DESIGN AND ANALYSIS OF AN UNDERWATER PROPELLER BLADE

ZAIMA SYAFIQAH BINTI MUHAMMAD ZAILAN

A report submitted

in fulfilment of the requirement for the degree of

Bachelor of Mechanical Engineering (with Honours)

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2018

DECLARATION

I declare that this project report entitled "Design and Analysis of an Underwater Propeller Blade" is the result of my own except as cited in the references.

Signature : ……………………………………………

Name : Zaima Syafiqah Binti Muhammad Zailan

Date : ……………………………………………

APPROVAL

I hereby declare that I have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature : ……………………………………….

Supervisor's Name : Dr. Shamsul Anuar bin Shamsuddin

Date : ……………………………………….

DEDICATION

To my beloved mother (Mariyani Binti Mohd Isa) and

father (Muhammad Zailan bin Tugimin)

ABSTRACT

The details about the designing and simulating of an Autonomous Underwater Vehicle (AUV) propeller blade is shown in this final year project report. ROV is a tethered underwater mobile device which would able to work in any depths of water and has more abilities to perform mechanically, inspection and recording tasks. Propulsion plays a significant role in the movement of the ROV. The objective of this paper is to determine the important design factors that affecting propeller efficiency and to identify the velocity and thrust in order to determine the ROV propeller blade efficiency. Based on the literature review study, the number of blades, propeller diameter, rake angle, blade design or blade area and ducted propeller are very important parameter in order to design the optimum propeller. The best conceptual design was selected as it achieved highest value in concept evaluation. The new design of the propeller blade is based on the specification parameter of the existing thruster (T200 thruster by Blue Robotics) so that the diameter of the propeller blade of the new design same and it would able to fixed into existing T200 thruster. Solidworks is used to design the propeller blade and analyse some performance of it. In addition, ANSYS Fluent is also being used to analyse the meshing size and structural meshing of the propeller blade with its propeller domains. The efficiency of the propeller blade is calculated by using the mathematical formulation. It was found that from the calculation, the efficiency of the propeller blade is different with different rpm value. In addition, it was known that as the rpm, thrust coefficient and velocity inlet decreased, the efficiency of the propeller blade is increasing. At the speed of 31.42 rad/s, the efficiency of the propeller blade is 93.63% while at 183.25rad/s, the propeller efficiency is 92.91% and for 397.94 rad/s, the propeller efficiency is about 92.42%.

ABSTRAK

Reka bentuk dan simulasi mengenai bilah kipas Autonomi Kenderaan Bawah Air (AUV) ditunjukkan dengan lebih terperinci di dalam laporan projek akhir ini. ROV adalah peranti mudah alih bawah laut yang dapat bekerja di semua jenis kedalaman air dan mempunyai kemampuan untuk melaksanakan tugas mekanikal, pemeriksaan dan rakaman. Pengusiran memainkan peranan penting dalam pergerakan ROV. Objektif projek sarjana muda ini adalah untuk menentukan faktor reka bentuk penting yang mempengaruhi kecekapan bilah kipas dan untuk mengenal pasti hadlaju, teras dan kuasa untuk menentukan kecekapan bilah kipas ROV. Berdasaran kajian literiture, bilangan bilah, diamater kipas, sudut rake,reka bentu bilah atau luas kawasan bilah dan kipas ducted merupaan parameter yang sangat penting untuk mereka bentuk kipas yang lebih optimum. Reka bentuk konseptual terbaik dipilih kerana ia mencapai nilai tertinggi dalam penilaian konsep.Reka bentuk bilah kipas yang baru adalah berdasarkan parameter spesifikasi thruster yang sedia ada (T200 thruster oleh Blue Robotics) supaya nilai diameter baling kipas reka bentuk baru sama dan ia dapat disesuaikan ke dalam T200 thruster itu. Solidworks digunakan untuk mereka bentuk bilah kipas dan menganalisis beberapa prestasi. Di samping itu, ANSYS Fluent juga digunakan untuk menganalisis prestasi lain untuk menyelesaikan nilai parameter yang tidak diketahui untuk mencari kecekapan bilah kipas. Kecekapan bilah kipas dikira dengan menggunakan formulasi matematik.Berdasarakan projek ini, dapat diketahui apabila nilai rpm,pekali teras dan hadlaju masuk berkurang, nilai kecekapan bilah kipas semakin meningkat.Berdasarakan pengiraan yang dilakuan, kecekapan bilah kipas adalah berbeza dengan nilai rpm yang berbeza. Pada kelajuan 31.42 rad/s, kecekapan bilah kipas adalah sebanyak 93.63%. sementara untuk rpm 183.25 rad/s , nilai kecekapan bilah kipas adalah kira-kira sebanyak 92.91%. Sementara itu, pada 397.94 rad/s, kecekapan bilah kipas adalah 92.42%

ACKNOWLEDGEMENT

First and foremost, all praise to Allah the Almighty, the Benevolent for His blessing and Guidance for giving me opportunity for learn and giving me strength to through the entire final year project 1 and 2. Many people contributed to the creation and completion on this final year project.

This project report is the sign that I have almost completed my study journey as Bachelor of Mechanical Engineering student in Universiti Teknikal Malaysia, Melaka. Therefore, I want to express my sincere appreciation to my Final Year Project supervisor, Dr Shamsul Anuar bin Shamsuddin, for the guidance, encouragement and positive feedback to help me whenever I need an advice and comment throughout the project . I am sincerely thanks to my second examiner and panels which are En.Masjuri Bin Musa @ Othman and Dr Shafizal Bin Mat for their guidance, advices and motivation along this final year project.

I am grateful to my family, especially my mother and father for their endless. I am also extending my appreciation to my friends for the positive support and always be there to help me. Last but not least, to UTeM staffs, thank you for always reminding the final year students with the important updates. Finally, we would like to thank everyone who is involved in this program directly and indirectly for their help.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF APPENDICES

LIST OF ABBREVIATIONS

ROV - Remote Operated Underwater Vehicle

AUV - Autonomous Underwater Vehicle

CFD - Computational Fluid Dynamics

BEM - Boundary Element Method

BAR - Blade Area Ratio

2D - Two Dimension

3D - Three Dimension

LIST OF SYMBOLS

D, d - Diameter

R, r - Radius

$\dot{m}$ - Mass Flow Rate

T - Thrust of the Propeller

Q - Torque Produced by the Propeller

A_o - Surface Area of the Propeller Blade

V_A - Inlet velocity

V_B - Velocity against the Thrust of the Propeller

V_C - Outlet velocity

n - Rotational Speed

P_D - Power obtained by the propeller

J - Advance Ratio

C_T - Thrust Coefficient

C_Q - Torque Coefficient

ρ - Density of the Water

η_o, η_i = Propeller Blade Efficiency

CHAPTER 1

INTRODUCTION

1.1 Background

Nowadays, scientists always find the best solution to get themselves or their instruments into a very specific parts in order to understand the ocean. They have to develop the underwater vehicle to make the research in an easy way. Traditionally, the researchers need to use ships to take a photo of the depths and to collect samples of water, rock, and marine life. However, in modern life, they do some innovation such as remote-controlled vehicles, autonomous, and towed robots.

Underwater vehicles are classified as either Manned, Remotely Operated Underwater Vehicles (ROV) and Autonomous Underwater Vehicle (AUV). They are submersible vehicle that can be assigned to do certain task underwater. The underwater vehicle also provides the functions necessary to serve as undersea tools for science and industry.

Manned submersibles are underwater vehicles that are controlled by human operators. These allow scientists to dive into the deep ocean and collect data or make observations first hand.

A remotely-operated underwater vehicle (ROV) is safe and controlled by an operator who remains out of the water which is this robot was controlled by a person on the surface. An ROV may contain a video camera, lights, sonar systems, and an articulating arm.

However, different with autonomously-operated underwater vehicle (AUV) which is designed to work without an operator and without a direct connection to the surface. These AUV are normally designed for a particular application and are pre-programmed to operate definite particular tasks. Figure 1.1, 1.2 and 1.3 show the example design of the manned submersible, Remote Operated Underwater Vehicle (ROV) and autonomous operated underwater vehicle (AUV) respectively.

All these underwater vehicles need a propulsion from the thruster to move themself. The design of the propeller blade inside the thruster is very important to ensure the underwater vehicle would able to move in efficient way. S. O. Oladokun (2015) believes that the propeller blade would able to move the ship against resistant of water which is this propeller converts engine torque into propulsive force or thrust and produce forward motion. In propeller design, it is important to ensure that it drives the vessel efficiently.

Figure 1.1: Manned Submersible Underwater Vehicle

Figure 1.2: Remote Operated Underwater Vehicle (ROV)

Figure 1.3: Autonomous Operated Underwater Vehicle (AUV)

.

1.2 Problem Statement

Lately, the ROV evolution has focussed on smaller size and higher efficient energy technology. The propeller blade of ROV is very important in producing a more efficient horizontal and vertical manoeuvre. However, the number of research on designing and manufacturing marine propeller is less. Most of them depend on the propeller of the airplane for propulsion as off the shelf propeller is more sensible because it is cheaper and easy to replace. Yet the use of airplane propeller in marine vehicle is not suitable because the density of air is 1000 times less than water (D'Epagnier, 2006).

The thruster and propeller blade itself have an electric motor that attached to it which it gives the underwater robot as a propulsion device. These enable the robot to make a movement and have the ability to against sea water resistance. If the efficiency of the propeller blade is low, the underwater vehicle would not able to move faster in an efficient way. Plus, underwater vehicle will have a low ability to against the sea water resistance (Maghareh, 2017)

Therefore, it is important for this research to clearly show the steps of the advance method in ROV propeller development which will promote engineers to adapt and continue the new design. The research of the propeller is important in order to know the best design based on the previous lesson. So, based on the research and process learning, the improvement of the propeller blade can be made referring to the standard criteria and parameter consideration.

1.3 Objective

The goals of this final year project are as follows:

1. To determine the important design factors that affect propeller efficiency.
2. To identify the velocity and total thrust of propeller blade in order to determine the ROV propeller blade efficiency.

1.4 SCOPE OF PROJECT

This research focuses on the hydrodynamic performance of the ROV where an efficient propeller is needed to be found using the designing and stimulating of the propeller blade in order to produce a reliable propulsion system.

The scopes of the study on ROVs propeller blade are:

i. To study the design process of ROVs propellers
ii. To design and modelling a propeller blade for ROVs referring to design parameters.
iii. To analyse the velocity and total thrust of the propeller blade using Computational Fluid Dynamics (CFD)
iv. To calculate the propeller efficiency using mathematical modelling.

CHAPTER 2

LITERITURE REVIEW

2.1 Introduction

Most of the literature review on propeller blade is full with experimental and analytical data that related with performance information and the design criteria. A propeller turns the engine power into the operating force of the ship and usually the propellers are used as propulsions and used to progress significant dive to propel the vehicle at its operational speed and RPM (B. Harish et al. 2015). I. Anthony et al. (2013) state that the propeller is a type of a fan that transmits power by converting rotational motion into thrust.

Propeller dynamics can be represented by both Bernoulli's principle and Newton's third law. A propeller is sometimes known as screw which it is actually can have a single blade, but in practice there always more than one in order to make the forces involved is balanced (Anthony et al. 2013). These authors have explored their study about the propeller blade efficiency and knowing that a propeller with blade area ratio of 0.55 has the open water efficiency about 73%.

Usual propeller blade geometry is shown in Fig.2.1.1 which profiles some of the terms used in designing an underwater propeller system. The propeller type, geometrical and performance parameters like radial velocity, thrust coefficient and others impressed the ship propeller.

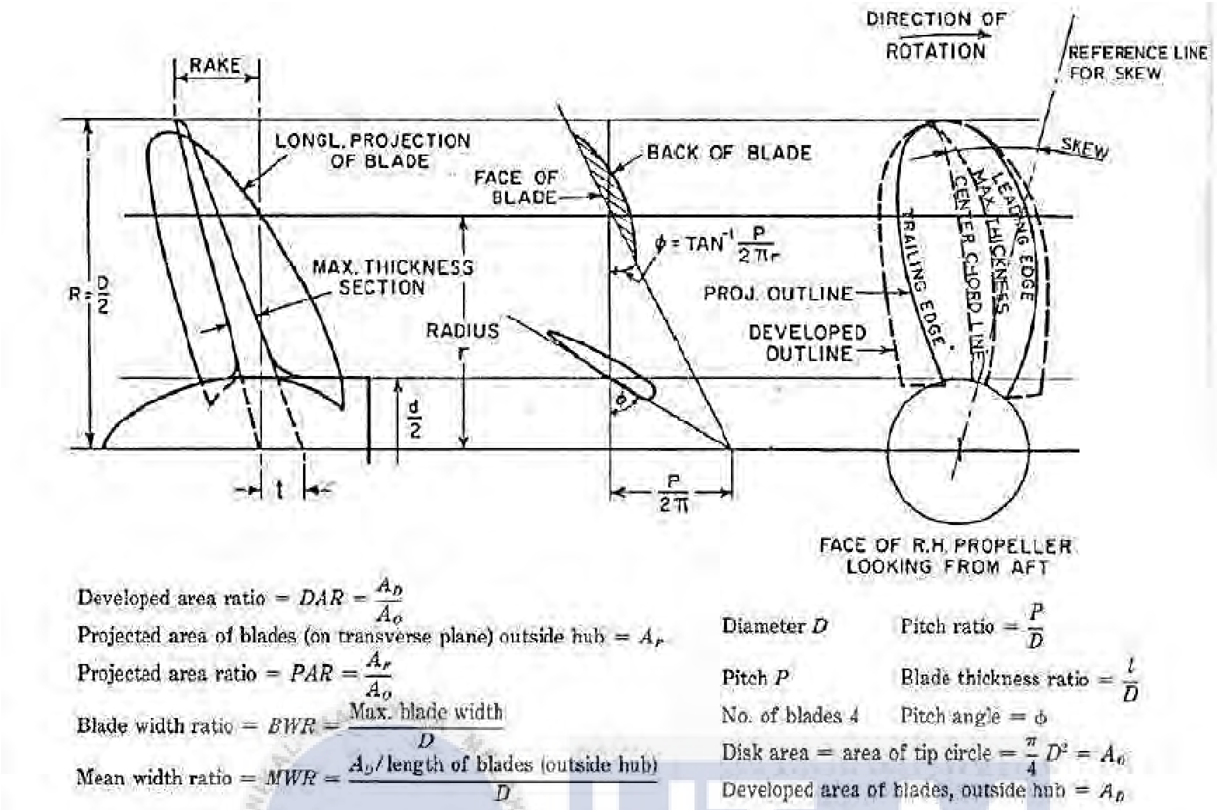


Figure 2.1.1: Typical propeller drawing (Anthony et al. 2013)

2.2 Design Analysis and Considerations

Anthony et al. (2013) focused the propeller design analysis at obtaining least power requirements, cavitation, noise, vibration and supreme efficiency conditions at a sufficient revolution. Ekinci (2011) believed that there are two methods are commonly used in propeller design which are by using the diagrams obtained from open water propeller experiments for systematic propeller series and mathematical methods (lifting line, lifting surface, vortex-lattice, BEM (boundary element method) based on circulation theory.

2.3 Parameter Analysis

Propeller diameter, number of blades and propeller speed is the parameters that supply foundation for the propeller blade. Meanwhile, the multi-combination of these three parameters will result in different efficiencies. Therefore, D'Epagnier et al. (2007) make a further study of the parameter that is important to know the propeller optimization.

2.3.1 Number of blades

Mutalib et al. (2015) mention that the most frequent marine propellers are designed to decrease the noise sound such as three or four bladed marine propellers. The number of blades selected will impact the level of unsteady forces working on them. Optimum open water potency will increase with increase within the range of blades by considering the efficiency of propeller blade.

2.3.2 Propeller diameter

Matt (2015) claims that the selected propeller diameter and pitch is important where it contribute in the propeller's ability to absorb power, the engine's ability to produce power, and either it properly matched at the engine's rated rpm.

