THE INFLUENCE OF BLADE PARAMETERS CONFIGURATIONS ON THE PERFORMANCE OF THRUST POWER FOR AUV APPLICATION

PHARAWUS A/L ACHUAN

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

THE INFLUENCE OF BLADE PARAMETERS CONFIGURATIONS ON THE PERFORMANCE OF THRUST POWER FOR AUV APPLICATION

PHARAWUS A/L ACHUAN

This report is submitted
in fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering (Hons.)

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2019

DECLARATION

I declare that this project report entitled "The Influence of Blade Parameters Configurations on The Performance of Thrust Power for AUV Application" is the result of my own work except as cited in the references.

Signature : ....

Supervisor's Name : PHARAWUS A/L ACHUAN

Date : 14 JUNE 2019

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 (Hons).

Signature : ....

Name : DR. MASJURI BIN MUSA@OTHMAN

Date : 14 JUNE 2019

DEDICATION

To my beloved mother and father

Eput A/P Din Chom & Achuan A/L Din Proom

ABSTRACT

In general, the project revolves about the configuration and analysis of propeller blade parameters on the performance of thrust power for Autonomous Underwater Vehicle (AUV) application. The study and analysis carried out is intended to find the optimum propeller blade configuration for the AUV performance in terms of stream velocity output and thrust power. The technique of modeling and simulation by using Computer Aided Design (CAD) software to manipulate the parameters configurations that come out with 32 designs of blade propeller is the best methods to get the effective optimum designs. From the study conducted, there are only research on each parameters of propeller blade but no research on the blade parameters configuration was found. Therefore, Computer Fluid Dynamic (CFD) has been used in this project as a medium for analyzing new parameters configuration for optimum performance of propeller blade. The study goes on to produce a propeller blade with new parameters configuration for AUV which the optimum thrust power produced. In additional, the data generated by the CFD analysis and calculated thrust power has been studied and the design with optimum thrust power has been identify while further work is to run the experiment as a method to test theories and hypotheses about how physical processes of this analysis work.

ABSTRAK

Secara umum, projek ini menyentuh tentang konfigurasi dan analisis parameter bilah kipas mengenai prestasi kuasa tujah untuk aplikasi Kenderaan Bawah Air Autonomi (AUV). Kajian dan analisis yang dijalankan bertujuan untuk mencari konfigurasi bilah kipas yang optimum untuk prestasi AUV dari segi output aliran dan daya tujahan. Teknik pemodelan dan simulasi dengan menggunakan perisian Reka Bantu Peranti Komputer (CAD) untuk memanipulasi konfigurasi parameter yang keluar dengan 32 reka bentuk kipas adalah kaedah terbaik untuk mendapatkan reka bentuk optimum yang berkesan. Dari kajian yang dijalankan, hanya terdapat penyelidikan pada setiap parameter bilah kipas tetapi tidak ada penyelidikan pada konfigurasi parameter yang dijumpai. Oleh itu, Dynamic Fluid Computer (CFD) telah digunakan dalam projek ini sebagai medium untuk menganalisis konfigurasi parameter baru untuk prestasi optimum bilah kipas. Kajian ini menghasilkan bilah kipas dengan konfigurasi parameter baru untuk AUV yang dihasilkan oleh daya teras yang optimum. Di samping itu, data yang dihasilkan oleh analisis CFD dan kuasa tujahan dikira telah dikaji dan reka bentuk dengan kuasa teras optimum telah dikenalpasti manakala kerja selanjutnya adalah untuk menjalankan percubaan sebagai kaedah untuk menguji teori dan hipotesis tentang bagaimana proses fizikal analisis ini bekerja.

ACKNOWLEDGEMENT

This thesis was prepared for the Faculty of Mechanical Engineering (FKM), Universiti Teknikal Malaysia Melaka (UTeM). This thesis report is submitted in fulfill of the requirement for the degree of Bachelor of Mechanical Engineering.

Firstly, I would like to express my deepest appreciation to my supervisor Dr. Masjuri Bin Musa@Othman for giving me this opportunity to do final year project with him. He never hesitated to give me advice and guidance whenever I confronted problems. I am thankful for his patience and advice while leading me in this project.

In additional, I would like to thank my course mates for giving me their support, patience and encouragement to complete this project. Lastly, I would like to thank my family who always on my side and giving their full support from the beginning till the end.

CONTENT

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS

T	-	Thrust
D	-	Propeller diameter
v	-	Velocity of incoming flow
ρ	-	Density of fluid
Δv	-	Additional velocity, acceleration by propeller
P	-	Thrust power
N	-	No. of blade
Θ	-	Blade angle
HTR	-	Hub to tip ratio (hub radius/propeller radius)
L	-	Nose cone length
R	-	Nose cone radius
v_o	-	Stream velocity outlet
g	-	Gravitational gravity
z	-	Elevation
p	-	Pressure
F	-	Thrust force
A	-	Surface area

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Autonomous Underwater Vehicle (AUV) was developed in 1957 by Stan Murphy and Bob Francois at the University of Washington to overcome the limitation of human capabilities into water especially deep sea. Deep sea in the ocean are very dangerous that sometimes can cause death to human. The AUV have created to explore and discover underwater world without make harm to human and at the same time reduce the limitation and increase the capabilities of human. Most of this vehicle have been used in military field, inspection of cracks on the underwater section of the ship, oil and gas industry, telecommunication and mineral exploration and the expedition to explorer under the deep ocean.

The AUV in military has been created and plays an important job to counter the danger from underwater by using advanced control systems, navigation systems and sonars to protect and detect any harms or enemy. The vehicle can operate in both shallow and deep sea, and allows exact detection, classification and coordination of any searching objects while the controller and others crew remain in a safe place and distance. After the AUV has completed the mission, military officers can enter the area with detailed information of the threat.

Though still a very small part of AUV usage in commercial sector like oil and gas industry that has been operated over 100 years. Oil and gas industry supplies energy to human in form of petroleum and gaseous by drilling to the resource cover up underground such as deep sea and transport it by using pipelines. In order to work in deep water that has a dangerous environment and can harm the human to operate, a robot plays an important job. An

Autonomous Underwater Vehicle (AUV), which is a kind of a robot is used for equipment installation, operation and maintenance.

AUV can operate automatically and travels underwater by diving from the surface to deep sea and reverse. System of AUV can be compact, portable and small or they can be long-endurance and very large systems. Autonomous Underwater Vehicle basically bring power such as batteries, fuel cells or rechargeable solar power onboard in the body that design aerodynamically like a sub-marine. The power onboard are uses to operate all sensors on the AUV and to move the propellers or thrusters. Certain AUV minimize energy demands by allowing buoyancy and gravity to impel them and can reach shallow sea that boats can't and deep that human divers or other vehicles can't. AUV are safe from bad weather once deployed and underwater.

To enable AUV move in water, propellers or thrusters play an important role. Propellers is a mechanical device consists of several twisted and air-foil shaped blades that rotates and produce pressure differential between the blades to accelerate the fluid and provide thrust in opposite direction to propel the AUV. There are several parameters of blades will affect the performance of propeller need to be consider such as the radius of blade, the chord of the propeller and radius of hub.

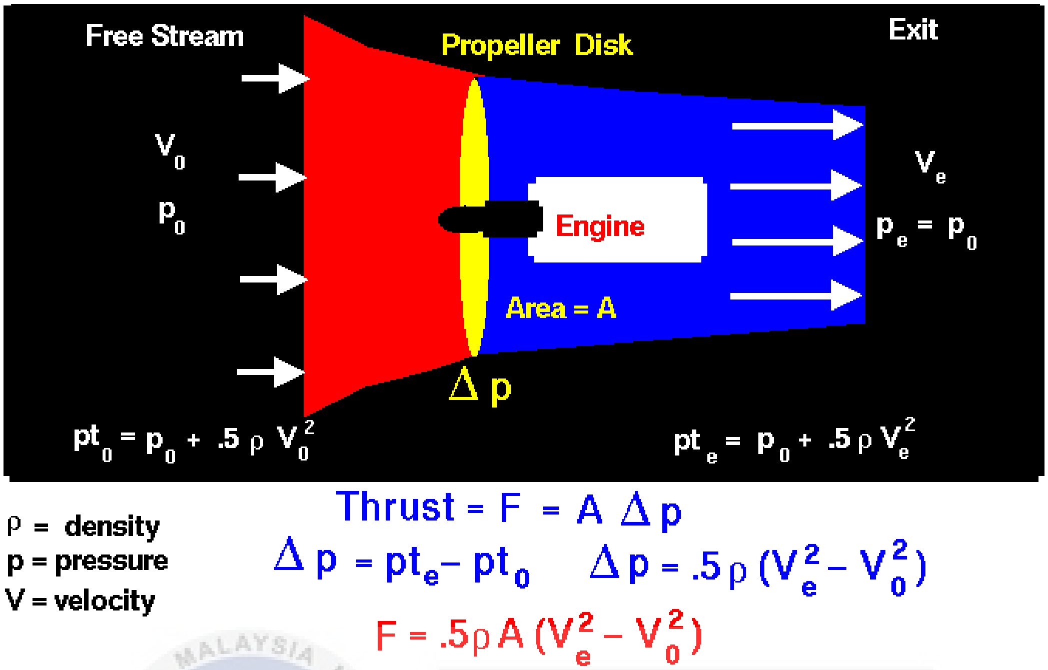


Figure 1.1: Propeller Thrust (Source: grc.nasa)

There are several terms used to describe performance properties and characteristics of propeller of AUV. It is important considering all terms to ensure high system efficiency to provide great performance to the AUV. Bernoulli's principle and Newton's third law are used to modelled the operation of propeller.

Bernoulli's principle,

$$\frac{v^2}{2} + gz + \frac{p}{\rho} = \text{constant}$$

Where:

- **v** is the fluid flow speed,
- **g** is the gravitational gravity,

- **z** is the elevation,
- **p** is the pressure,
- **ρ** is the density of the fluid.

While Newton's third law states that all forces between two objects exist in same magnitude but opposite direction. In the Figure 1.2, a boy is pulling a rope that is attached to an elephant with force F_A then elephant at the same time pulling a rope that is attached to a boy with force F_B, and the two forces are equal in magnitude but opposite in direction, $F_A = -F_B$.

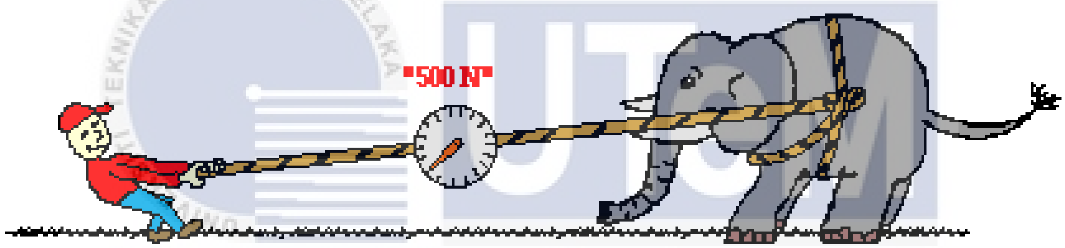


Figure 1.2: Example of Newton's Third Law (Source: physicsclassroom.com/class/newtlaws/Lesson-4/Newton-s-Third-Law)

1.2 PROBLEM STATEMENT

Nowadays, the development of AUV have been focus more on how to design the AUV with more portable and more compact with more energy efficiency innovation. To move the AUV, propeller been used to produce the efficient vertical and horizontal movement. There are a lot of research on the airplane propeller but development research on designing and manufacturing new marine propeller is still low. While the data or design from airplane propeller research is not suitable to use in marine vehicle because the different of the density of medium which is the density of water is 1000 times higher than air (D’Epagnier, 2006). Meanwhile the fabrication of a customize propeller is much more expensive compare to buying the existing propeller.

On the other hand, the development of new concepts in propeller designing has shown a significant improvement in the design method that could enhance the process of propeller designing. However, the impact of development has yet not shown due to the innovators and designers end at the traditional technique that has gone through loads of researches and were validated, which is much more reliable and were implemented for plenty of times.

The velocity of propeller is the main key to improve the movement performance of AUV. The design of blades with all parameters on propeller is most important to achieve a efficient movement of AUV. Therefore, it is important for this research to clearly show the best parameters of blades for optimum performance of AUV.

1.3 OBJECTIVE

The objective of this project is:

1. To identify the average outlet flow velocity based on blade parameters configurations which affected the performance (thrust power) of AUV.

1.4 SCOPE OF PROJECT

The scopes of this project are as follows:

1. To develop a blade parameters configurations for Autonomous Underwater Vehicle (AUV) by using Computer-Aided Design (CAD).
2. To do Computational Fluid Dynamic (CFD) simulation analysis on the blade parameters configurations to identify the average outlet flow velocity based on the blade parameters such as number of blades, blade angle, hub to tip ratio and nose cone.
3. To identify the optimum blade configuration for the AUV performance (thrust power).

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This chapter is devoted to a review of literature. The past developments in propeller for AUV will be discussed. Different methods will be covered, their shortcomings and advantages addressed, all with the goal of selecting the appropriate performance prediction method to use in this analysis. This section also will be focused on the review of propeller optimization methods with the same goal in mind of selecting the appropriate method for this present study. Past work performed in the field specifically focusing on research performed on type of blade, number of blades, blade angle, hub to tip ratio, nose cone of propeller and design of duct is covered. This section will also provide a brief introduction into a variety of materials and present their capabilities and limitations and CFD analysis. A summary of the conclusions reached in the process of the review will be presented.

2.2 Propeller Physical

The radius of the propeller is the distance between the tip of the blade and the centre of the hub (Duelley, 2010). The chord of the propeller is a straight-line distance between the leading edge and the trailing edge at a blade section of the propeller (Duelley, 2010). Figure 2.1 shows the nomenclature of the propeller as described.

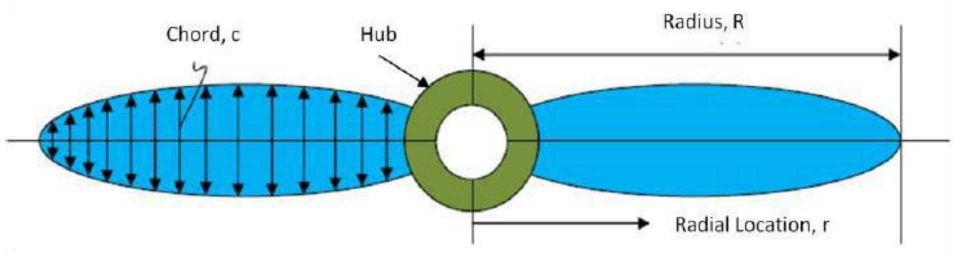


Figure 2.1: Propeller nomenclature (Source: Duelley, 2010 and Schultz, 2009)

Camber of the propeller is the maximum distance from the mean line to the chord lines. The maximum thickness of the blade is measured in a normal to the chord line (Ferrando, 2012). The optimum performance of ideal propeller would be infinitely thin so that it is more hydrodynamic (Duelley, 2010). However, this is unrealistic and impossible to be achieved because it is not manufacturability, durability and efficient. Low thickness profile has the risk of breaking or bending when the blade is handled roughly as shown in Figure 2.2 (Duelley, 2010).

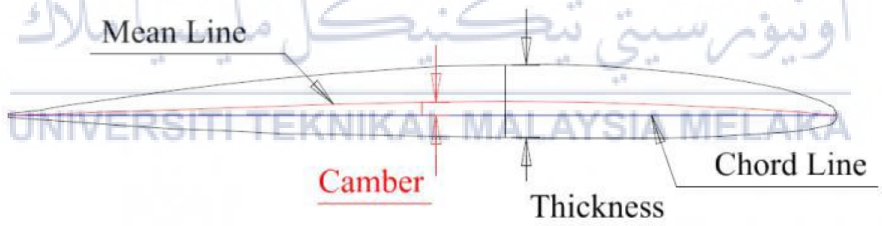


Figure 2.2: The cross section of the blade. (Source: International Towing Tank Conference, 2011)

Besides, the blade is classified into two, which is the leading edge and the trailing edge which is outlined on the Figure 2.3. Leading edge is the first part that contact with water when it is rotating while trailing edge is the last part of the blade that contact with the water. From Figure 2.4, it can clearly be seen that the pressure face of the propeller where it has high pressure at that part. The suction back has low pressure which is the cause of the AUV to move forward.

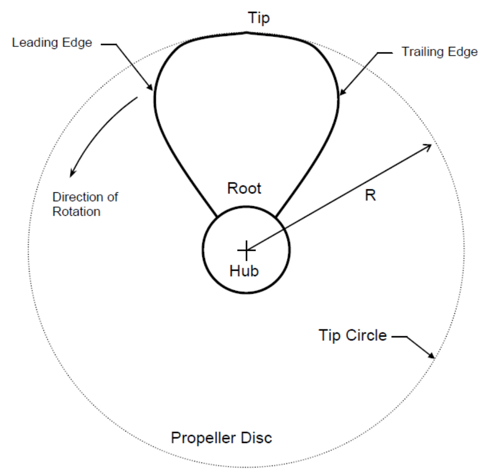


Figure 2.3: Basic propeller geometry. (Source: US Navy)

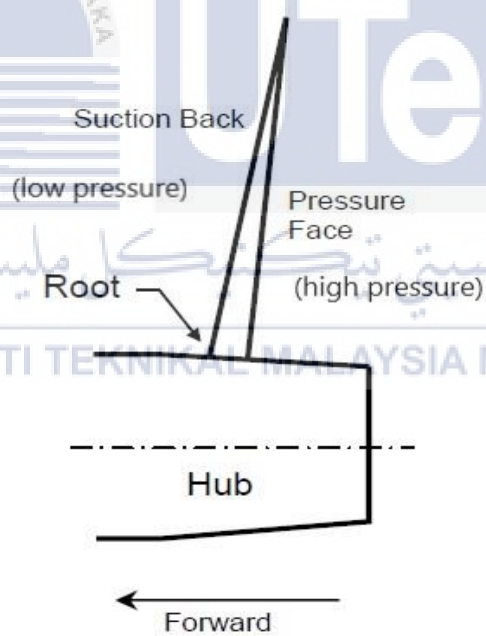


Figure 2.4: The pressure face and suction back (Source: US Navy)

Pitch is a measurement of the distance of propeller would move parallel to the direction of motion when it rotates at one revolution (Duelley, 2010). High pitch over diameter ratio can increase the propeller efficiency (D'Epagnier, 2006). Pitch angle is the angle of the blade that is perpendicular to the water flow. Figure 2.5 explain about the pitch of a propeller.

