EFFECT OF RUDDER DESIGN ON HYDRODYNAMIC FORCES OF AUTONOMOUS UNDERWATER GLIDER

MOHAMAD SYAMIM BIN ZAKARIA

MECHANICAL ENGINEERING
UNIVERSITI TEKNOLOGI PETRONAS

JANUARY 2017

Effect Of Rudder Design On Hydrodynamic Forces Of Autonomous Underwater Glider

by

Mohamad Syamim bin Zakaria

18287

Dissertation submitted in partial fulfillment of

as a Requirement for the

Bachelor of Engineering (Hons)

(Mechanical)

JANUARY 2017

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

32610 Seri Iskandar

Perak Darul Ridzuan

Malaysia

CERTIFICATION OF APPROVAL

Effect Of Rudder Design On Hydrodynamic Forces Of Autonomous Underwater Glider

by

Mohamad Syamim bin Zakaria
18287

A project dissertation submitter to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
In partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL)

Approved by,

____

(Dr. Mark Ovinis)

UNIVERSITI TEKNOLOGI PETRONAS
BANDAR SERI ISKANDAR, PERAK
January 2017

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertake or done by unspecified sources or persons.

____

MOHAMAD SYAMIM BIN ZAKARIA

Abstract

An autonomous underwater glider (AUG) is a vehicle with fixed-wing. They move through the ocean by changing the buoyancy to follow a "Saw-tooth" pattern of motion. In this project, a part of AUGs which is rudder design is studied. Specifically, the relationship between rudder design and lift and drag of hydrodynamic forces is investigated. The comparative study is done between conventional rudder and schilling/ fishtail rudder. For the modelling works, SOLIDWORKS is used to create the model. ANSYS code "Fluent" is used to do analysis on hydrodynamic performance such as lift force of AUGs. The numerical simulation is done at various angle of attack from 0˚- 20˚. The result from the simulation shown that the fishtail rudder has higher lift coefficients and can produce better maneuvering performance compare to conventional rudder.

Acknowledgements

In the name of Allah, thank to him for giving me the opportunity to carry on all the things that I did in my life. May Allah blesses all the time and effort that I have spent throughout the completion of this project. Amin.

Special appreciation goes to my supervisor, Dr. Mark Ovinis for the endless support and guidance. I would like to express my gratitude to him for all the motivations and time to the completion of this project.

Finally, I would like to express my gigantic appreciation to my family members especially to my parents for their unconditional love and supports. Lastly, thank you to my friends who support me through all these past 5 years in university. I am so thankful for our friendship and thank you for being the shoulder I can always depend on.

Table of Contents

List of Figures

Chapter 1 INTRODUCTION

1.1 Background study

An autonomous underwater glider (AUG) is buoyancy- propelled and having fixed-wing AUGS[1]. This vehicle normally travel in the ocean by changing their buoyancy for vertical motion and make use of their wings which convert the vertical motion to inclined motion (vertical and horizontal motion)[2]. Nowadays, the application of using AUG has increased in demand due to their advantages in many ways and they also help in reducing the human life risk because of their ability in reaching location that beyond human limit which allow critical information gathering being done. Besides, AUGS also capable for long-range mission and high- endurance deployments as well.[3]

According Bingham, he shared his words by recommending the future AUGS to be more effective in terms of fast speed, limited climate dependence and ability to travel more longer for underwater operation[4]. However, there are some limitation in the current design of AUGS that prevent AUGS from accomplishing their ideal execution.[5] One of them is the selection of rudder profile.

Rudder can be considered as one of the most important hydrodynamic control surface which control the movement of the AUGS in horizontal motion. Their importance in generating forces for maneuvering is no doubt. Meanwhile in some cases, rudder also used as emergency brake and roll stabilization as well.[6] The performance of rudder depends on the rudder hydrodynamic characteristic, which are affected the design choices. Therefore, the proper design of rudder will surely produce better performance in maneuverability, fuel consumption and increase efficiency which means having minimum drag at the required lift force.[7]

A study done by Yoshiho Ikeda et all on the performance of various kind of fishtail rudder (schilling type) and hydrodynamic mechanism to generate larger lift force due to the fish tail. As the result, higher lift force is produced when larger trailing edge is applied. The high lift force is created due to high pressure produced at the concave face side of the rudder. The result obtained shown as below in Figure 1.1

In this project, comparative analysis on hydrodynamic forces like lift and drag were performed on different type of rudder designs.

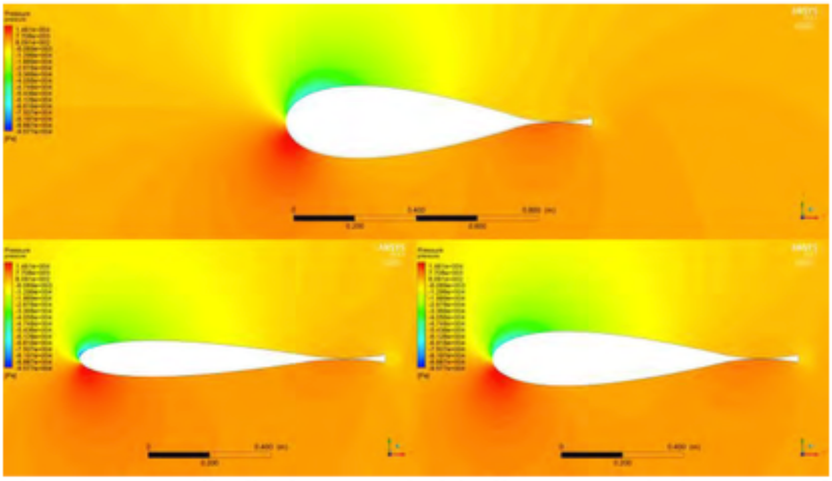

Figure 1-1: Example of Pressure distribution flow around fishtail forms [8]

1.2 Problem statement

Various form of rudder profile is used to create better manueverability and enhance performance of the AUV. However, the effectiveness of using them still have been not clarified yet. So this study is done to propose the standard rudder design that can be used at low speed and having better performance as well.

1.3 Objectives

- To propose and analyse rudder designs which is to optimize the AUV performance
- Simulating the lift and drag formed using different profile of rudder
- To evaluate different rudder designs for better performances by establishing relationship between lift and drag forces to angle of attack (AOA)

1.4 Scope of study

For this project, the investigation will be focused on the numerical study on the rudder of underwater vehicle and deal with computational fluid dynamics(CFD) using ANSYS Fluent software. The flow field is focused on the x-direction only which represented the flow velocity. Y-direction and Z- direction is neglected in this project. Several designs which are conventional rudder (NACA 0020) and schilling/ fish tail rudder with different value of trailing edge length are compared. The comparison work will focus on the hydrodynamic characteristic generated by the rudder.

