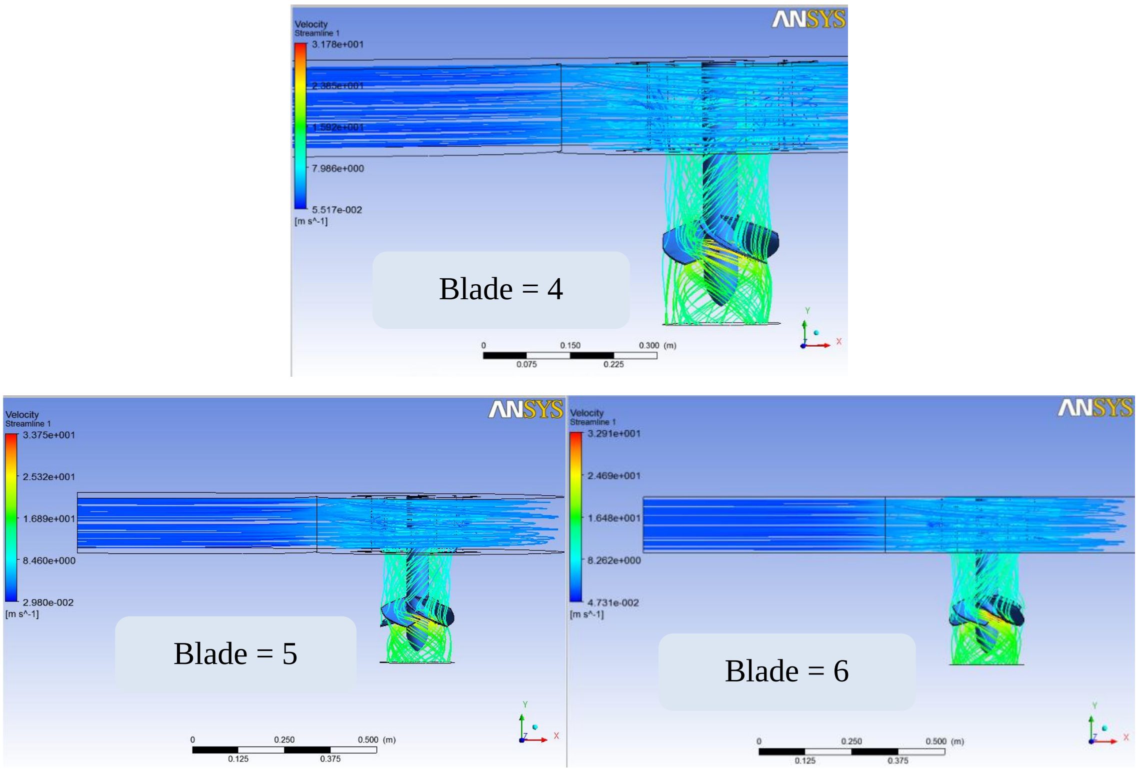



**Figure 4.7:** Simulation on propellers with differences of number of blades

**Table 4.5:** Tip angles of ($38^0$inlet $85^0$exit) propeller with differences number of blades

| | $Discharge$ | | $Effective\ Head$ | | $Shaft\ Power$ | | $Efficiency$ | |
|---|---|---|---|---|---|---|---|---|
| | $(\frac{m^3}{s})$ | $\%\ change$ | $(m)$ | $\%\ change$ | $(kW)$ | $\%\ change$ | $(\%)$ | $\%\ change$ |
| 4 blades | 0.5001 | +6.1 | 14.8222 | +12.2 | 72.7239 | +19.1 | 49.4 | +12.1 |
| 5 blades | 0.5306 | | 16.6339 | | 86.5852 | | 55.4 | |
| 6 blades | 0.5177 | +2.5 | 15.8645 | +4.8 | 80.5754 | +7.5 | 52.9 | +4.7 |

The Table 4.5 shows the result of simulation of propellers with difference numbers of blade. Five blade propeller produces the highest shaft power that is 86.585kW with 55.4% of efficiency.

## 4.3 COMPARISONS OF SIMULATION AND EXPERIMENT

Figure 4.8 shows the comparison of simulation and experiment result that have done. It is illustrates that the propeller with 38 degree inlet tip angle and 74 degree outlet tip angle have produce more shaft power and water discharge for simulation compared to experiment. This is because of some different factors that exist in simulation and experiment.