These author also state that the huge diameter and a shallow pitch of propeller blade may be used, or a smaller diameter and a steeper pitch, to produce the equal load on the engine. However, if it going so much in either method through, the efficiency of the propeller will be decrease.

In a terrible tiny diameter, steep pitch propeller tends to place a lot of work into spinning the water flow, and fewer into accelerating fast away from the boat. Besides that, even in fixed rpm, if the diameter is increase too much, the pitch should be created terribly shallow, and efficiency once more decrease.

2.3.3 Blade area and blade design

Blade area ratio (BAR) is the magnitude relation or ratio of the whole area of the blades, divided by the whole area of the propeller. This blade area ratio might affect the propeller efficiency and thrust making performance. Blade design may become the minor consideration in selecting the propeller design. However, the blade area ratio may effected by the blade design. Therefore, the different blade design may affect the efficiency of the propeller blade itself. Lee et al. (2010) had design technique for greater performance of the marine propellers with an optimum propeller performance. These authors was adjusting expanded areas of the propeller blade. Results show that efficiency can be grow up to over 2% by decreased the blade area. Table 2.3.3 shows the comparison result for parent propeller, design propeller (7% decreased) and design propeller (14% decreased).

Figure 2.3.3 (a) Parent propeller (Lee et al. 2010)

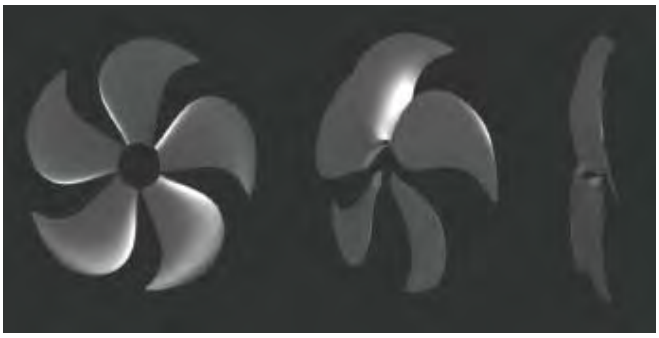


Figure 2.3.3 (b): Design Propeller (7% decreased) (Lee et al. 2010)

Figure 2.3.3 (c): Design Propeller (14% decreased) (Lee et al. 2010)

Table 2.3.3 Comparison of different area design propellers. (Lee et al. 2010)

Propeller	J_A	K_T	$10K_Q$	$\eta_{propeller}$	$\Delta\eta_{propeller}$ (%)
Parent	0.5790	0.2240	0.3330	0.6090	-
7% decreased	0.5790	0.2240	0.3280	0.6180	1.50
14% decreased	0.579	0.2240	0.3250	0.6240	2.40

2.3.4 Rake Angle

Rake angle is the quantity of degrees the propeller blades angle perpendicular to the propeller hub. The vary angle could be from -5 to 30 degrees. The median rake angle for many propellers is 15 degrees. Figure 2.3.4 below show the various rake angle which are 0 degree, 15 degree and 30 degree respectively.

(a) 0 degree rake angle. (Scott Reichow, 2011)

(b) 15 degree rake angle. (Scott Reichow, 2011)

(c) 30 degree rake angle. (Scott Reichow, 2011)

Figure 2.3.4

Scott Reichow (2011) mention that the blade rake increases as a blade face slope back near to propeller rear. Rake can be divided into two categories which are flat (straight) and curved (progressive). Boat that has high rake angle may enhance a better performance by holding the bow higher and make the boat lighter and fast. However, some cases for light boat, the high rake angle may cause the bow elevate too much. It shows that high rake angle also can cause too much elevate bow. This will make the boat less stable. Therefore, a medium rake propeller (or a suites style for outboard) could be a more sensible choice.

2.3.5 Ducted propeller

In this recent years, many authors make a research of the dynamics performance of the ducted propeller. This is because they conduct a study and knowing that the ducted propeller is actually more efficient in producing thrust than a conventional propeller of same diameter, especially at low speed and high static thrust level. On the other hand, the ducted fan also provides a better safety on the ground. The comparison between this both type of propeller had been analysed by (Baltazar et al.2012).

Figure 2.3.5 below shows the profiles of duct for duct type 19 A and 19 Am. The differences between the duct inner surface and the blade tip is steady and become 0.83 % of the radius of the propeller.

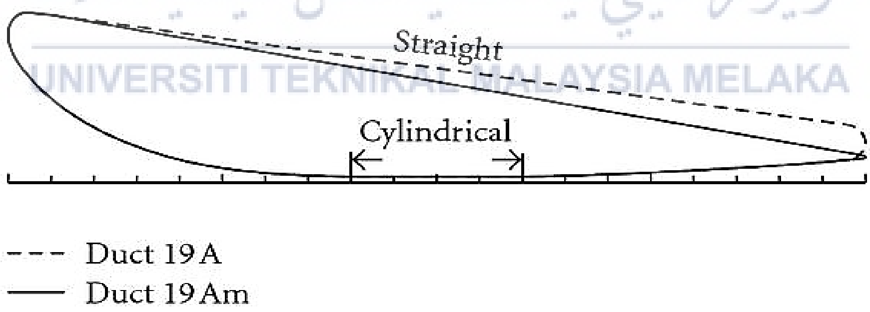


Figure 2.3.5: Profiles of duct type 19 A and 19Am. (Baltazar et al. 2012)

However, through Xueming et al. (2015), the flow field distribution characteristics of the ducted propeller has been analysed where table 2.4.5 shows geometry parameters of the propeller.

Table 2.3.5: Geometry parameters of the propeller. (Xueming et al. 2015)

Parameter	Value
Diameter (m)	0.06
Hub Diameter Ratio d_o/D	0.2
Number of Blades	4
Gap between the Blade and Duct (mm)	2

2.4 CFD Model

The application of greater speed computers and development of advanced software, the simulation of the propeller hydrodynamics is now possible in behind underwater vehicle condition. The calculation of the water flow around the underwater vehicle accompanied with the speeds that reach the propeller disk can be found using CFD (Computational Fluid Dynamics). B. Harish et al. (2015) trust that CFD is usually used in broad type of applications such as flow analysis, thermal analysis of various equipment used in industry. By giving the necessary boundary conditions, the forces acting on the model would able to analyse.

2.4.1 Geometry and Propeller Domain

Chruściel et al. (2014) conclude that the main objective of the construction of CFD models was to gain knowledge on how to model the discussed issue, and to approach the simulation results with the theoretical and experimental results. Next, the loading force on the propeller was assessed using the pressure generated in the CFD analysis of the free propeller. Figure 2.4.1(a) shows the domain of the free propeller model.

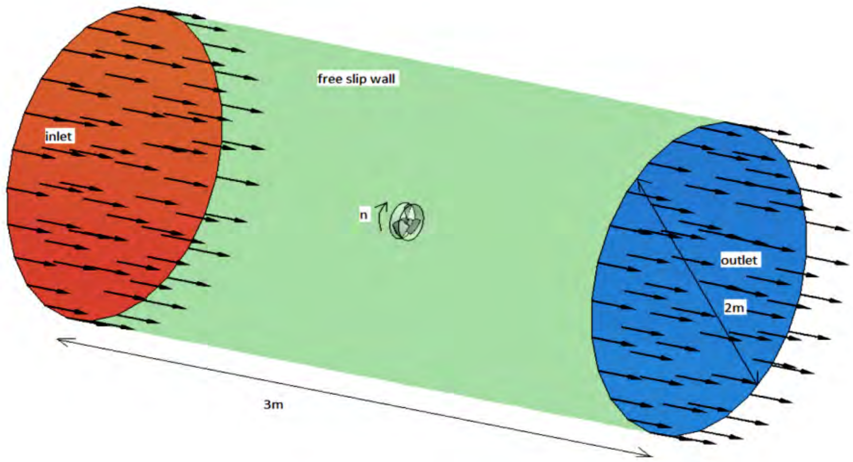


Fig. 2.4.1(a): Domain of the free propeller model. (Chruścinael et al. 2014)

Maghareh and Ghassemi (2017) state that the computational domain is required to be separated and to turn the partial differential equations into sequence of algebraic equations. The propeller is placed into two different size of domain which is the first domain cylinder diameter is about 0.1D while the second domain radius is around 0.5D. However, both domains depends on the size of the designed propeller and the suitability. The geometrical dimensions of the propeller are presented in Table 2.4.1 while the computational domain is shown in Figure 2.4.1(b).

Table 2.4.1: Main geometry of the propeller. (Maghareh and Ghassemi, 2017

Propeller diameter (m)	0.300
First domain diameter (m)	0.340
Second domain Radius (m)	1.50
Propeller upstream (m)	1.00
Propeller downstream (m)	4.00

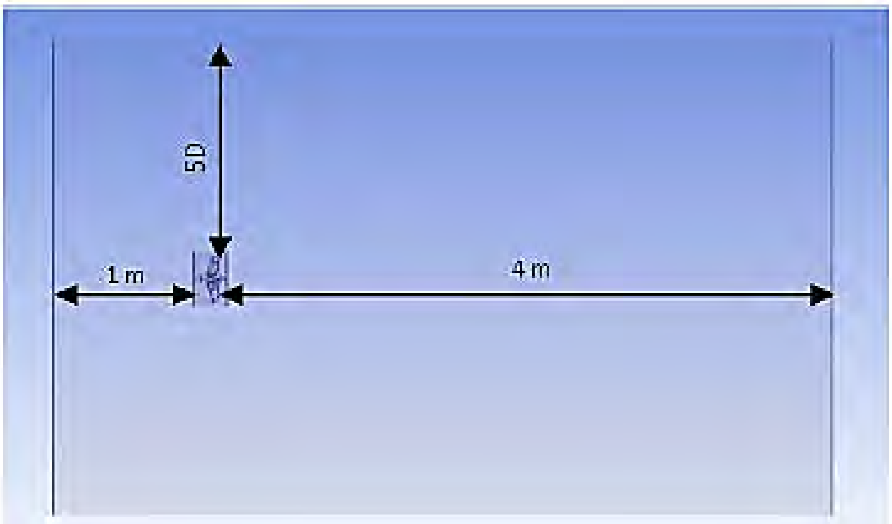


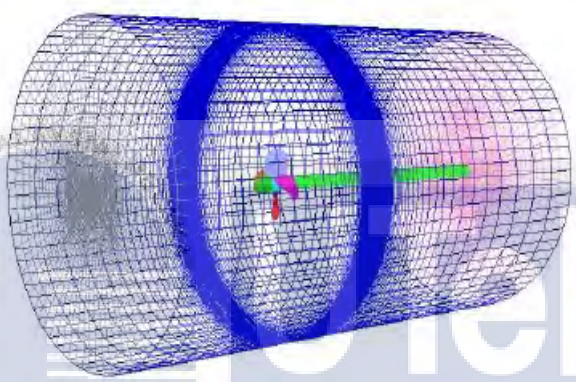
Figure 2.4.1(b): Computational domain for the propeller blade. (Maghareh and Ghassemi, 2017)

2.4.2 Grid Generation or Meshing

Ramakrishna et al. (2012) claim that by generate the structural grid or meshing in ANSYS is important as it produces accurate mesh for efficient multi-physics solutions. The propeller domain was set to 3D (where D is diameter of the propeller) from mid of the chord of the root section while the other domain was set to 4D from same point at downstream.

As a result, the mesh caused the cell sizes near the blade wall small and increased near outer boundary. The convergence total number of cells build for whole grid was 1.3 million. Fig. 4 and Fig. 5 show the grid over propeller and entire domain of propeller respectively by using Fluent 6.3 software

(a) Grid for the propeller. (Ramakrishna et al. 2012)



(b) Grid over the entire domain. (Ramakrishna et al. 2012)

Figure 2.4.2

2.4.3 Setup and Solver

Husaini et al. (2014) had run CFD simulation by set the boundary condition of the inlet velocity at 2 m/s as a resistance in order to know the thrust required to move 2m/s in water. Muljowidodo et al. (2009) had designed and analysis of the ducted propeller using CFD simulation. The parameter of the designed propeller is shown as table 2.4.3.1 below. The 3D design for the ducted propeller is shown in figure 2.4.3.2 while table 2.4.3.3 shows the variation speed, thrust and torque of the propeller blade.

\

Table 2.4.3.1: Parameter of ducted propeller designed. (Muljowidodo et al. 2009)

Propeller Diameter (mm)	161
Hub Diameter (mm)	50
Blade Number	5
Nozzle inner diameter (mm)	163
Outer nozzle diameter (mm)	220
Nozzle length (mm)	82
Propeller speed (rpm)	1300
Propeller motor power (Watt)	400

Figure 2.4.3.2: 3D design for ducted propeller (Muljowidodo et al. 2009)

Table 2.4.3.3: Variation speed, thrust and torque of the propeller blade. (Muljowidodo et al. 2009)

Rotation (rpm)	Thrust (N)	Torque (Nm)	Power (Watt)
1300	4.34	0.1047	60.48
600	14.45	0.382	132.762
900	37.5	0.817	293.44
1300	74.4	1.56	393.84

2.5 Propeller Theory

2.5.1 Actuator Disk Theory

The disc actuator theory is an ideal momentum model which has no friction, incompressible, and irrotational. This theory is originally to supply an analytical means that evaluating ship propellers (Rankine, 1865) and (Froude, 1885). Blade Element Momentum Theory Blade element momentum theory (BEMT) is used in design and analysis of the ship propeller (Benini, 2004).Regarding to the figure 2.5.1 below, the ship direction towards left direction while V_A is the inlet velocity, V_B is the velocity against the thrust of the propeller and V_C is the outlet velocity.

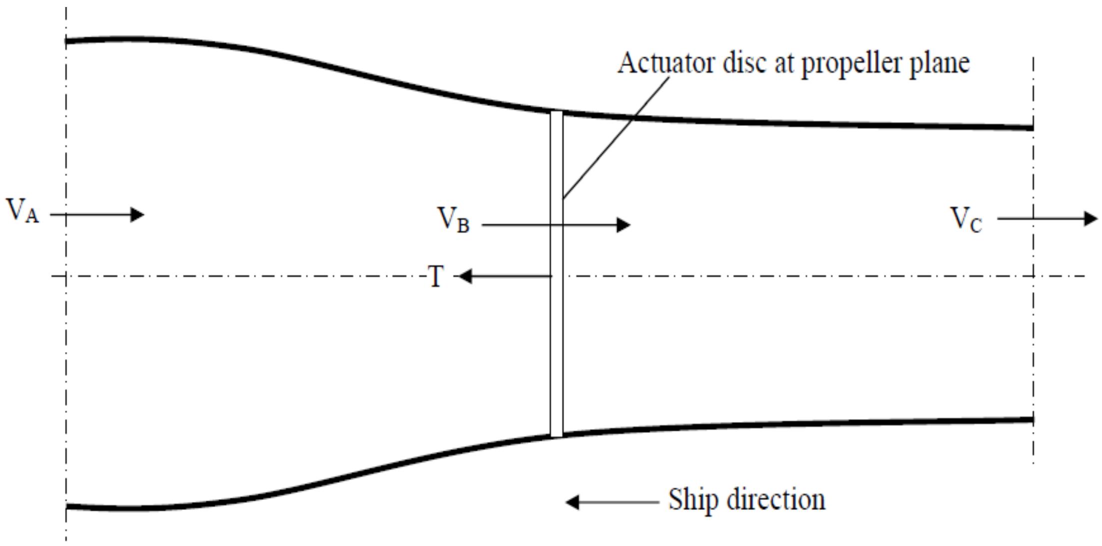


Figure 2.5.1: Disc Actuator Model. (Rankine, 1865) and (Froude, 1885).

2.5.1.1 Mathematical Modelling

Based on the basic thrust equation, the mass flow rate through the propeller will affect the amount of thrust besides, the velocity differences through the propulsion system. Regarding to figure 2.5.1, it shows the flow from left to the right. The equation for the thrust is:

$$T = \dot{m}\,(V_c - V_A) \quad (1)$$

where T is the thrust of the propeller blade [N], $\dot{m}$ is the mass flow rate [kg/s], V_C is the outlet velocity [m/s^2], V_A is the inlet velocity [m/s^2].