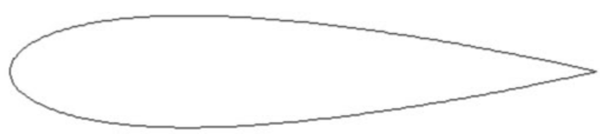

Figure 1-2 Example of Conventional rudder design, NACA

Chapter 2 : LITERATURE REVIEW

This chapter review some of the previous studies and research papers that have been designed and developed.

2.1 Rudder design

Generally, this project will focus on the hydrodynamic forces behavior on conventional rudder and fishtail rudder. Thus, it is important to know the knowledge behind it first. For different shape of rudder, their hydrodynamic forces characteristic of lift and drag coefficient, slope of the lift curve should be different too. Liu proposed to estimate the rudder force generated of different design and investigate their effect towards ship maneuverability by regression formula [10]. Besides, the selection of rudder used should be depend on few characteristics including operational requisites of maneuvering performance, and fuel conservation. [10]

Fishtail rudder

Fishtail rudder also known as schilling rudder, it is designed based on NACA, HSVA or IFS with trailing edge. The curve part at the back, is said to have better pressure distribution that believed to slow down the stalling process. Stalling process happen when the critical angle of attack is exceeded the limit. This kind of rudder design has potential to create more lift thus improve the maneuverability of the ship especially at slow speeds. [10]. That's why this rudder is normally used on inland vessels. However, only few studies being done on the fishtail rudder based on experimental and numerical test. [8]

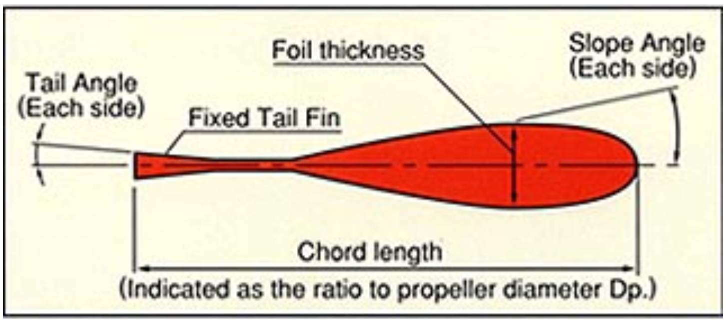

Figure 2-1:Fishtail rudder design specification

NACA (National Advisory Committee for Aeronautics)

Meanwhile for conventional rudder, NACA profiles are the most commonly used applied rudder[10]. This rudder design also used during investigation purpose of aerodynamic and hydrodynamic [10]. In 1958 and 1965, a group of researcher carried out experiment to validate the performances of NACA design shape and it showed that they were able to generate enough force for maneuvering with high percentage of efficiency.[10]

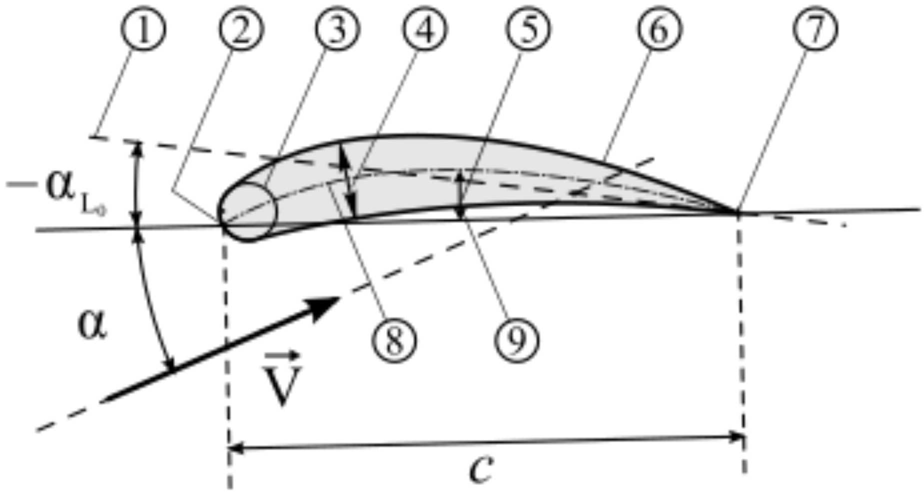

Figure 2-2: Rudder profile terminology [10]

1. Zero lift
2. Leading edge
3. Nose circle
4. Max thickness
5. Chamber
6. Upper surface
7. Trailing edge
8. Camber mean line
9. Lower surface

2.2 Hydrodynamic performance

Hydrodynamic forces and the moment of the body and wings is considered as one of the major factor that contribute to the motion of AUGs[9]. Under hydrodynamic forces, which are lift and drag force, this both force are important to determine the AUGs abilities and help the movement of the vehicle in the X-direction (horizontal motion). [6]

When there is a pressure difference, surely the lift force will be generated. [10] In other word, it is a force that act . Meanwhile stall angle is the condition where the maximum lift occur when it is at critical AOA. Typically, in open water, the stall angle will range from 15-20 degree. But the one that normally used for propeller only range about 30-40 degree [10]

Drag is defined as the force generated and exerted on the body in opposite. [9] Drag normally divided into two elements, which are skin friction drag and pressure drag. Generally, the value of drag increase when AOA increase.[11] The friction drag is formed due to viscous drag in the boundary layer around the shape and it is controlled by the extent of the wetted surface. Meanwhile pressure drag, it will behaved depend on the shape of rudder itself. [10] For rudder that has same wetted surface, the friction drag generated is assumed to be the same. However, the pressure drag is very small. [12]

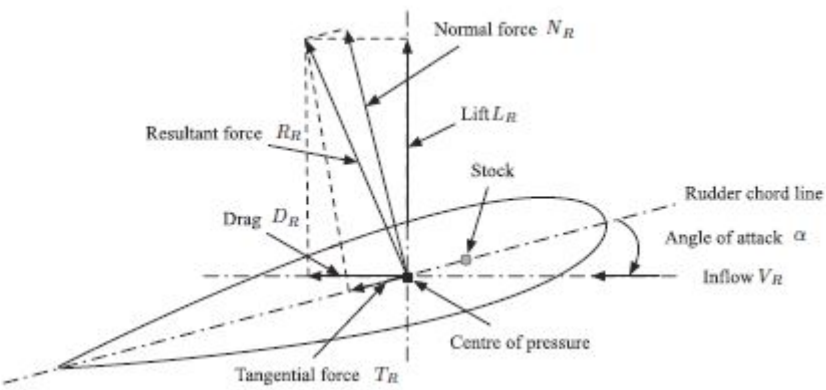

Figure 2-3 : Rudder induced force [10]