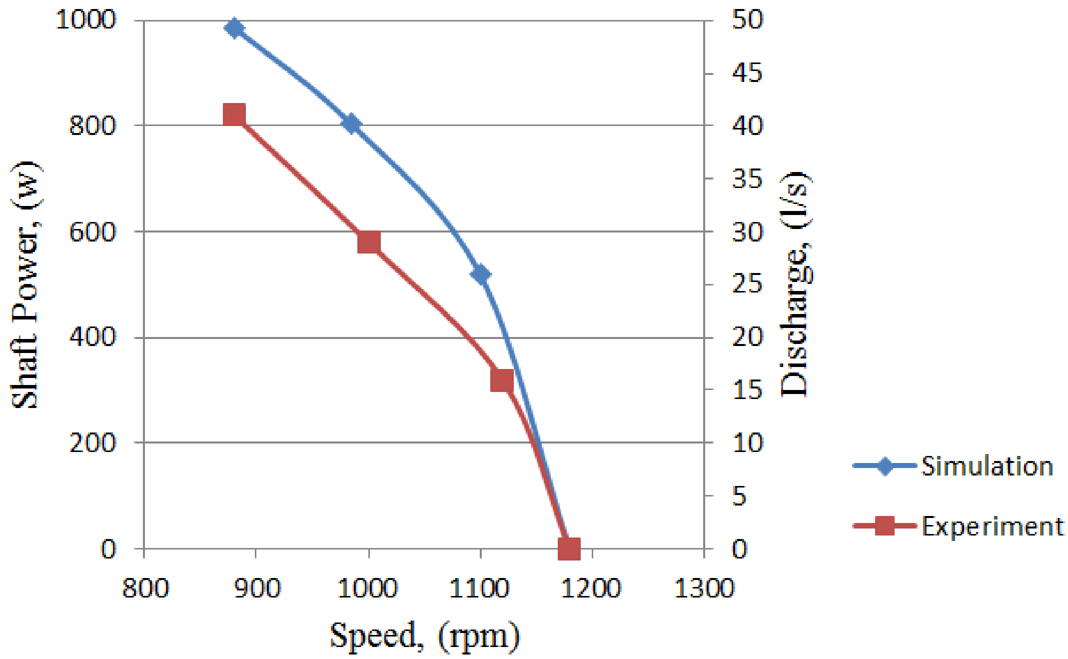



**Figure 4.8:** Graph of propeller speed versus shaft power and discharge for propeller with 38 degree inlet tip angle and 74 degree outlet tip angle

The comparison of computational fluid dynamics simulations with experiment results shows a reasonable fit of the shaft power curve and a worse adjustment in the efficiencies due to scale effects, torque measure accuracy due to mechanical friction in the balance brake system and in bearings, and in leaks between the propeller and the external envelop. CFD simulations are not able to take into account these effects.

# CHAPTER 5

# CONCLUSION AND RECOMMENDATION

## 5.1 Conclusion

In this research work, an understanding on the micro-hydro electric especially on propeller turbine was archived through the research and simulation studies that have been done.

The goal of this project is to optimize the existing design of propeller turbine by using Ansys CFX software. The design criteria of the propeller turbine such blade angle, nose cone, and tip to tip length can increase performance of the propeller turbine.

In conclusion for this project, from the simulation studies that have been done, it is proven that the propeller with $38^0$ inlet tip angle, $85^0$ exit tip angle, with nose cone, and 94mm high of blade was the most reliable design as it gives maximum shaft power that is 86.5852kW. It also has the highest efficient that is 55.4% compared with the other designs. Therefore, the objective of this project has achieved and successfully fulfilled.

## 5.2 Recommendation

The percentage of improvement of the propeller turbine performance is slightly low. For future works, some recommendations have been listed based on the problems in order to improve the performance. First recommendation is about domain. This project only used single domain which is rotating domain in doing the analysis of propeller turbine. To get high accuracy of the result, it recommended using multiple domains that consisted stationary and rotating domain in doing the analysis. Secondly is mesh, because the high quality mesh is highly influence the accuracy of the simulation result which can provide better result compare to by using low quality mesh. The design criteria also should be considered. There are so many design criteria that can be use to optimize the performance of propeller turbine such variation of blade numbers, blade shape, thickness of blade and so on. For future works it is recommended using these criteria to optimize the propeller turbine. In simulation, the computer is a important device. CFD software is the software that uses a lot amount of memory (RAM) and CPU processes especially for complicated simulation. It is suggest that for future works, using high performance computer to doing analysis or using computer that connected to server that have parallel processing that combine  a few computer. By doing this, the computational time can be save. Lastly is about the training. Mostly student have hard time to learn the CFD software due to unfamiliar with this software or lack of expert staff in this field.  It is recommended to university to provide CFD training for student.

## REFERENCES


[1] Alexander, K. V., Giddens, E. P., & Fuller, A. M. (2009). Axial flow turbines for low head microhydro systems. . Journal of Renewable Energy Reviews 34 (2009) 35-47.

[2] Balat, H. (2005). *A renewable perspective for sustainable energy development in Turkey: The case of small hydropower plants*. Journal of Renewable and Sustainable Energy Reviews 11 (2007) 2152–2165.

[3] Carlton, J. (2007). *Marine propellers and propulsion,* 2nd ed. United States of America, USA: Elsevier Ltd.

[4] Choi, H. J., Zullah, M. A., & Roh, H. W. (2012). *CFD validation of performance improvement of a 500kW Francis turbine.* Journal of Renewable Energy 1(2012)pp1-13.

[5] Gerr, D. (2001). *The propeller handbook*. United States of America, USA: McGraw-Hill.

[6] Helena, M. R., Mariana, S., & Borga, A. (2013). *Experiments and CFD analyses for a new reaction microhydro propeller with five blades*, American Society of Civil Engineers : ASCE.

[7] Herzog A. V., Lipman, T. E., & Kammen, D. M. (2000). *Renewable energy sources*. Berkeley, USA: University of California
.

[8] Kontras, E., Gould, K., & Maffeo, D.(2001). *NACA Airfoil Evaluation*. MAE 4440/7440 Aerodynamics: University of Missouri.