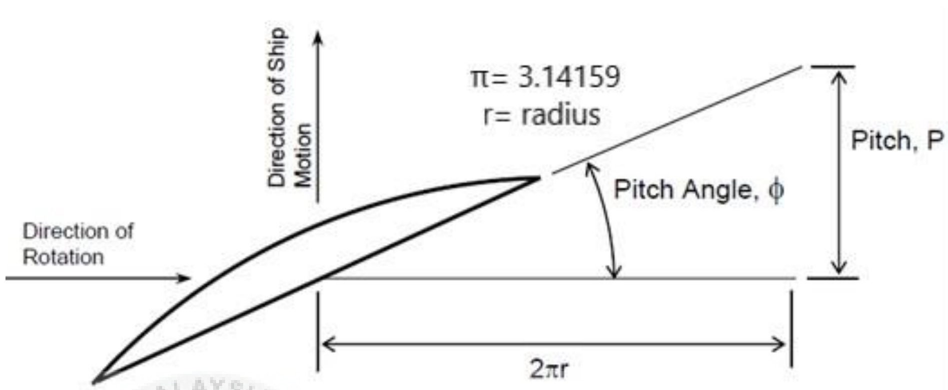


Figure 2.5: Pitch and pitch angle (Source: US Navy)

2.3 Type of Propeller Blade

There are two common type of blade, flat blade and twisted blade. Flat blade means the blade angle is the same at all points from the hub to the tips (Vesting et al., 2014). While twisted blade means the angle of blade starts high at the hub and progressively decreases to the tips (Xiang et al., 2018).

Figure 2.6: Example of Flat Blade Propeller (Source: chinabrands)

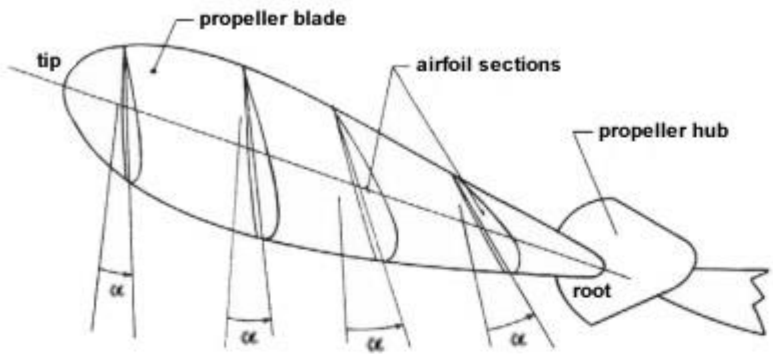


Figure 2.7: Example of Twisted Blade Propeller (Source: aviation.stackexchange)

Flat blades normally can perform reversible rotation because of constant blade angle but generally required more horsepower to drive than twisted blade and not as clean cut or as good a windrow (Vesting et al., 2014). If one out of several blades is lost, flat blade produces less vibration.

While twisted blades need for a different blade angle along the blade from the fact that angular speed is constantly increase from the hub to the tips means that the speed of blade is much higher at the tip than at the hub (Xiang et al., 2018). The uses of twisted blade is necessary to maintain a constant angle of attack between the fluid flow and surface along the length of the blade (Prasad et al., 2017). Propeller performance is degraded when it is not at its optimum angle like a wing. Combine this with any forward speed the propeller, the relative waterflow is different from the hub to the tips. To keep thrust equal along the blade, the design is that the blade is thick at the hub with a large blade angle and thin at the tip with a low blade angle.

Choosing type of blades very important to get a good performance of propeller. Flat and twisted blades can use in different condition with their own advantage and disadvantage. From the information, twisted blade gives more equivalent thrust along the blade, so it will generate better forward speed than flat blades. While flat blades produce higher thrust at the tip of blade and low thrust at the hub.

2.4 Number of Propeller Blade

Propeller blades may vary from 3 blade propeller to 4 blade propeller and sometimes even 5 blade propellers (Hong, 2018). The most commonly used are 3 and 4 blades propeller. By increasing number of the blades, it caused to increase pressure in front of the propellers. So, the mass flow rate and the rotational speeds decreased (Ahmadi, 2016). Adding more blades means increasing the surface of the blades that will provide more force. Total force and thrust are increase but force on each blade and rotational speeds are reducing by adding blades. Having more blades moving through the water would generate more flow (Monfared, 2016). However, this ignores the effect of drag or water resistance. It is this drag that slows down motion, reduces waterflow, and increases energy consumption of a propeller. Having fewer blades may reduce drag but may also generate little waterflow (Dekanski, 1993). But fitting more blades to a propeller may not generate more waterflow because of the larger drag. This increased in drag would also mean a more powerful and a more energy consumption propeller motor is required (Dekanski, 1993). Having more blades also adds weight to the propeller and will need a more powerful propeller motor. The selection of the number of blades is a function of several important parameters, namely uniformity of output power, propeller loads and vibrations, cost as well as the propeller aerodynamic performance (Rezaeiha et al., 2018).

The number of blades in a propeller has little effect on performances. Three blades seem to be the best compromise between aerodynamic and structural concerns in low-speed applications such as boats and small aircraft but for higher spinning rates. Four blades are better, since they accelerate quicker and have less vibrations. In general, more propeller blades required in a powerful engine. The number of blades has a small effect on the efficiency. Usually a propeller with more blades will perform better as it distributes its power and thrust more evenly in its wake or start (Shaikh, 2015).

Figure 2.8: Front views of Twisted Propeller with different number of blades (Source: dreamstime)

Table 2.1: Characteristics of 3 and 4 blades

Number of Blades	3	4
Characteristics	• The manufacturing cost is lower than other types. • Gives a good high-speed performance. • The acceleration is better than other types. • Low speed handling is not much efficient.	• The manufacturing cost is higher than 3 blade propellers. • Have better strength and durability. • Gives a good low speed handling and performance. • Has a better holding power in rough seas. • 4 blade propeller provides a better power economy

2.5 Size of Propeller Blade

Propeller size is expressed with two numbers, diameter and pitch (Boatus, 2009). Diameter is two times the distance from the centre of the hub to the tip of blade (Schubel et al., 2012). Smaller diameters generally go with smaller engines or with fast high performing boats because it will generate less thrust than bigger diameter of propeller. The propeller size has a big impact on performance. Usually a larger propeller diameter will have a higher efficiency, as it catches more incoming fluid and distributes its power and thrust on a larger fluid volume (Shaikh, 2015). Pitch is the theoretical forward distance that a propeller travels during one revolution.

The size of propeller also can be in term of hub to tip ratio which mean there are two different diameters in this form, diameter or radius of the hub and diameter or radius of propeller. The radius of propeller is the distance from the tip of the blade to the centre of the hub. The hub to tip ratio is radius of the hub per radius of propeller. In the case, the diameter of the propeller is constant while the diameter of the hub is increasing, or the diameter of the hub and propeller has been increased with the same ratio. The hub effect will reach its minimum at a certain hub ratio (Ghassemi et al., 2012).

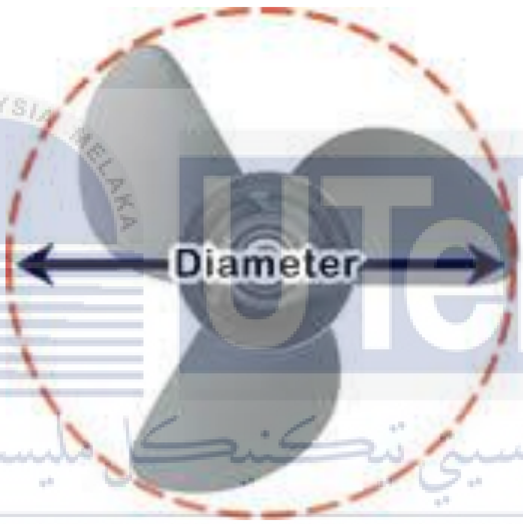


Figure 2.9: An example of twisted propeller diameter (Source: bblades)

Figure 2.10: Example of different size of propeller (Source: getfpv)

A larger diameter means that the vessel is can achieve higher thrust, and so higher forward speed for the same rotational speed of the shaft (Asimakopoulos, 2016). The propeller diameter determination is a complex procedure involving various empirical formula for optimum efficiency. The rotational speed of the propeller could be reduced but still achieve the required forward movement in larger diameter condition (Asimakopoulos, 2016). A reduced rotational speed mean that the pressure imbalance on the blades be reduced with the reduction of the forces and so cavitation would be decreased or alleviated.

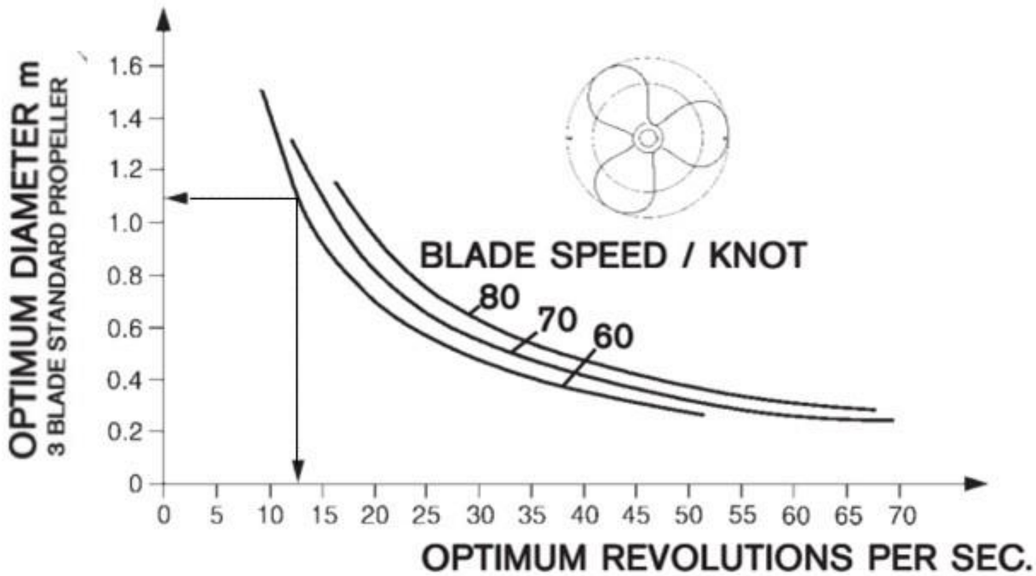


Figure 2.11: Optimum diameter selection graph for twisted propeller (Source: Asimakopoulos, 2016)

By considering type or condition need to be use, size of propeller should suitable in each condition. For example, there are unnecessary to use a big propeller that will need more power from motor in small boat or use a small propeller that will generate low thrust in big boat. Propeller should generate suitable thrust and force to move a marine vehicle. The size and weight of marine vehicle have effect for choosing the optimum size of propeller.

2.6 Design of Ducted Propeller

Ducted propellers are present in modern rotor systems ranging from marine/offshore propulsion systems, ducted wind turbine, tidal turbines to aerospace turbomachinery, drones and flying cars (Zhang, 2018). A ducted propeller is surrounded by a static nozzle which provides several attractive benefits, namely physical protection, greater power density, cavitation reduction, and expanded operating environment. Based on the commercial duct propeller, a kind of vector propulsion system, including a deflectable duct propeller can only produce thrust without any additional control force. However, when the propeller is fixed, and the duct is deflected, the deflectable duct propeller will generate the lateral force and moment controlling AUV to change its states and the thrust to drive AUV. The design of a streamlined underwater vehicle utilizing an entirely enclosed propulsion and steering system (Kopman et al., 2012)

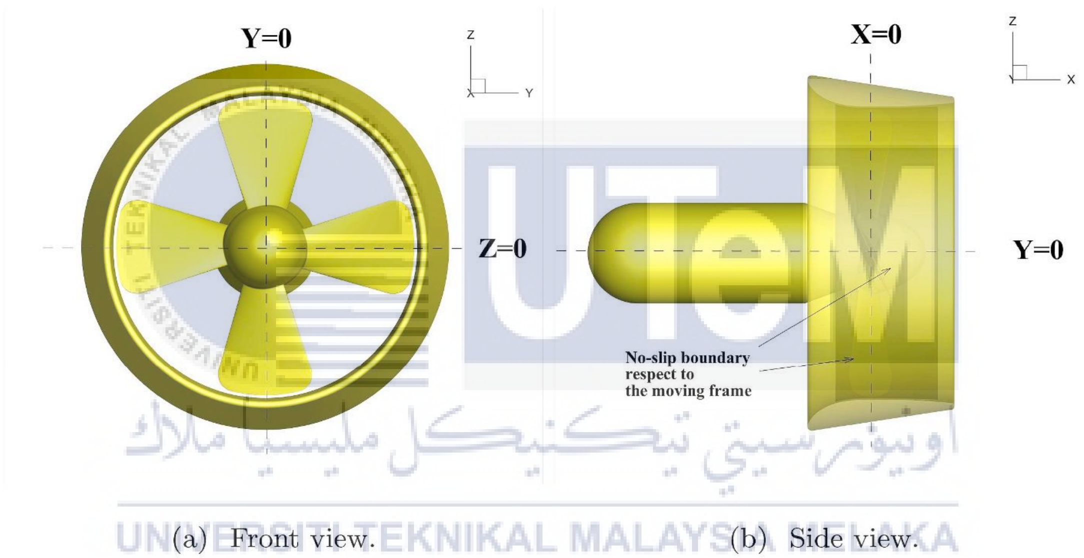


Figure 2.12: Geometry of ducted twisted propeller with four blades. (Source: Zhang, 2018)

The blades will produce high pressure areas behind them and low-pressure areas in front when propeller rotates. The different of pressure provides the excessive thrust to move the vessel in the fluid (Bhattacharyya, 2015). However, losses will occur as water escapes from the high-pressure side of the blade to the low-pressure side at the tip of each blade. A ducted propeller will stop the escaping water by restricting water flow to the propeller tips. The entrance diameter is bigger than the trailing throat of the nozzle that designed as a funnel (Go, 2017). The mass flow rate must be constant as the water density is constant. The mass must compensate with a higher velocity at the trailing end when the area becomes smaller, thus accelerating throughout the nozzle. Thrust at the same input power when more water is moved by the ducted propeller (Krasilnikov, 2015). The nozzle will be increasing the inflow velocity, lower the pressure contribution on the propeller and the efficiency. The risk of cavitation will increase when lowering the pressure in the propeller disc. The inflow velocity will reduce when increasing the pressure through the propeller disc, but in turn reduce the cavitation risk.

The design of duct will help to prevent or decrease the losses or water escape at the tip of blade. The inlet diameter of nozzle generally bigger than outlet diameter of nozzle to give more acceleration and thrust at the same input power as no-duct propeller. The size of nozzle should suitable with the size of propeller to get the better performance of propeller. If the gap between the nozzle and the tip of blade is big, the water escape will still occur, the situation will be same as no-duct propeller.

2.7 CFD analysis of a propeller flow

CFD analysis used to simulate the behaviour of the model by running the mathematical equation in the analysis. The 3-dimension (3D) solid modelling for each propeller been used in CFD simulation will be exported from the computer aided design (CAD) software (Othman, 2014). 3D solid modelling of propeller is carried out using CATIA.

In CFD analysis, the speed rotation of propeller has been set because the simulation of propeller is using a low flow application (Othman, 2014). This value normally used a minimum speed rotation needed by a propeller. When all initial value needed under CFD has been fulfilled, the simulation and end results can be obtained. The purpose of implementing CFD analysis is to determine the stream velocity for each propeller involved as well as to look at the flow patterns of the water at inlet, between the blades and at the outlet section (Subhas, 2012). The values of the stream velocity for each design later will be substituted in the power output equation in order to obtain the most optimum results of power output.

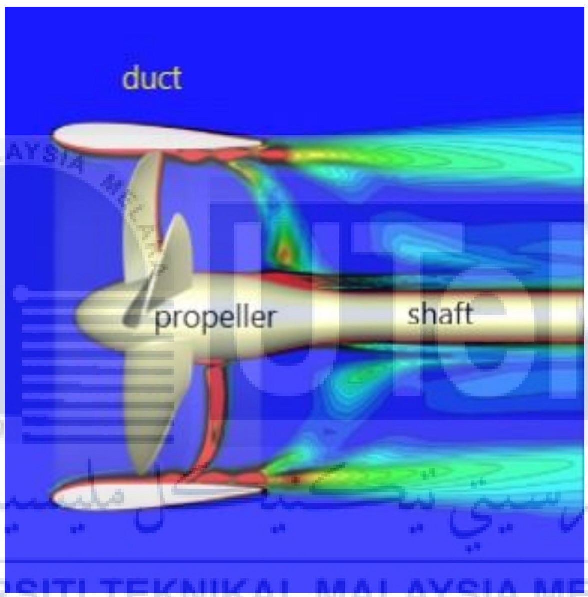


Figure 2.13: An example of CFD analysis on ducted propeller (Source:marin.nl)

Propeller performance in open water can calculate in the design stage. The thrust and torque coefficient can be compute using CFD. CFD was used to get the performance of the propeller when the water was moving because of difficulty in testing the propeller under axial flow (Husaini,2005). Computational Fluid Dynamics program that uses numerical methods to solve and analyse fluid problems (Colley, 2012). Pre-processing, processing (solving) and post-processing are generally three parts to solving a problem computationally. The boundary conditions are defined as specifying the way the fluid should move, the object and surrounding surfaces and volumes are then meshed during pre-processing. While the processing stage is done via a solver. OpenFOAM, an open source CFD, allows many different solvers to be used. The solver that will be examined is steady-state solver for incompressible laminar flow. The objects and surrounding surfaces and volumes are then meshed. Various types of problems can be solved in different dub modules. ParaView is another open source tool in post-processing is where the analysis of the solution is done, to visualise and investigate the problem.

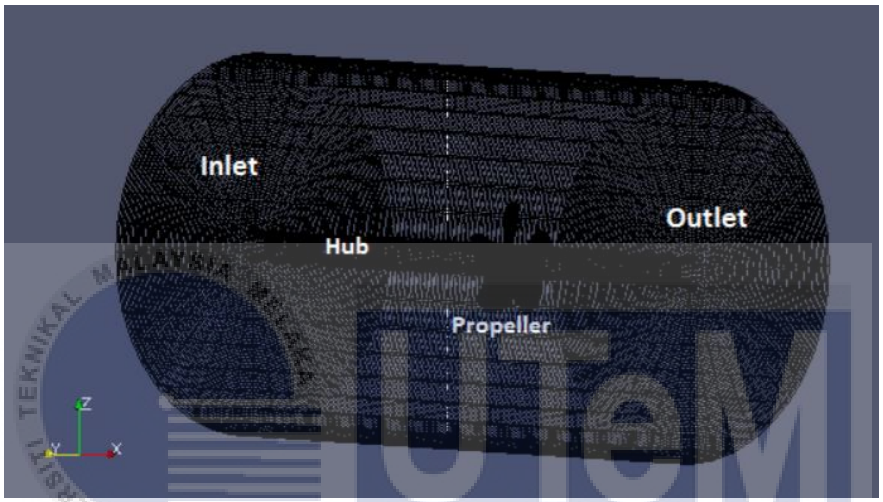


Figure 2.14: Example of the blockMesh in paraView (Source: Colley, 2012)

2.8 Summary

This chapter provided a literature review of propeller physical, type, size and number of propeller blade, design of ducted propeller, as well as CFD analysis of a propeller flow. The purpose of the CFD analysis was to choose the appropriate methods for this study. It was concluded that the CFD analysis is a sufficiently accurate and efficient method for analysing the velocity and power of propeller after drawn a 3D solid modelling in CAD. Type, size and number of propeller blade of this review was devoted to determining the best parameters. Twisted blade will be used because will give the equal thrust along the blade. So, there will be two angles needed in this type of blade which are inlet angle and outlet angle. While having more blades will generate more flow of water but also will increase the drag force. The number of blades must be suitable to use with AUV. The objective was to select an appropriate value of every parameters of blade to consider. Several possible values were discussed, and it was also concluded that shape of ducted should be able to provide better thrust and performance of propeller.