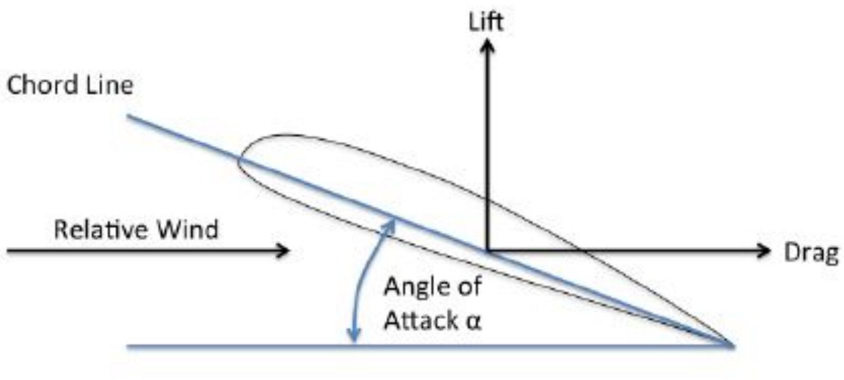

Figure 2-4: Lift and Drag diagram [10]

Lift and drag forces are defined as shown in Fig. 5. The lift coefficient C_l and drag coefficient C_d are commonly defined by:

$$C_l = \frac{L}{\frac{1}{2}\rho v^2 S} \qquad (1)$$

$$C_d = \frac{D}{\frac{1}{2}\rho v^2 S} \qquad (2)$$

Where *L* is the lift force, *D* is the drag force, ✕ is the fluid density, *v* is fluid velocity and *S* is projected area of the foil (Chord length x Width).

The sectional lift coefficient and drag coefficient can be expressed as follow:

$$c_l = \frac{l}{\frac{1}{2}\rho v^2 c} \qquad (3)$$

$$c_d = \frac{d}{\frac{1}{2}\rho v^2 c} \qquad (4)$$

Where *l* is the lift force, *d* is the drag force acting on two dimensional foil, and *c* is the chord of the foil.

2.3 Computational Fluid Dynamics

A CFD commercial code, ANSYS FLUENT, is used to calculate the forces and moments of rudder sections. The k-omega SST turbulence model is applied.

There was a study done by Van Nguyen, he made a cuboidal space whose measurement are 12m x 8m x 6m separately using AUTOCAD.[13] The critical perspective while generation of domain is to assure that, the wall has little impact on distribution flow around the rudder. The distance from the wall should be 3-4 times the size or rudder. Meanwhile for the leading edge, at least has to be 1.5m from inlet boundary condition.

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient solutions. ANSYS Mesh tool is used for the meshing of computational domain. In order to separate domain into smaller volume, non-uniform unstructured mesh elements is used. Meanwhile, at the area around rudder, it was set to be denser and as the element move far away from it, the element will increase in size. The approach method allows the simulations to run economically.
The physical and boundary conditions for simulation are listed in Tables 2.5 and 2.6

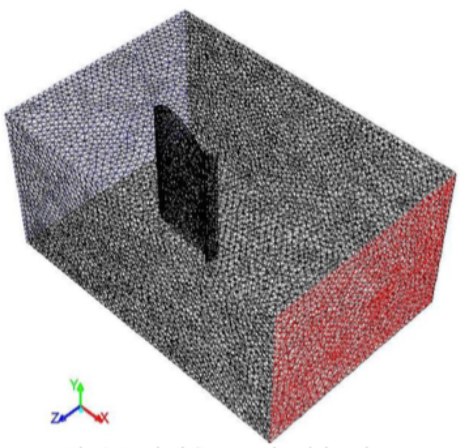

Figure 2-5 :Example of Meshed computational domain [13]

Boundary	Type
Left	Velocity inlet
Right	Pressure outlet
Rudder profile	No slip wall
Remaining sides	No slip wall

Table 2.5 **Example of boundary condition set up**

Parameter	Value	Unit
Fluid	Water	
Density	998.2	Kg/m3
Viscosity	0.001003	Kg/m-s
Velocity	6	m/s
Reynolds number	6000000	

Table 2.6 **Physical condition for CFD calculation**

Chapter 3 : METHODOLOGY

3.1 Project Flowchart

The process flow chart is shown in Figure 3.1 below.

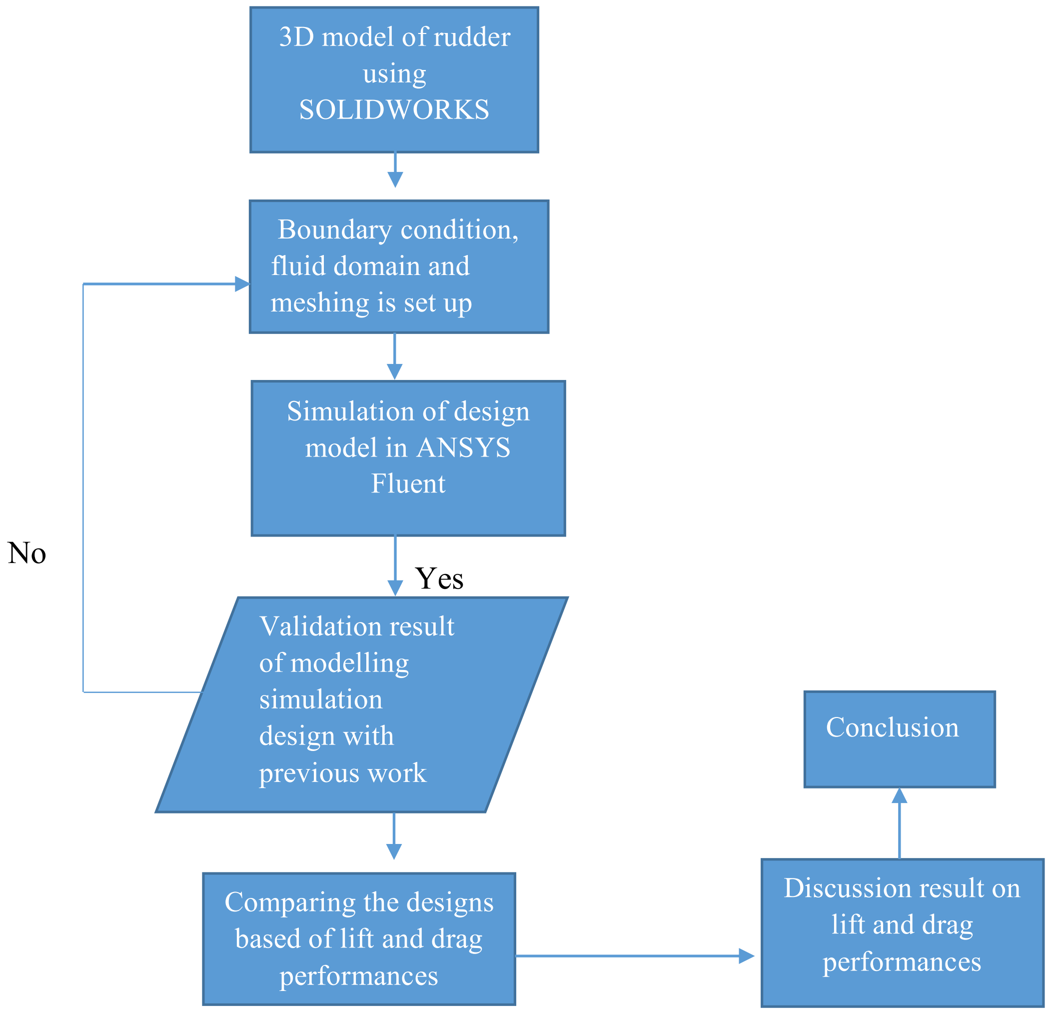

Figure 3.1: **Project Flow**

3.2 Tools and software required

In order to perform this project, the tools and software used are:

1. **SOLIDWORKS**

   This software specifically used to create the 3D modelling. By using this software, datum design and another two designs, conventional rudder and schilling rudder were created.

2. **ANSYS Fluent**

   ANSYS is basically used to do the simulation work once after the design being developed in the SOLIDWORKS. Boundary condition, fluid domain or even meshing work is being done here as well

3. **Microsoft Office (Word, PowerPoint, et al)**

   For Microsoft Word, the main purpose is to create text document which can be saved electronically, printed or even saved as PDF files. Meanwhile for PowerPoint, it is used to create slide show for enhancing presentation.

3.3 Modelling

The modelling work is done using SOLIDWORKS software. Roughly there are about 2 designs choose, which are fishtail / schilling rudder and conventional rudder (NACA 0020). Figure below shows the design of rudder with different size of trailing edge and shape as well.

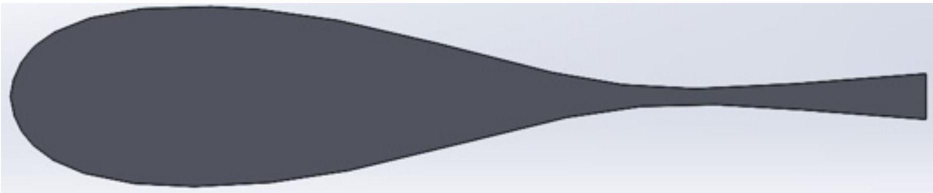

Figure 3-1 : 20 cm length, 5cm trailing edge (Top view)

Figure 3-2: 20cm length, 10cm trailing edge (Top view)

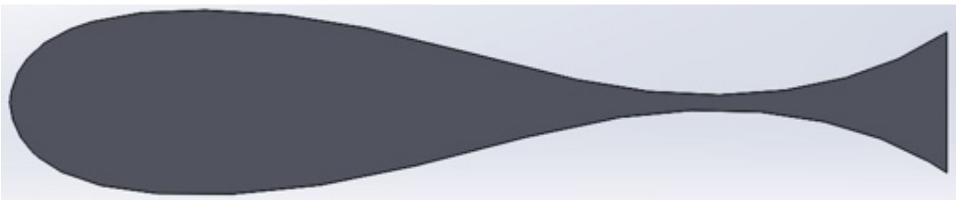

Figure 3-3: 20cm length, 15cm trailing edge (Top view)

Figure 3-4: NACA 0020 (Top view)

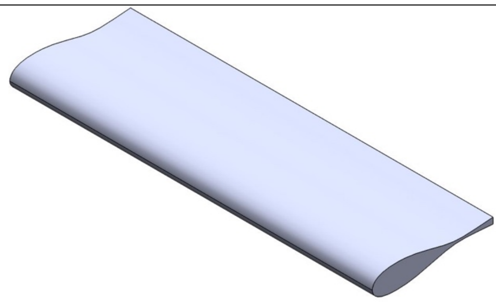

Figure 3-5 :Isometric view of 20cm, 5cm trailing edge

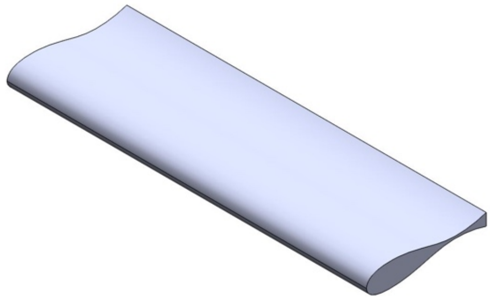

Figure 3-6: Isometric view of 20cm, 10cm trailing edge

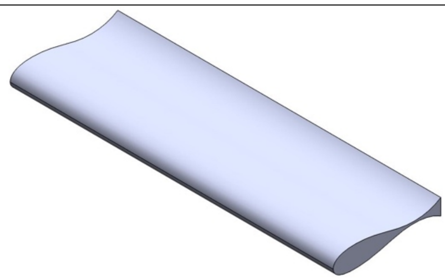

Figure 3-7: Isometric view of 20cm, 15cm trailing edge

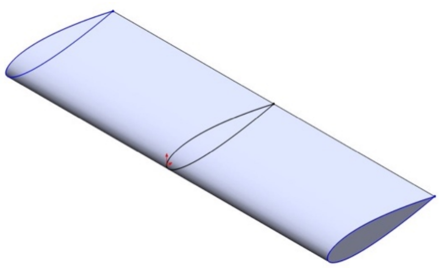

Figure 3-8: Isometric view of NACA 0020

3.2 Boundary Conditions

For this work, rudder model will be placed in the square box (boundary condition) which later created in ANSYS Fluent. The rudder model is set to be as no slip condition. The velocity inlet boundary condition will be applied at inlet and meanwhile the outflow boundary condition will be applied at outlet part. Model is placed in the center of the square box so that to make sure that the velocity will fully developed at the exit entrance.

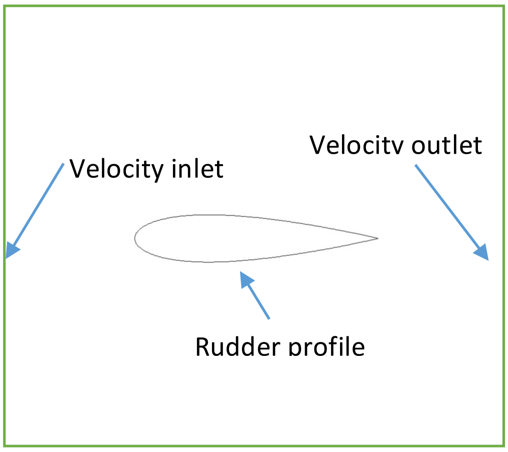

Figure 3-9 : Fluid domain with boundary conditions in CFD [15]

3.3 Fluid Domain Meshing

Accuracy and complexity of the analysis is depend on the selection of mesh size affects the. The mesh of models conducted using ANSYS Fluent, a Computational Fluid Dynamic (CFD) software. The result of this entire simulation study will depend highly on the various mesh parameters (i.e. distance of the first layer of the cell to the hull, mesh size and domain size) Fine mesh was selected for this project. The elements size for the mesh is about 0.054m. Fine size mesh is used so that the result yield will be more accurate. Other than that, for advanced size function, proximity method is used for conventional rudder, NACA 0020. Meanwhile for fishtail, the method selected is proximity and curvature. Figure 3.8 shows the fluid domain meshing.