[9] Lam, C. and Bremhorst, K., “A Modified Form of the k-e Model for Predicting Wall Turbulence,” ASME *Journal of Fluids Engineering*, Vol. 103, pp. 456-460, 1981.

[10] Leras, P.E., & Hally, D. (2010). *RANS calculations of the flow past inclined propellers.* Canada: Defence Research and Development.

[11] Lipson, H., & Shipitalni, M. (2008). *Conceptual design and analysis by sketching.* USA: Mechanical Engineering Department, Massachusetts Institute of Technology.

[12] Maher, P. and Smith, N. P. A. (2001). “*Micro Hydro For Village Power; A Practical Manual for Schemes up to 100 kW in Hilly Areas*.” 2nd Ed.

[13] Maher, P., Smith, N. P. A. and Williams, A. A. (2003). “Assessment of Micro Hydro as an Option for Off-Grid Electrification in Kenya”. Renewable Energy. 28. pp1357-1369.

[14] Mckinney, J. D., & Warnick, C.C. (1983). *Microhydropower handbook*. United States: U.S. Department of Energy.

[15] Morgut, M., & Nobile, E. (2010). *Influence of grid type and turbulence model on the numerical prediction of the flow around marine propellers working in uniform inflow*. Journal of Ocean Engineering, pp. 26-34.

[16] Mulligan, S. and Hull, P. (2002).*Design and optimization of a water vortex hydropower plant*. Sligeach: IT Sligo.

[17] Nichols, R.H. (2007). *Turbulence models and their application to complex flows*. Birmingham: University of Alabama.

[18] Practical Action, The Schumacher Centre for Technology & Development. (2000). Micro-Hydro Power (MHP).

[19] Ramos, H. M., Borga, A., and Sim˜ao, M.(2010) “*Energy production in water supply systems: computational analysis for new design solutions*,” in *Environmental Hydraulics*, P. A. L´opez-Jim´enez et al., Ed., Taylor & Francis, London, UK.

[20] Simpson, R. G. and Williams, A. A. (2006) *Application of computational fluid dynamics to the design of pico propeller turbine*. Nottingham Trent University.

[21] Singh, P., & Nestmann, F. (2009). *Exit blade geometry and part-load performance of small axial flow propeller turbines: An experimental investigation.* Journal of Experiment Thermal and Fluid Science, 34 (2010) 798–811.

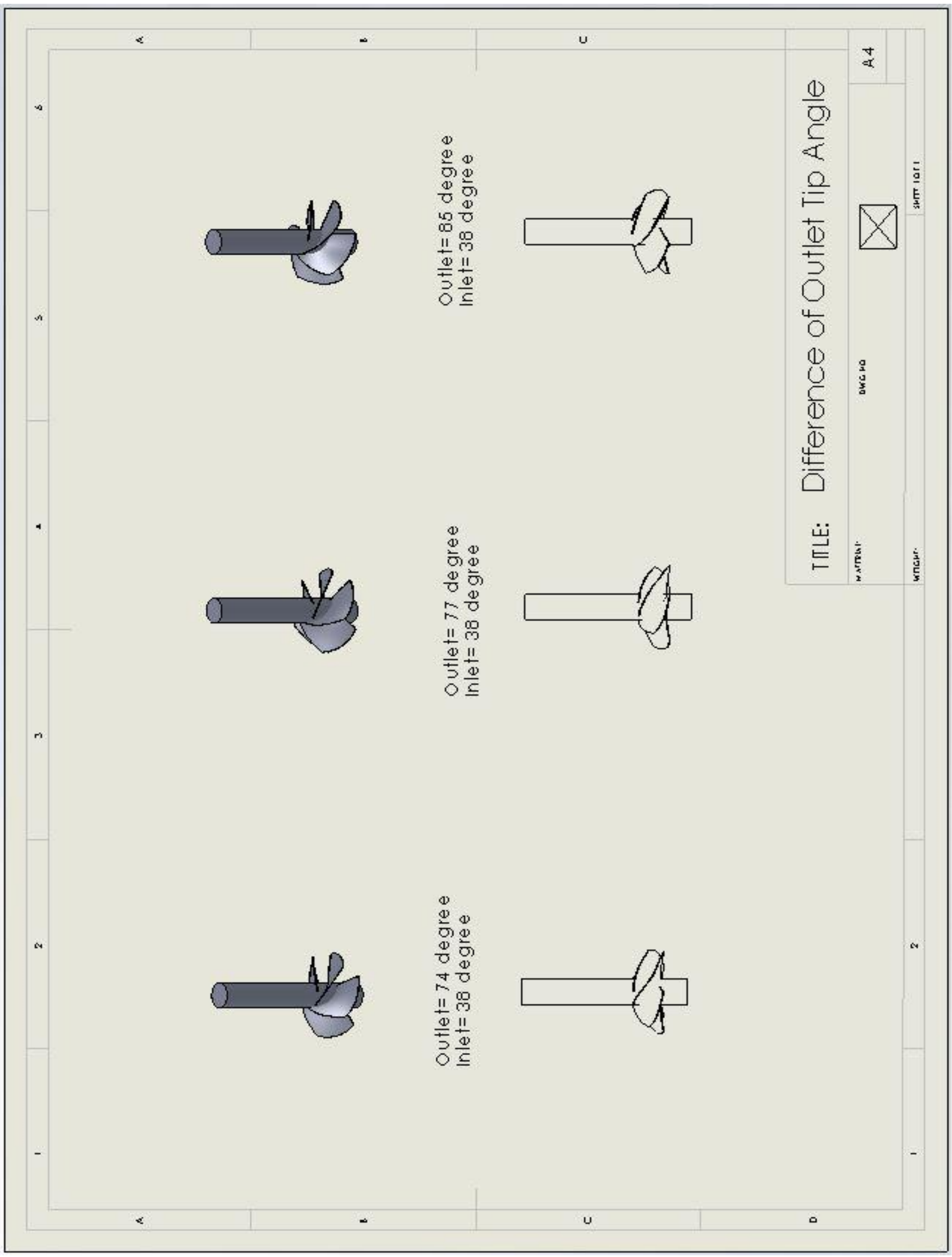
[22] Singh, P., & Nestmann, F. (2009). *Experimental optimization of a free vortex propeller runner for micro hydro application.* Journal of Experiment Thermal and Fluid Science, 33 (2009) 991–1002.