As the pressure at C is same to pressure at A, therefore, there is no pressure difference term. The power PD absorbed by the propeller is stated as:

$$P_D = \frac{1}{2}\dot{m}\,(V_c^2 - V_A^2) \quad (2)$$

where P_D is the Power obtained by the propeller [W], $\dot{m}$ is the mass flow rate [kg/s], V_C is the outlet velocity [m/s^2] and V_A is the inlet velocity [m/s^2].

Yet, the delivered power PD is also equal to the work done by thrust force:

$$P_D = TV_B \quad (3)$$

where P_D is the Power obtained by the propeller [W], T is the trust of the propeller [N] and V_B is the velocity against the thrust of the propeller [m/s^2]

By solving the equation (2) and (3), the velocity at the propeller position turn:

$$V_B = \frac{1}{2}(V_C + V_A) \quad (4)$$

where V_B is the velocity against the thrust of the propeller [m/s^2], V_C is the outlet velocity [m/s^2] and V_A is the inlet velocity [m/s^2].

The hydrodynamic propeller operation can be modelled by the following non-dimensional equations for Advance Ratio (J), Trust Coefficient (C_T) and Torque Cofficient (C_Q) respectively :

$$J = \frac{V_A}{nD} \quad (5)$$

where V_A is the inlet velocity, n is the revolution of the propeller blade (rad/s), D is the diameter of the propeller blade [m]

$$C_T = \frac{T}{\frac{1}{2}\rho {V_A}^2 A_o} \quad (6)$$

where T is the trust of the propeller [N], D is the diameter of the propeller blade [m], ρ is the density of the water [997 kg/m^3] , V_A is the inlet velocity and A_o is the surface area of the propeller blade [m^2]

$$C_Q = \frac{Q}{\rho n^2 D^5} \tag{7}$$

where Q is the torque produced from the propeller blade , ρ is the density of the water [997 kg/m^3], n is the revolution of the propeller blade (rad/s) and D is the diameter of the propeller blade [m].

The propeller thrust is made non-dimensional with the propeller area and the inflow velocity, V_A. Then, the efficiency of the propeller can be written as:

$$\eta_o = \eta_i = \frac{2}{1+\sqrt{1+C_T}} \tag{8}$$

where η_o and η_i is the propeller blade efficiency and C_T is the thrust coefficient.

2.6 Propeller Performance

Xueming et al. (2015) state that propeller efficiency is used to define how well a propeller transmits its rotational force or energy into thrust. The amount of energy it takes to rotate the propeller is almost always greater than the thrust from the propeller.

Oladokun, (2015) is conducted a study on the performance and the efficiency of the propeller where it was determined using JavaProp. Table 2.6.1 and 2.6.2 below show propeller geometry specification and computational estimation of thrust and torque respectively.

Table 2.6.1: Propeller geometry specification. (Ramakrishna et al. 2012).

Propeller diameter (m)	1.560
Hub diameter (m)	0.210
Number of Blades	4
Blade Area Ratio	0.70
Propeller (RPM)	446.650

Table 2.6.2: Computational estimation of Thrust and Torque. (Ramakrishna et al. 2012).

Rotational speed n (rps)	Velocity of Advance (m/s)	Advance Coefficient,J	Thrust force, T (N)	Torque ,Q (N-m)
25	6.22	1.092511	57.9089	5.7258
30	6.22	0.910426	322.7728	16.3267
40	6.22	0.682819	1075.962	46.5109
50	6.22	0.546256	2102.149	87.0225

Figure 2.6.3 below shows graph plotted for propeller efficiency against advance ratio while Figure 2.6.4 shows graph plotted of propeller efficiency against propeller diameter with various rpm. Regarding to Figure 2.63, the highest efficiency that they can be achieved is about 0.91 or 91% while in Figure 2.6.4 the highest value efficiency is about 81 % at 900 rpm.

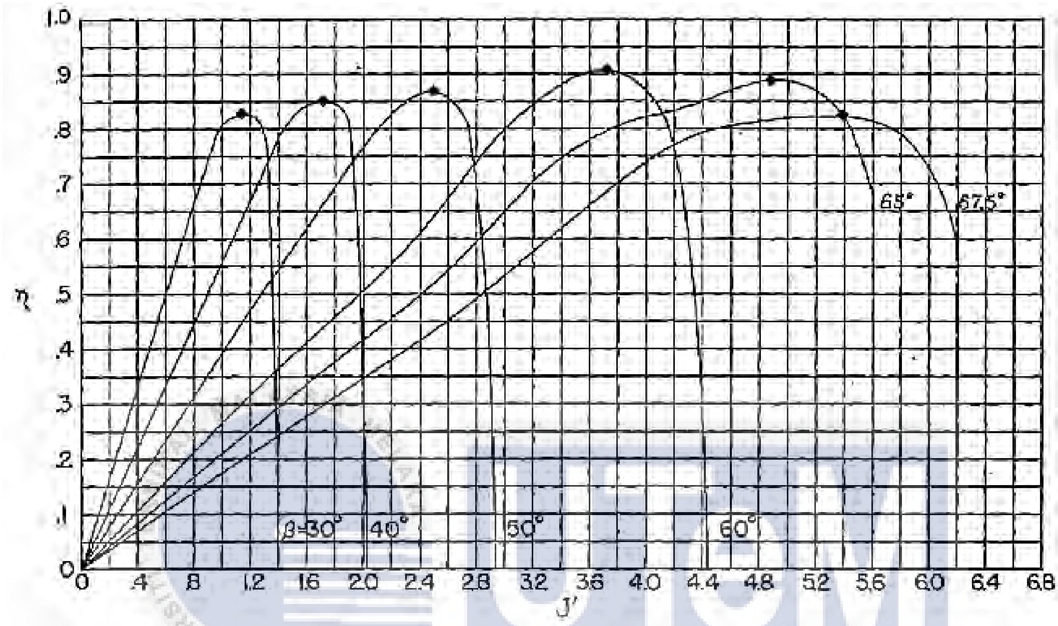


Figure 2.6.3: Propeller with four blade efficiency map. (Gur, 2016)

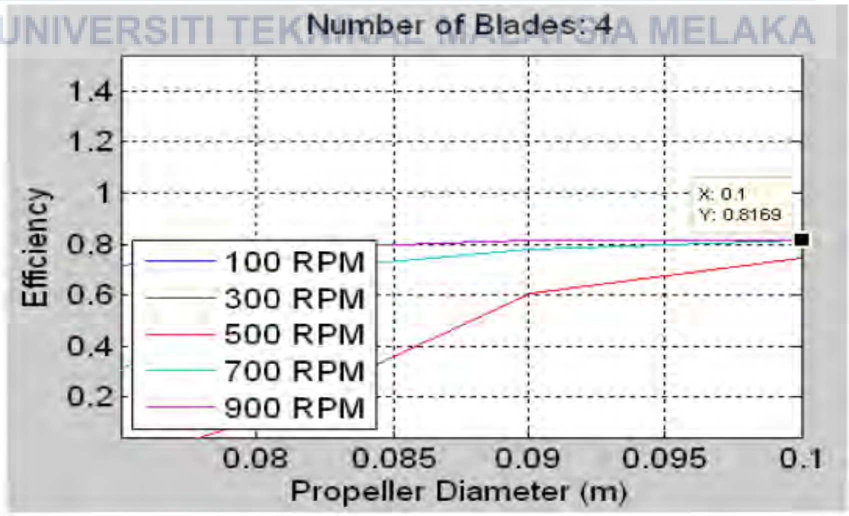


Figure 2.6.4: Plotted graph efficiency of propeller against propeller diameter with various rpm. (Husaini et al. 2014)

CHAPTER 3

METHODOLOGY

3.1 Steps of Methodology

This subtopic will cover the details explanation of methodology that is being used to make this project complete and run smoothly. To accomplish the best result, the need to construct the methodology is important to know the flow of the project. Moreover, this method helps the researcher to achieve the objectives or goals. Figure 3.1 below shows the steps of the methodology for this final project.

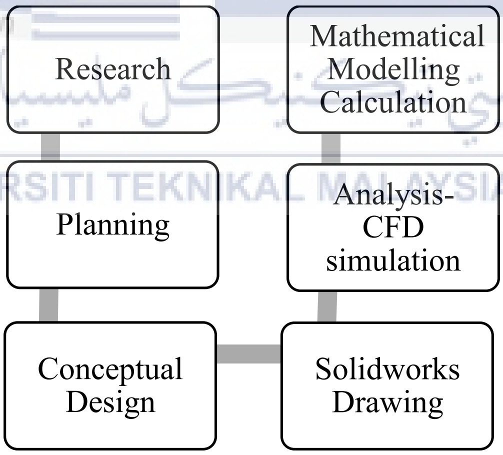


Figure 3.1: Step of Methodology

Regarding the Figure 3.1 above, the first step is by doing a research. The research is done by doing a study on the existing journal about the ROV, underwater thruster and propeller blade. Based on the research, there is a lot of knowledge that can be obtained. Thus, from the gathered information, there are some criteria that can be considered when designing and making an analysis of the propeller blade.

Then, the gained knowledge can be applied when to make the planning. The important data collected from the journals, research paper and from the internet as well. While planning, the research that related to the project is done, which is including the parameter analysis that can be considered when designing the propeller blade. If the propeller blade fulfils the best criteria, the underwater vehicle itself would able to run at optimum efficiency.

After planning, the conceptual design can be obtained based on the morphological chart. The design makes is done by considered the all the knowledge and data collection from the literature study. Therefore, from the conceptual design, the best design can be made.

Then, from the design selected, the drawing and the dimension of the propeller blade is done in Solidworks. The drawing is important as we can know perfectly the dimension and we can avoid the problem before proceeding to CFD analysis and mathematical modelling calculation.

The next process is to analyse the geometry in Computational Fluid Dynamics (CFD) simulation. The software that used to simulate the CFD are Solidworks and ANSYS Fluent. In CFD, the geometric modelling is carried out using 3D modelling software starting from one blade to build the entire propeller. The boundary condition of the propeller blade also can be known. In addition, this software also estimated the thrust and velocity for the different rotational speeds (rps) of the propeller blade.

3.2 Design Input Flow Chart

The main part of this project is to design the underwater propeller blade. Therefore, the methodology throughout this part is a very important which act as an outlines to solve the problem besides the plan and analysis the data collected. After making a research through the journal, there are a lot of outcomes and knowledge obtained. The most critical thing is the parameter of the propeller blade before proceed to the propeller blade design. Figure 3.1 below shows the methodology flow of the propeller blade design process

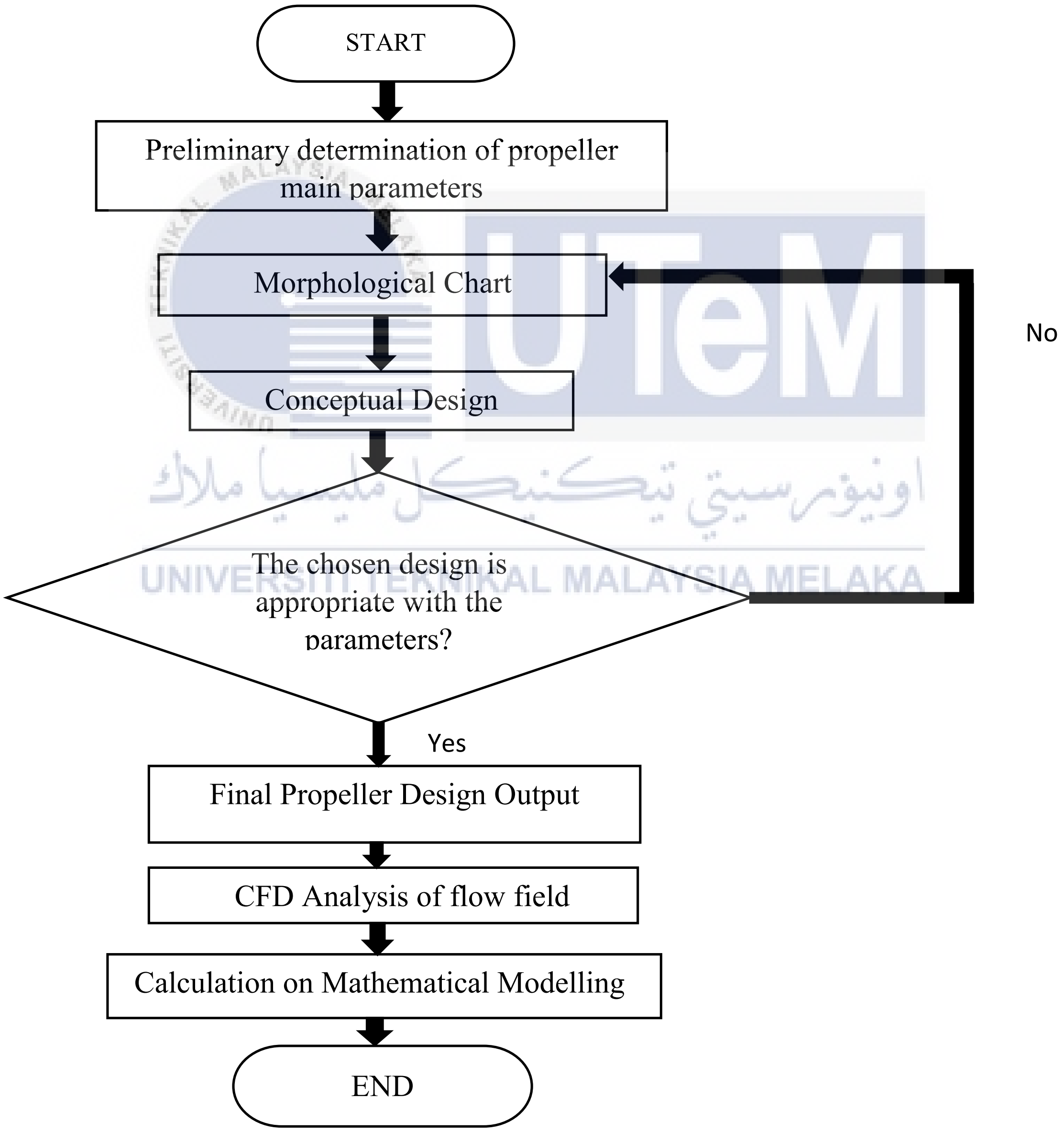


Figure 3.2: Methodology flow of blade design.

3.2.1 Preliminary Determination of Propeller Main Parameter

The study of the propeller blade parameter is conducted through the research study through journal and web surfing. The main reason why the parameter of the propeller blade is important is the propellers have to be designed in a way to decrease noise and vibrations besides set the cavitation to the lowest possible level in order to achieve the propeller efficiency. In addition, the consideration parameter is also important in order to know the characteristics of the propeller to increase the propeller blade efficiency. Therefore, the modifications made in the basic propeller geometries would able to determine and analyse propeller performance.

3.2.2 Morphological Chart

The morphological chart is a method to generate ideas in an analytical and systematic manner or a visual way to capture the necessary product functionality. Thus, after determined the parameter of the propeller blade, the next step is to make a morphological chart. The purpose of making the morphological chart is to finalize the parameter design in order to provide the structured approach to concept generation to find a solutions to a design problem.

3.2.3 Conceptual Design

The conceptual design is the one of the critical design process where in this step, the knowledge to choose the best design is very important. The knowledge that had gained through research study is very important to select the best design. If the true design based on the all requirement needed is made, the problem of the project have the best solution. However, if the wrong design selection is made, the problem of this project cannot be solved. In advance, the step of the methodology need to start again from the morphological chart analysis and continue until the best design that followed the chosen parameter is find out.

3.2.4 Final Propeller Design Output

The existed design underwater thruster of T200 blue robotics is used as a referred design to make a new propeller design. Figure 3.2.4 below shows the specification and dimension of T200 thruster. The propeller diameter is 76mm or 3.0 inch.