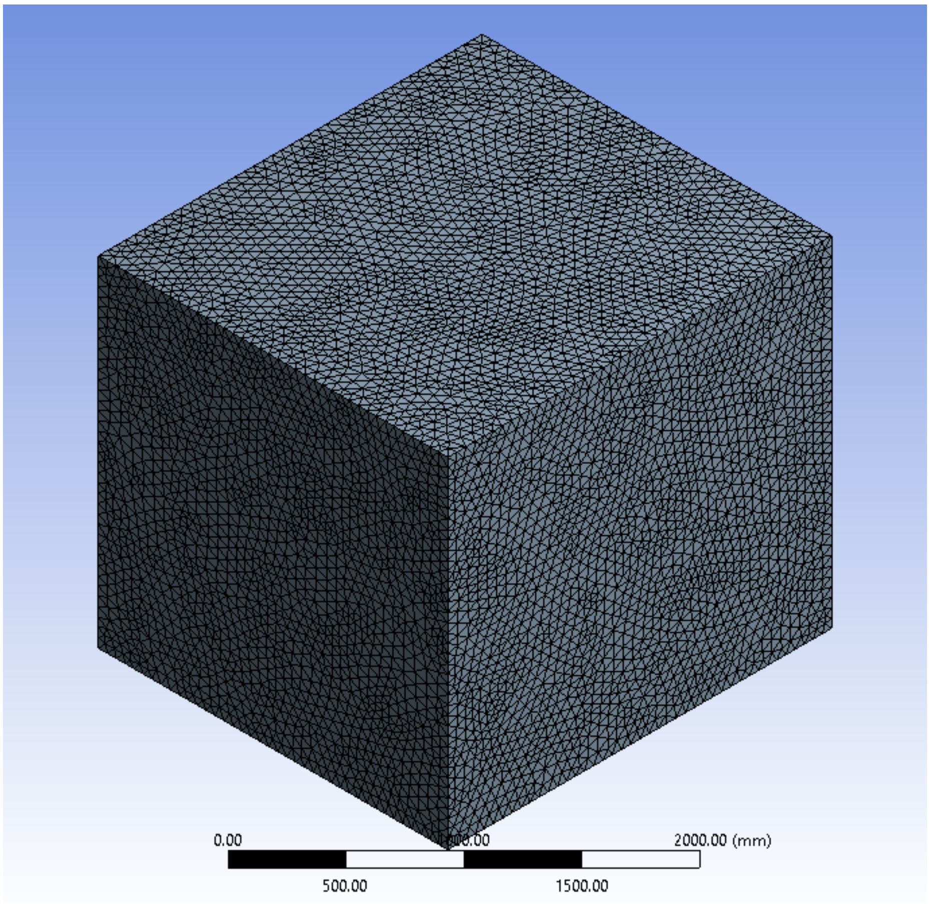

Figure 3-10 : Meshing process ANSYS Fluent

Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Num Cells Across Gap	Default (3)
Min Size	Default (0.54290 mm)
Proximity Min Size	Default (0.54290 mm)
Max Face Size	Default (54.290 mm)
Max Size	Default (108.580 mm)
Growth Rate	Default (1.850)
Minimum Edge Length	209.080 mm

Figure 3-11: Meshing sizing setting for Fishtail profile

3.4 ANSYS Fluent Simulation

In this work, rudder model is kept fixed at their position and only flow will move from inlet boundary condition. The velocity is set to constant at 0.45m/s (average speed of AUGs). Simulation also done at various angle of attack starting from 0-20 degree's. Meanwhile for turbulence model, Re-Normalization Group (RNG) k- epsilon model with non-equilibrium wall function was used instead of standard model to optimize the accuracy of the result. The lift and drag results obtained by each simulation were recorded and compared using graph to represent the result.

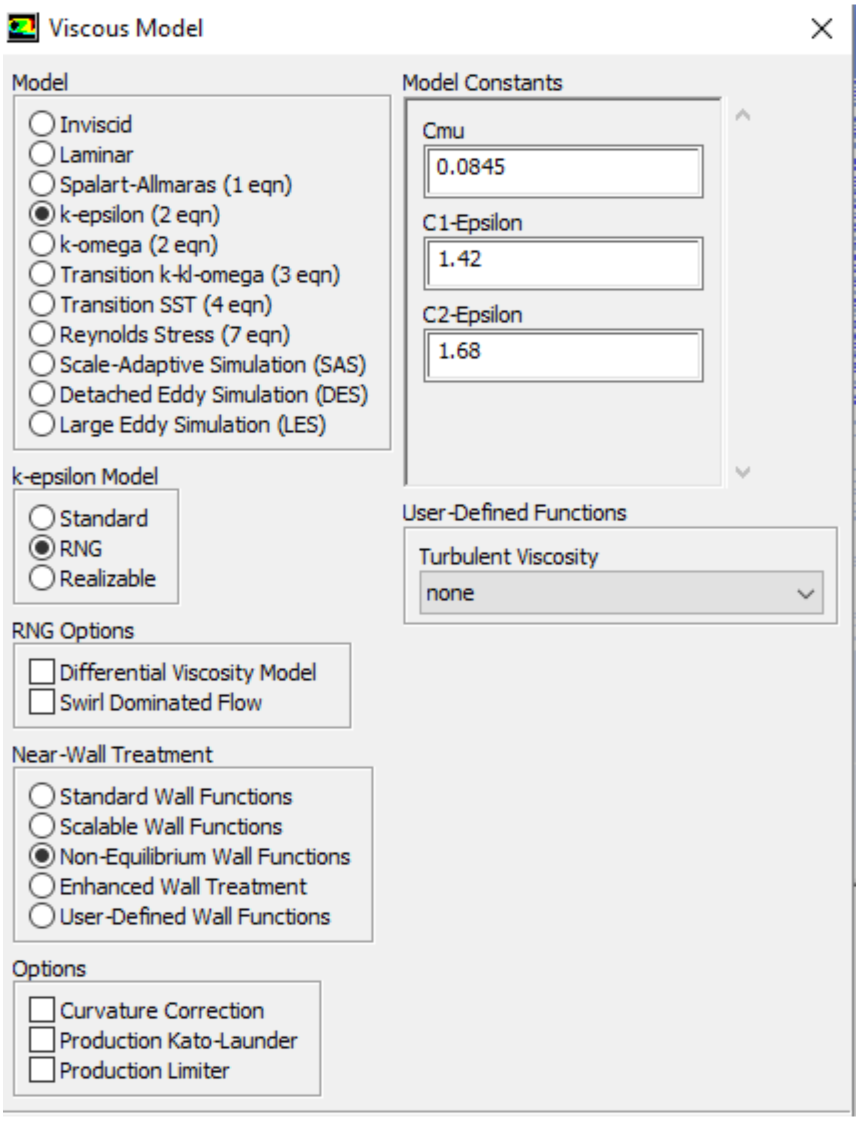

Figure 3-12: Example of turbulence model setting for Fishtail profile

3.5 Gantt Chart

Table 3.1 shows the Gantt Chart of this project.