CHAPTER 3

METHODOLOGY

3.1 Introduction

This chapter will cover the details explanation of methodology that is being used to make this project complete and working well by using three major steps to implement project starting from planning, design and analyse based on System Development Life Cycle (SDLC). All the methods used for finding and analysing data regarding the project related to achieve the objective of the project that will accomplish a perfect result. A detailed discussion of the parameters of blade used for the purpose of propeller performance prediction will be given. It provides information on the propeller, what the parameters and how they were sampled and the reasons for this choice. The instrument that was used for data collection is also described and the procedures that were followed to carry out this study are included. This chapter also discuss the methods used to analyse the data. Lastly, the methods used to analyse the data are also discussed.

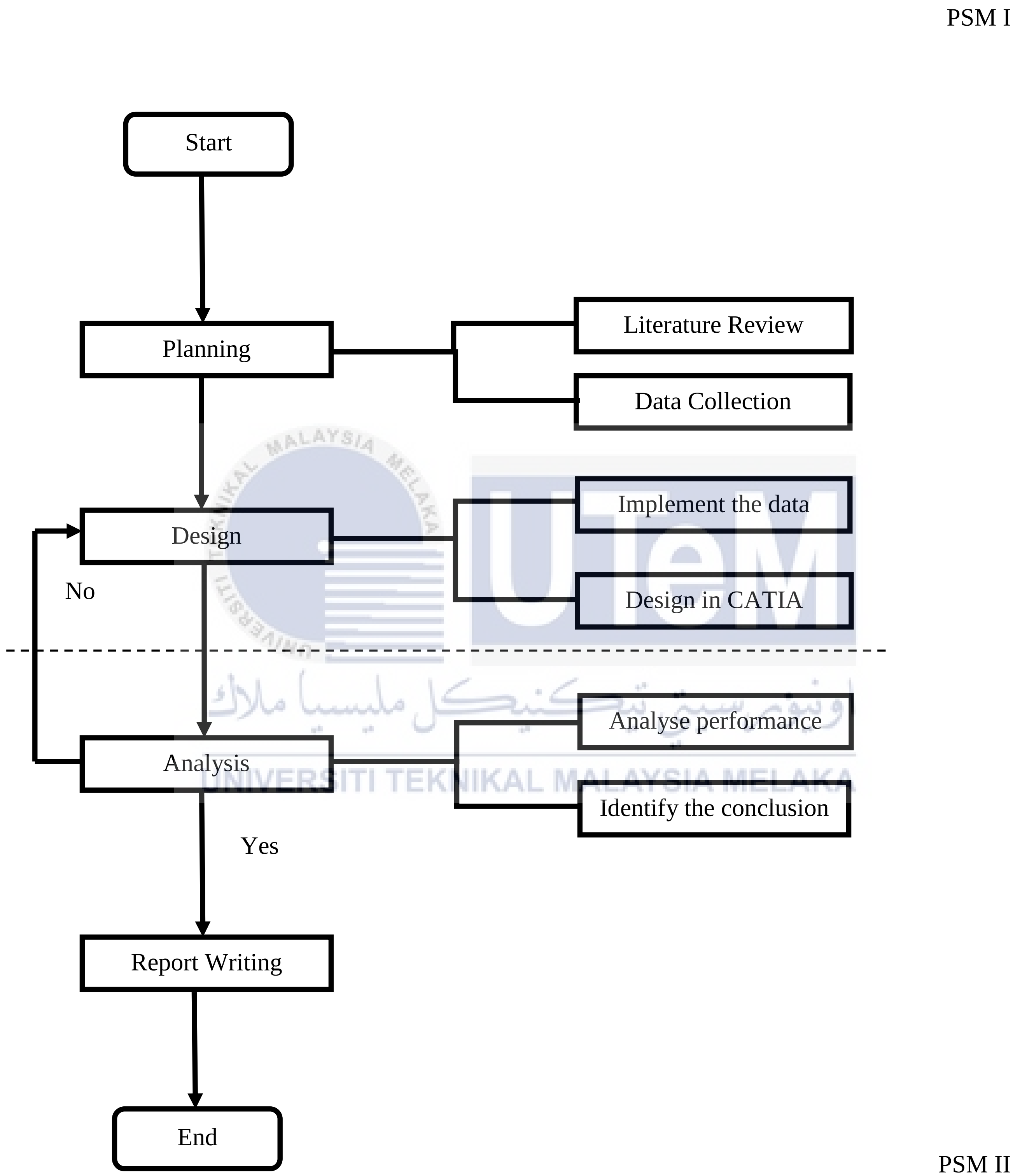


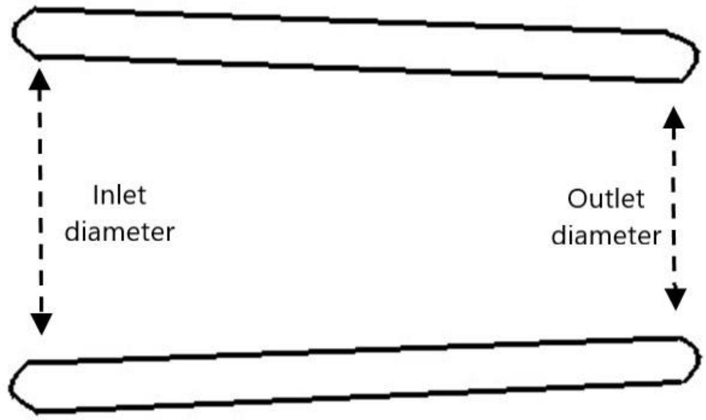
Figure 3.1: SLDC Phase and Steps of Methodology

3.2 Blade Parameters in Analysis

To identify all the information and requirement such as software, planning must be done in the proper manner. The planning phase have two main elements namely data collection and the requirements of software. The twisted shape of blade will be designed with parameters shown in Table 3.1. This data values gain from referring to existing project and a was the best values. The design of this ducted is the standard design used in propeller of AUV.

Table 3.1: Parameters of Blade

Parameters	**Details**
Type of Blade	Twisted Blade (Source: mecafluxHeliciel, 2018)
Number of Blades (N)	3 and 4 (Source: Aerotrash, 2012)
Blade Angle (θ)	Inlet angle: I_1= 60˚, I_2= 64˚ Outlet angle: 0_1= 16˚, 0_2= 20˚ (Source: Xiang, 2018)
Hub to Tips Ratio (radius of hub / radius of blade) of Blade (HTR)	HTR_1 : (2.0cm/5.2cm)=0.48 HTR_2 : (2.0cm/4.8cm)=0.53 (Source: BlueRobotics, 2018)
Nose Cone	Design 1: 2.5cm in length, 26.7mm of radius Design 2: 3.5cm in length, 47.0mm of radius (Source: Ranjan, 2015)

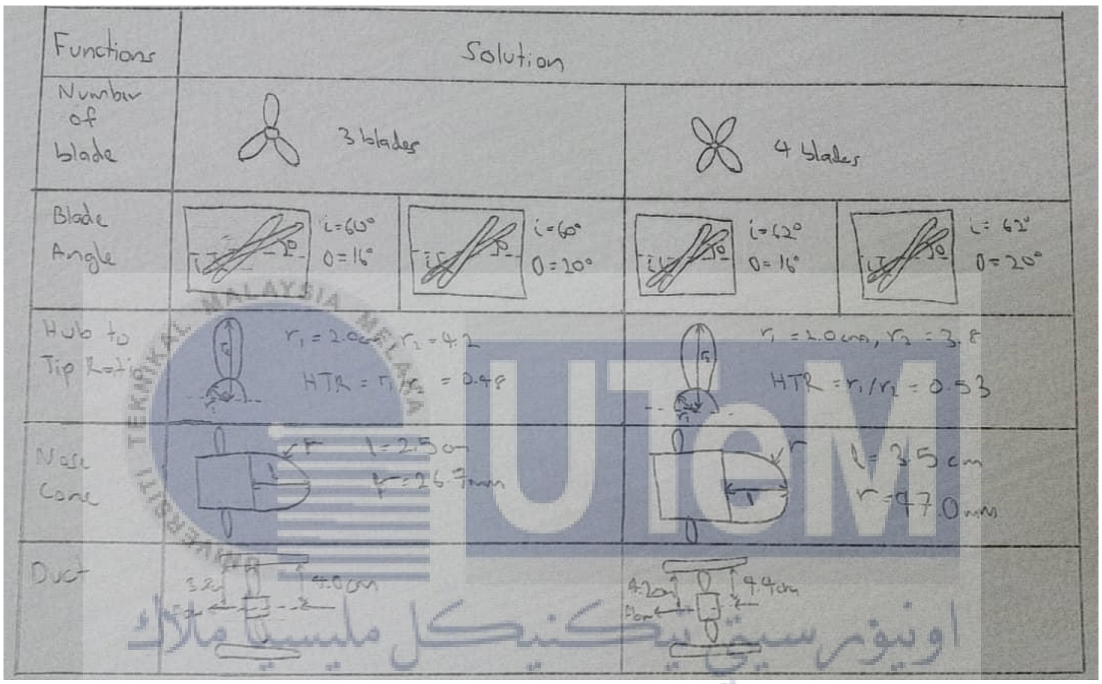
Duct	 For HTR_1 , Inlet radius, IR_1 = 5.8cm / Outlet radius, OR_1 = 5.0cm For HTR_2 , Inlet radius, IR_2 = 5.4cm / Outlet radius, OR_2 = 4.6cm (Source: BlueRobotics, 2018)

For software requirement, I have chosen CATIA to draw 3D solid modelling. CATIA is a software tool suite used primarily for 3D design. The software is used mainly to create 3D solid modelling for manufacturing of printed model. While analysis will be done using CFD simulation to gain the velocity of the water flow. That velocity will be using to calculate the power output of propeller performance.

3.3 Parameters Configurations by Matrix Multiplication

At this stage, planning about the project's resources and requirements, literature studies and schedule to get more information in this study. All the materials are collected from journal, texts book and research papers gathered from libraries and Internet. The morphological chart used to generate analytical and systematic ideas. Functions of the propeller are taken as a starting point and subfunctions which is parameters of blade can be established through a function analysis.

Table 3.2: Morphological chart



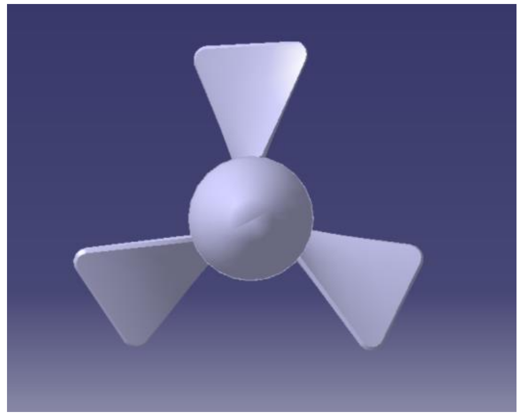
All data of blade parameters by using twisted blade with 3 and 4 blades. Twisted blade consists of inlet and outlet angle which is 64˚, 60˚ for inlet angle and 16˚, 20˚ for outlet angle. The size of propeller is measure using radius of blade and radius of hub. In this case, radius of hub is fix with 2.0cm while radius of blade is 4.8cm and 5.2cm. For nose cone, there are 2 design with different length and radius. First design is 2.5cm in length, 26.7mm in radius while second design is 3.5cm in length, 47.0mm in radius. By using matrix multiplication, all data was combined to get 32 designs in Table 3.2. Before proceeding with CFD analysis, the 3-dimension solid modelling of each data pairing have been designed and drawn in the computer aided design (CAD) software which is CATIA.

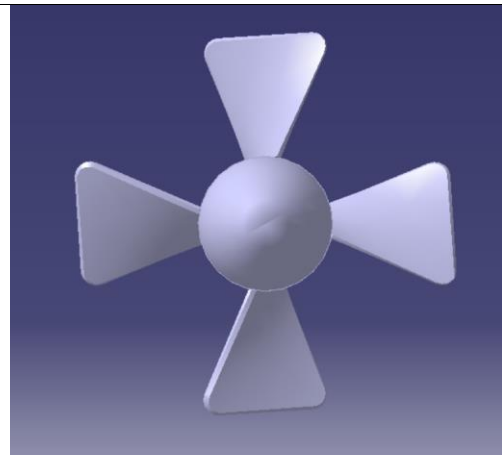
Table 3.3: Data Combination

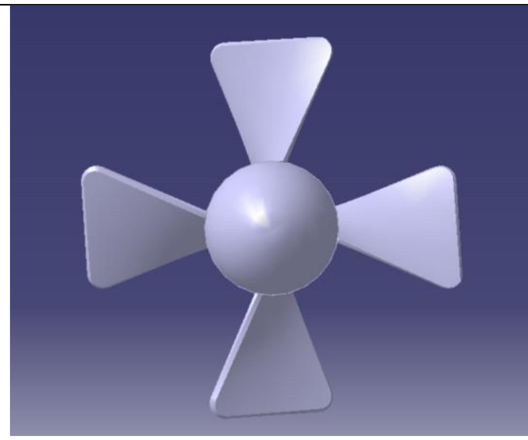
No. of Design	No. of Blade, N	Blade Angle, Θ	HTR	Nose Cone	Velocity, v(m/s^2)	Power, P(Watt)
Design 1	3	I_1, 0_1	HTR_1	Design 1	V_1	P_1
Design 2	3	I_1, 0_2	HTR_1	Design 1	V_2	P_2
Design 3	3	I_2, 0_1	HTR_1	Design 1	V_3	P_3
Design 4	3	I_2, 0_2	HTR_1	Design 1	V_4	P_4
Design 5	3	I_1, 0_1	HTR_2	Design 1	V_5	P_5
Design 6	3	I_1, 0_2	HTR_2	Design 1	V_6	P_6
Design 7	3	I_2, 0_1	HTR_2	Design 1	V_7	P_7
Design 8	3	I_2, 0_2	HTR_2	Design 1	V_8	P_8
Design 9	3	I_1, 0_1	HTR_1	Design 2	V_9	P_9
Design 10	3	I_1, 0_2	HTR_1	Design 2	V_{10}	P_{10}
Design 11	3	I_2, 0_1	HTR_1	Design 2	V_{11}	P_{11}
Design 12	3	I_2, 0_2	HTR_1	Design 2	V_{12}	P_{12}
Design 13	3	I_1, 0_1	HTR_2	Design 2	V_{13}	P_{13}
Design 14	3	I_1, 0_2	HTR_2	Design 2	V_{14}	P_{14}
Design 15	3	I_2, 0_1	HTR_2	Design 2	V_{15}	P_{15}
Design 16	3	I_2, 0_2	HTR_2	Design 2	V_{16}	P_{16}
Design 17	4	I_1, 0_1	HTR_1	Design 1	V_{17}	P_{17}

Design 18	4	I_1, 0_2	HTR_1	Design 1	V_{18}	P_{18}
Design 19	4	I_2, 0_1	HTR_1	Design 1	V_{19}	P_{19}
Design 20	4	I_2, 0_2	HTR_1	Design 1	V_{20}	P_{20}
Design 21	4	I_1, 0_1	HTR_2	Design 1	V_{21}	P_{21}
Design 22	4	I_1, 0_2	HTR_2	Design 1	V_{22}	P_{22}
Design 23	4	I_2, 0_1	HTR_2	Design 1	V_{23}	P_{23}
Design 24	4	I_2, 0_2	HTR_2	Design 1	V_{24}	P_{24}
Design 25	4	I_1, 0_1	HTR_1	Design 2	V_{25}	P_{25}
Design 26	4	I_1, 0_2	HTR_1	Design 2	V_{26}	P_{26}
Design 27	4	I_2, 0_1	HTR_1	Design 2	V_{27}	P_{27}
Design 28	4	I_2, 0_2	HTR_1	Design 2	V_{28}	P_{28}
Design 29	4	I_1, 0_1	HTR_2	Design 2	V_{29}	P_{29}
Design 30	4	I_1, 0_2	HTR_2	Design 2	V_{30}	P_{30}
Design 31	4	I_2, 0_1	HTR_2	Design 2	V_{31}	P_{31}
Design 32	4	I_2, 0_2	HTR_2	Design 2	V_{32}	P_{32}

Table 3.4: Examples of twisted propellers with different blade parameters.

Design 16 N= 3 $\Theta = I_2, 0_2$ HTR= HTR_2 Nose Cone= Design 2 Duct= Inlet diameter, IR_2 / Outlet diameter, OR_2	 (a) Front view  (b) Side view

Design 32	
N= 4 Θ= I_2, 0_2 HTR= HTR_2 Nose Cone= Design 2 Duct= Inlet diameter, IR_2 / Outlet diameter, OR_2	 (a) Front view  (b) Side view

Design 24 N= 4 Θ= I_2, 0_2 HTR= HTR_2 Nose Cone= Design 1 Duct= Inlet diameter, IR_2 / Outlet diameter, OR_2	 (a) Front view  (b) Side view

3.4 Computational Fluid Dynamics Analysis of Propeller Blades

To do the CFD analysis, all the 3D solid modelling from the CAD be exported to the CFD analysis software. Since this study is about low flow application for the propeller, therefore the speed rotation of the propeller has been set in the CFD analysis by using estimated value for water flow rate. Once when all initial condition needed under CFD has been fulfilled, the simulation and the end results can be obtained. The purpose of implementing CFD analysis is to determine the stream velocity for each propeller involved as well as to look at the flow patterns of the water at inlet, between the blades and at the outlet section. The values of the stream velocity for each propeller design will be substituted in the thrust power equation in order to obtain the most optimum results.

$$T = \left(\frac{\pi}{4}\right) . D^2 . \left(v + \frac{\Delta v}{2}\right) . \rho . \Delta v$$

and

$$P = T . v$$

Where,

T = Thrust (N)

D = Propeller diameter (m)

v = Velocity of incoming flow (m/s)

ρ = Density of fluid (kg/m^3)

Δv = Additional velocity, acceleration by propeller (m/s)

P = Thrust power

For propeller that working in fixed medium at constant speed, thrust will depends on the velocity of flow at the outlet of propeller and propeller diameter only. While thrust power defined as work done per unit time. Using the available thrust and velocity of incoming flow, the propulsive power can be calculated.

3.4.1 ANSYS 16.0 Software

Ansys software is a general-purpose software used to design products and semiconductors, as well as to create simulations that test a product's durability, temperature distribution, fluid movements, and electromagnetic properties. Therefore, Computational Fluid Dynamics (CFD) tools in ANSYS can be used to analyse and simulate tests on propeller blade on AUV in virtual environment before processing of prototypes of the products. In this project, ANSYS Fluent platform is used to carry out an external flow of the fluid motion around the propeller. Then, the performance of the propeller can be determined, and the stream velocity of the propeller blade can be analysed. Fluent software contains the broad, physical modelling capabilities for model flow, turbulence, heat transfer and reactions for industrial applications. These range can be used to generate the fluid flow over a propeller that also offers highly scalable, high-performance computing (HPC) to help solve complex, large-model computational fluid dynamics (CFD) simulations quickly and cost-effectively.