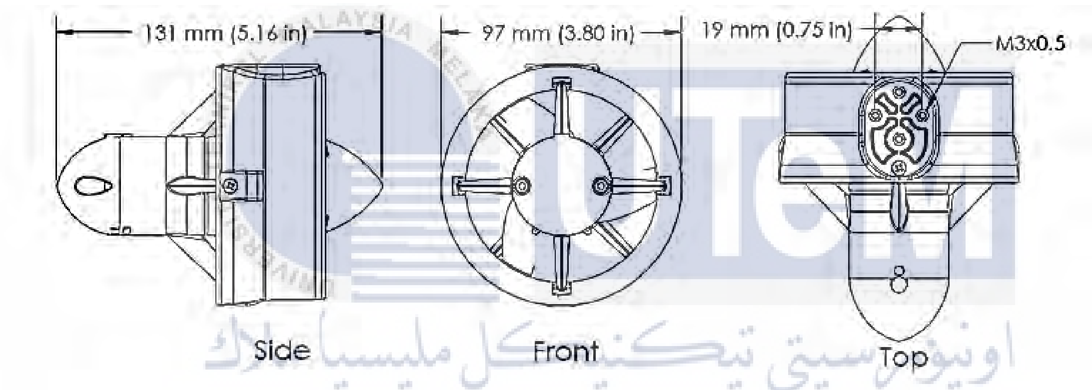


Figure 3.2.4: Specification dimension for T200 thruster (Blue Robotics, 2018)

After the design selected in conceptual design, the final propeller design output can be proceed. In this step, the correct dimension and the drawing of the final design is applied in the engineering drawing software which is Solidworks.

3.2.5 CFD Analysis of Flow Field

The final design of the propeller blade then is analyse through Computational Fluid Dynamics (CFD) fluid flow capabilities to obtain greater insight into product performance that is critical to design success. In fact, CFD would able to illustrate the water flow around the propeller blade and the force acting on the model as well.

Then, the CFD simulation reliability depends on many aspects, including computational domain sizing, mesh size and quality as well chosen solution type (steady state or transient) and physical formulation (type of equation solved, additional wall formulations and corrections).

After the geometry is solved in the meshing part, the next step in ANSYS Fluent is to setup and solve the solver using the computational while the last step is to execute the data and the result is obtained. Before the result is executed, the boundary condition was set. The selected boundary condition is shown in results and discussion chapter. The focus in this study is to find the velocity and the total thrust that made by the propeller blade in order to fulfil the requirement in mathematical modelling calculation later on.

3.2.6 Calculation on Mathematical Modelling

The objective of this study is to determine the efficiency of the propeller blade. Regarding to the mathematical formulation in literature review subtopic, the thrust coefficient need to define in order to determine the efficiency of the propeller. In this step, the using of Microsoft Excel is the best way to compute, analyse and obtain the data and results. In addition, by using this software, the graph can be easily plotted.

3.3 Work Breakdown Structure

A work breakdown structure is a chart or organize which examine the work elements, such as tasks, of a project are present to show their correlation to each other throughout the whole project. This work breakdown structure helps to organize and interpret the entire work scope of the project. In fact, the benefits of a work breakdown structure is to decrease complex activities to a collection of tasks. This is necessary for the project manager because the tasks can be supervised more effectively than the complicated activities. Figure 3.3 below shows the work breakdown structure for propeller blade design.

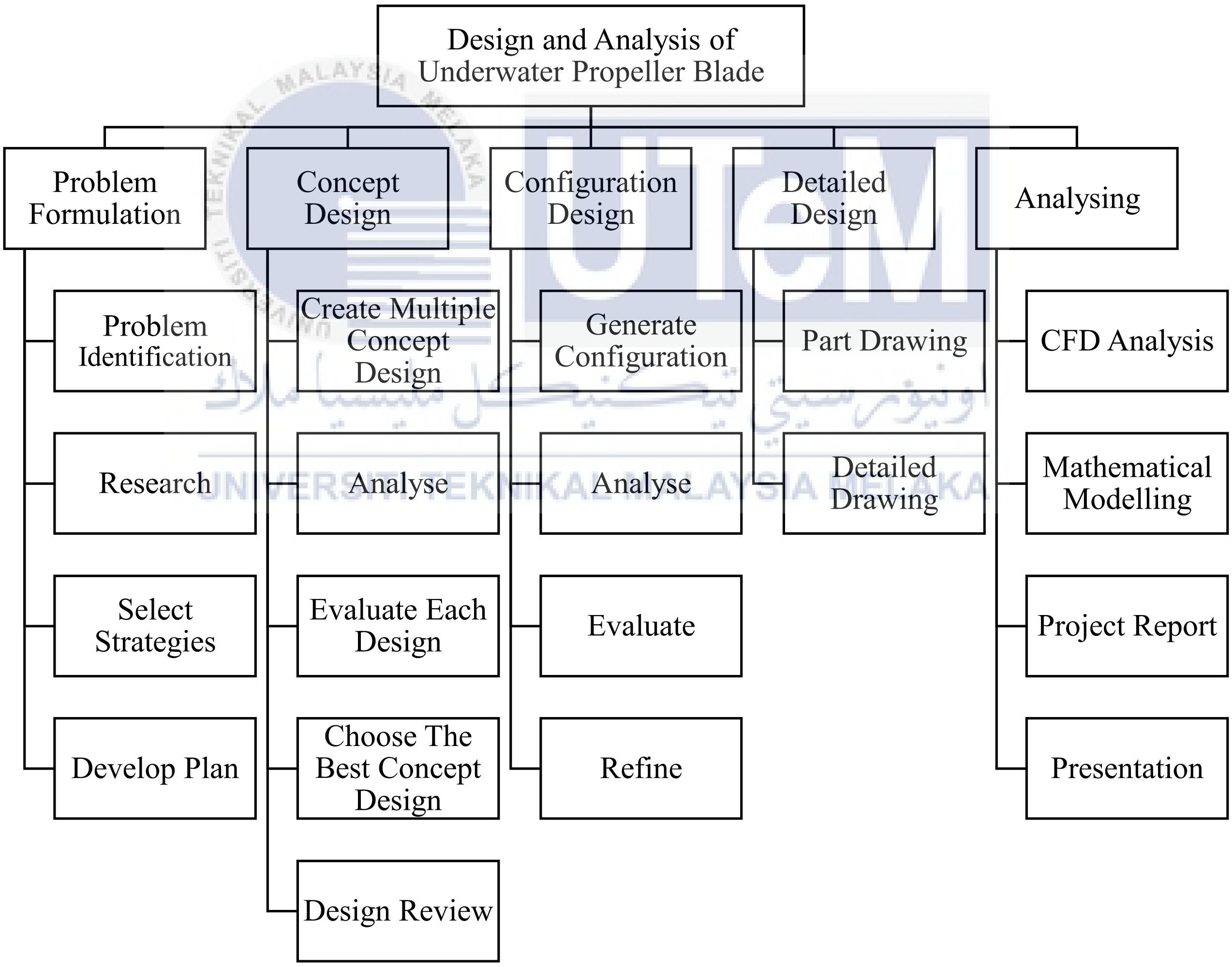


Figure 3.3: Work Break Structure of the project.

3.4 Gantt Chart of the Project.

The Gantt chart below shows the amount of work done or production completed in certain periods of time of this final project. Figure 3.4 and figure 3.5 below shows the Gantt charts of final year project 1 and final year project 2 respectively.

No	Task	Week														
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Problem Statement & Objectives															
2	Research & literature studies															
3	Morphological Chart															
4	Conceptual Design															
5	Concept Evaluation															
6	3D design															
7	Part Design															
8	CFD Analysis															
9	Report Submission															
10	Presentation															

Figure 3.4: Gant Chart of the Final Project 1

No	Task	Week														
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Part Design															
2	CFD Analysis															
3	Data Collection & Analysing															
4	Mathematical Modelling Calculation															
5	Report and Slide Presentation Making															
6	Report Submission															
7	Presentation															

Figure 3.5: Gant chart of final year project 2.

Regarding the Figure 3.4, for the first and second weeks, the problem and the objectives of the project will be determined based on the project title given. Next, the research and literature review is done throughout three weeks after that. Week 5 is a middle semester break.

After getting some knowledge on the research about the propeller design and some analysis, the morphological chart will be done based on the best criteria. The conceptual design is done after that which is in week 8 and 9. Then, 3D design of the propeller blade on the Solidworks is completed around week 10 until week 11.

Next, after making the geometry drawing, the learning process and trial and error on CFD analysis is on week 13 and 14. The report submission for final year project 1 is on week 14. Lastly, the presentation 1 is on week 15.

Based on figure 3.5 above, the first task during Final Year Project 2 is by doing the part design on the first and second week. Some mistake occurs when making the design in Final Year Project 1. The old parameter is not suitable to be fixed into the existing T200 thruster. Therefore, the new design is made with a parameter that suitable with T200 thruster.

CFD analysis in Final Year Project 1 is not valid. Therefore, CFD analysis is continue starting on week 2 until week 4 in Final Year Project 2. The duration of making this CFD analysis is quite long because of the learning process and to make sure the result obtain is accurate. The data collection and analysing is done throughout week 6 and 7.

The mathematical modelling calculation is done on week 10. The mathematical modelling calculation is based on the result obtain from the CFD simulation. The report is submitted on week 14 and the presentation is done on week 15.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Conceptual Design

This chapter presents the results of the work based on the literature review study in Chapter 2 and the step taken in Chapter 3. The evaluation in this Chapter is very important in order to know the validation results based on previous study.

4.1.1 Morphological Chart

Parts	Option 1	Option 2	Option 3
Number of Blades	 Two blades	 Three blades	 Four blades

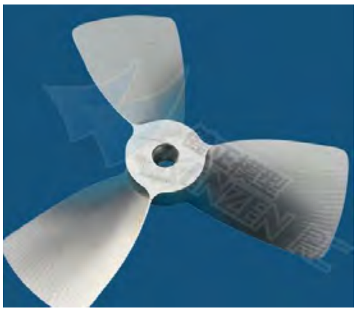
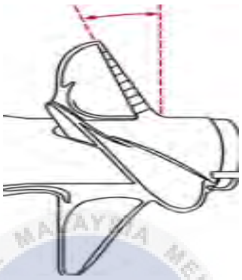
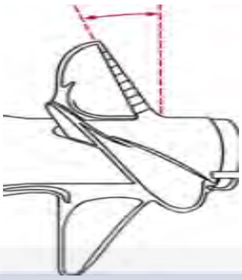
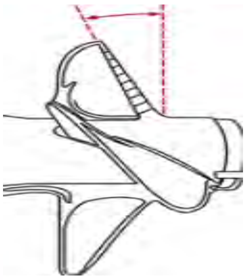
Blade design			
Rake angle	 5 degree rake angle	 0 degree rake angle	 15 degree rake angle
Ducted propeller			None
Propeller Diameter	Fixed to 76 mm	Fixed to 76 mm	Fixed to 76 mm

Figure 4.1.1: Morphological Chart of propeller design

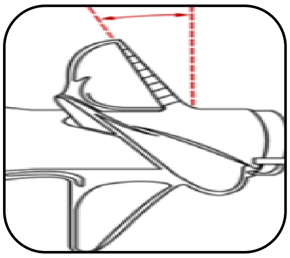
Figure 4.1.1 above shows the morphological chart of the propeller design. The characteristics or the parts is chosen based on the research study. The option for the parts contain a different option as the generated idea to make a better propeller design

4.1.2 Concept Generation

4.1.2.1 Concept 1

Number of Blades: 2

Blade Design



5 degree of rake angle

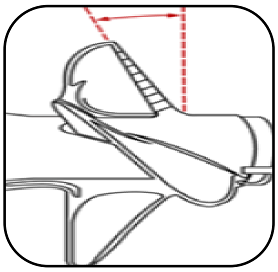
Unducted Propeller

Figure 4.1.2(a): The first conceptual design

4.1.2.2 Concept 2

Number of Blades: 3

Blade Design



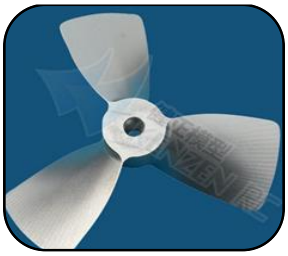
0 degree of rake angle

Ducted Propeller

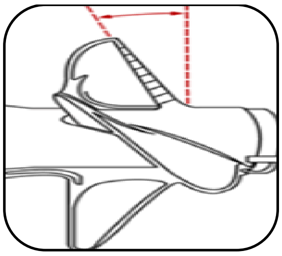
Figure 4.1.2(b): The second conceptual design

4.1.2.3 Concept 3

Number of Blades: 4



Blade Design



15 degree of rake angle

Ducted Propeller

Figure 4.1.2(c): The third conceptual design

The concept generation of disable car seat product as shown in Figure 4.1.2(a), figure 4.1.2 (b) and Figure 4.1.2(c). The design three type of concept generation to expand the idea of the real product. Based on this three concept, the used of different option regarding to morphological chart in Figure 4.1.1 of the parts involve to form the function product.

4.1.3 Concept Evaluation

The concept evaluation of this propeller blade is due to some factors based on finding research. However, the important factors that affect efficiency of the propeller are the number of blades, propeller diameter, rake angle, blade design or blade area and ducted propeller. Figure 4.1.3 below shows the mind maps of important design factors that affecting the propeller efficiency.

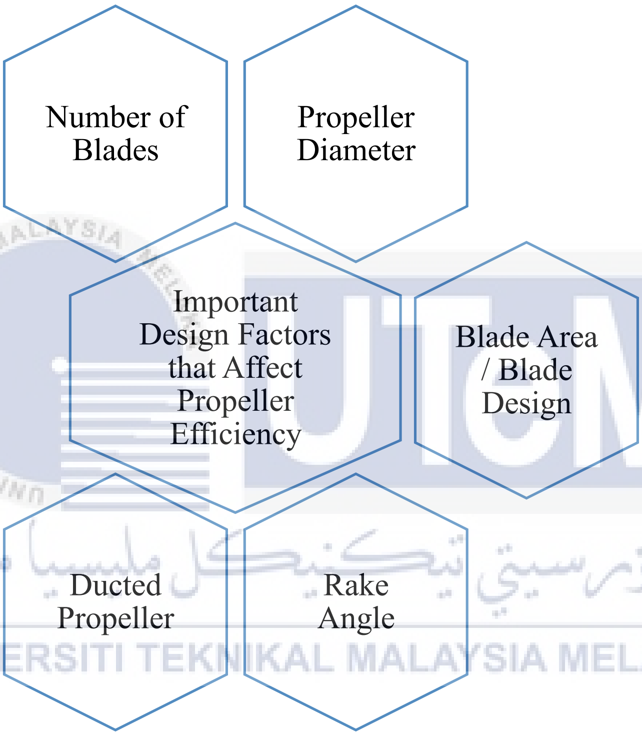


Figure 4.1.3: The mind maps of important design factors that affecting the propeller efficiency.

The optimum open water efficiency is increasing with increasing of number of blades. Besides that, the increasing of propeller diameter, the more efficiency of the propeller blade. However, the optimization of the propeller design is done to meet the balance in order to reduce the vibration and noise. Besides that, the efficiency of the propeller blade also depends on the blade are. The more reduced the blade's area, the more efficient the propeller becomes. This is because the frictional drag will be increase as the blade area increase. The blade area is calculated based on its blade design. Therefore, the blade design also affected the propeller diameter. Next, the propeller that has the high or medium rake angle has a better efficiency compared to low rake angle. The ducted propeller also one of the factor that has to be considered n efficiency of the propeller blade.

The weighted decision matrix method is irrelevant to be used as there is no survey making. The concept alternatives and evaluation is based on the research study on the literature review and web surfing. Therefore, the different concept alternatives and evaluation is used. The evaluation scheme for the design is shown in Table 4.1.3.1 below and the concept alternatives is shown in table 4.1.3.2 below.

Table 4.1.3.1: Evaluation scheme for the design

Rating (R)	Value (V)
Unsatisfactory	0
Weak	1
Satisfactory	2
Good	3
Excellent	4

Table 4.1.3.2: Concept Alternatives of the design.