No	Detail/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Selection of project topic														
2	Preliminary research work														
3	Literature review														
4	Submission of Extended Proposal					x									
5	Methodology														
7	Proposal Defense							x							
8	Preliminary result (SOLIDWORKS)													x	
9	Submission of Interim Report														x
Semester break (3 Jan 2017 – 15 Jan 2017)															
1	Continue research work														
2	Result gathering and collection on ANSYS Fluent														
3	Progress Report submission							x							
4	Analyzing and completion the project report														
5	Pre-SEDEX										x				
6	Final report draft submission											x			
7	Soft-bound dissertation submission												x		
8	Technical paper submission												x		
9	Viva														x
10	Hard-bound dissertation submission														x

Key Milestone

1. Submission of Extended Proposal: 3rd Oct – 9th Oct 2016
2. Proposal Defence: 18th Oct 2016
3. Submission of Interim Report (Draft): 5th Dec – 11th Dec 2016
4. Submission of Interim Report (Final): 18th Dec 2016

Chapter 4 : RESULT AND DISCUSSION

For this section, the numerical results of lift and drag forces generated are shown. Fishtail models are compared with the conventional rudder, NACA0020. The effect of Angle of attack(AOA) on rudder hydrodynamic performances will be investigated as well.

i)Lift force

A. Lift generation

Computational Fluid Dynamic (CFD) is used to simulate the Conventional rudder and fishtail rudder at 0°, 5°, 10°, 15°,20° are shown, which gives us a visualization behavior or the lift and drag of the rudder. Figure below shows the comparison of lift generated against angle of attack. Basically it is obvious that the lift value will increase as if angle of attack increase. The lift force of fishtail is slightly higher than the conventional rudder, NACA 0020 due to the additional pressure force created on the surface of rudder due to their converging and diverging nature of rudder surface.