[23] Study spots. (2004). Retrieved November 15, 2012, Retrieved from National Aeronautic and Space Administration (NACA): http://history.nasa.gov/naca/

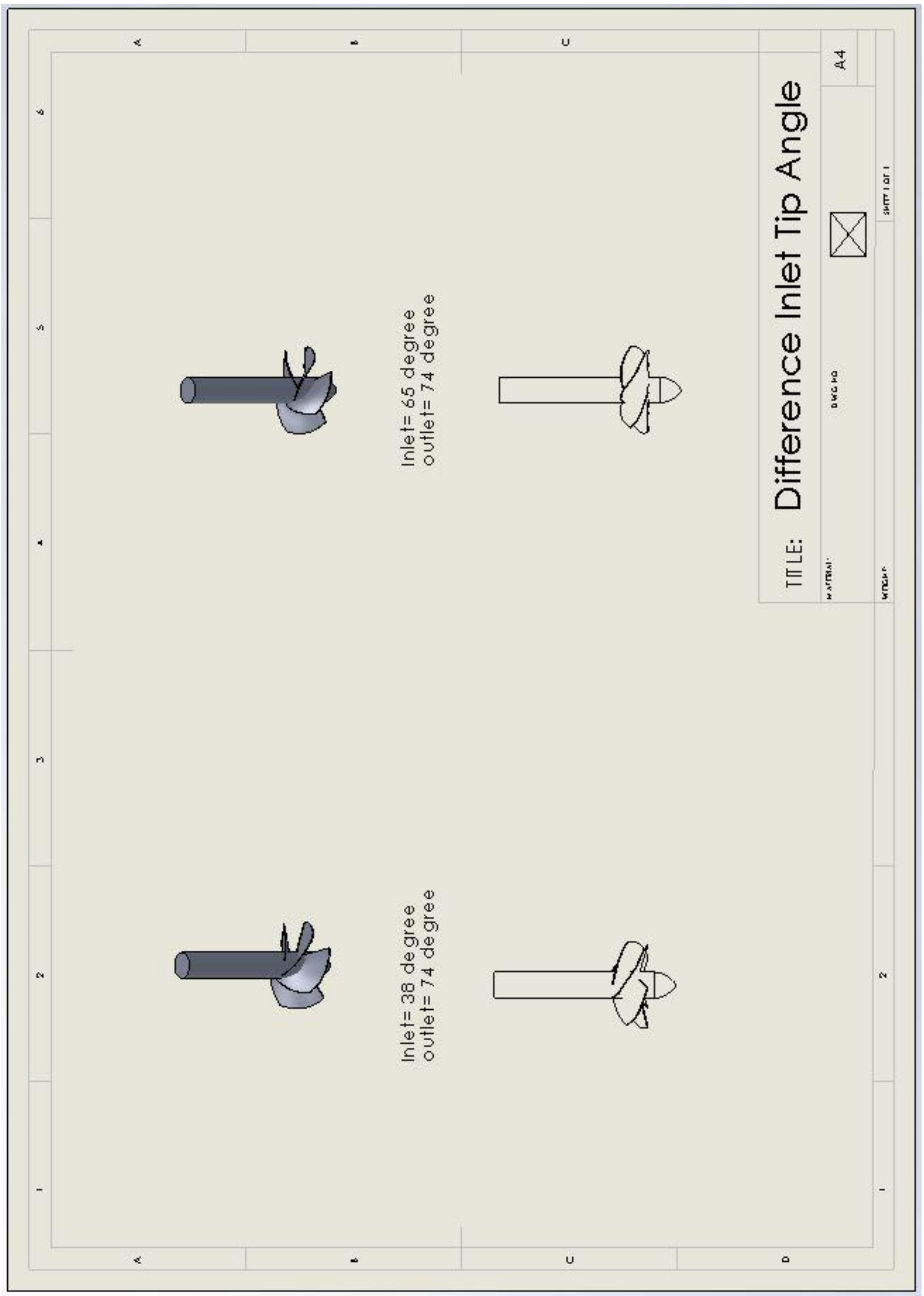
[24] Wang, X., & Walters, K. (2012). *Computational analysis of marine-propeller performance using transition-sensitive turbulence modeling.* Journal of Fluids Engineering Vol. 134, Mississippi State University.