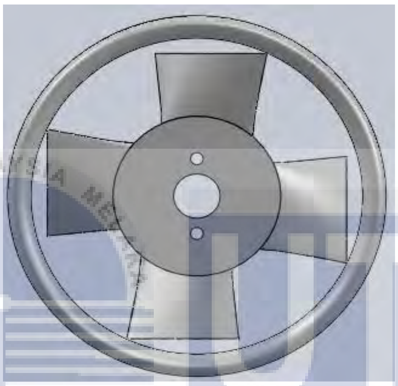
	Concept Alternatives		
Design Criteria	Concept 1	Concept 2	Concept 3
Number of blades	2	2	3
Blade Design/ Blade Area	2	1	2
Rake angle	1	2	3
Ducted Propeller	2	3	3
Total	7	8	11

Regarding concept alternatives shows in Table 4.7 above, Concept 3 is chosen where Concept 1 and Concept 2 have same rating below Concept 3. Concept 3 fulfil the requirement of the number of blades and rake angle of the propeller. Besides that, for the others design criteria, the rating is in the medium rating.

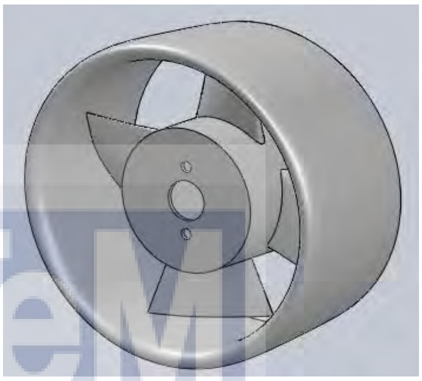
4.2 Geometry Design

(a) Top view

(b) Side view



(c) Front view



(d) Isometric view

Figure 4.2.1

Figure 4.2.1 (a), 4.2.1(b), 4.2.1(c) and 4.2.1(d) above show the top, side, front and isometric view of final propeller blade design in Solidworks drawing respectively. This final propeller design is choose regarding to literature review study, morphological chart analysis, conceptual design selection and the concept evaluation. The final propeller blade design has 4 blade design with 16.5 degree of rake angle (slightly different from concept 3's rake angle), 76mm of propeller diameter and has ducted around the propeller.

4.3 Computational Fluid Dynamics Analysis

4.3.1 Geometry Design with Propeller domain

Two types of software is used to analyse CFD of the propeller blade which are Solidworks and ANSYS Workbench student version. The reason of making CFD analysis using this two software is to compare the results obtained. However, ANSYS workbench student version are not able to execute the final result since the meshing element for the propeller blade is more than 900,000 elements. ANSYS Workbench student version cannot execute more than 512,000 elements. Therefore, the using of ANSYS workbench only can analyse until meshing step only.

The geometry design is the first step before proceed to other steps for CFD analysis purpose. The second step is meshing step. Before the start the meshing step. The geometry of the propeller blade design must have its propeller domain. Figure 4.3.1(a) and figure 4.3.1(b) below show the details view for both first and second propeller domain.

Details View

Details of Enclosure1	
Enclosure	Enclosure1
Shape	Cylinder
Cylinder Alignment	Z-Axis
Number of Planes	0
Cushion	Non-Uniform
FD1, Cushion Radius (>0)	0.015 m
FD2, Cushion (>0), +ive Direction	0.008 m
FD3, Cushion (>0), -ive Direction	0.008 m
Target Bodies	All Bodies

(a)

Details View

Details of Enclosure2	
Enclosure	Enclosure2
Shape	Cylinder
Cylinder Alignment	Z-Axis
Number of Planes	0
Cushion	Non-Uniform
FD1, Cushion Radius (>0)	0.19 m
FD2, Cushion (>0), +ive Direction	0.35 m
FD3, Cushion (>0), -ive Direction	0.35 m
Target Bodies	All Bodies

(b)

Figure 4.3.1: (a) The details view of the enclosure 1 (first propeller domain), and (b) the details view of the enclosure 2 (second propeller domain)

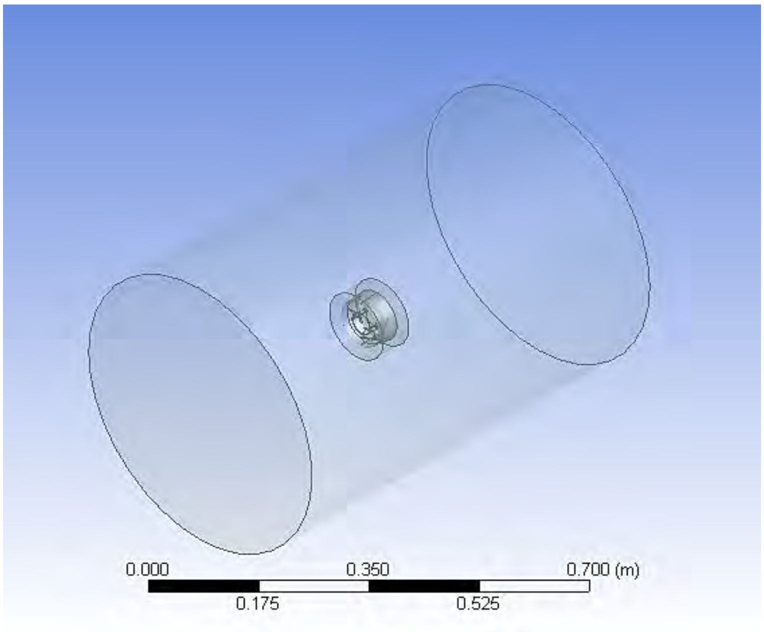
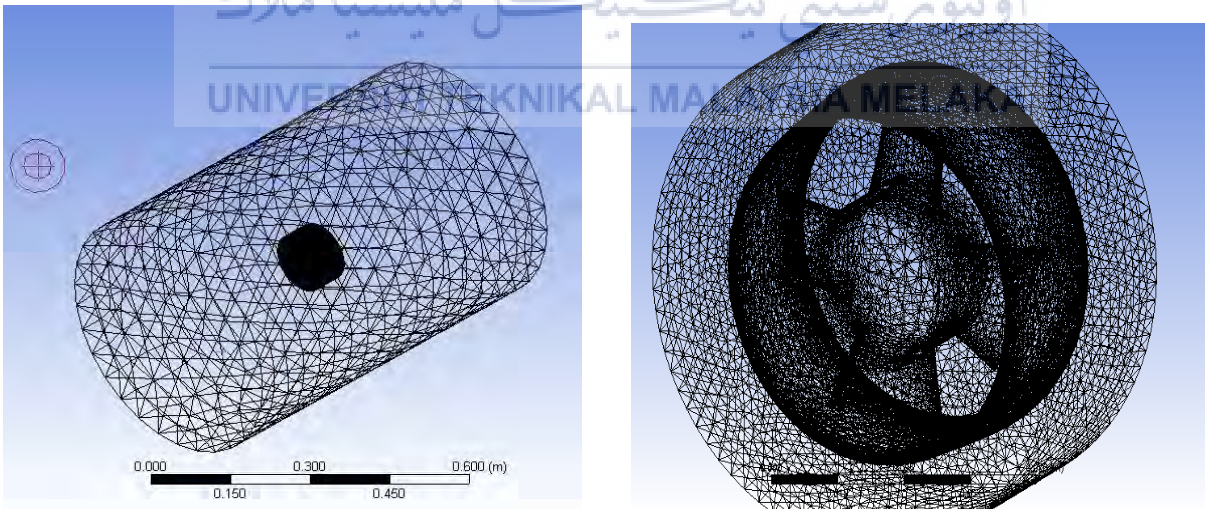


Figure 4.3.1(c): The view of the propeller blade with enclosures.

4.3.2 Grid Generation or Meshing

The mesh tool in ANSYS Fluent is used to form the grid generation. The grid is vital as a result of it provides the individual illustration of geometry interest.



(a) (b)

Figure 4.3.2(a): Meshing of propeller blade with both first and second propeller domains, and (b) Meshing of propeller blade with first propeller domain.

Details of "Mesh"	
Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Min Size	Default (5.1667e-004 m)
Max Face Size	Default (5.1667e-002 m)
Max Size	Default (0.103330 m)

Details of "Mesh"	
Assembly Meshing	
Method	None
Patch Conforming Options	
Patch Independent Options	
Advanced	
Defeaturing	
Statistics	
Nodes	176243
Elements	941082
Mesh Metric	None

Figure 4.3.2: The details view meshing

Based on figure 4.3.2, it shows the meshing of the propeller blade with both first and second propeller domains. From that figure it obviously seen that the second domain meshing size is quite bigger than the first propeller domain. The meshing sizes near the blade wall and the outer boundary is structured. The relevance center that had been used is coarse type. The grid is fully structured tetrahedral in both stationary and rotating domains. Regarding to figure 4.3.2, the statistics value for modes and elements are 176,243 and 941,082. It shows that the value of elements is bigger. This is because the default setting for min size, max face size and max size is quite small.

4.3.3 Setup and Solver

Setup and solution is made using Solidworks software. Given the specification for the motor rpm is around 300 rev/min – 3800 rev/min or 31.42 rad/s – 397.94 rad/s. The rotating region was set into three rpm value which are low rpm value, medium rpm value and High rpm value in order to make the relevant analysis. Table 4.3.3 below shows the exact value for low, medium and high RPM of the rotating region.

Table 4.3.3: The Value of Various rpm of the Rotating Region

Low Rpm Value	31. 42 rad/s
Medium Rpm Value	183.25 rad/s
High Rpm Value	397.94 rad/s

4.3.4 Results

Figure 4.3.4.1 below shows the solver result for the low rpm value (31.42 rad/s) while table 4.3.4.1 shows the important data that obtained from the figure 4.3.4.2.

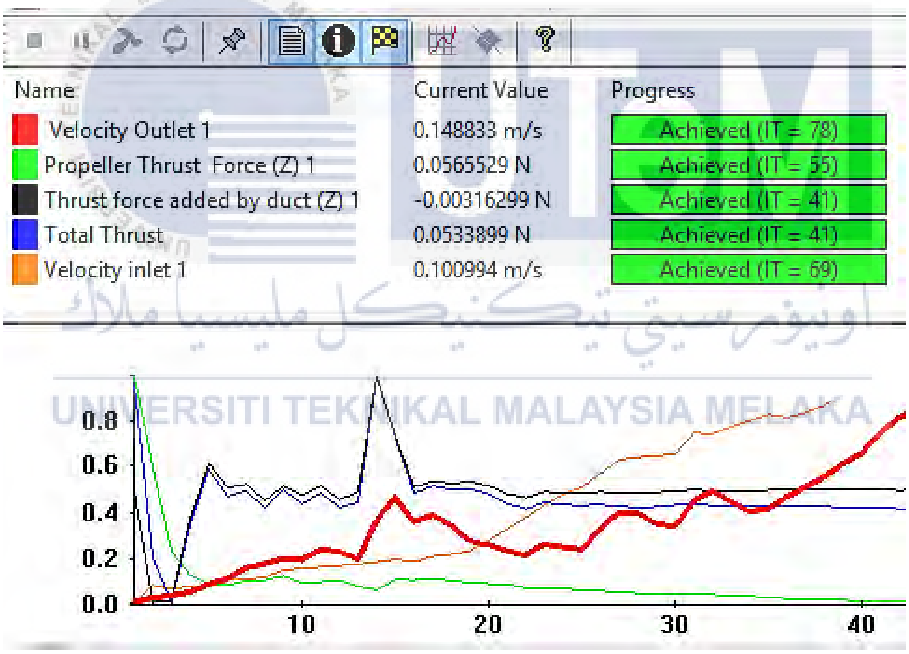


Figure 4.3.4.1: Solver Result for 31.42 rad/s of propeller blade in CFD.

Table 4.3.4.1: Solver Result Data for Low Rpm Value

Propeller Thrust Force [N]	0.0565529
Thrust Force Added by Duct [N]	-0.00316299
Total Thrust [N]	0.0533899
Inlet Velocity [m/s]	0.100994
Outlet Velocity [m/s]	0.148833

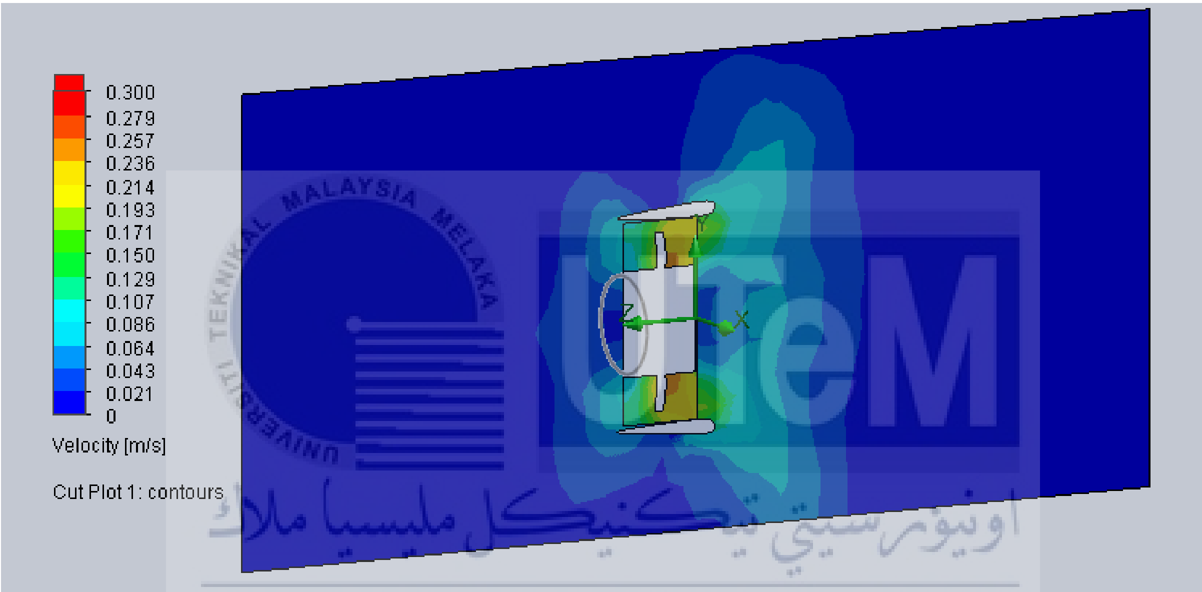


Figure 4.3.4.2: Velocity Distribution for Low Rpm Value of Propeller Blade in CFD

Based on the Figure 4.3.4.2 above, it shows the velocity distribution of the propeller blade in the propeller domain for 31.42 rad/s of rotating region. The maximum velocity that can be achieved from this rotating region is about 0.2 m/s which is at the corner of the rotating blade. The outlet velocity of the propeller blade is obviously seen higher than the inlet velocity.

Figure 4.3.4.3 below shows the solver result for the medium rpm value (183.25 rad/s) while table 4.3.4.3 shows the important data that obtained from the Figure 4.3.4.4.