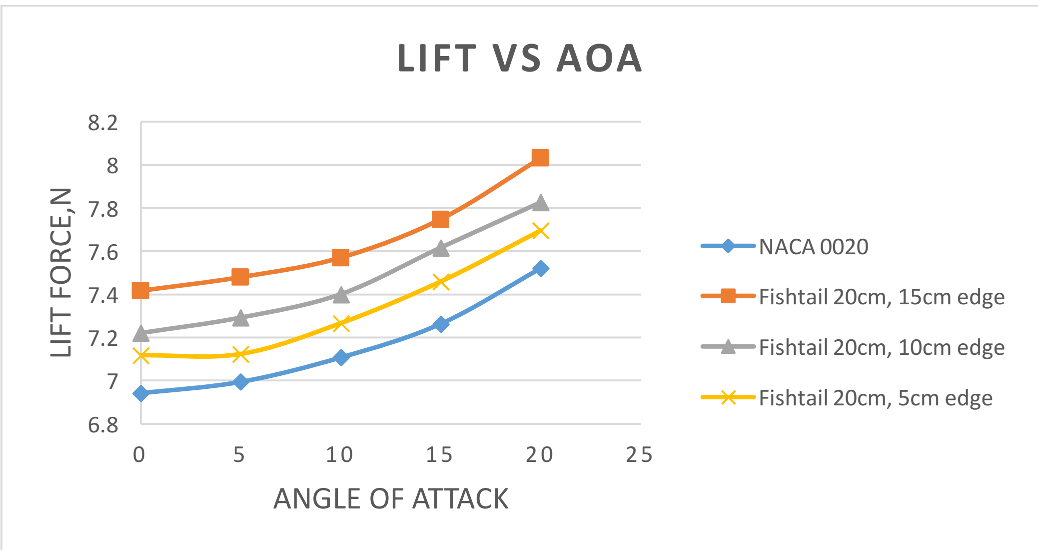

Figure 4-1: Comparison of lift generated against AOA for fishtail models

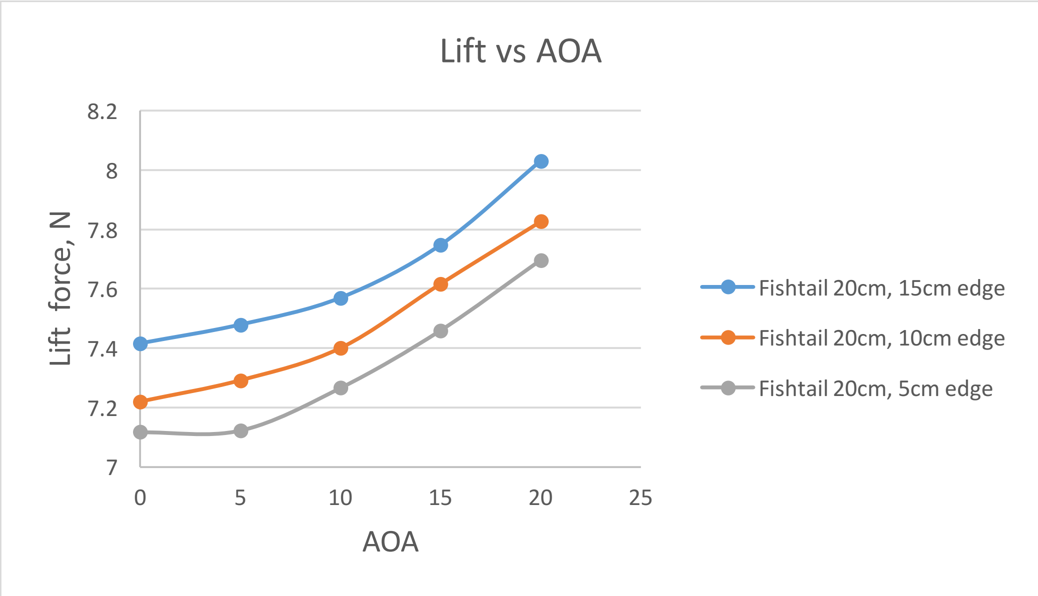

Figure 4-2 : Comparison of lift generated against AOA for fishtail models

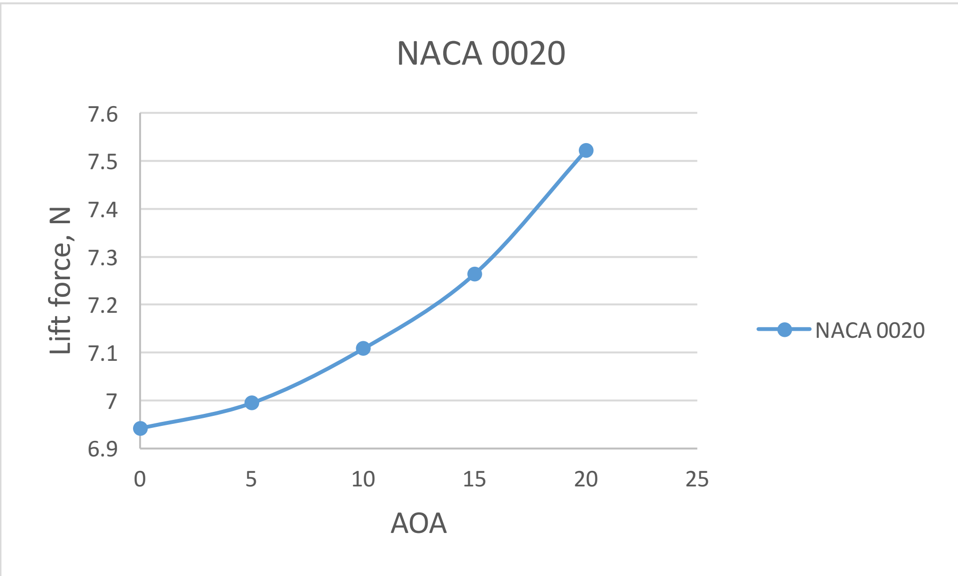

Figure 4-3: Lift generated against AOA for NACA 0020

ii) Drag force

From the figure 4.4, in general, it is clearly shown that the drag force of the fishtail rudder is also higher compare to conventional rudder. The reason behind it that due to huge variations on surface along the flow, more pressure force is generated which has a component in the drag force direction. It also shown that the size of trailing edge influenced the drag generated.

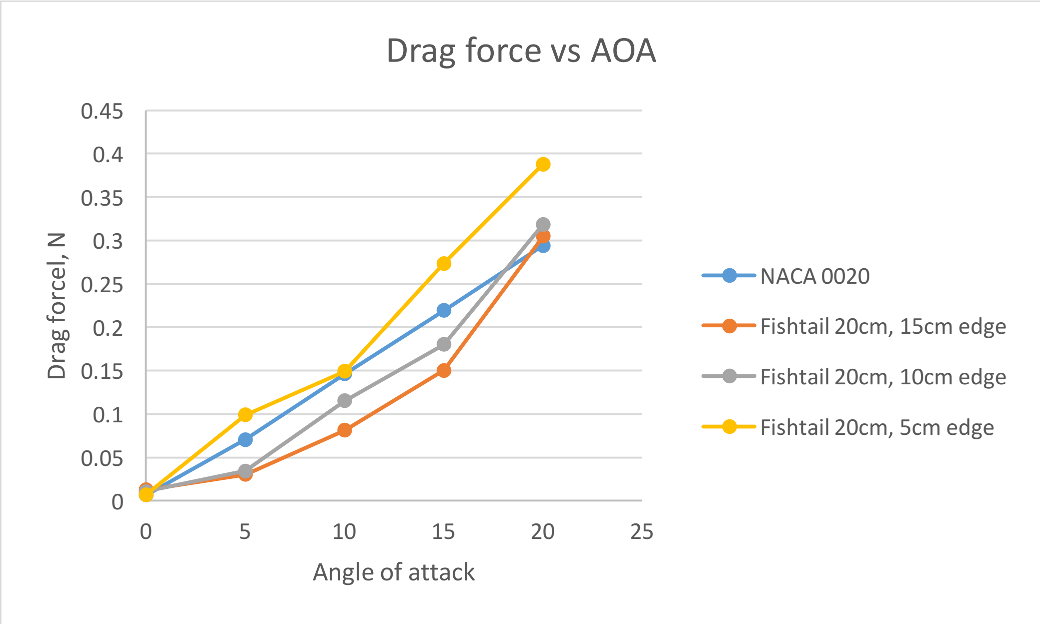

Figure 4-4: Comparison of drag generated against AOA for fishtail models

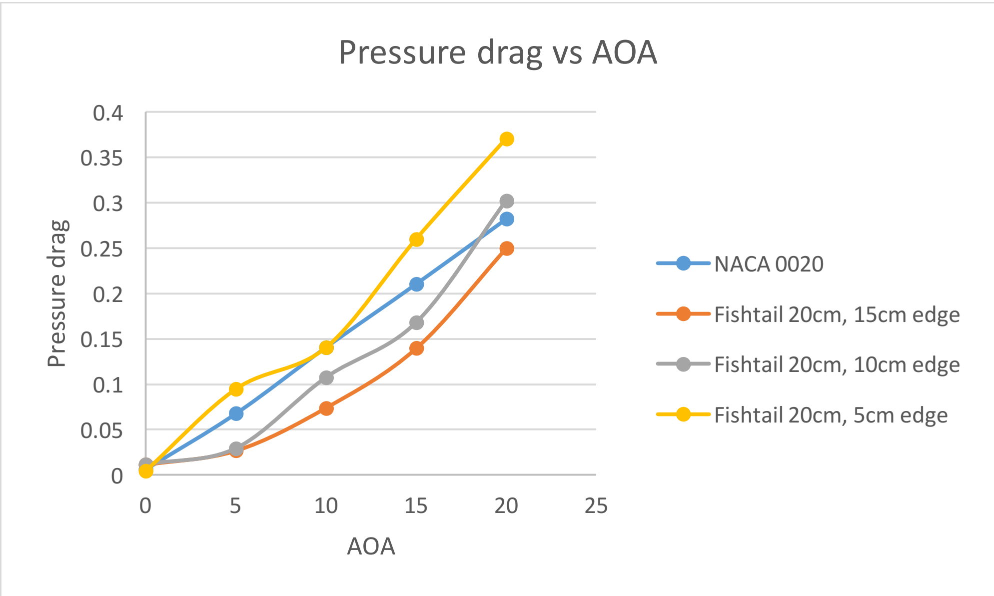

Figure 4-5 : Comparison of pressure drag generated against AOA for fishtail models and NACA 0020