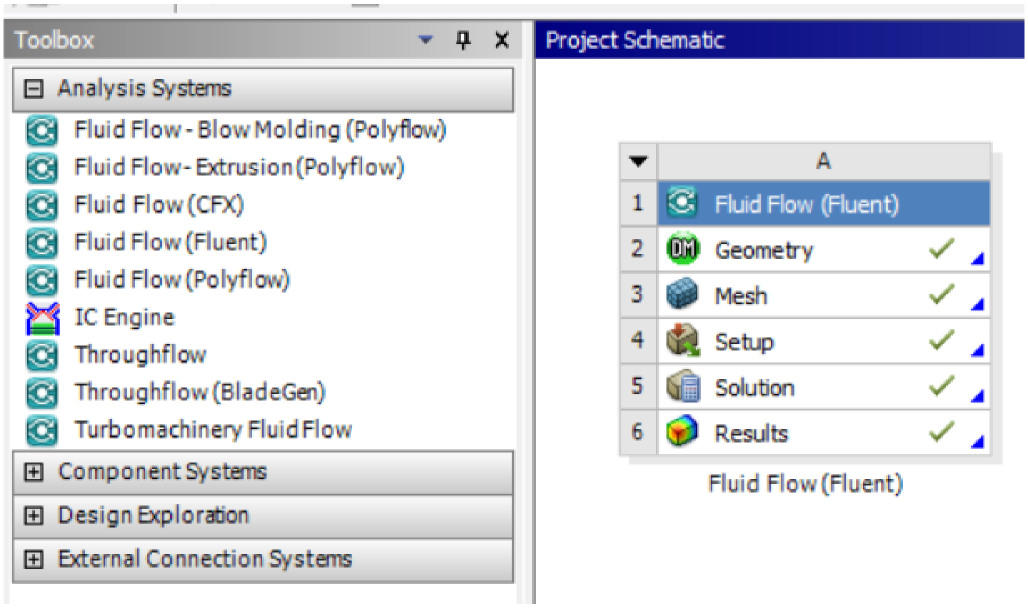


Figure 3.2: ANSYS Fluent platform

3.4.2 Pre-processing

In ANSYS Fluent platform, the first step is the preparation of the model by import the propeller design from CAD using Standard for the Exchange of Product (STP). STP is a file extension for a 3-D graphic file used by CAD software. The simulations are performed inside a fluid tunnel in the shape of a cylinder of 60 mm width in diameter and 60 mm length using the Enclosure tools along the y-axis. The surfaces of the cylinder need to be renamed with inlet, wall and outlet for boundary conditions setup. The propeller and nose cone inside the cylinder are combine using the Boolean tools of unite because these two parts will be move in rotation. The second Boolean tools is used to determine which part will be use in the simulation which the combination of propeller and nose cone with duct acted as a tool bodies, and the cylinder of fluid flow act as a target body. The tool bodies need to preserve for the cell zone conditions setup.

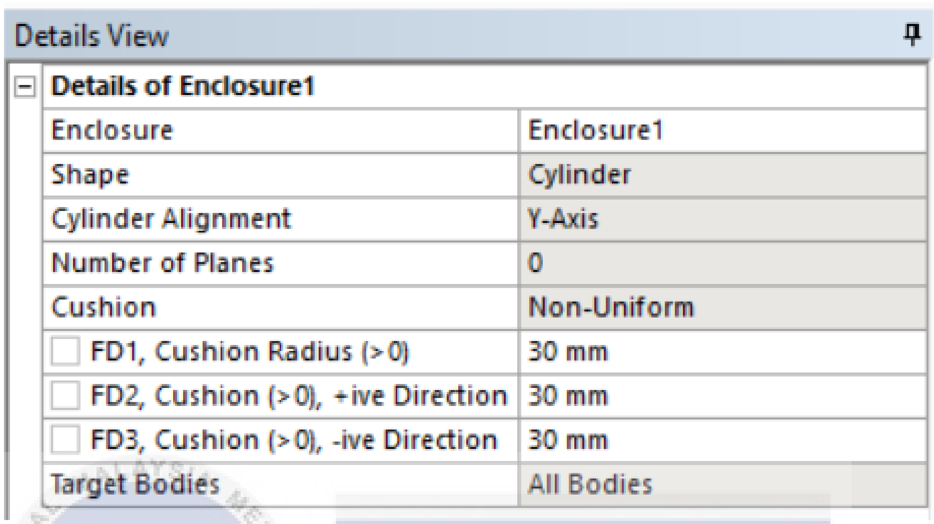


Figure 3.3: Details of Enclosure.

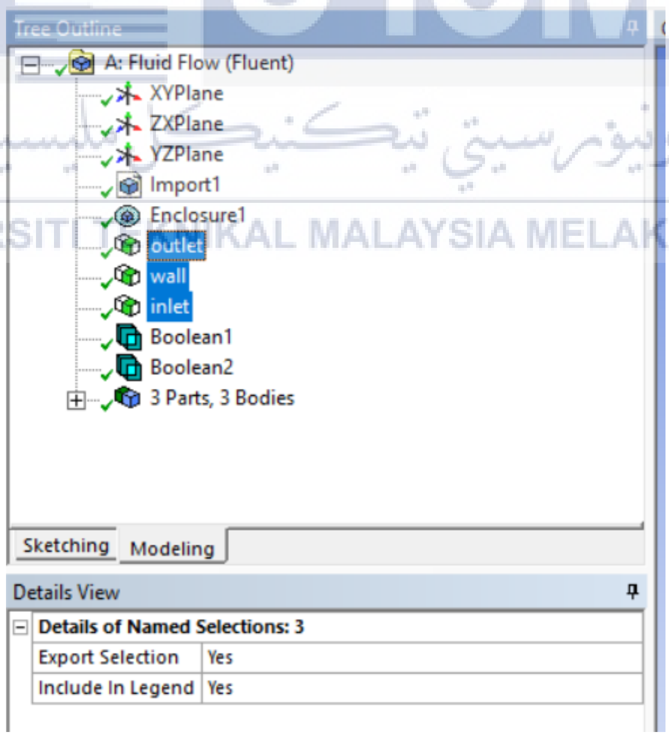


Figure 3.4: Details of Named Selections for Cylinder Surfaces.

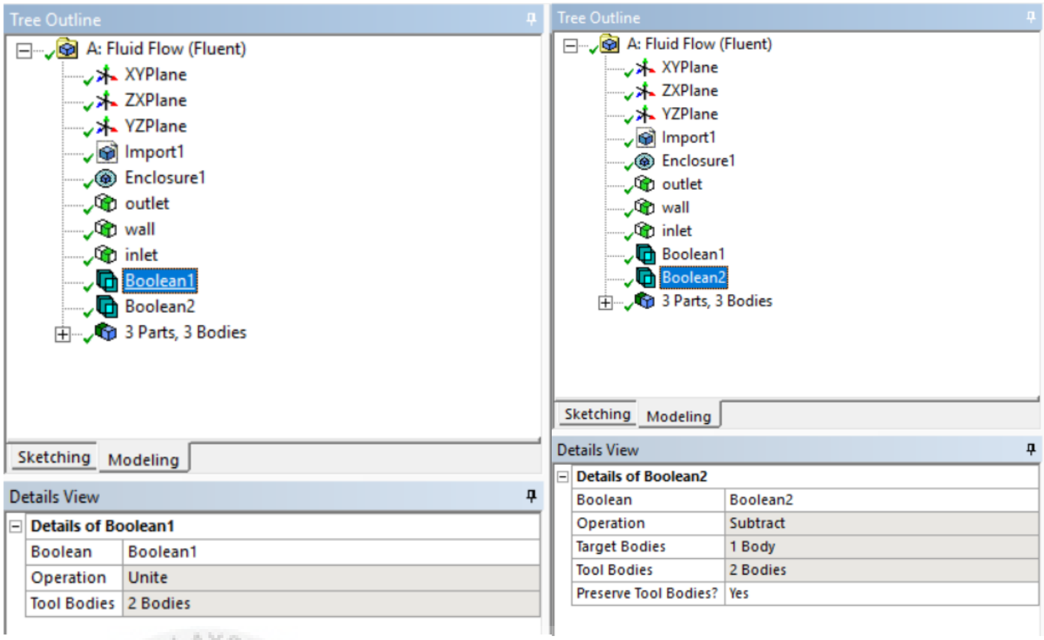


Figure 3.5: Details of Boolean 1 and Boolean 2.

In meshing setup, the first step is to delete all the contacts in the connections tab to avoid manifold in the model. Next is the Proximity and Curvature option is chosen in the Advanced Size Function tab in Mesh details to control the number of mesh elements employed in the gaps between two geometric entities and to specify the minimum number of elements layers created in regions that constitute as gaps in the model. The Proximity and Curvature also been selected because the maximum tetrahedral size defined on surface parts is larger than a geometric entity in the specified part. Then, generate the mesh and the final output of meshing process with statistics of nodes and elements for mesh will be auto generated.

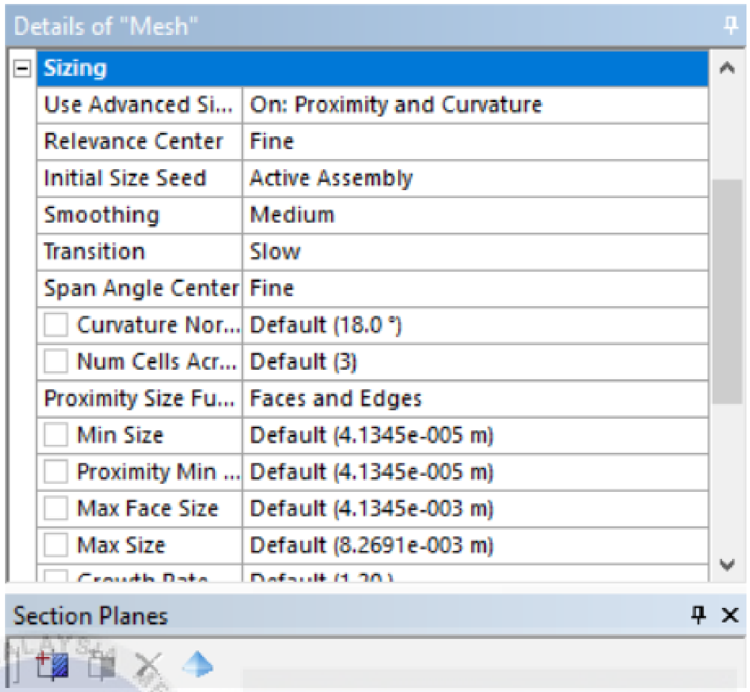


Figure 3.6: Setup of Mesh.

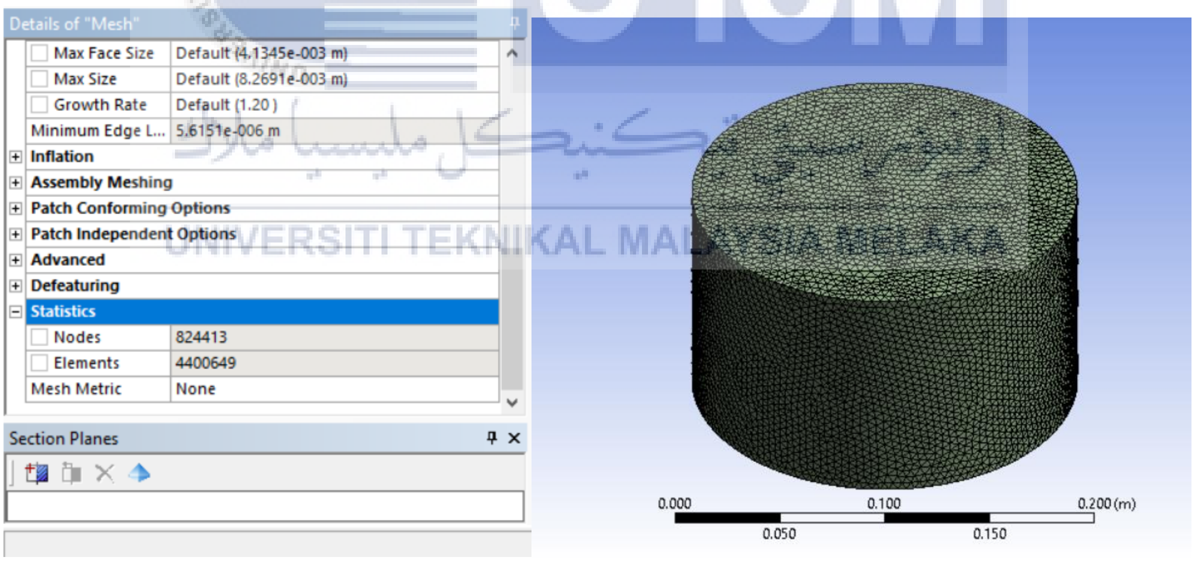


Figure 3.7: Final output in the meshing process.

3.4.3 Processing

For General Setup, the solver with Pressure Based type been selected because the non-premixed flow model is available only with this solver. The Time is set to be Steady for frame motion of propeller while Velocity Formula is absolute based on the type of solver.

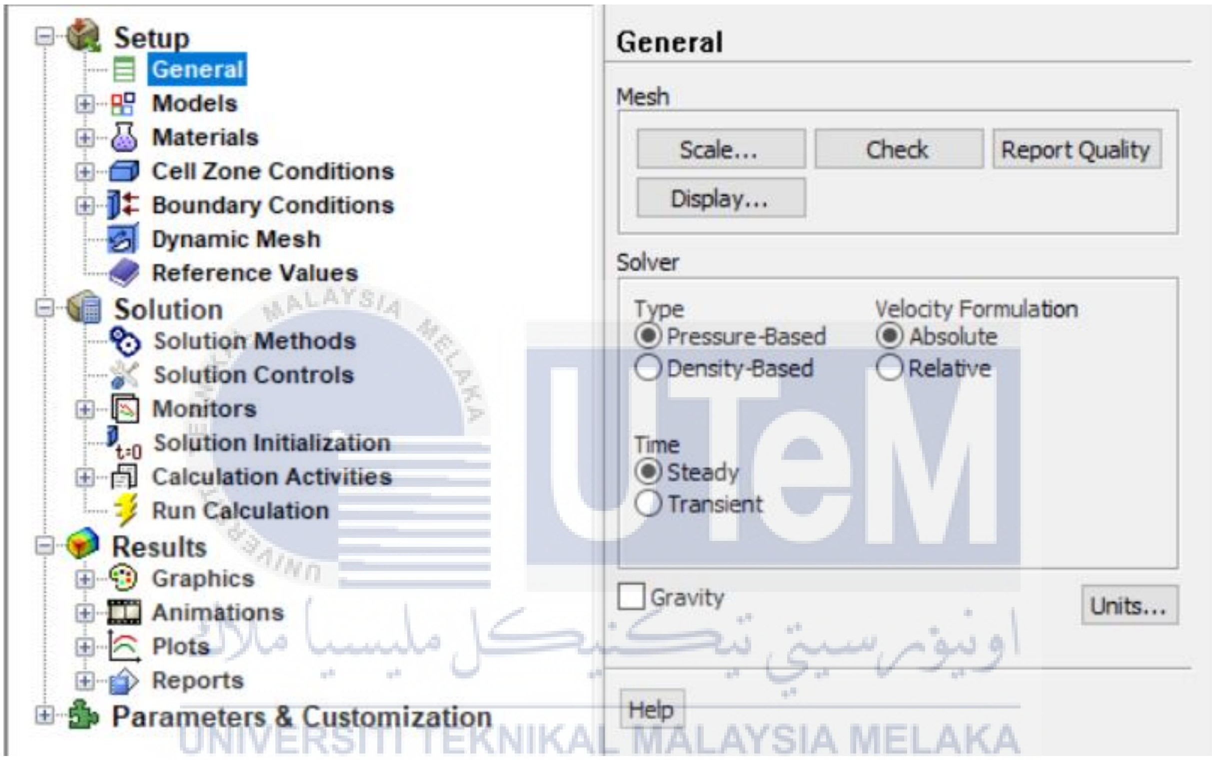


Figure 3.8: General Condition in Setup.

In materials setup, the water-liquid will be used as the fluid flow. So, the new material needs to be created from the Fluent Database.

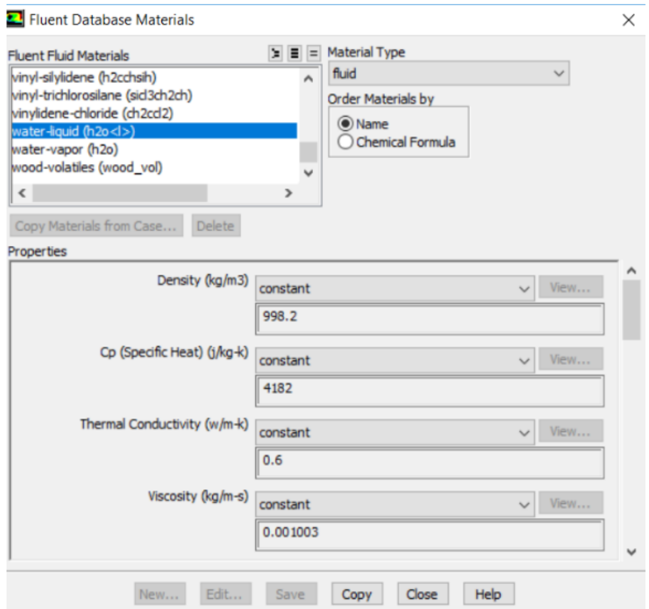


Figure 3.9: Materials Setup.

The Cell Zone Conditions of duct, propeller and cylinder zone need to be setup. The material that will be used for duct and propeller zone is aluminium because this material is a currently been used in the simulation and it is also can be used for these zones without affected the output of simulation. In propeller zone, the frame motion with rotational velocity of -600 RPM and rotate at y-axis. The negative sign of rotational velocity indicates to the rotation in counterclockwise in y-axis. While in solid (cylinder) zone, the material is referring to fluid flow which is water-liquid. The different of frame motion and mesh motion are the frame motion is a coordinate system that moves during simulation, e g. rotation. Instead to put all mesh and paddles to rotates, the coordinate system can do it. Mesh motion is more general and can deal with rotation, shape changes or normal displacements of wall that changes the domain's volume, e.g. gear pumps and car engines cylinders.