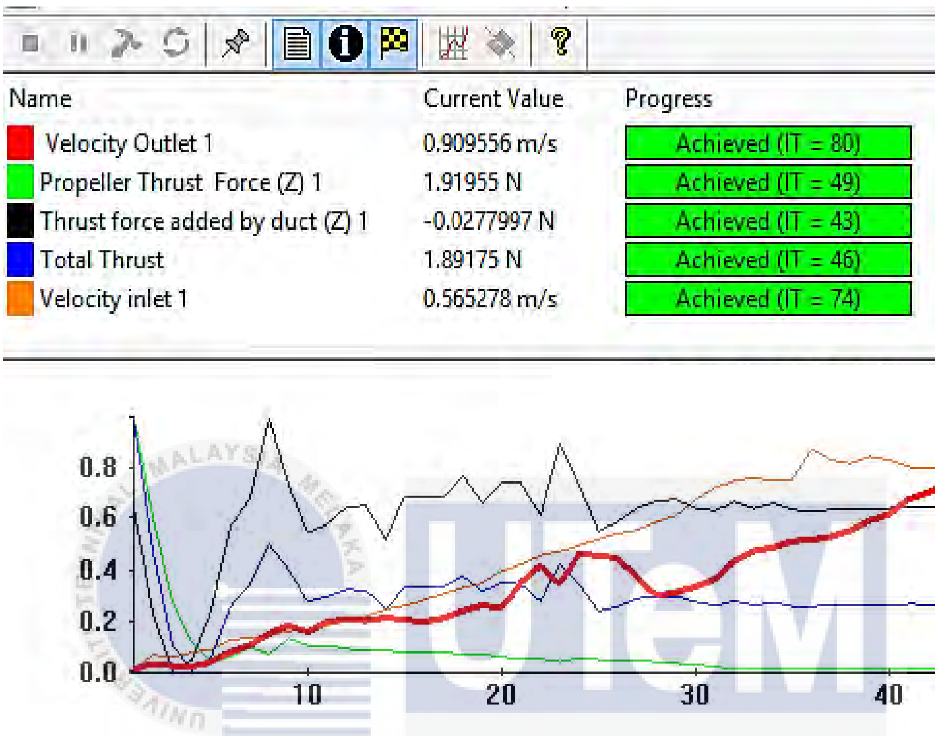


Figure 4.3.4.3: Solver Result for 183.25 rad/s of Propeller Blade in CFD.

Table 4.3.4.3: Solver Result Data for Medium Rpm Value

Propeller Thrust Force [N]	1.91955
Thrust Force Added by Duct [N]	-0.0277997
Total Thrust [N]	1.89175
Inlet Velocity [m/s]	0.565278
Outlet Velocity [m/s]	0.909556

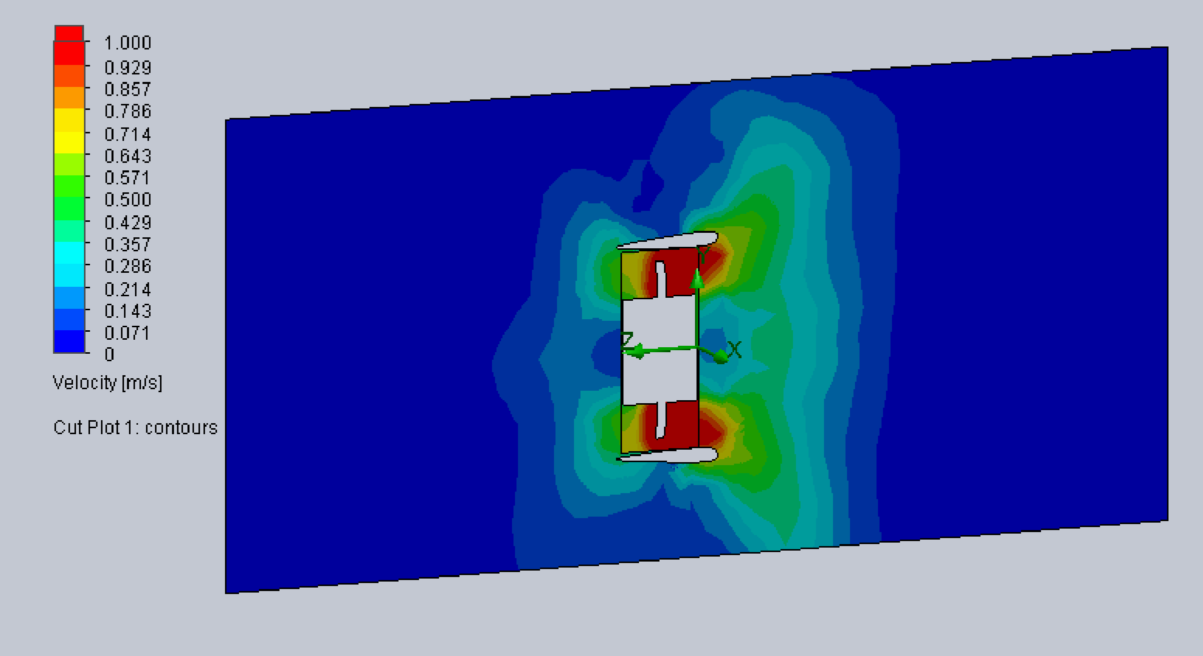


Figure 4.3.4.4: Velocity Distribution for Medium Rpm Value of Propeller Blade in CFD

Based on the figure 4.3.4.4 above, it shows the velocity distribution of the propeller blade in the propeller domain for 183.25 rad/s of rotating region. The maximum velocity that can be achieved from this rotating region is about 1 m/s. The red contour (high velocity) is seen accumulated at the rotating blade towards the outer part of the propeller blade. The velocity distribution is spread out away to the outer part of the propeller blade. The outlet velocity for 183.25 rad/s of rotating region almost approached 1 m/s of velocity.

Figure 4.3.4.5 below shows the solver result for the high rpm value (397.94 rad/s) while table 4.3.4.5 shows the important data that obtained from the figure 4.3.4.6

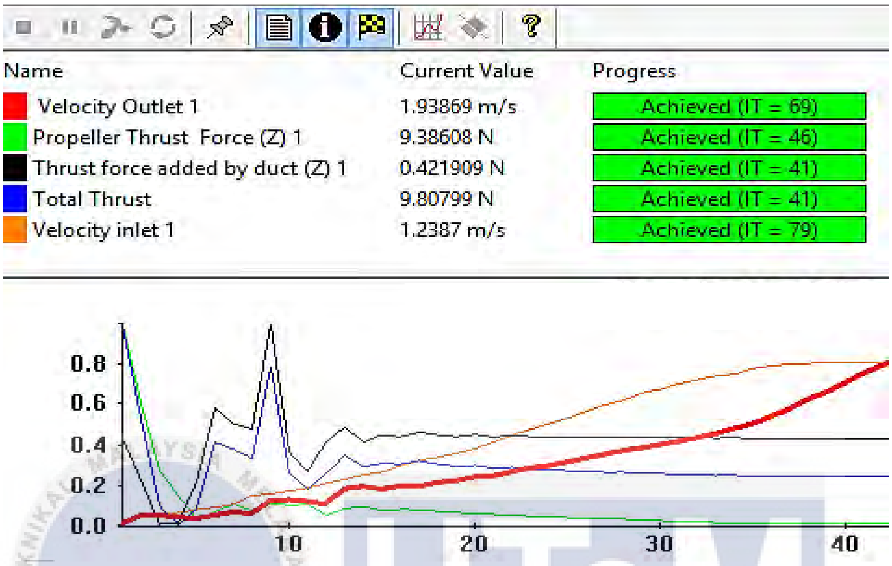


Figure 4.3.4.5: Solver Result for 397.94 rad/s of Propeller Blade in CFD.

Table 4.3.4.5: Solver Result Data for High RPM Value

Propeller Thrust Force [N]	9.38608
Thrust Force Added by Duct [N]	0.421909
Total Thrust [N]	9.80799
Inlet Velocity [m/s]	1.2387
Outlet Velocity [m/s]	1.93869

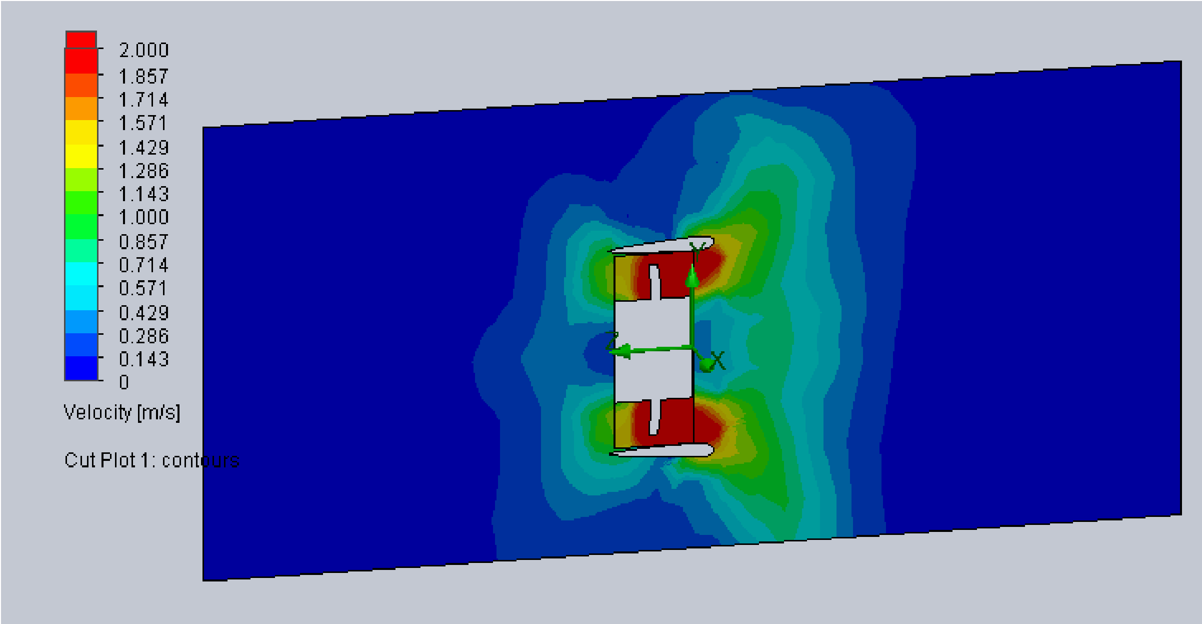


Figure 4.3.4.6: Velocity Distribution for High RPM Value of Propeller Blade in CFD

Based on the figure 4.3.4 (f) above, it shows the velocity distribution of the propeller blade in the propeller domain for 397.94 rad/s of rotating region. The maximum velocity that can be achieved from this rotating region is about 2 m/s. The bright red contour (high velocity) and yellow contour (quite high velocity) is seen accumulated at the rotating blade towards the outer part of the propeller blade. The velocity distribution in high RPM value is spread out away more than low RPM value and medium RPM value.

4.4 Mathematical Modelling Calculation

4.4.1 Tabulated Data

The result obtained from CFD analysis is then transferred into Microsoft Excel to form a tabulated data. In addition, the Microsoft Excel is a good platform to calculate an accurate data from the formula.

Table 4.4.1: Important data from CFD simulation analysis in different Rpm Value.

Speed (rad/s)	Velocity inlet , V_A (m/s)	Propeller Thrust Force (N)	Propeller Thrust by duct (N)	Total Thrust (N)	Thrust Coefficient, C_T	Propeller Efficiency	Efficiency propeller (%)
31.42	0.100994	0.0565529	-0.00316299	0.0534	0.290655	0.9363	93.63
183.25	0.565278	1.91955	-0.0277997	1.8918	0.328738	0.9291	92.91
397.94	1.2387	9.38608	0.421909	9.8080	0.354942	0.9242	92.42

Regarding to Table 4.4.1 above, the result for propeller efficiency is obtained using Microsoft Excel as the range rpm is around 300 – 3800 rev/min, the density of water is 997 kg/m^3 while the area of the propeller blade is 0.036018 m^2.

4.4.2 Example Calculation

RPM Range for the motor = 300 rev/min -3800 rev/min

= 31.42 rad/s – 397.94 rad/s

Density of the water, ρ = 997 kg/m^3

Cross sectional area of the propeller blade, A_o = 0.036018 m^2

At low RPM = 31.42 rad/s,

Result obtained from CFD,

Where V_A= 0.100994 m/s, Total thrust = 0.0534 N.

$$\eta_{propeller} = \frac{2}{1+\sqrt{1+C_T}} \qquad C_T = \frac{T}{0.5\rho {V_A}^2 A_O}$$

$$C_T = \frac{(0.0534\ N)}{0.5\left(997\frac{kg}{m^3}\right)\left(0.100994\frac{m}{s}\right)^2(0.036018)} = 0.290655$$

$$\eta_{propeller} = \frac{2}{1+\sqrt{1+(0.290655)}} = 0.9363 \text{ @ } 93.63\ \%$$

4.4.3 Graph Plotting

Figure 4.4.3(a), 4.4.3(b) and 4.4.3(c) below show the various graph for efficiency of the propeller against RPM, efficiency of the propeller against thrust coefficient, efficiency of the propeller against velocity inlet, V_A respectively and total thrust against velocity inlet, V_A respectively.

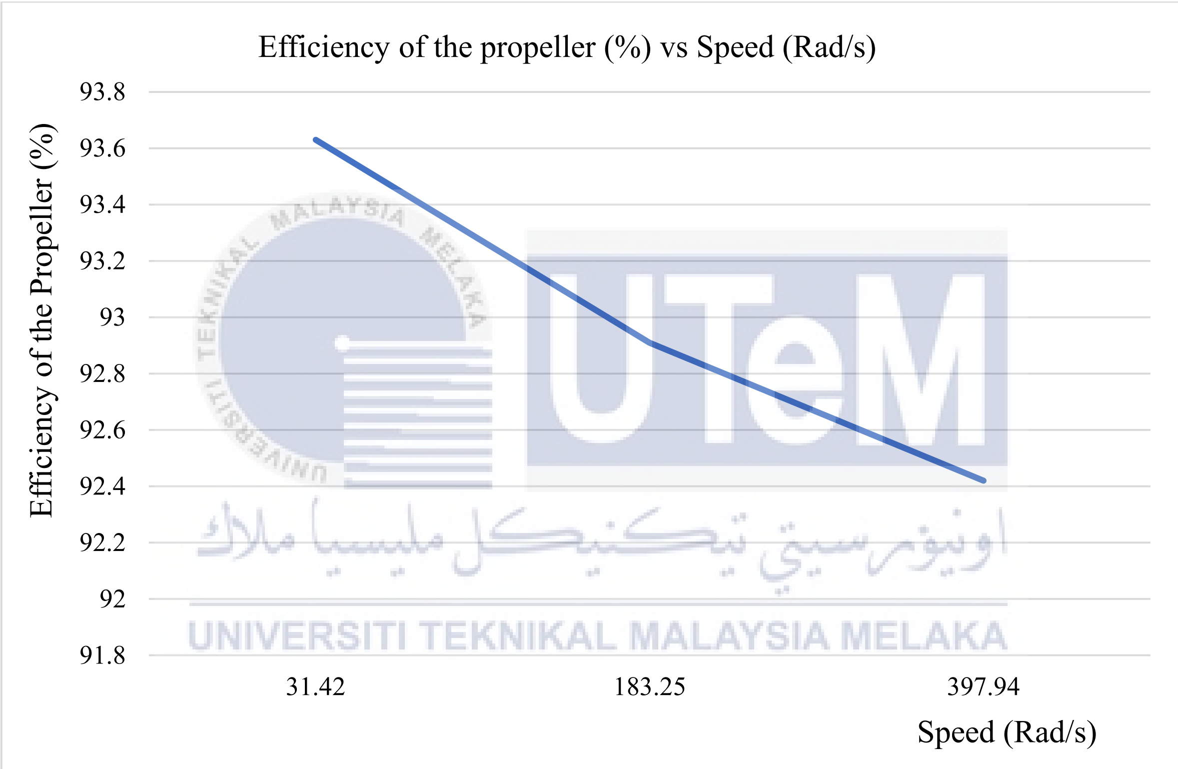


Figure 4.4.3(a): Graph of Efficiency of the Propeller (%) against Rpm (rad/s)