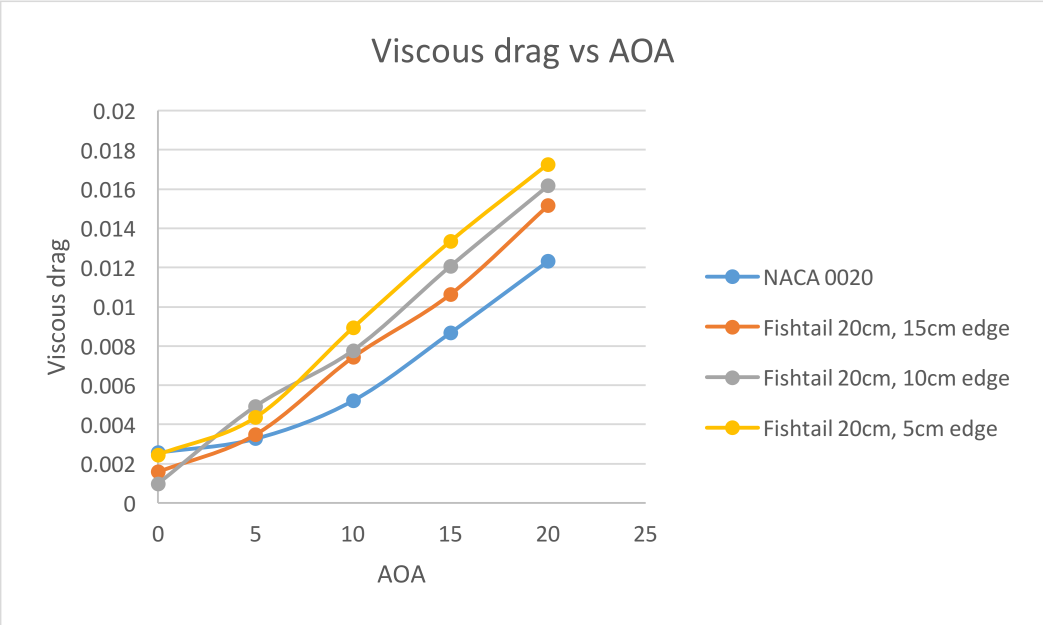

Figure 4-6 : Comparison of viscous drag generated against AOA for fishtail models and NACA 0020

Based on Figure 4.5 and Figure 4.6, it can be seen that the pressure drag has much higher contribution to total drag of rudder compare to viscous drag which is slightly less important on the total drag.

Effects of Angle of Attack (AOA) on the lift force

Relationship between lift force and AOA are show in figure 4.1, figure 4.2 and figure 4.3. The analysis is done only through simulation in ANSYS Fluent. Generally as the AOA increased from 0°- 20° , the values of lift force increased as well. The AOA increased gradually due to the low cruising speed of all the AUGs which was set at 0.45 m/s. Based on the figure 4.2, fishtail of 15cm trailing edge achieved the highest lift value at 8.12N when the angle of attack is at 20 °. This shows that larger trailing edge also possible to produce more lift as well.

Effects of Angle of Attack (AOA) on the drag force

Meanwhile for drag force, generally it is force generated in opposite direction movement. In this simulation, velocity in vertical direction is kept constant. 0.45m/s is set to be the velocity on horizontal direction. The drag force increased as the angle of attack increased either in the positive angle or the negative angles.

Based on Figure 4.6, skin friction drag is highest for all models when angle of attack at 20 degree. The highest is achieved by fishtail of 5cm trailing edge. The viscous drag is basically increased when the surface area of the body is larger. Thus, this shows that skin friction drag is affected by the flow of the fluid around the glider body.

Chapter 5 : CONCLUSION

5.1 Conclusion

In this paper, the hydrodynamic characteristics of the three fishtail rudder models are investigated and compared with the NACA 0020 one.

Based on the simulation result obtained in ANSYS Fluent:

- The force generated increased when the angle of attack is increased
- Fishtail rudder exhibit higher value of lift and drag force compare to conventional rudder, NACA 0020
- Appropriate size of the trailing edge can increases the lift while the drag does not increase so much.

REFERENCES

[1] J. G. Graver, R. Bachmayer, N. E. Leonard, and D. M. Fratantoni, "Underwater glider model parameter identification," in *Proceedings of the 13th International Symposium on Unmanned Untethered Submersible Technology*, 2003.

[2] D. Joe, R. S. Shankar, R. Vijayakumar, and A. Das, "Concept Design of Autonomus Underwater Glider," *International Journal of Innovative Research and Development|| ISSN 2278–0211,* vol. 1, pp. 176-189, 2012.

[3] N. A. Ali Hussain, T. M. Chung, M. R. Arshad, R. Mohd-Mokhtar, and M. Z. Abdullah, "Design of an underwater glider platform for shallow-water applications," *International Journal of Intelligent Defence Support Systems,* vol. 3, pp. 186-206, 2010.

[4] D. Bingham, T. Drake, A. Hill, and R. Lott, "The application of autonomous underwater vehicle (AUV) technology in the oil industry–vision and experiences," in *Proceedings of the International Federation of Surveyors' 22nd Congress*, 2002.

[5] P. Stevenson, M. Furlong, and D. Dormer, "AUV shapes-combining the practical and hydrodynamic considerations," in *Oceans 2007-Europe*, 2007, pp. 1-6.

[6] A. Molland, "Rudder design data for small craft," 1978.

[7] P. Stevenson, M. Furlong, and D. Dormer, "AUV design: shape, drag and practical issues," *Sea Technology,* vol. 50, pp. 41-44, 2009.

[8] T. Van Nguyen and Y. Ikeda, "Development of fishtail rudder sections with higher maximum lift coefficients," in *The Twenty-fourth International Ocean and Polar Engineering Conference*, 2014.

[9] D. Paster, "Importance of hydrodynamic considerations for underwater vehicle design," in *OCEANS'86*, 1986, pp. 1413-1422.

[10] J. Liu and R. Hekkenberg, "Sixty years of research on ship rudders: effects of design choices on rudder performance," *Ships and Offshore Structures,* pp. 1-18, 2016.

[11] H.-J. Kim, S.-H. Kim, J.-K. Oh, and D.-W. Seo, "A proposal on standard rudder device design procedure by investigation of rudder design process at major Korean shipyards," *Journal of Marine Science and Technology,* vol. 20, pp. 450-458, 2012.

[12] M. Sadraey, "Design of Control Surfaces," *Aircraft Design: A Systems Engineering Approach,* pp. 631-753, 2012.

[13] T. Van Nguyen, "A Numerical Study on 3-D Effects of Marine Rudder by Using CFD."

[14] A. Phillips, M. Furlong, and S. Turnock, "Virtual planar motion mechanism tests of the autonomous underwater vehicle Autosub," 2007.

[15] A. Phillips, M. Furlong, and S. R. Turnock, "The use of computational fluid dynamics to determine the dynamic stability of an autonomous underwater vehicle," 2007.