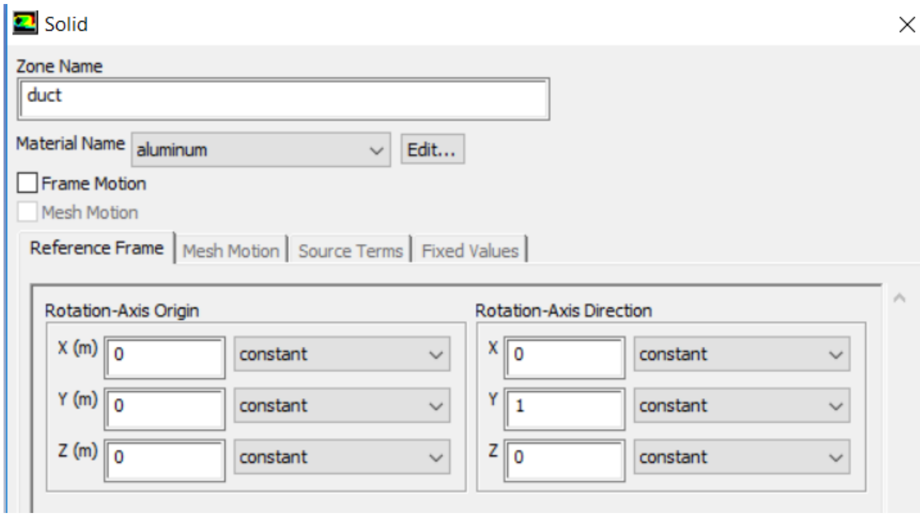


Figure 3.10: Duct Zone Setup in Cell Zone Conditions.

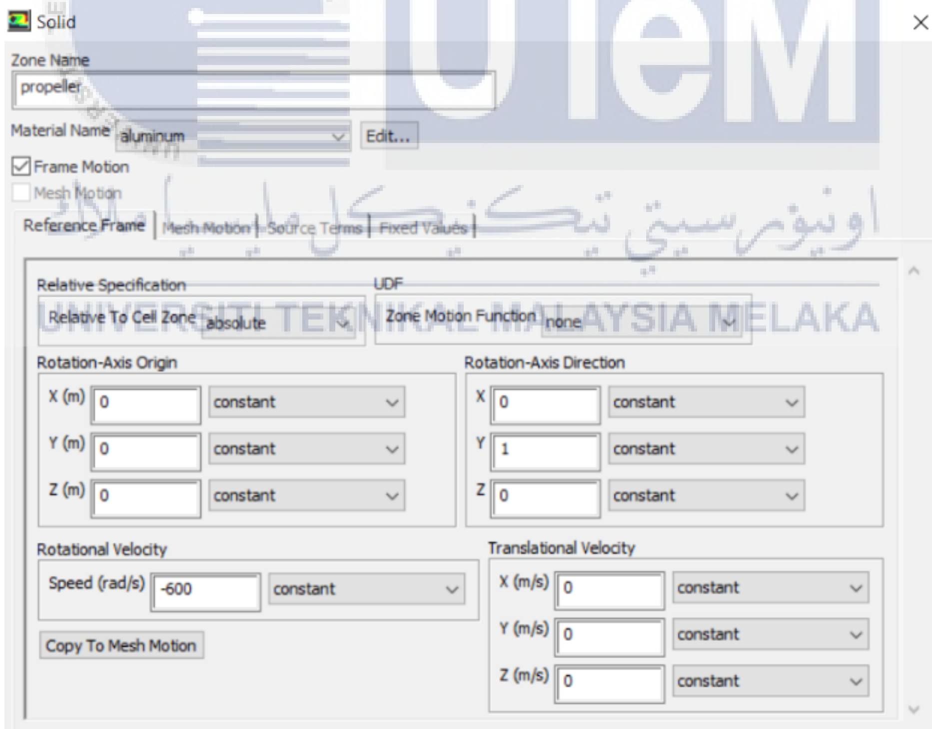


Figure 3.11: Propeller Zone Setup in Cell Zone Conditions.

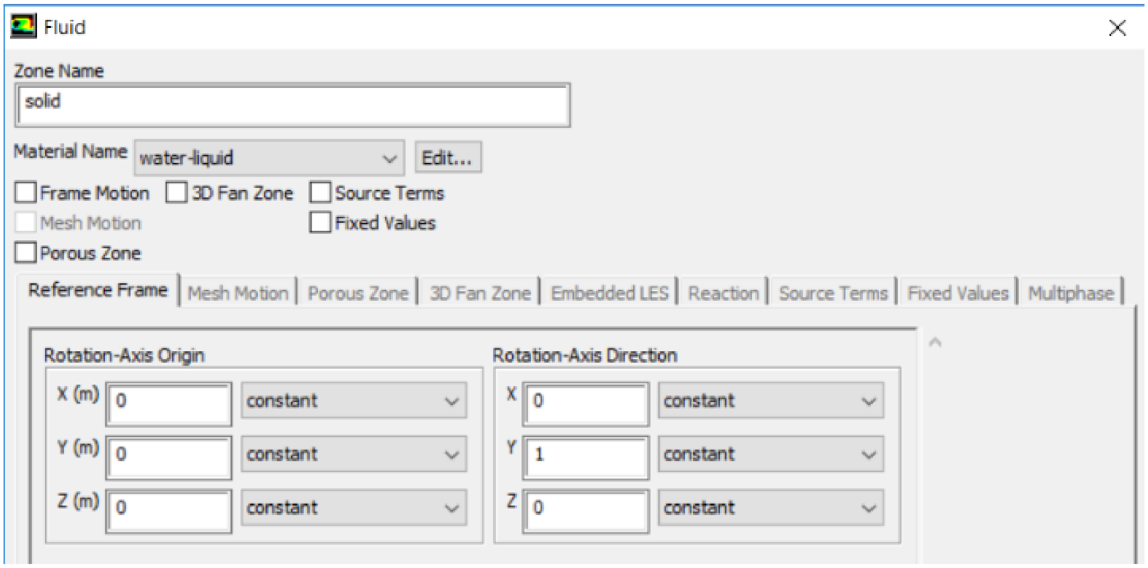


Figure 3.12: Solid Zone Setup in Cell Zone Conditions.

In Boundary Conditions Setup, the inlet zone act as a velocity inlet with the fluid flow velocity of 0.3m/s (Kobeissi, 2017) in y-axis direction which is suitable for water flow in the pool that used in AUV competition. While the outlet zone is a pressure outlet with the pressure of 1 atm. This pressure is referring to the pressure inside the pool. There are just a little different of pressure at the surface and bottom of the pool that can be ignore.

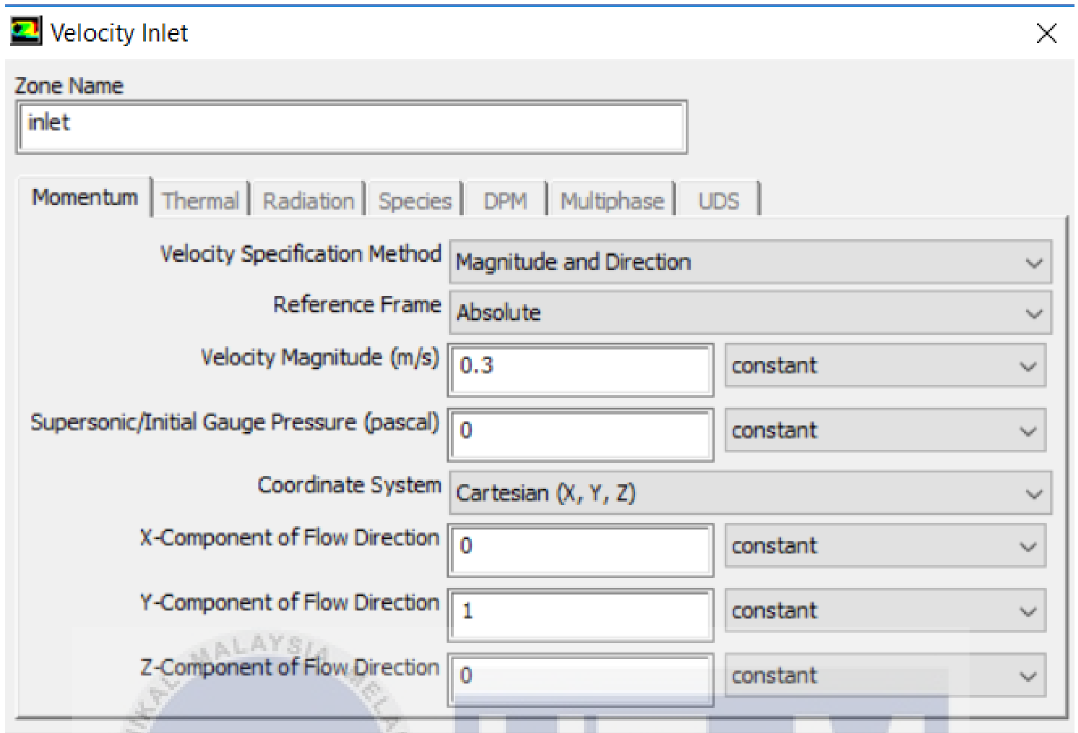


Figure 3.13: Inlet Zone Setup in Boundary Conditions.

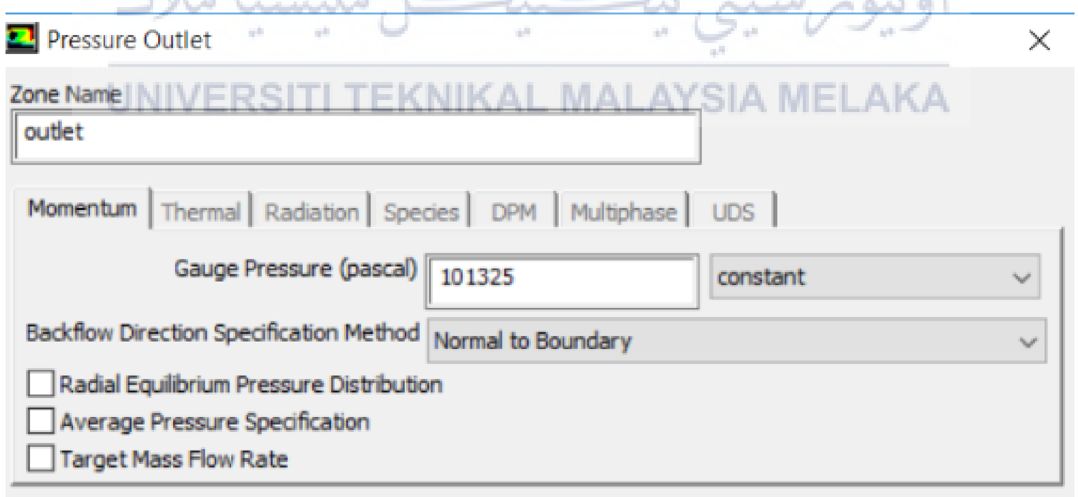


Figure 3.14: Outlet Zone Setup in Boundary Conditions.

The reference values need to be computed from inlet and outlet to get the density of water, pressure and inlet velocity.

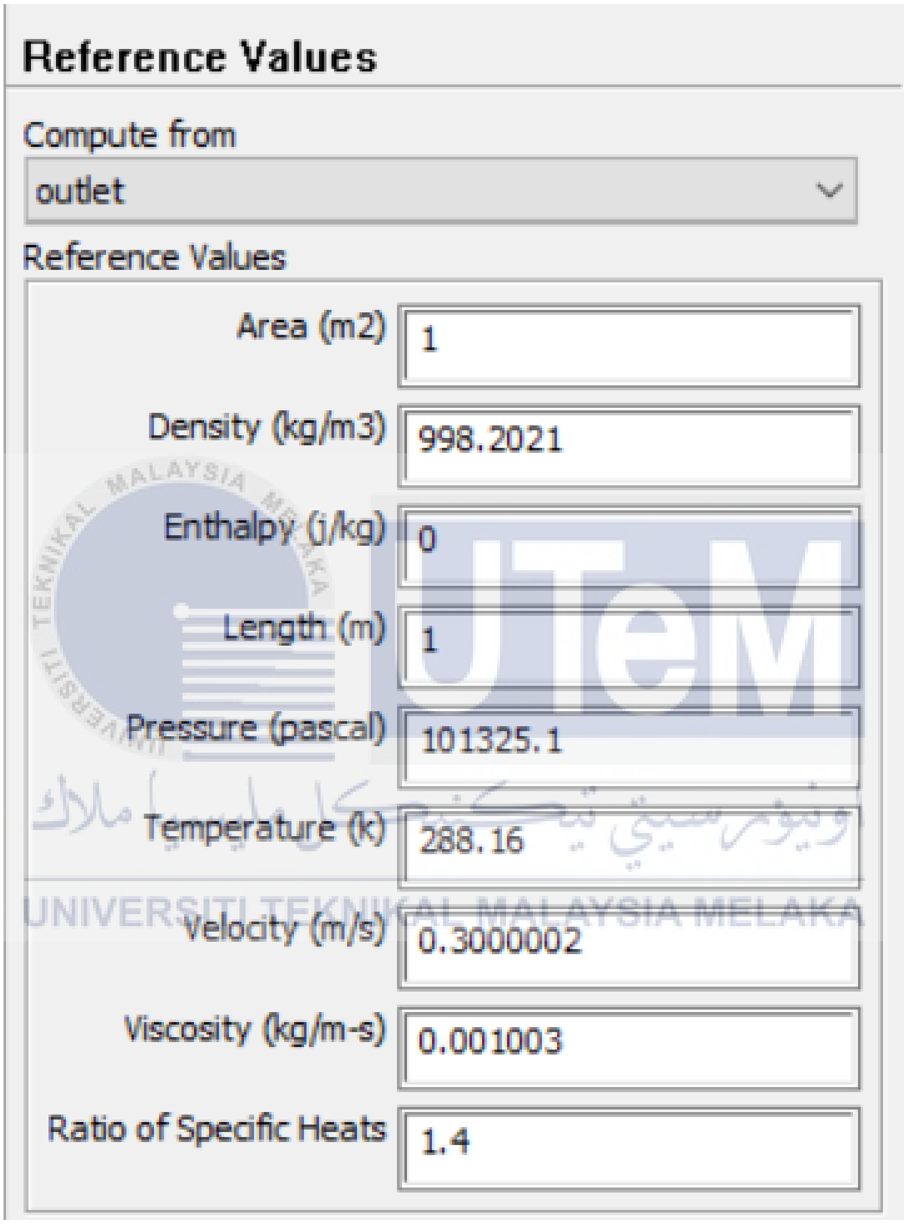


Figure 3.15: References Values Setup.

In Solution Initialization, the standard initialization was selected because the initial values of gauge pressure and inlet velocity need to be computed from inlet and outlet. Then, this setup needs to be initialized before proceeding to run calculation.

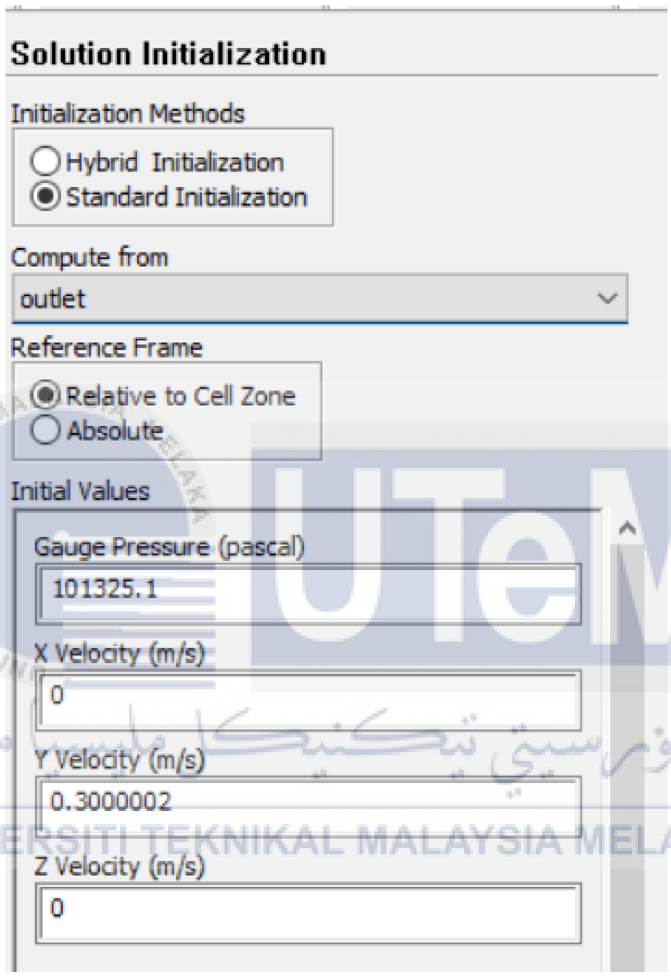


Figure 3.16: Solution Initialization Setup.

In Run Calculation Setup, the number of iterations is set to 100 iterations. Iterations mean the number of times the same equations will performed on the different boundary values. When define a system of simulation through the stand-alone system (CFD), a specific mathematical problem on the specific geometry have been define. Further, it is divided into the mesh, the mesh is comprised of nodes and element. On the first iteration, the entire problem will be executed once on given mesh. Then, the second iterations on the same problem with different mesh will give more proper results than the first iteration. Then this loop goes on until the last iteration.

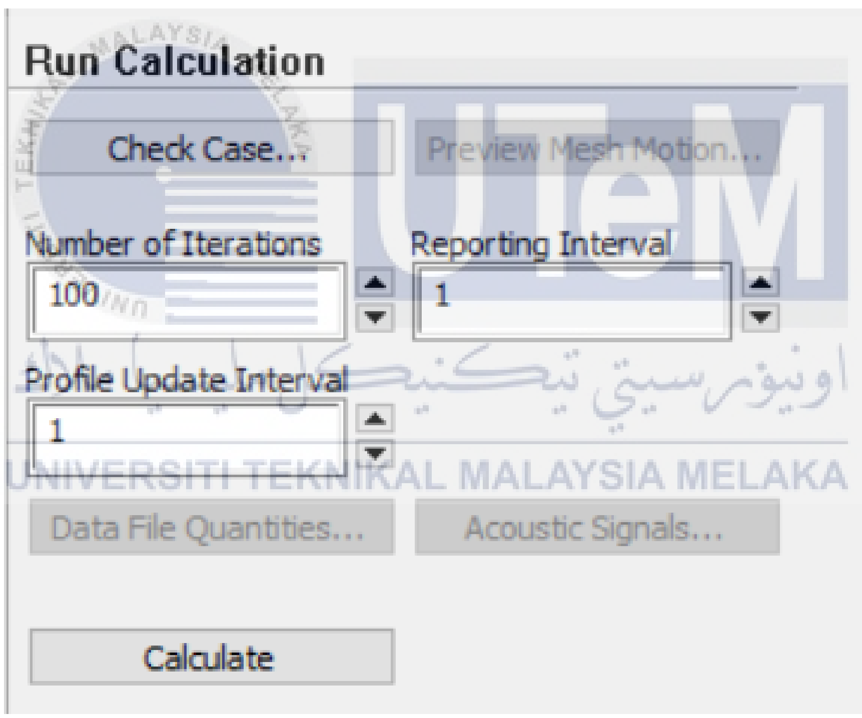


Figure 3.17: Run Calculation Setup.

3.4.4 Post-processing

After Run Calculation is finish, the result of the simulation can be view in the Result. By apply the streamline of velocity from the inlet with 500 points, the flow patterns or the streamline of the water from the inlet, between the propeller blades and at the outlet section can be view and analyse.