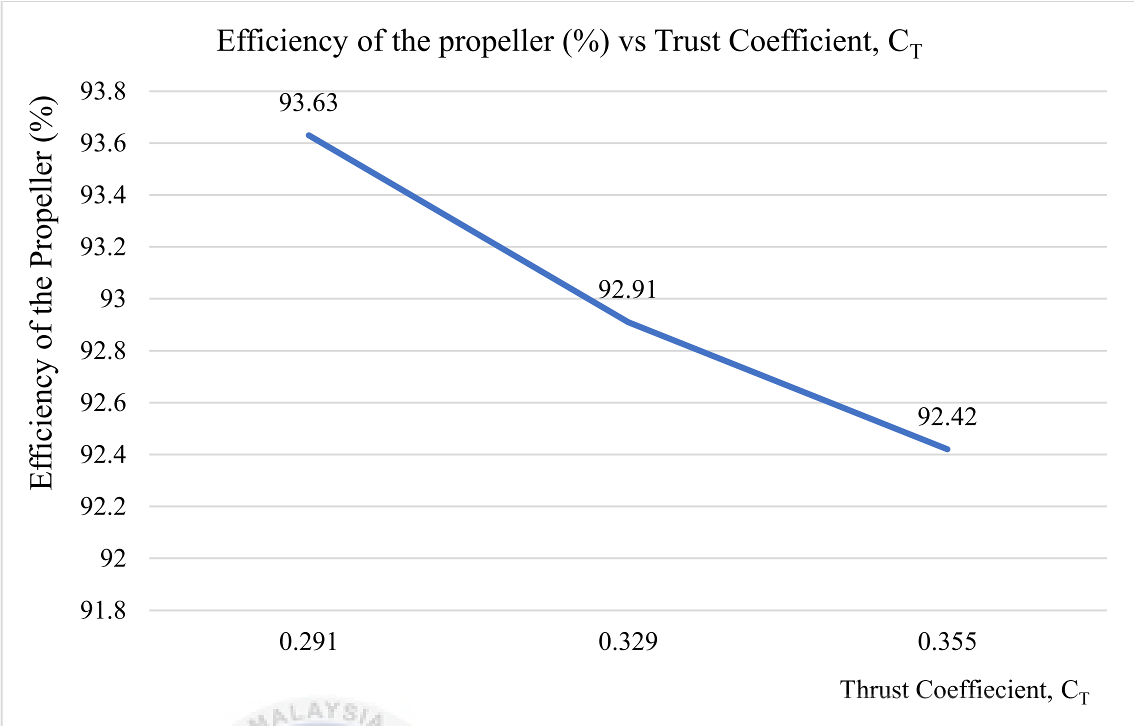


Figure 4.4.3(b): Graph of Efficiency of the Propeller (%) against CT (Thrust Coefficient)

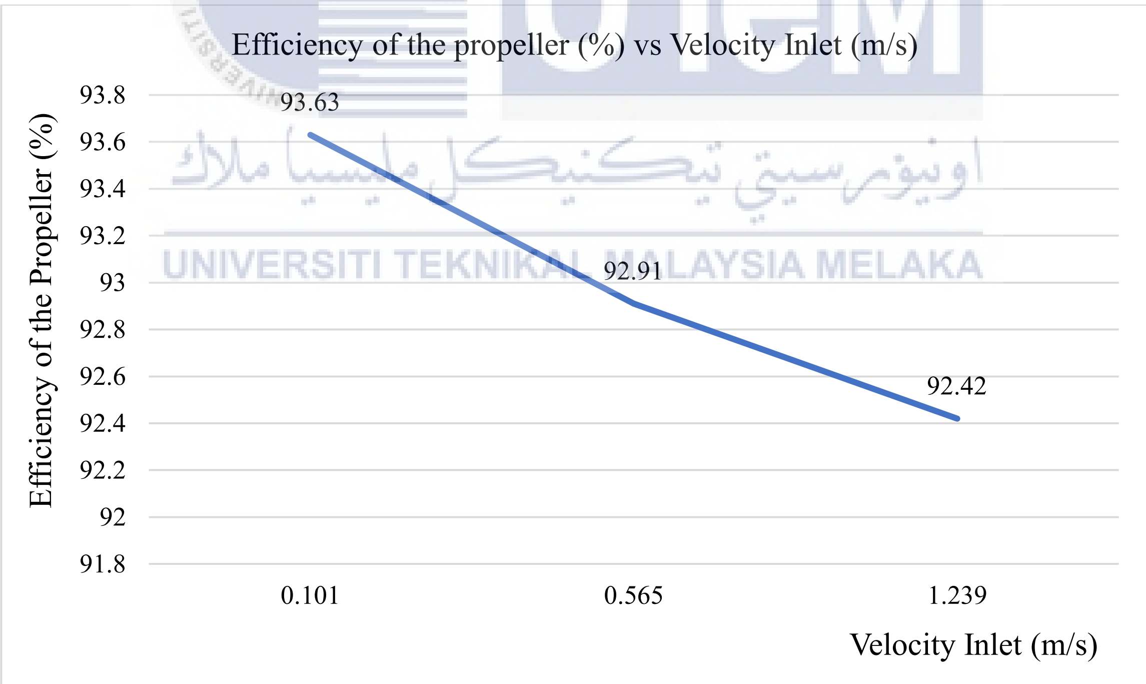


Figure 4.4.3(c): Graph of Efficiency of the Propeller (%) against Velocity Inlet

Figure 4.4.3(a), 4.4.3(b) and 4.4.3(c) show the same graph pattern. The efficiency of the propeller blade is inversely proportional to the RPM, thrust coefficient and velocity inlet. As the RPM, thrust coefficient and velocity increase, the efficiency is decrease and vice versa. Meanwhile figure 4.4.3(d) below shows graph of total thrust against velocity inlet. From this graph it can be seen that as velocity increase, the total thrust also increase.

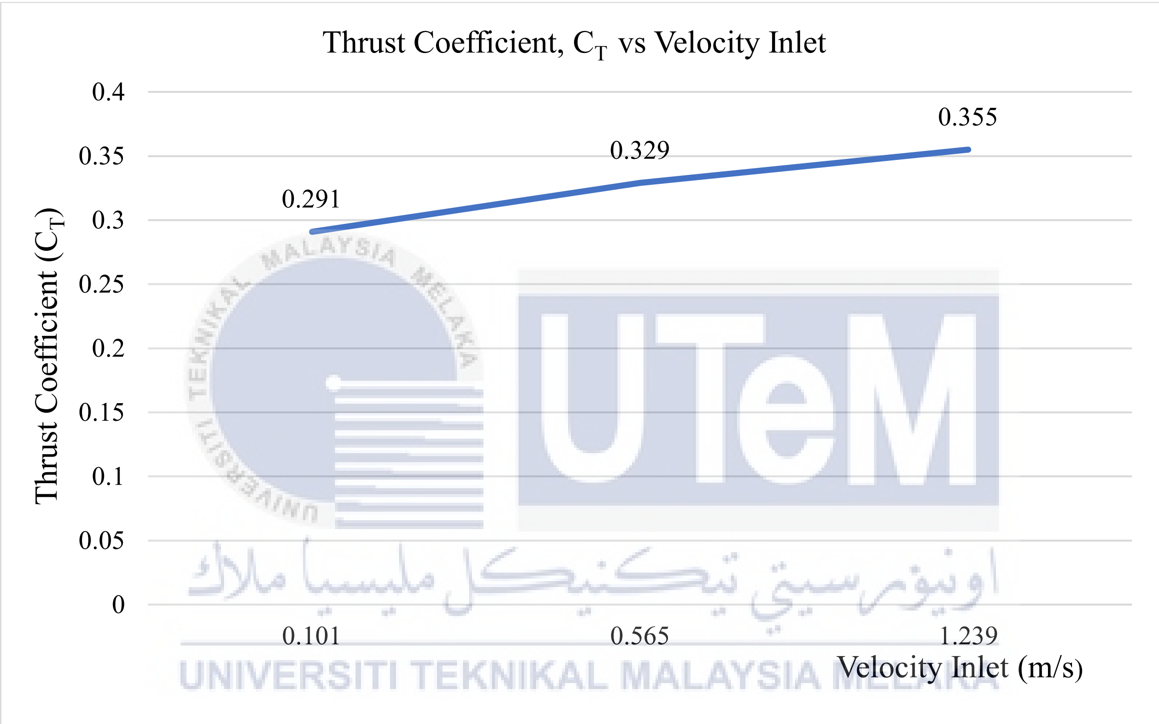


Figure 4.4.3(d): Graph of Total Thrust against Velocity Inlet.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS FOR FUTURE RESEARCH

5.1 Conclusion

There are a few factor that contributes to propeller efficiency regarding to the research study. The parameters of the propeller blade need to be considered in order to design a good propeller blade. There are a few authors that agreed the ducted propeller blade will contribute more efficient performance compare to unducted propeller blade. This is because the ducted fan is more efficient in producing thrust than a conventional propeller of similar diameter by reducing its propeller blade tip losses, especially at low speed and high static thrust level. Besides that, a proper method while making the design and analysis of the propeller blade to ensure the objective can be achieved. The Solidworks software is used to design the chosen conceptual design. The computational analysis is also analyse from Solidworks and ANSYS Fluent 16.0 software. Throughout this project, the optimum propeller AUV is developed. The efficiency of this propeller is up to 92.42%. The propeller blade diameter is 76mm while the hub diameter is 40.13mm and the operating speed that around 31.42 rad/s – 397.94 rad/s is analysed. This propeller has four blades and has 16.5 degree of rake angle. In order to avoid over expecting of the result, thrust value is very important whereas the data form CFD simulation is vital to reduce value of most appropriate speed required. This is because the speed required is proportional with thrust required.

5.2 Recommendation

In the future, this project need to be extended until manufacturing process and continue with the experiment setup to know the validation based on the CFD simulation results. Next, pressure distribution of the propeller blade in CFD analysis may be performed in order to know the failure mode of the propeller. Besides that, the limitation of time on this project are not able to analyse the other two conceptual design. Therefore, in future research, the other conceptual designs must also undergo CFD analysis. This is important to know validation of the conceptual election in conceptual design whereas to determine the optimum design that can achieve high efficiency of propeller blade.

REFFERENCES

T. Chruściel, E. Ciba, and J. Dopke,2014. "CFD and FEM Model of an Underwater Vehicle Propeller," *Polish Marit. Res.*, vol. 21, no. 3, pp. 40–45.

LTJG Kathryn P. D'Epagnier,USN*†, Hsin-Lung Chung, Michael J. Stanway, and Richard W. Kimball, 2007. An Open Source Parametric Propeller Design Tool, pp. 1-9.

Xueming He1, Hecai Zhao, Xuedong Chen, Zailei Luo, Yannan Miao, 2015. Hydrodynamic Performance Analysis of the ducted propeller based on the combination of Multi-Block Hybrid Mesh and Reynolds Stress Model. *Journal of Flow Control, Measurement & Visualization*, pp. 67-74.

Muljowidodo K., SaptoAdiN., Nico Prayogo, and Agus Budiyono, 2009. Design and testing of underwater thruster for SHRIMP ROV-ITB. *Indian Journal of Marine Sciences, 38*, pp. 338-345.

M. Bhanu Priya, K. M. Krishna, and P. Giribabu, 2015. "Design and Analysis of a Propeller Blade," *Int. Res. J. Eng. Technol.*, pp. 45–56.

B. Harish, K. S. Prasad, and G. U. M. Rao, 2011 "Static Analysis of 4-Blade Marine Propeller Static Analysis of 4-Blade Marine Propeller," pp. 30-38.

S. Ramakrishna, V. Ramakrishna, A. Ramakrishna, and K. Ramji,2012. "CFD Analysis of a Propeller Flow and Cavitation," *Int. J. Comput. Appl.*, vol. 55, no. 16, pp. 26–33.

P. Prasad and L. B. Babu, 2017. "Design and analysis of the propeller blade ," no. 2, pp. 14–19.

J. Klesa, "Increasing of Aircraft Propeller Efficiency by Using Variable Twist Propeller Blades."

C. Kong, H. Park, K. Lee, W. Choi, and P. Blade, 2012. "A Study on Structural Design and Analysis of Composite Propeller Blade of Turboprop for High Efficiency and Light Weight," no. June, pp. 24–28.

S. A. Mutalib, S. Suresh, and S. J. Kishore,2015. "Design and Analysis of Composite Marine Propeller using Ansys WorkBench ," vol. 4, no. 9, pp. 3224–3228.

I. Anthony, W. Ekwere, E. Ogbonnaya, and K. Ejabefio, 2013. "Design Procedure of 4-Bladed Propeller," *West African J. Ind. Acad. Res. West African J. Ind. Acad. Res.*, vol. 8888, no. 1111, pp. 13–24.

S. Ekinci, 2011. "A Practical Approach for Design of Marine Propellers with Systematic Propeller Series," *Brodogradnja*, vol. 62, no. 2, pp. 123–129.

M. Maghareh and H. Ghassemi, 2017.“Propeller Efficiency Enhancement by the Blade’s Tip Reformation,” *Am. J. Mech. Eng.*, vol. 5, no. 3, pp. 70–75.

H. Cai, C. Ma, K. Chen, Z. Qian, and C. Yang,2013. “An Integrative Design Method of Propeller and PBCF (Propeller Boss Cap Fins),” no. May, pp. 320–323.

J. Noosomton and J. Charoensuk,2015. “The New Propeller Design 3 Blades Type on 18o Skew Angle and Testing Cavitation of Tailing Thai Boat,” vol. 5013, no. 4, pp. 257–263.

R. K. Tripathi, V. Tiwari, I. Khunger, and P. Berwal, 2015. “Development of Propeller Test Rig and Evaluation of Propeller Performance,” pp. 1161–1167.

S. A. Rahaman,2014. “Design , Development and Analysis of Advanced Airboat Propeller by Using Bamboo Composite Fibre Material,” vol. 2, no. 4, pp. 3475–3481.

K. P. S. Babu and S. Padmanabhan, 2017. “Design of Marine Propeller Blade with Different Blade Sequences Analyse The Hydro Formation Under Pressure Hydrodynamic Fill ”, vol. 12, no. 17, pp. 4996–5002.

B. Moffitt, T. Bradley, D. Parekh, and D. Mavris, 2008. “Validation of Vortex Propeller Theory for UAV Design with Uncertainty Analysis,” *46th AIAA Aerosp. Sci. Meet. Exhib.*, no. January, pp. 1–19.

S. Ramakrishna, V. Ramakrishna, A. Ramakrishna, and K. Ramji, 2012. “CFD Analysis of a Propeller Flow and Cavitation,” *Int. J. Comput. Appl.*, vol. 55, no. 16, pp. 26–33.

K. V. R. & K. K. S. Sudhakar Uppalapati, 2016. “Design and Analysis of Propeller Blade Using Catia & ANSYS Software”, *BEST Int. J. Manag. Inf. Technol. Eng. (BEST IJMITE)*, vol. 4, no. 4, pp. 83–96.

M. Stajuda, M. Karczewski, and D. Obidowski,2016. “Development of a CFD model for propeller simulation,” vol. 20, no. 4, pp. 579–593.

Baltazar, J., Falcão De Campos, J. and Bosschers, J. (2012) Open-Water Thrust and Torque Predictions of a Ducted Propeller System with a Panel Method. *International Journal of Rotating Machinery.*

S. O. Oladokun,2015. “Study of Efficiency and Environmental Performance of Propeller,” *J. Coast. Zo. Manag.*, vol. 18, no. 2.

Dunna Sridhar et al,2010. “ Frictional resistance calculation on a ship using CFD”, *computer applications,* International journal on computer applications, Vol 11-No. 5.

Dunna Sridhar et al,2010. “ Frictional resistance calculation on a ship using CFD”, *computer applications,* International journal on computer applications, Vol 11-No. 5.

J. Quang, T. Huynh, and M. Ludvigsen, 2016. "Detailed Design of a Thruster Solution for a Small Mass-Market Remotely Operated Underwater Vehicle,".

M. Husaini, Z. Samad, and M. R. Arshad, 2004. "Autonomous Underwater Vehicle Propeller Simulation using Computational Fluid Dynamic,".