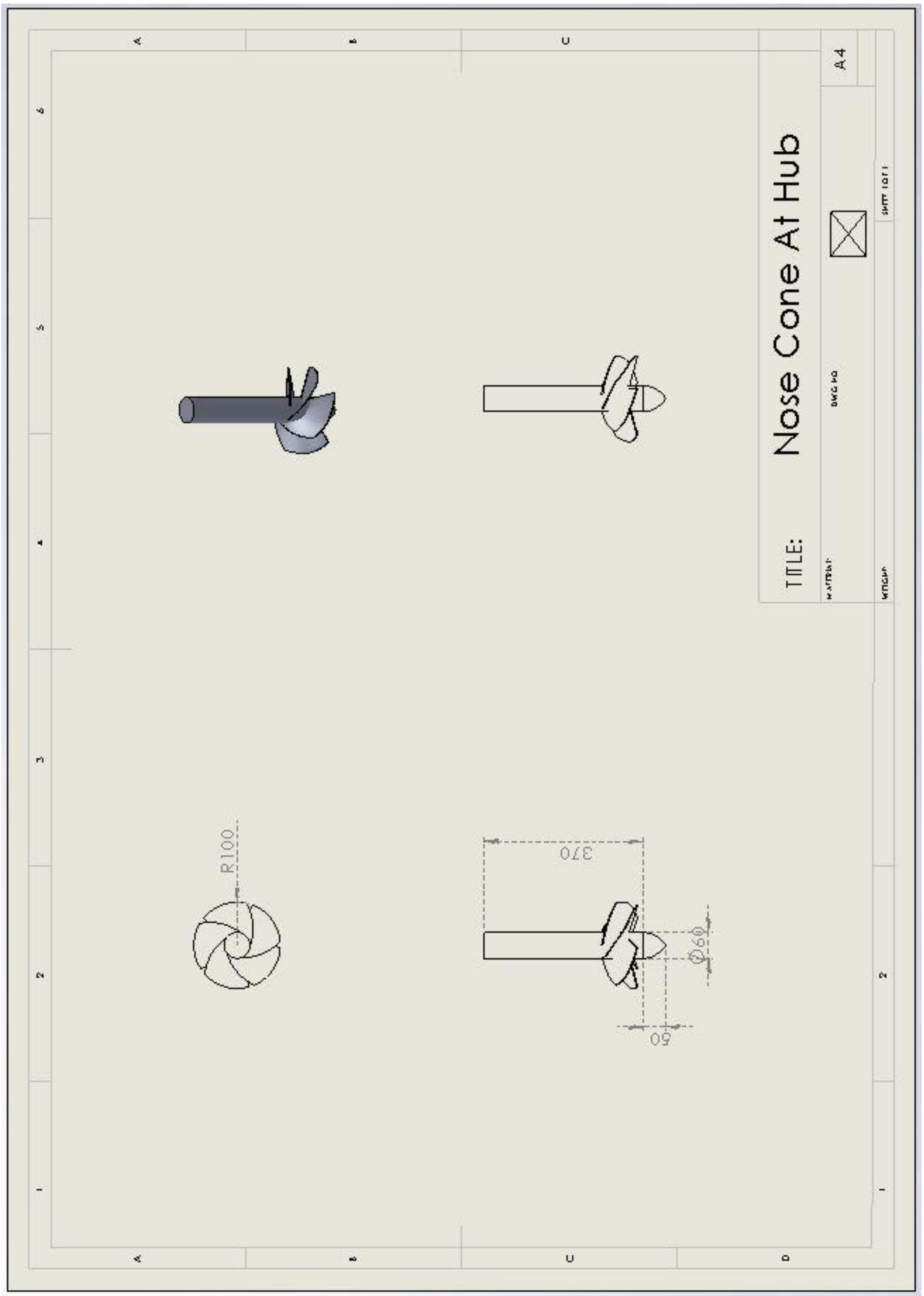
[25] Water Turbines. (2010). “The energy bible”. Retrieved September 8, 2010, from http://energybible.com/water_energy/water_turbines.html