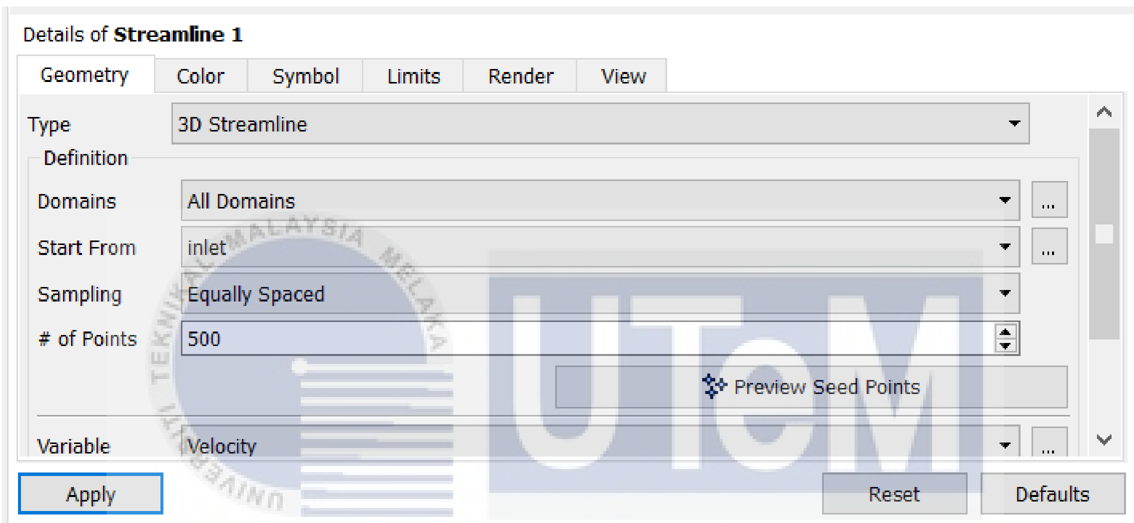


Figure 3.18: Details of Velocity Streamline Setup.

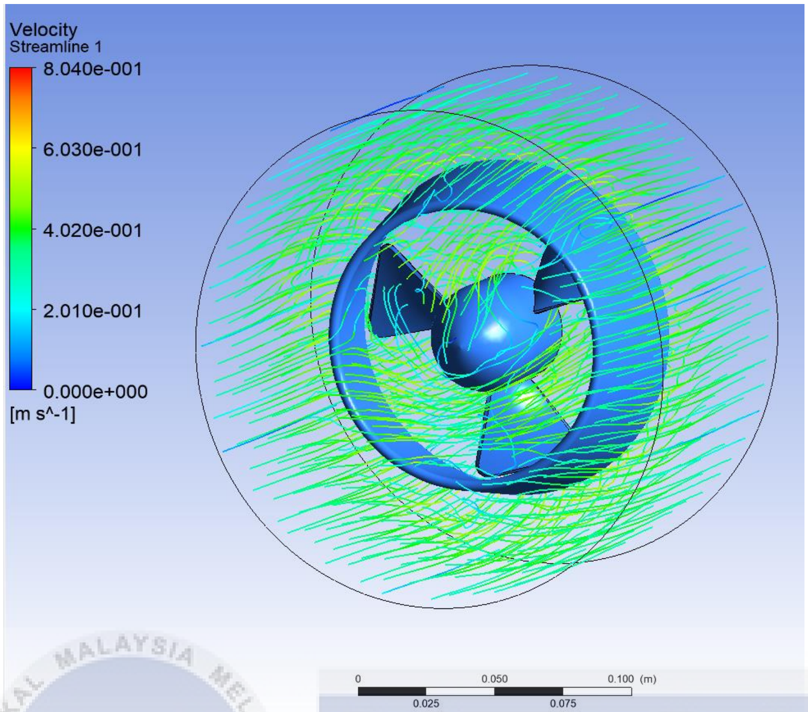


Figure 3.19: Pattern of Velocity Streamline.

Then, apply the velocity contour at the middle of propeller by using plane to see the graphic in combination of constant magnitude of velocity.

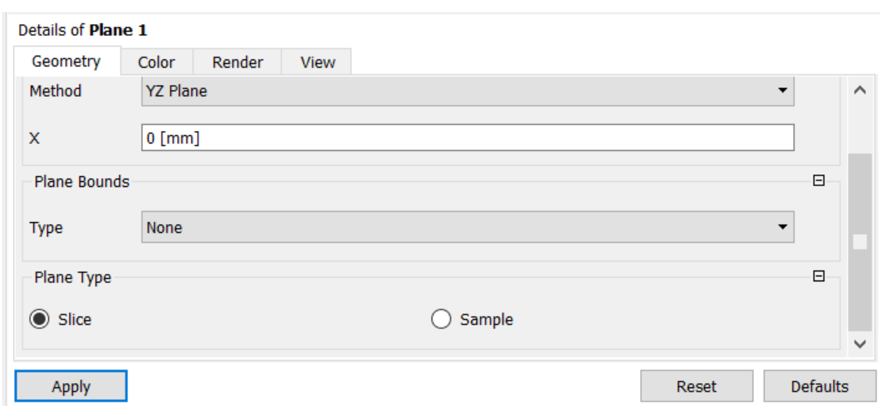


Figure 3.20: Setup of Plane.

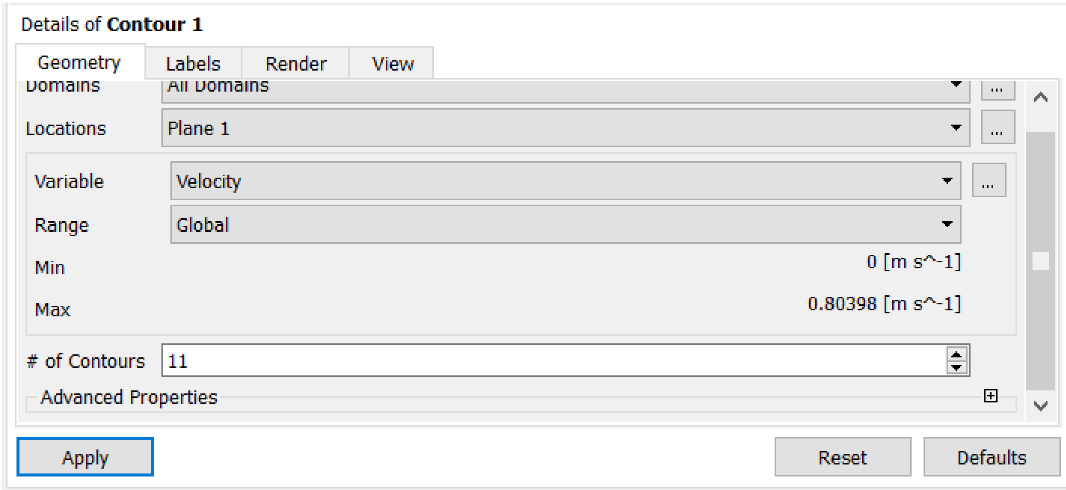


Figure 3.21: Setup of Velocity Contour.

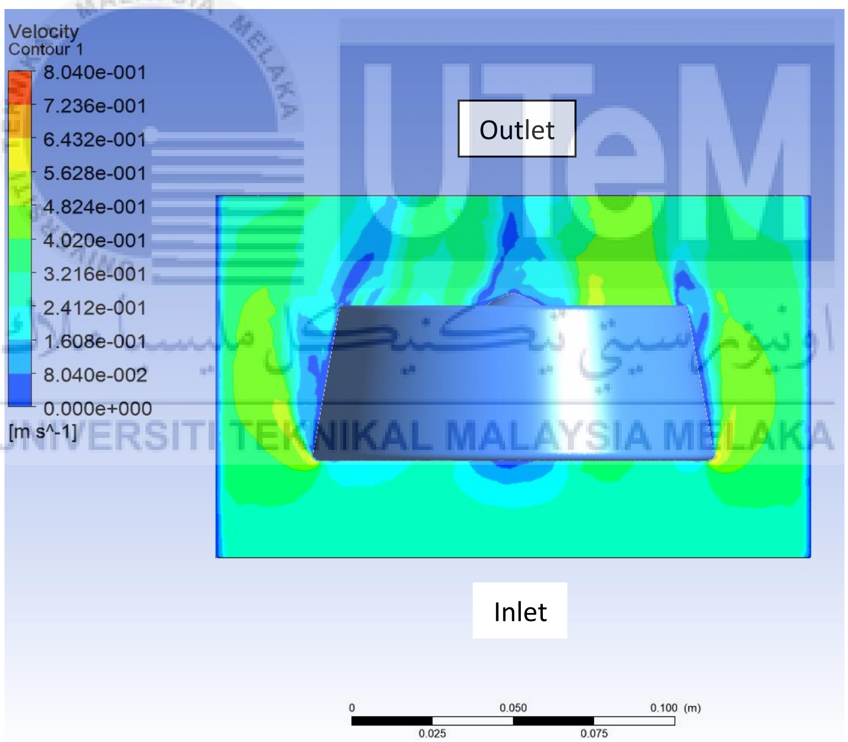


Figure 3.22: Output of Velocity Contour.

Finally, calculate the stream velocity at the outlet by using the Function Calculator. Set the function with Maximum Value of Velocity of v on Outlet. The stream velocity is located at the outlet in y-axis.

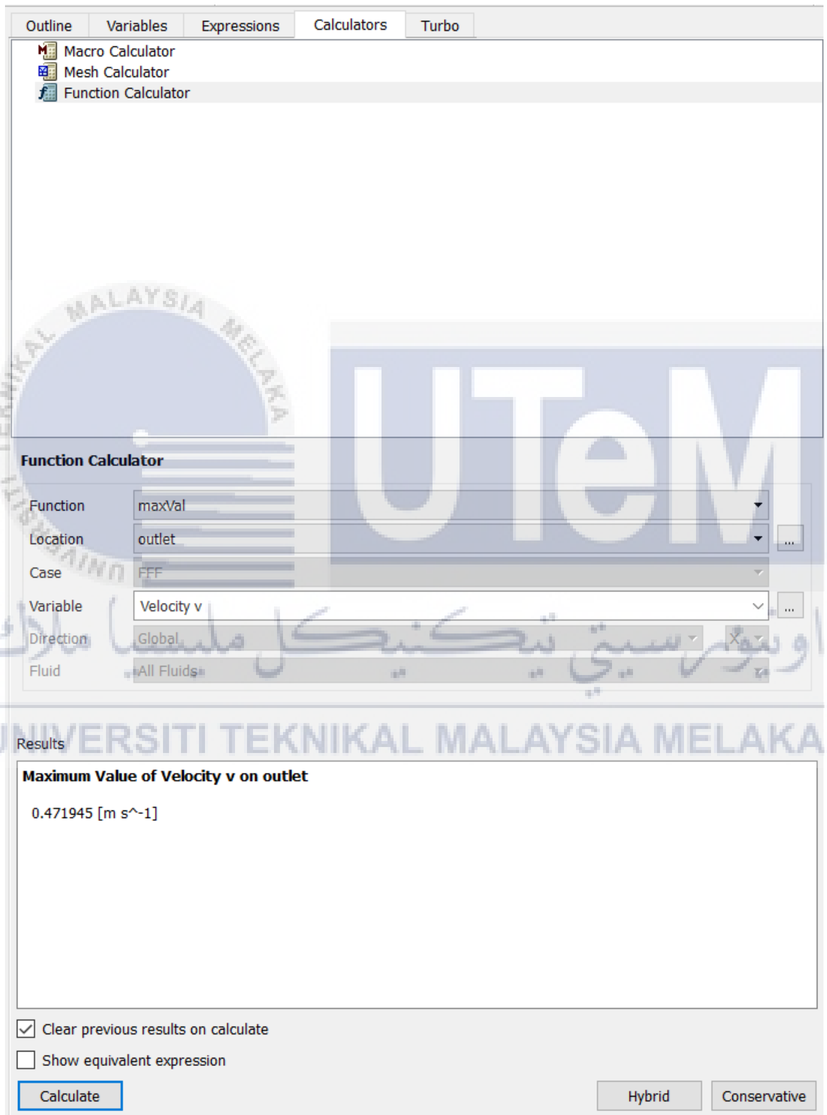
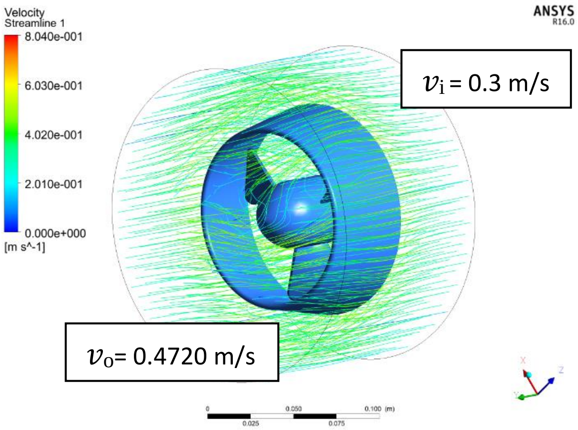
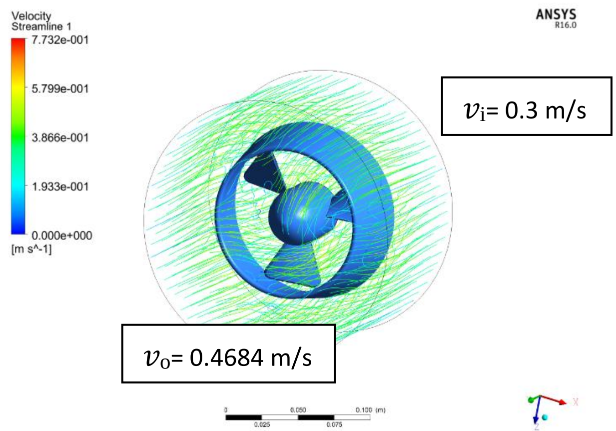


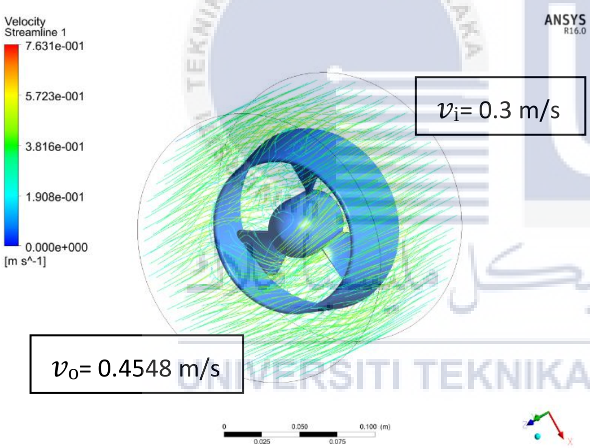
Figure 3.23: Calculation of Fluid Velocity Outlet.



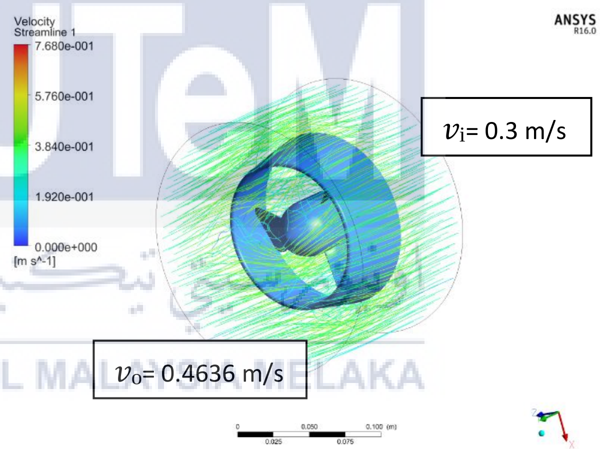
(a) CFD image of Design 1



(b) CFD image of Design 2



(c) CFD image of Design 3



(d) CFD image of Design 4

Figure 3.24: Examples of CFD analysis images with stream velocity.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 RESULTS

All the result data for each design in CFD analysis has been collected and compiled in the Table 4.1. The value of the stream velocity for each configuration of related parameter from the CFD analysis will be substituted into the equation of thrust power.

Thrust power equation,

$$T = \left(\frac{\pi}{4}\right).D^2.\left(v + \frac{\Delta v}{2}\right).\rho.\Delta v$$

and

$$P = T.v$$

Where,

Thrust, T (N)

Propeller diameter, D (m) = 0.104m for HTR_1 and 0.096m for HTR_2

Velocity of incoming flow (velocity inlet), v or v_i (m/s) = 0.3 m/s

Density of fluid, ρ_{water} (kg/m^3) = 998.2 kg/m^3

Additional velocity, acceleration by propeller, Δv (m/s) = ($v_o - v$)

Thrust power, P (W)

4.1.1 Thrust Power

Table 4.1: Data of Thrust Power for all designs.