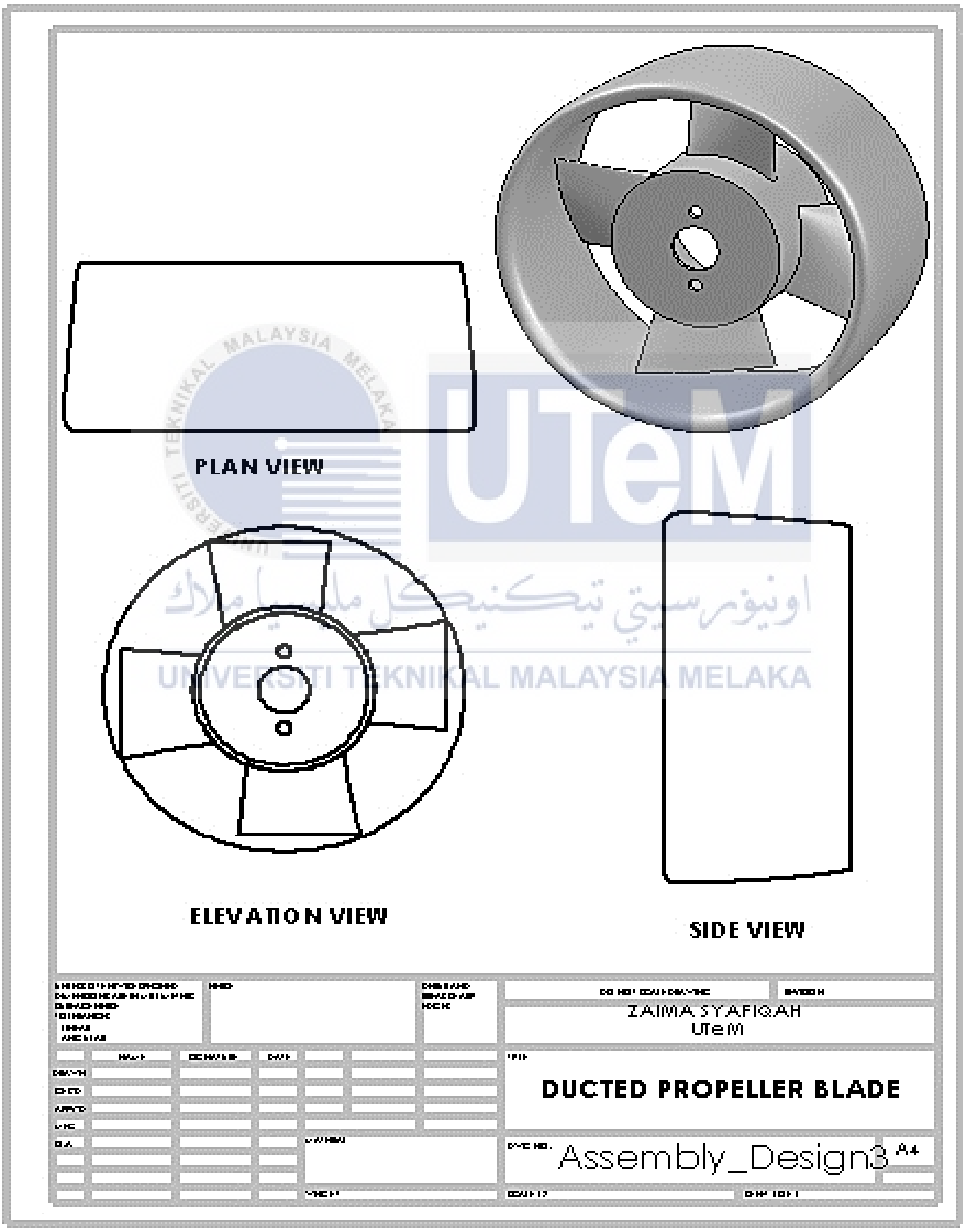
H. Ghassemi, A. Mardan, and A. Ardeshir, 2012. "Numerical analysis of hub effect on hydrodynamic performance of propellers with inclusion of PBCF to equalize the induced velocity," *Polish Marit. Res.*, vol. 19, no. 2, pp. 17–24.

M. Husaini, Z. Samad, and M. R. Arshad, 2014. "Optimum Design of URRG-AUV Propeller Using PVL,".

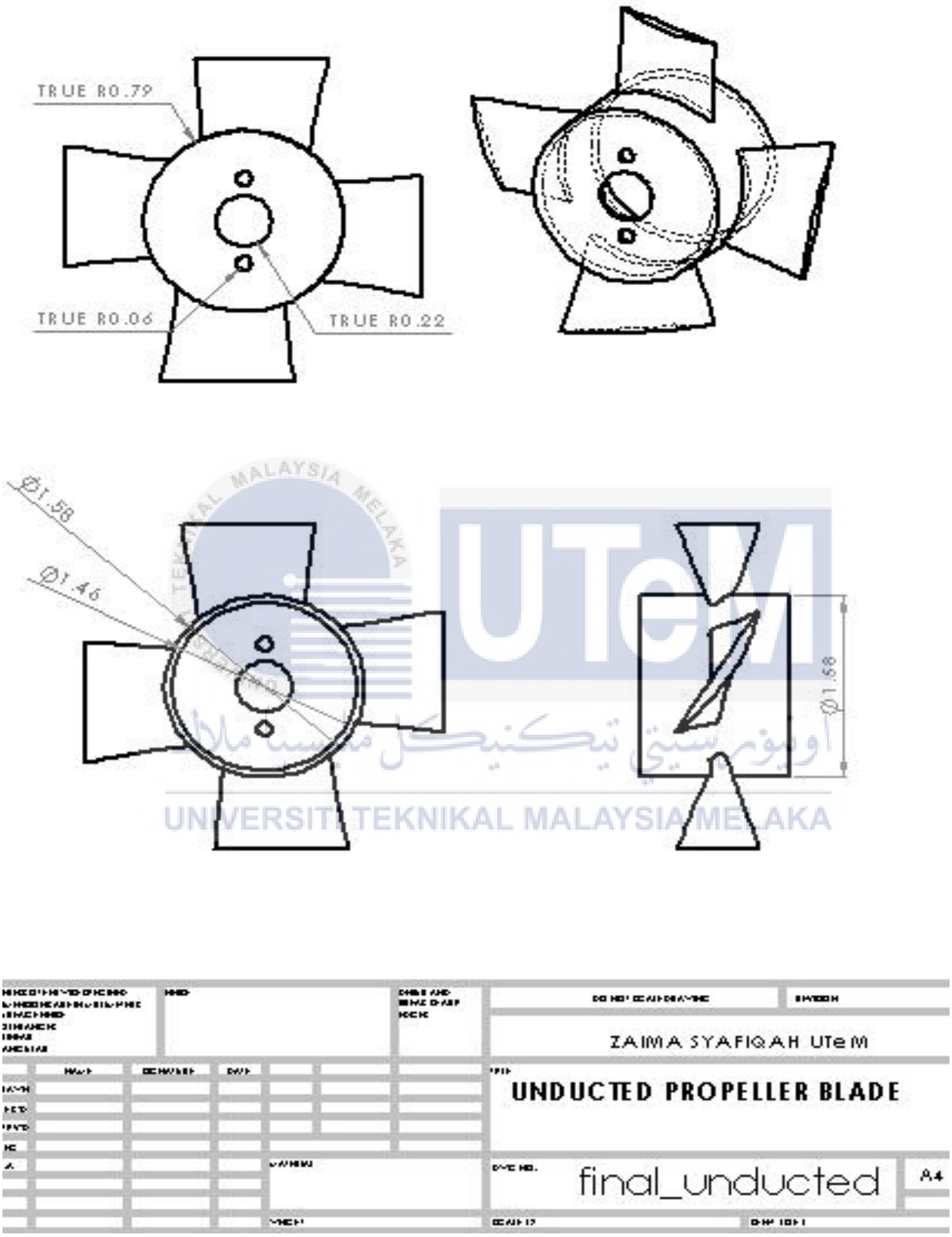
C. S. Lee, Y. D. Choi, B. K. Ahn, M. S. Shin, and H. G. Jang, 2010. "Performance optimization of marine propellers," *Int. J. Nav. Archit. Ocean Eng.*, vol. 2, no. 4, pp. 211–216.

APPENDICES

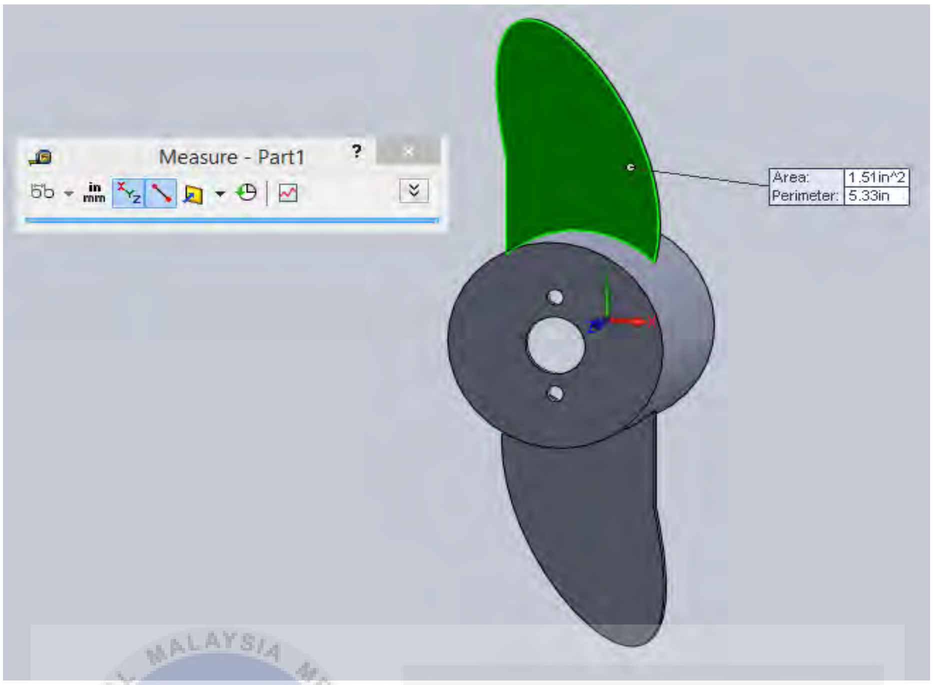
A – Detail Drawing of Ducted Propeller Blade



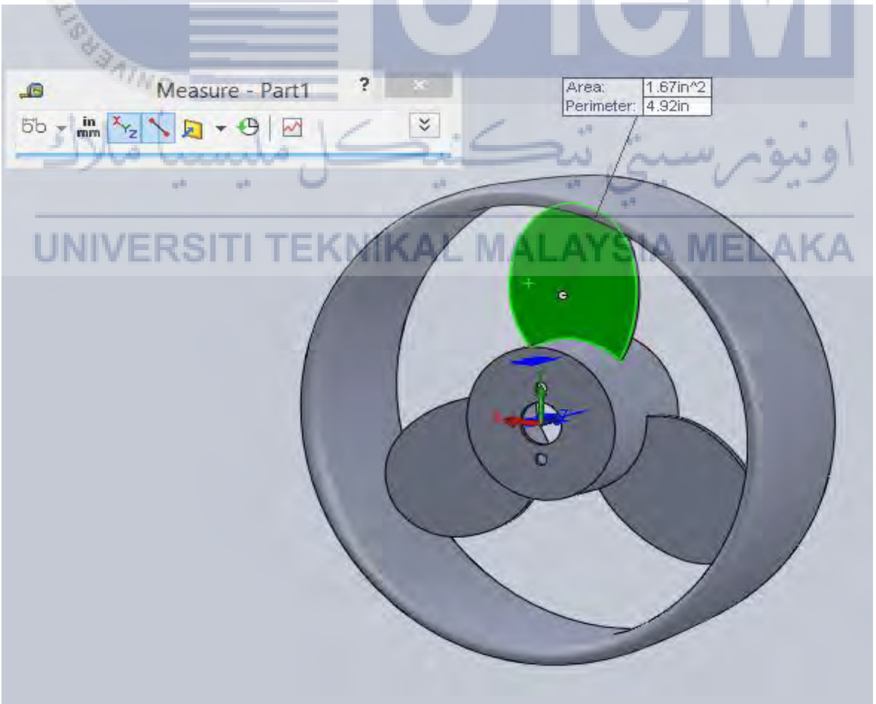
B –Detail Drawing of Unducted Propeller



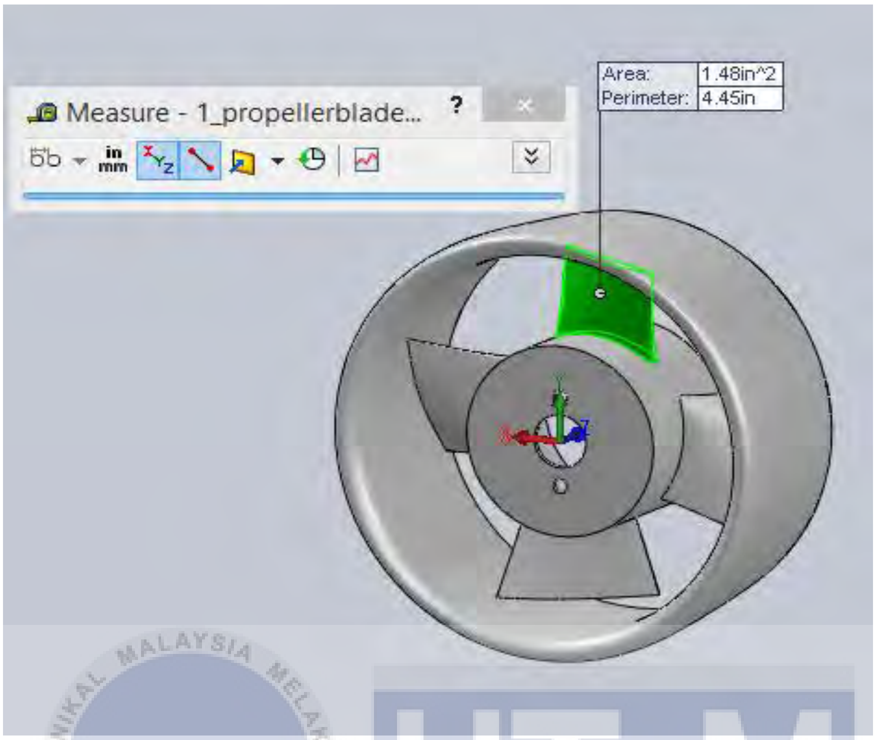
C- First Conceptual Design Choice with Surface Area Value



D – Second Conceptual Design Choice with Surface Area Value



E – Third Conceptual Design Choice with Surface Area Value



F – Calculation of Mathematical Modelling in Excel

B	C	D	E	F	G	H	I
Range RPM = 300-3800 rev/min = 31.42 - 397.94 rad/s			Density [kg/m^3]	Area [m^2]			
			1000	0.036018			
Rad/s	Velocity inlet (VA)	Propeller Thrust Force [N]	Propeller Thrust by duct [N]	Total Thrust [N]	CT (Thrust Coefficient)	Efficiency of propeller	Efficiency propeller (%)
31.42	0.100994	5.66E-02	-0.00316299	5.34E-02	0.290655432	0.936298829	93.62988294
183.25	0.565278	1.91955	-0.0277997	1.8917503	0.328738104	0.929061928	92.90619278
397.94	1.2387	9.38608	0.421909	9.807989	0.354942576	0.924205873	92.42058732

G– Specification Details of T200 Thruster by Blue Robotics.

Specifications

	Parameter	
Max Thrust – Forward @16V	11.2 lbf	5.1 kgf
Max Thrust – Reverse @ 16V	9.0 lbf	4.1 kgf
Max Thrust – Forward @12V	7.8 lbf	3.55 kgf
Max Thrust – Reverse @ 12V	6.6 lbf	3.0 kgf
Min Thrust	0.02 lbf	0.01 kgf
Rotational Speed	300-3800 rev/min	
Operating Voltage	6-20 volts	
Max Current	25 Amps	
Max Power	350 Watts	
Length	4.45 in	113 mm
Diameter	3.8 in	96.5 mm
Propeller Diameter	3.0 in	76 mm
Cable Length	39 in	1.0 mm
Mounting Hole Threads	M3x0.5	
Mounting Hole Spacing	0.75 in	19 mm
Weight in Air (with 1m cable)	0.76 lb	344 g
Weight in Water (with 1m cable)	0.34 lb	156 g