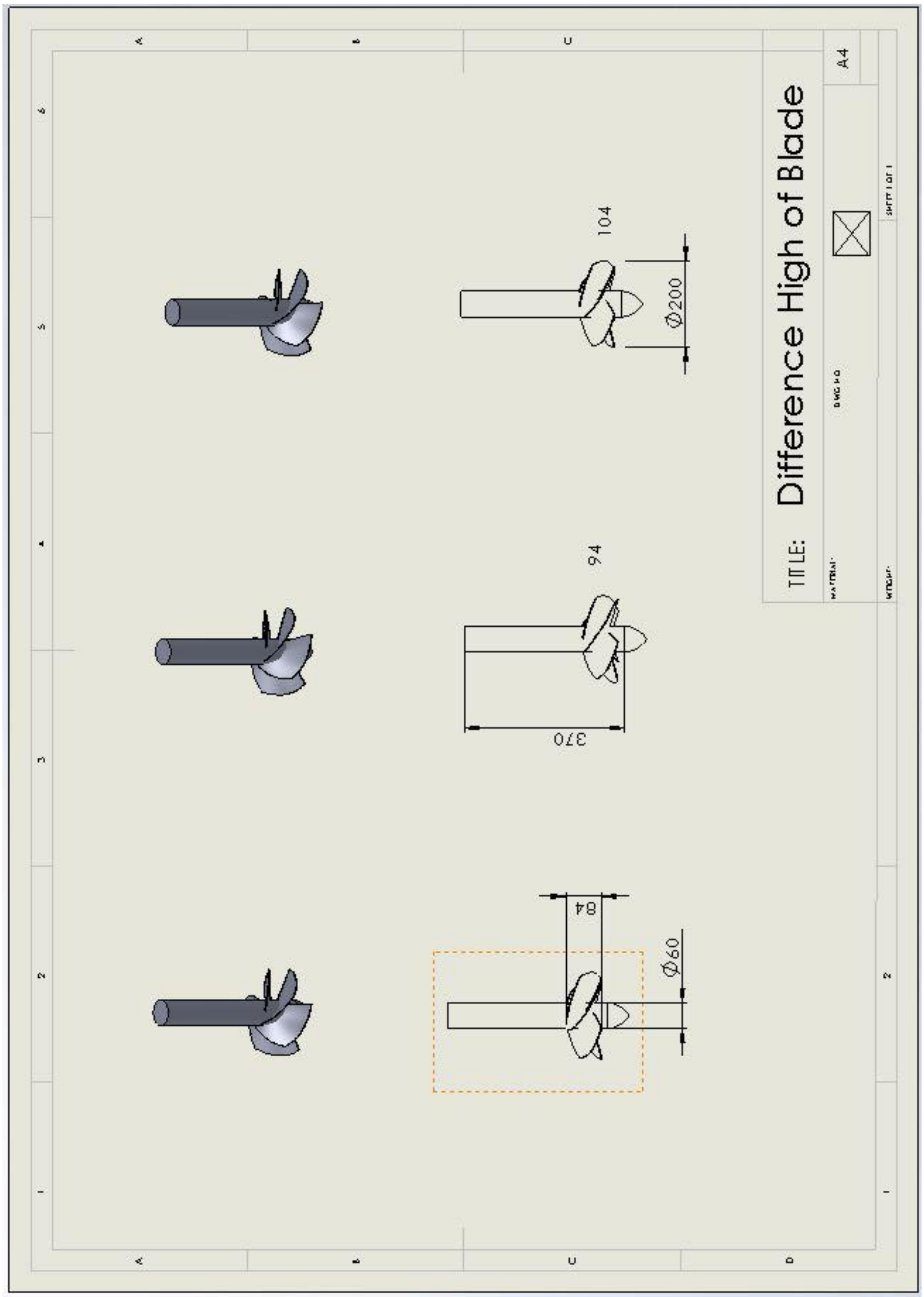
# APPENDICES

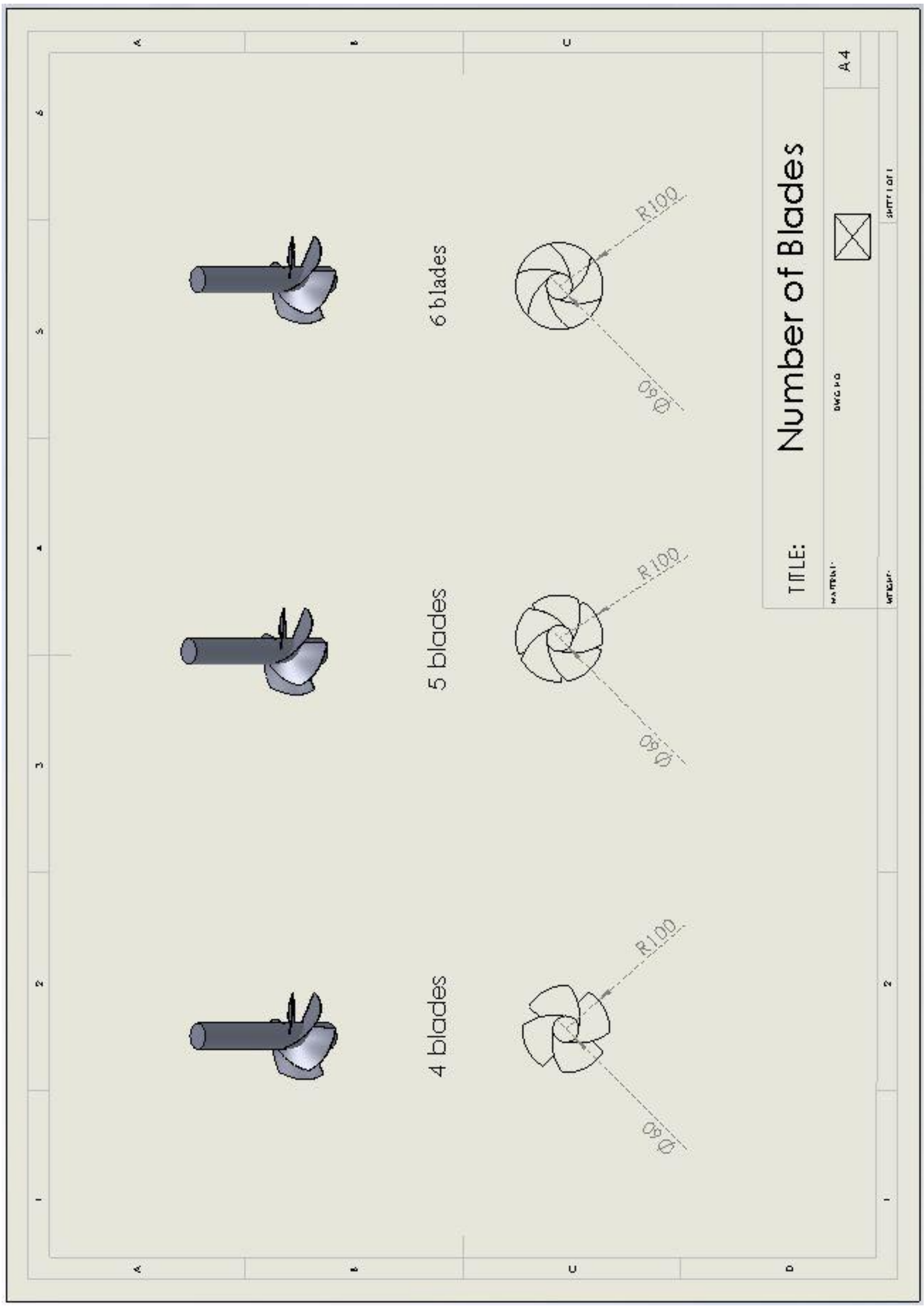


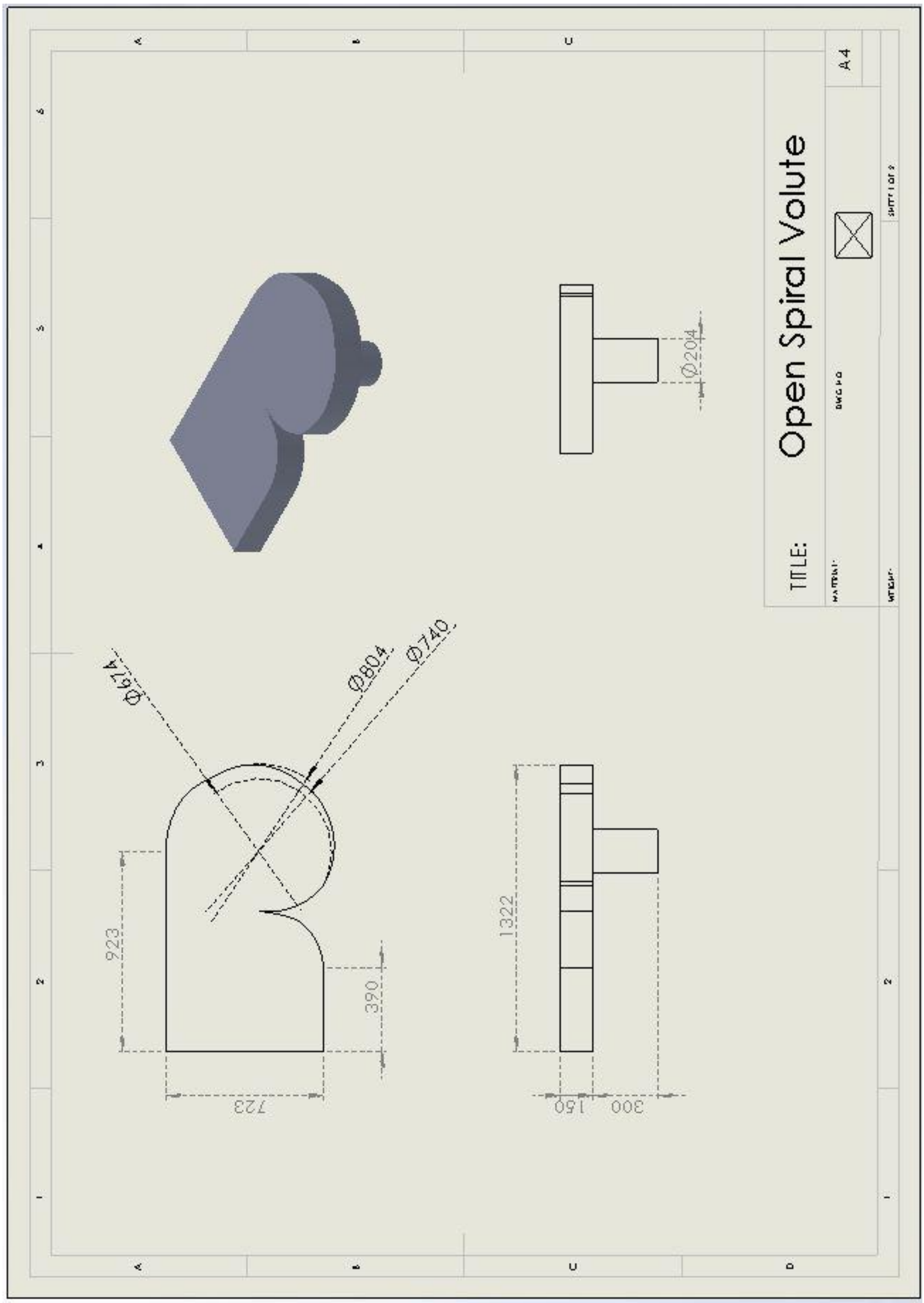
Outlet= 85 degree
Inlet= 38 degree
Outlet= 77 degree
Inlet= 38 degree
Outlet= 74 degree
Inlet= 38 degree
TITLE:
Difference of Outlet Tip Angle
A4