No. of Design	No. of Blade, N	Blade Angle, θ (˚)	Hub to Tip Ratio (hub radius/propeller radius), HTR	Nose Cone, (m) Length=L Radius=R	Thrust Power, P (Watt)
Design 1	3	$I_1 = 60$ $0_1 = 16$	0.48	L = 0.025 R = 0.0267	0.2657
Design 2	3	$I_1 = 60$ $0_2 = 20$	0.48	L = 0.025 R = 0.0267	0.2569
Design 3	3	$I_2 = 64$ $0_1 = 16$	0.48	L = 0.025 R = 0.0267	0.2253
Design 4	3	$I_2 = 64$ $0_2 = 20$	0.48	L = 0.025 R = 0.0267	0.2456
Design 5	3	$I_1 = 60$ $0_1 = 16$	0.53	L = 0.025 R = 0.0267	0.2159
Design 6	3	$I_1 = 60$ $0_2 = 20$	0.53	L = 0.025 R = 0.0267	0.2181

Design 7	3	$I_2 = 64$ $0_1 = 16$	0.53	L = 0.025 R = 0.0267	0.2110
Design 8	3	$I_2 = 64$ $0_2 = 20$	0.53	L = 0.025 R = 0.0267	0.2072
Design 9	3	$I_1 = 60$ $0_1 = 16$	0.48	L = 0.035 R = 0.0470	0.2214
Design 10	3	$I_1 = 60$ $0_2 = 20$	0.48	L = 0.035 R = 0.0470	0.2596
Design 11	3	$I_2 = 64$ $0_1 = 16$	0.48	L = 0.035 R = 0.0470	0.2133
Design 12	3	$I_2 = 64$ $0_2 = 20$	0.48	L = 0.035 R = 0.0470	0.2276
Design 13	3	$I_1 = 60$ $0_1 = 16$	0.53	L = 0.035 R = 0.0470	0.2355
Design 14	3	$I_1 = 60$ $0_2 = 20$	0.53	L = 0.035 R = 0.0470	0.2126
Design 15	3	$I_2 = 64$ $0_1 = 16$	0.53	L = 0.035 R = 0.0470	0.1937
Design 16	3	$I_2 = 64$ $0_2 = 20$	0.53	L = 0.035 R = 0.0470	0.1997
Design 17	4	$I_1 = 60$ $0_1 = 16$	0.48	L = 0.025 R = 0.0267	0.3178
Design 18	4	$I_1 = 60$ $0_2 = 20$	0.48	L = 0.025 R = 0.0267	0.3217

Design 19	4	$I_2 = 64$ $0_1 = 16$	0.48	L = 0.025 R = 0.0267	0.2883
Design 20	4	$I_2 = 64$ $0_2 = 20$	0.48	L = 0.025 R = 0.0267	0.2898
Design 21	4	$I_1 = 60$ $0_1 = 16$	0.53	L = 0.025 R = 0.0267	0.2517
Design 22	4	$I_1 = 60$ $0_2 = 20$	0.53	L = 0.025 R = 0.0267	0.2768
Design 23	4	$I_2 = 64$ $0_1 = 16$	0.53	L = 0.025 R = 0.0267	0.2632
Design 24	4	$I_2 = 64$ $0_2 = 20$	0.53	L = 0.025 R = 0.0267	0.2822
Design 25	4	$I_1 = 60$ $0_1 = 16$	0.48	L = 0.035 R = 0.0470	0.3065
Design 26	4	$I_1 = 60$ $0_2 = 20$	0.48	L = 0.035 R = 0.0470	0.3817
Design 27	4	$I_2 = 64$ $0_1 = 16$	0.48	L = 0.035 R = 0.0470	0.2946
Design 28	4	$I_2 = 64$ $0_2 = 20$	0.48	L = 0.035 R = 0.0470	0.2835
Design 29	4	$I_1 = 60$ $0_1 = 16$	0.53	L = 0.035 R = 0.0470	0.2798
Design 30	4	$I_1 = 60$ $0_2 = 20$	0.53	L = 0.035 R = 0.0470	0.2698

Design 31	4	$I_2 = 64$ $0_1 = 16$	0.53	L = 0.035 R = 0.0470	0.2705
Design 32	4	$I_2 = 64$ $0_2 = 20$	0.53	L = 0.035 R = 0.0470	0.2594

4.1.2 Graph of Thrust Power

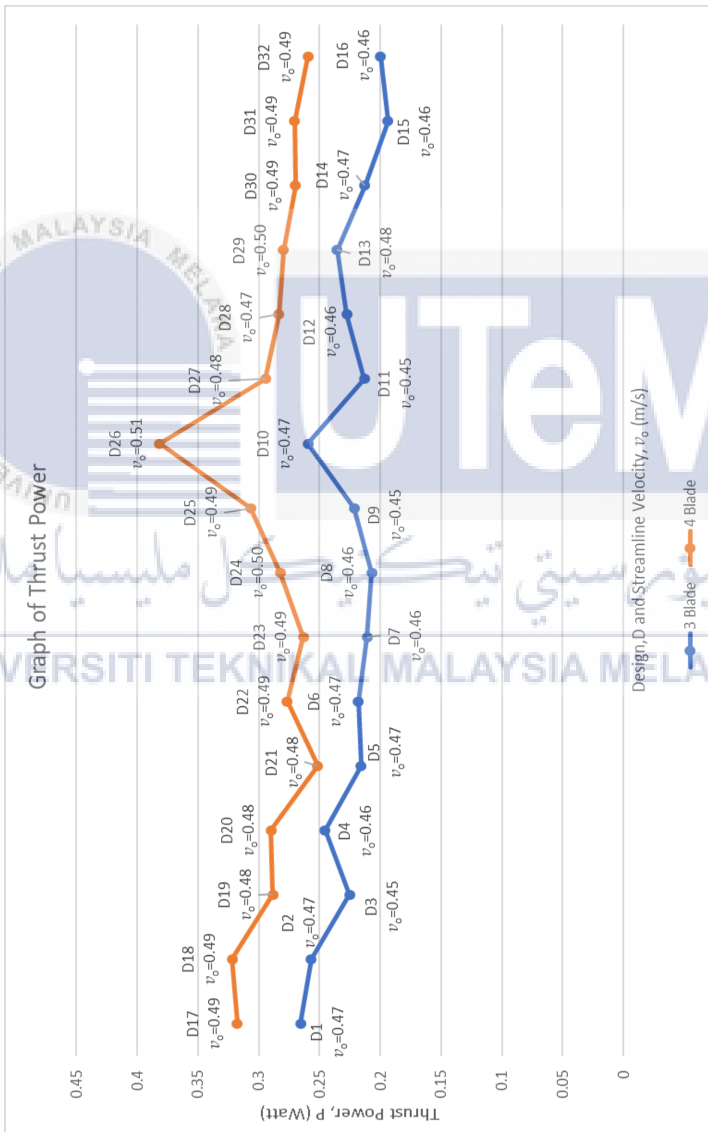


Figure 4.1: Graph of P against D and v_o.

4.2 DISCUSSION

Based on the results in Table 4.1 and graph in Figure 4.1, the parameters configuration of propeller blade in Design 26 indicated the highest value of thrust power, 0.3817W compared to others design. Design 26 are using 4 blades, blade angle with inlet angle of 60˚ and outlet angle of 20˚, hub to tip ratio of 0.48 and with nose cone of 0.035m in length and 0.0470m in radius. This show that design with smaller inlet angle, bigger outlet angle, smaller hub to tip ratio or bigger propeller blade diameter and longer nose cone can generate more stream velocity and thrust power. The longer nose cone will reduce drag force and make the stream flow more ease through the blade. while twisted blade with smaller inlet angle and bigger outlet angle will provide more equivalent thrust along the blade that will help to increase the stream flow velocity. The bigger diameter of propeller blade obviously will increase the velocity of stream flow by give more force but at the same time it is increase the friction and contact between the surface of propeller blade and flow of streamline. The overall results are randomly for each design which mean it cannot be conclude mathematically.

These results may influence by some conditions such as water hammer and friction between water flow and surface of propeller blade. Water hammer occur when the motion of water flow is forced to stop or force to change it direction suddenly. It will reduce the speed of water flow and change the momentum of water flow until the pressure in front of propeller increase. While the friction between the water flow and surface of propeller blade will depend on the size or length and the angle of propeller blade. The higher the size or length of propeller blade, the higher the friction between the water flow and surface of propeller blade. It will decrease the speed of water flow or stream velocity.

While based on Figure 4.1, the graph of thrust power for Design 1 until Design 32 with separated line for 3 blades and 4 blades presents both patterns or trends of thrust power for 3 blades and 4 blade from Design 1 to Design 32 are almost the same. This indicated that the stream velocity output from CFD analysis is acceptable while the calculated value of thrust power is in small-scale because it is a minimum thrust power of propeller for every parameters configurations that will be generated from 0.3m/s of inlet velocity at rotation of blade at 600RPM.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS FOR FUTURE WORK

The literature study to investigate the blade parameters of propeller has been presented in this report. Type of propeller, number of blades, blade angle, hub to tip ratio or size of propeller, design of nose cone and duct are all parameters that will be a manipulated data to find the optimum performance of propeller.

All the information and requirement such as data of blade parameters are gain from the best measurement values of existing project and experiment. The twisted shape of blade will be designed with parameters. While the design of ducted is the standard design used in propeller of AUV with bigger inlet diameter and smaller outlet diameter. By using matrix multiplication, all data was combined to get all designs in. The 3-dimension solid modelling of each data pairing have been designed and drawn in the computer aided design (CAD) software which is CATIA.

CFD analysis has been used in this study to analyse the flow patterns of the streamlines at the inlet and outlet of the propeller blade. The stream velocities data for each of blade's parameters configuration involved were also gained from this analysis. Therefore, from all these data, the estimated thrust power produced by the propeller blade can be calculated by substituted it with related thrust power equation. From the results in Table 4.1, indicated that the maximum thrust power which able to produce by the propeller blade with 4 blades, blade angle with inlet angle of 60˚ and outlet angle of 20˚, hub to tip ratio of 0.48 and with nose cone of 0.035m in length and 0.0470m in radius was 0.3817 W by Design 26. In this case, the minimum output value produced was about 0.1937 W by Design 15. From the Figure 4.1 also, increasing of number of blades will cause the value of the thrust power produced by the propeller became higher. Furthermore, by increasing the outlet angle and decreasing the inlet angle of blades, the thrust power produced by the propeller blade will also became higher because it generates higher pressure different around propeller. While nose cone with longer length and smaller radius will make the waterflow more efficient. From the analysis, 4 blades of propeller produced more higher thrust power than 3 blades propeller. While the average outlet flow velocity based on blade parameters configurations which affected the performance (thrust power) and the optimum blade parameters configuration for the AUV are in range of 0.4494 m/s to 0.5147 m/s from the velocity inlet of 0.3 m/s.

However, fabrication and field experiments are important to be actualized in which the parameters and specifications configurations included must be the same as what had been conducted in CFD analysis so that comparison can be made between both results later. The design should be fabricated, and experiments should be conducted as a primary component of the scientific method to test theories and hypotheses about how physical processes work under real conditions.

The research in designing and manufacturing of marine propeller especially for parameters of the blade need to be increase so the development of new concepts in propeller designing can be improve and the optimum performance of propeller can be obtained.

REFERENCES

Shaikh, A., 2015. Study of Propeller Design Parameters. *ICITDCEME'15 Conference Proceedings*. 2015. Maharashtra, India.

Pivano, L., 2008. *Thrust Estimation and Control of Marine Propellers in Four- Quadrant Operations*. Norwegian University of Science and Technology.

Chrusciel, T., Ciba, E., and Dopke, J., 2014. CFD and FEM model of an underwater vehicle propeller. *Polish Maritime Research*, 3(83), pp. 40-45.

Zhang, Q., & Jaiman, R. K. (2019). Numerical analysis on the wake dynamics of a ducted propeller. *Ocean Engineering*, 171, pp. 202-224.

Bhattacharyya, A., Krasilnikov, V., & Steen, S. (2016). A CFD-based scaling approach for ducted propellers. *Ocean Engineering*, 123, pp. 116-130.

Subhas, S., Saji, V.F., Ramakrishna, S., and Das, H.N., 2016. CFD Analysis of a Propeller Flow and Cavitation. *International Journal of Computer Applications*.

Husaini, M., Samad, Z., & Rizal, M. (2011). Autonomous Underwater Vehicle Propeller Simulation using Computational Fluid Dynamic. *Computational Fluid Dynamics Technologies and Applications*.

Song, Y. S., & Arshad, M. R. (2016). Thruster modeling for a Hovering Autonomous Underwater Vehicle considering thruster-thruster and thruster-hull interaction. *2016 IEEE International Conference on Underwater System Technology: Theory and Applications (USYS).*

Go, J., Yoon, H., & Jung, J. (2017). Effects of a duct before a propeller on propulsion performance. *Ocean Engineering*, 136, pp. 54-66

Ghassemi, H., Mardan, A., and Ardeshir, A., 2012. Numerical Analysis of Hub Effect on Hydrodynamic Performance of Propellers with Inclusion of PBCF to Equalize the Induced Velocity. *Polish Maritime Research* 2(73), pp. 17-24

Othman, M. M., Razak, J. A., Bashar, M. F., Muhammad, N. S., & Sopian, K. (2014). CFD Analysis on the Flat Runner Blades of Propeller's Turbine under Low Head and Low Flow Condition. *Applied Mechanics and Materials*, 699, pp. 437-442

Lee, C.S., Choi, Y.D., Ahn, B.K., Shin, M.S., & Jang, H.G., 2010. Performance optimization of marine propellers. *Inter J Nav Archit Oc Engng*, 2, pp. 211-216

Djodikusumo, I., Diasta, I. N., & Awaluddin, I. S. (2016). Geometric Modeling of a Propeller Turbine Runner Using ANSYS BladeGen, Meshing Using ANSYS TurboGrid and Fluid Dynamic Simulation Using ANSYS Fluent. Applied Mechanics and Materials, 842, 164-177.

Mutoh, H., & Maekawa, K. (2008). 119 CFD Analysis Example using "FLUENT for CATIA V5". The Proceedings of Conference of Kansai Branch, 2008.83(0)

Bothe, F., Friebe, C., Heinrich, M., & Schwarze, R. (2014). CFD Simulation of Incompressible Turbomachinery — A Comparison of Results From ANSYS Fluent and OpenFOAM. Volume 2B: Turbomachinery

Meng, X., Qin, Y., Zhang, M., & Xu, Y. (2010). Thruster Fault Diagnosis of An AUV Based on Multi-AUV System. Robot, 32(3), 314-320

Healey, A., Rock, S., Cody, S., Miles, D., & Brown, J. (n.d.). Toward an improved understanding of thruster dynamics for underwater vehicles. Proceedings of IEEE Symposium on Autonomous Underwater Vehicle Technology (AUV94)

Saunders, A., & Nahon, M. (n.d.). The effect of forward vehicle velocity on through-body AUV tunnel thruster performance. Oceans 02 MTS/IEEE

Guillaume. (n.d.). 2-blade vs 3-blade and 4-blade propellers. Retrieved from http://aerotrash.over-blog.com/2015/02/2-blade-vs-3-blade-and-4-blade-propellers.html

Ranjan, R. R., Parmar, D. V., Raipuria, H. K., & Singh, P. B. (2015). Innovative Nose Cone Design of Aircraft. Volume 1: Advances in Aerospace Technology

Software, M. H. (n.d.). Retrieved from https://heliciel.com/en/helice/Vrillage calage pale helice.htm

T200 Thruster. (n.d.). Retrieved from https://www.bluerobotics.com/store/thrusters/t100-t200-thrusters/t200-thruster/

Xiang, S., Yang, K., & Zhao, W. (2018). Aerodynamic Characteristics Test for Two Kinds of Propeller with Different Blade Angle. DEStech Transactions on Engineering and Technology Research, (Amee)

Joung, T., Sammut, K., He, F., & Lee, S. (2012). Shape optimization of an autonomous underwater vehicle with a ducted propeller using computational fluid dynamics analysis. International Journal of Naval Architecture and Ocean Engineering, 4(1), 44-56.

APPENDIX

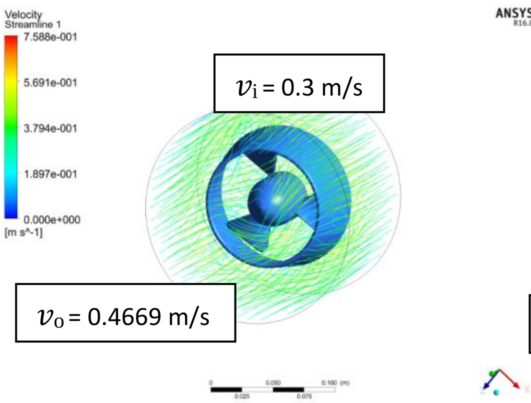
APPENDIX A

Table of stream velocity outlet from CFD analysis with Calculated Thrust.

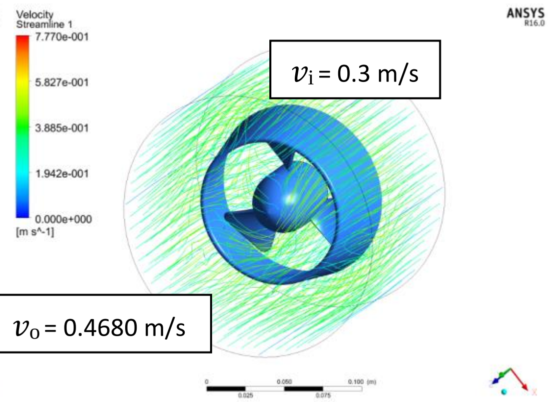
No. of Design	Stream Velocity Outlet, v_o (m/s^2)	Thrust. T (N)	No. of Design	Stream Velocity Outlet, v_o (m/s^2)	Thrust. T (N)
Design 1	0.4720	0.5630	Design 17	0.4922	0.6456
Design 2	0.4682	0.5486	Design 18	0.4937	0.6518
Design 3	0.4548	0.4954	Design 19	0.4810	0.5993
Design 4	0.4636	0.5297	Design 20	0.4816	0.6018
Design 5	0.4669	0.4624	Design 21	0.4838	0.5204
Design 6	0.4680	0.4661	Design 22	0.4948	0.5593
Design 7	0.4645	0.4543	Design 23	0.4889	0.5383
Design 8	0.4626	0.4480	Design 24	0.4971	0.5675
Design 9	0.4530	0.4885	Design 25	0.4880	0.6281
Design 10	0.4695	0.5530	Design 26	0.5147	0.7416
Design 11	0.4494	0.4747	Design 27	0.4834	0.6092
Design 12	0.4558	0.4993	Design 28	0.4791	0.5916
Design 13	0.4763	0.4944	Design 29	0.4961	0.5639
Design 14	0.4653	0.4570	Design 30	0.4918	0.5486
Design 15	0.4557	0.4251	Design 31	0.4921	0.5496
Design 16	0.4588	0.4353	Design 32	0.4872	0.5323

APPENDIX B

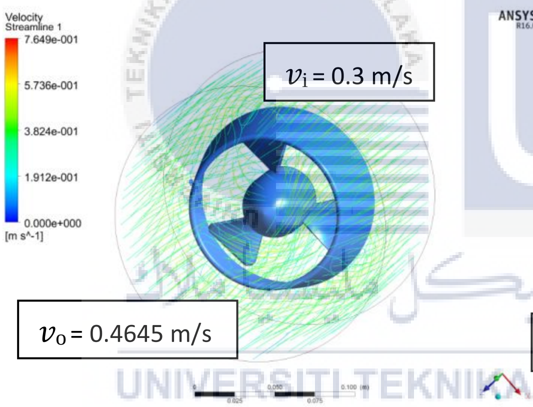
Figure of CFD analysis output with stream velocity.