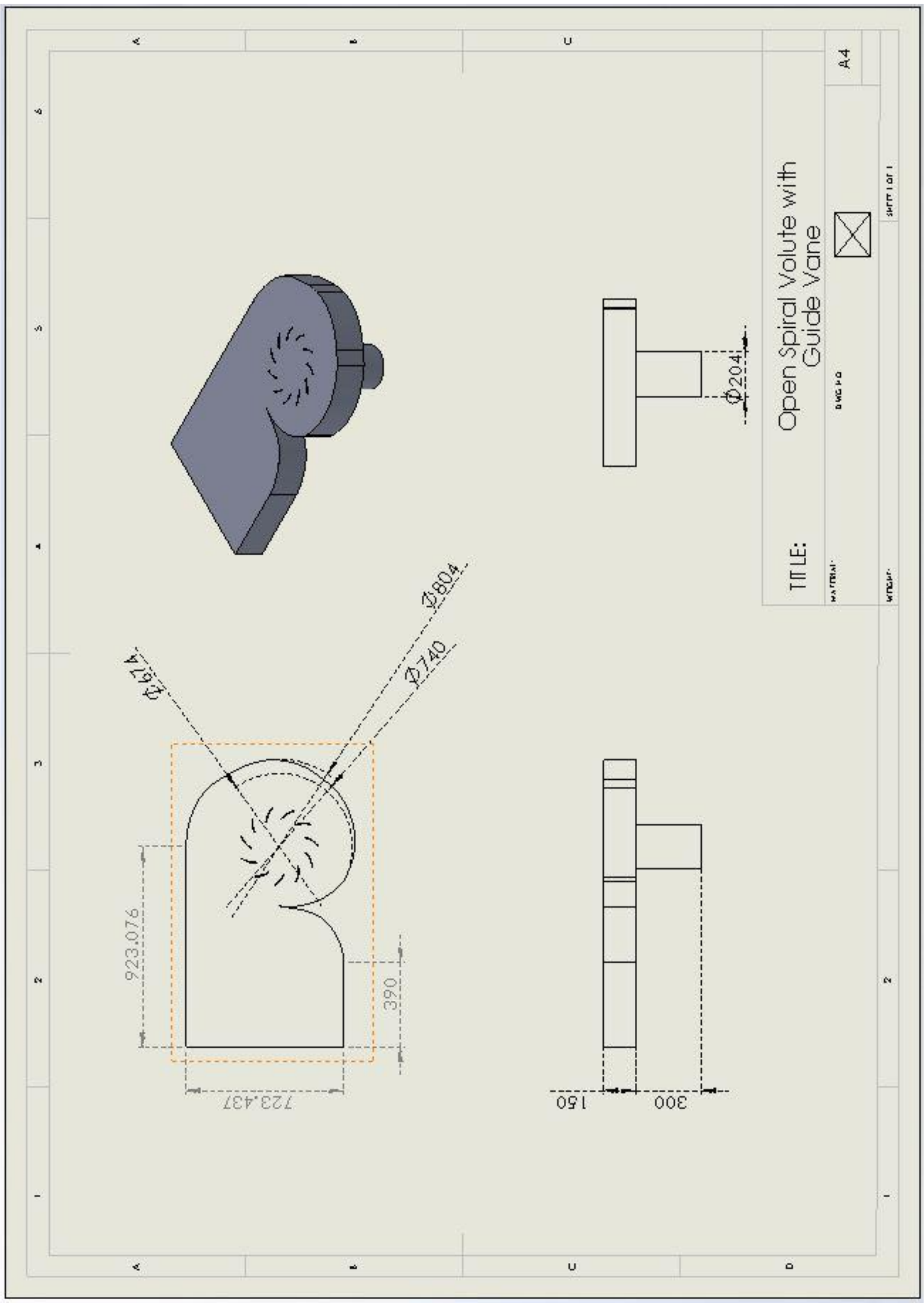

Inlet= 65 degree
outlet= 74 degree
Inlet= 38 degree
outlet= 74 degree
TITLE: Difference Inlet Tip Angle
A4


TITLE:
Nose Cone At Hub
A4
R100
370
Ø60
50


TITLE:
Difference High of Blade
A4
104
Ø200
94
370
84
Ø60


4 blades
5 blades
6 blades
R100
Ø60
R100
Ø60
R100
Ø60
TITLE:
Number of Blades
A4


TITLE:
Open Spiral Volute
A4
Ø674
Ø804
Ø740
923
390
723
1322
150
300
Ø204


TITLE:
Open Spiral Volute with Guide Vane
A4
Ø804
Ø740
Ø674
923.076
723.437
390
150
300
Ø204