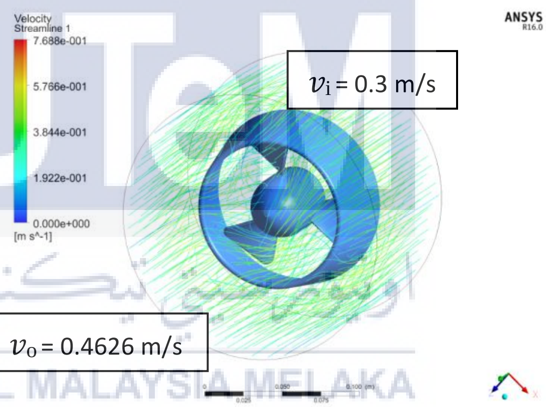
CFD image of Design 5



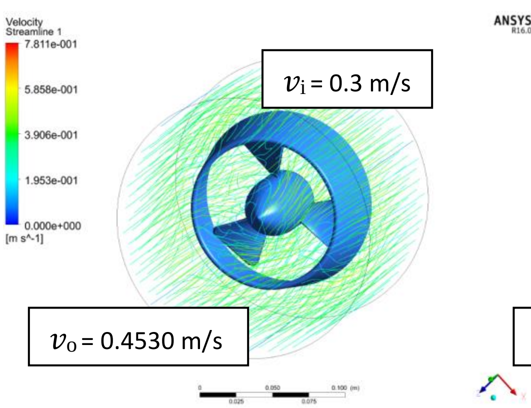
CFD image of Design 6



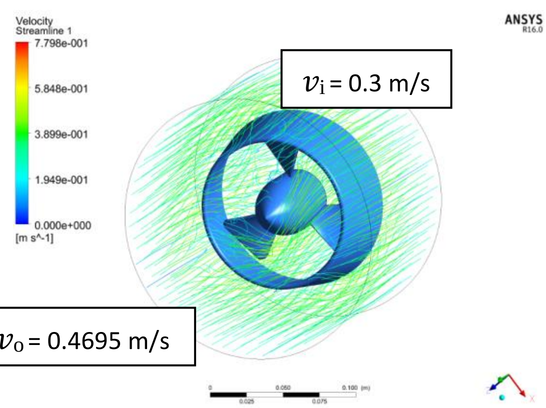
CFD image of Design 7



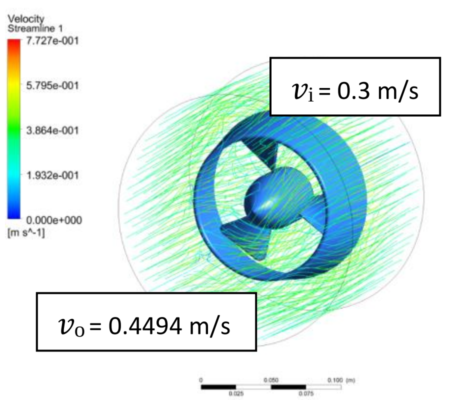
CFD image of Design 8



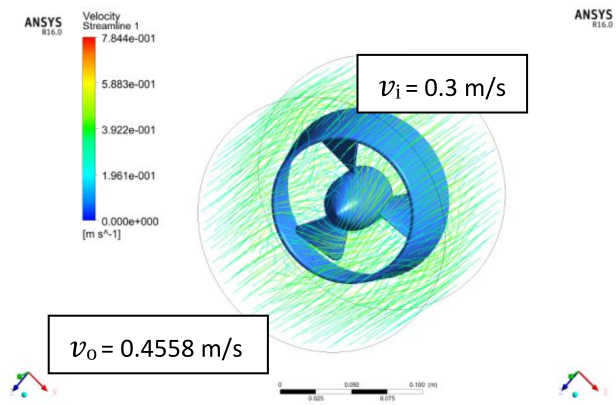
CFD image of Design 9



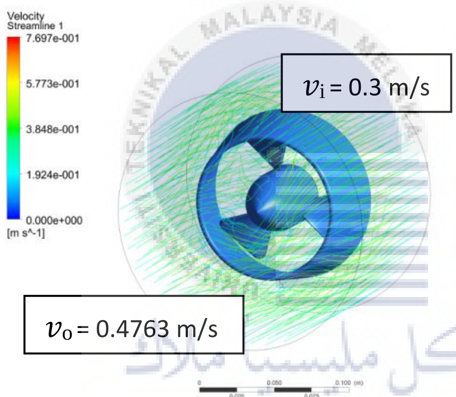
CFD image of Design 10



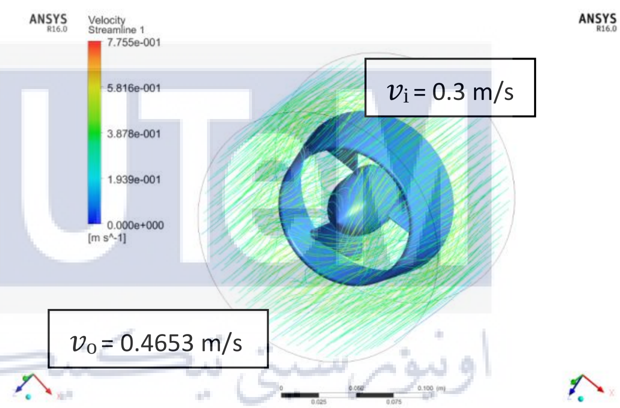
CFD image of Design 11



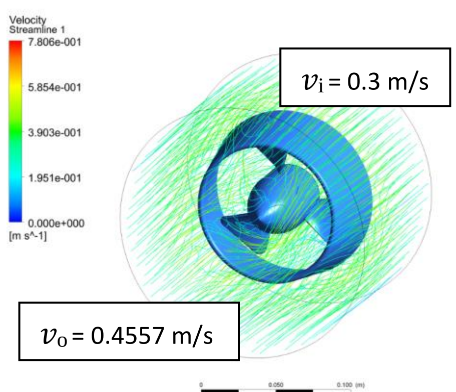
CFD image of Design 12



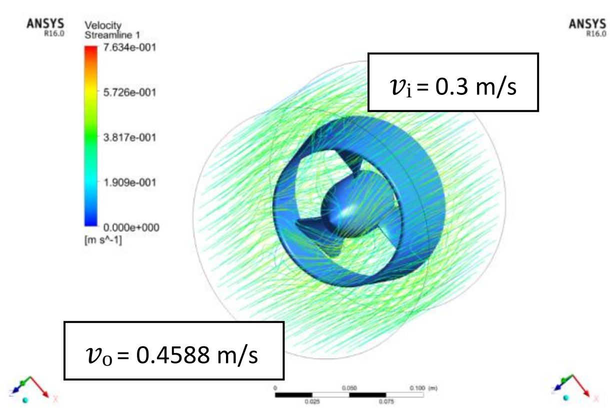
CFD image of Design 13



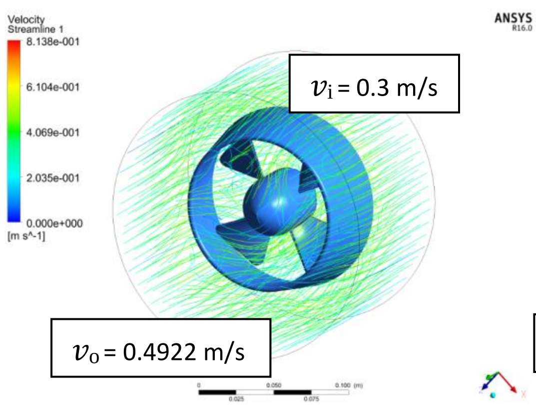
CFD image of Design 14



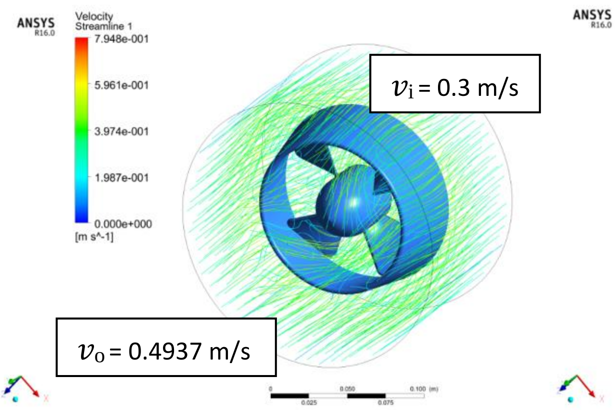
CFD image of Design 15



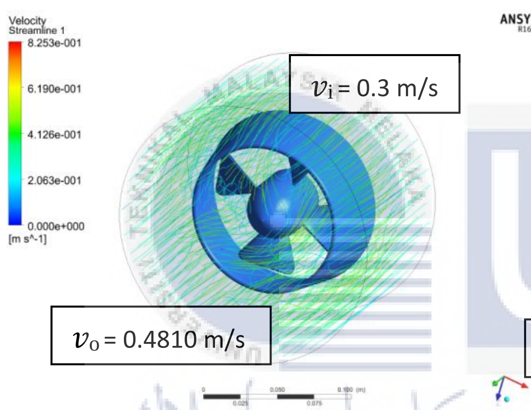
CFD image of Design 16



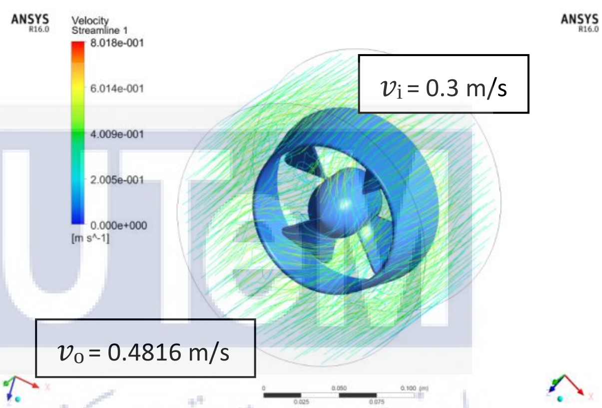
CFD image of Design 17



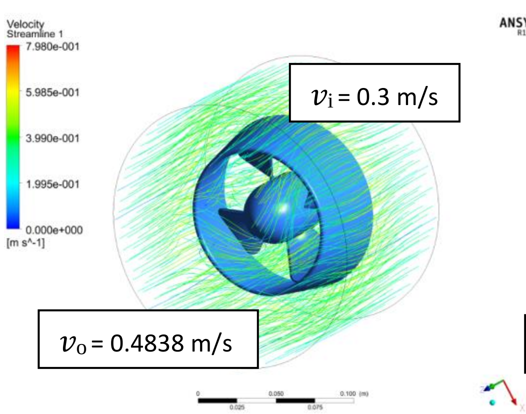
CFD image of Design 18



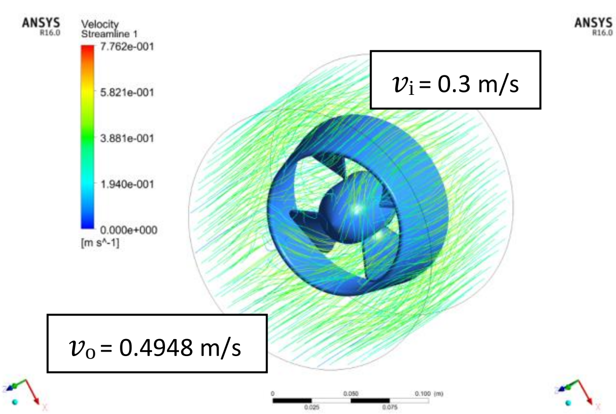
CFD image of Design 19



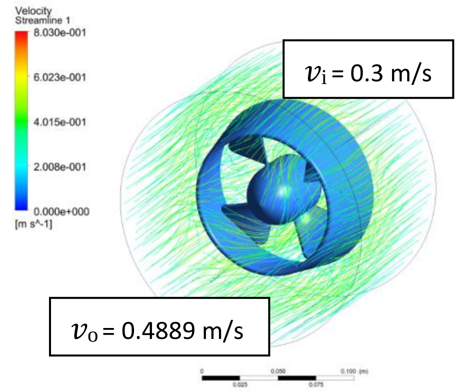
CFD image of Design 20



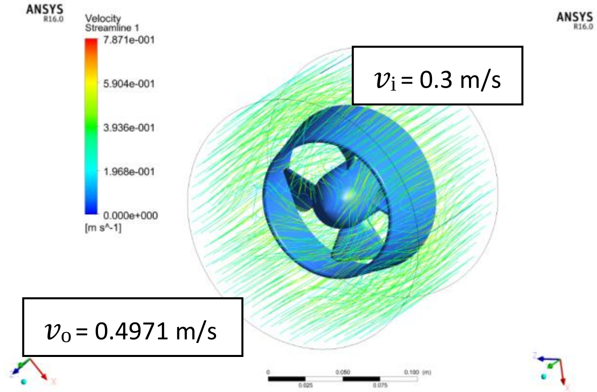
CFD image of Design 21



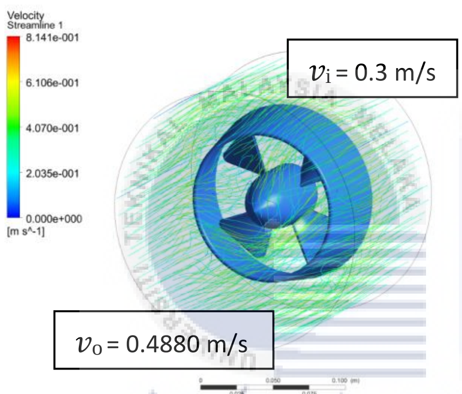
CFD image of Design 22



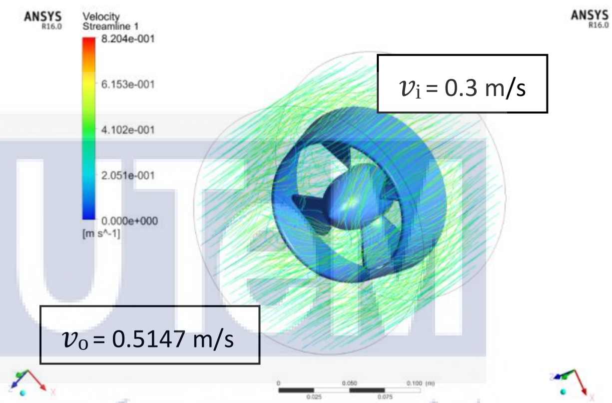
CFD image of Design 23



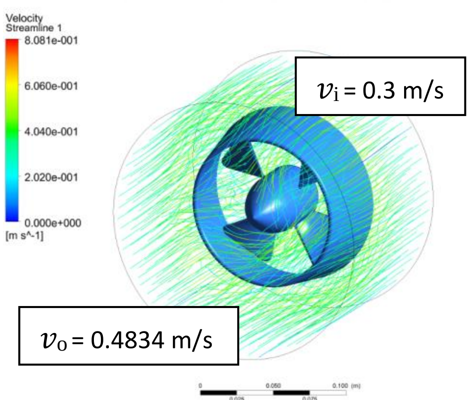
CFD image of Design 24



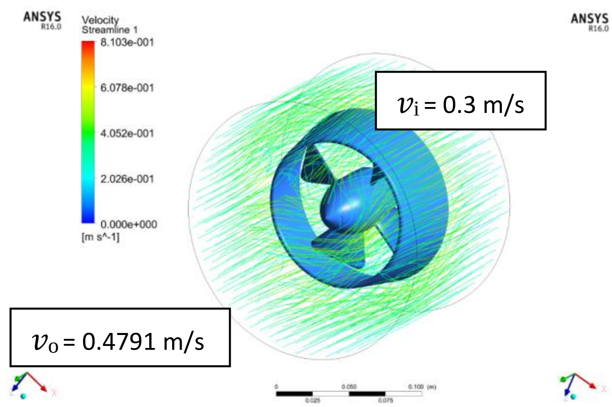
CFD image of Design 25



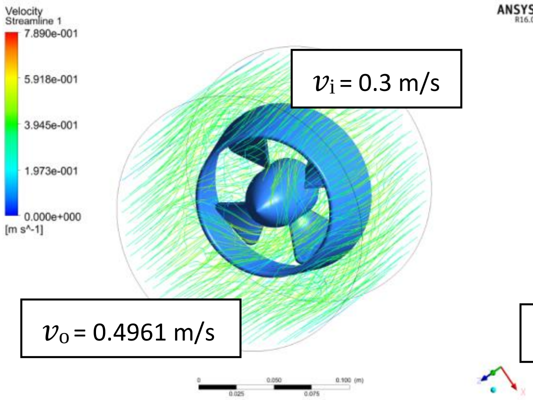
CFD image of Design 26



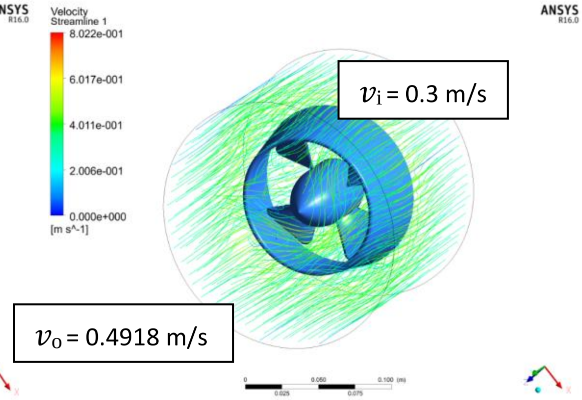
CFD image of Design 27



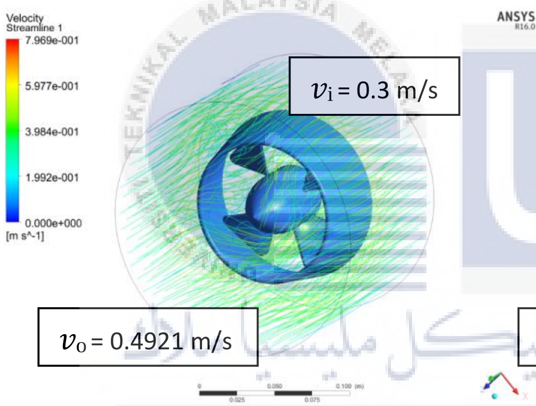
CFD image of Design 28



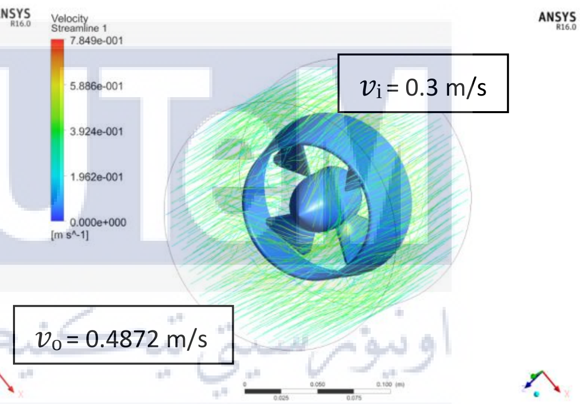
CFD image of Design 29



CFD image of Design 30



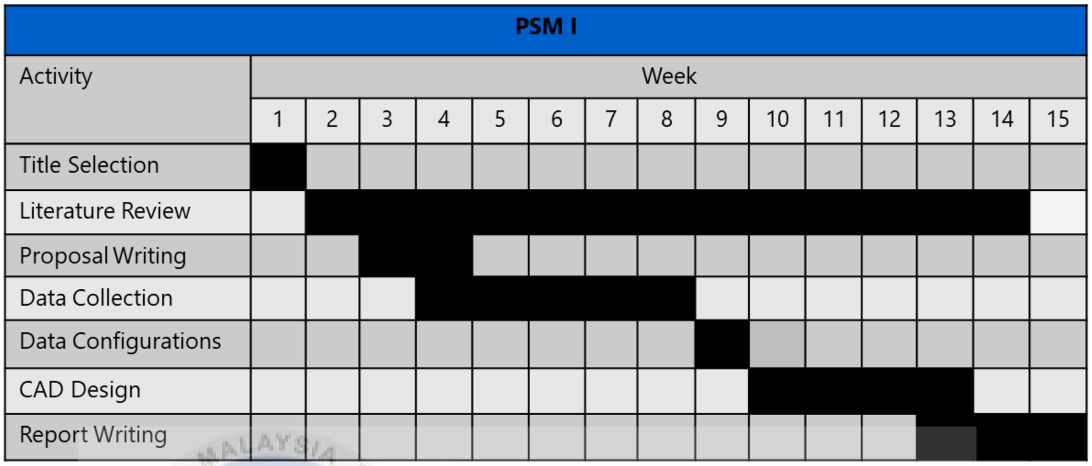
CFD image of Design 31



CFD image of Design 32

APPENDIX C1

Gantt Chart for PSM I (Semester 1 2018/2019)



APPENDIX C2

Gantt Chart for PSM II (Semester 2 2018/2019)