# Performance Analysis of a 2-Blades Propeller Design for USV (Unmanned Surface Vehicle)

Rochmad Eko Prasetyaning Utomo[1], Prabuditya Bhisma Wisnu Wardhana[1], Mega Lazuardi Umar[1], Agung Fauzi Hanafi[1]

[1] Politeknik Negeri Banyuwangi, Banyuwangi 68461, Indonesia
rochmad@poliwangi.ac.id

**Abstract.** USV (Unmanned Surface Vehicle) is a tool that can facilitate human work. Conducting surveys, searching in vulnerable areas, mapping, or even exploring for a long time can be done easily by USV. USV technology is currently experiencing very rapid progress. However, this sophistication cannot be separated from USV's main need, which is to move. To move the USV requires a force which is commonly referred to as thrust. The required thrust is usually generated through a certain instrument, in this case the propeller. Different propeller design would produce different thrust. Likewise with the obstacles from the shape of the USV building itself, it will also provide different obstacles. So it is necessary to do a study to find out which propeller fits a certain USV shape. Every USV needs special propeller design based on the USV design to optimize the energy consumption. In this observation an adjustment will be made to the pitch of the propeller. So that the maximum thrust required by USV can be obtained. In this study the performance of propeller will be observe. With the motor that can produced various rpm. Modeling will be carried out with measurable parameters for each design that has been made, then an analysis will be carried out to find out which design has the best thrust. The results of the observations show the increase of power are influenced by the increase of rpm. Maximum increase of power has been surpassed the power of stock model propeller

**Keywords:** Pitch Blade Analysis, 2 Blades Propeller, Unmanned Surface Vehicle.

## 1 Introduction

USV (Unmanned Surface Vehicle) is a vehicle that operates on surface water and can be controlled remotely. USV can be used to explore some place that cannot be reached normally by human. USV can be controlled to collect or record natural phenomena. USV can be operated automatically with the help of algorithm and program. Thrust is needed to move USV from one place to another. The bigger thrust generated would make the USV moved with more speed. More speed means the USV needed more powerful motor. Powerful motor and a fine propeller are needed to produce the speed of USV. Thrust can be produced from thruster or propeller. In this study we would conduct a simulation from a propeller design to determine how much the thrust can be generated.

Several research has been conducted to investigate performance of propeller. Some research were done using numerical approaches. The others used experimental approaches to investigate performance of propeller. Experimental approaches needed towing tank. Which is very expensive, needed more times and must be conducted with complex procedure for every experimental configuration. Based on the study that has been conducted []. Computational Fluid Dynamic (CFD) are the one of alternative that offer much advantages. Such as simulation can be conducted in actual geometry, ex- treme condition of model can be simulated too, harsh condition of flow can be simulated and the result can have error under 10% from experimental data. Using CFD can reduced the time required to do analysis. And with the support of the newest hardware the simulation can produced more detailed result.

In these study we used rotational speed from 1400 – 1800 (rpm) to do the simulation. The combination of rpm and pitch will produce effect. The effect will be shown and discussed. From the result we can determine the performance of 2 blades propeller design based on rpm and pitch of propeller.


F. Kurniawan et al. (eds.), *Proceedings of the Adisutjipto Aerospace, Science and Engineering International Conference (AASEIC 2024)*, Advances in Engineering Research 258,
https://doi.org/10.2991/978-94-6463-668-0_7

## 2 Research Methods

In this study, we try to simulate the flow around the propeller. From propeller design we got the propeller model. Next we would to simulate the water flow around the propeller when the propeller works. From the simulation we could determine the total thrust and torque from various rpm. The rpm that used in this study based on DC motor that used. In this study the DC motor can works up to 5000 rpm. We would use various rpm to determine how much thrust and torque can be produced from the combination of propeller and the DC motor. From the result we could take a conclusion which rpm that produce maximum thrust from the propeller.

### 2.1 Propeller Design

In this study we used 2 blades propeller model to get the best performance. The dimension of the propeller can be seen in table 1. From the dimension in table 1. We made the 3D model using CAD software. After that the 3D model propeller will be used to do the CFD (Computational Fluid Dynamic) analysis to get the phenomena that happen in the propeller. With different RPM we will study the phenomena that happen in the blades of propeller.

**Table 1.** Propeller Dimension

| **Parameter** | **Dimension** |
|---|---|
| Diameter (cm) | 6 |
| Pitch | 30,40,50 |
| Blade Number | 2 |

### 2.2 Model Design

Propeller 3D model made using CAD software. The 3D model were made based on data in table 1. The 3D model can be seen at Fig. 1 bellow. The 3D model bellow was used in the CFD simulation to acquire the performance of the propeller. Various pitch were used in the simulation. The result of the simulation shown that pitch gave influence in the performance of the propeller.

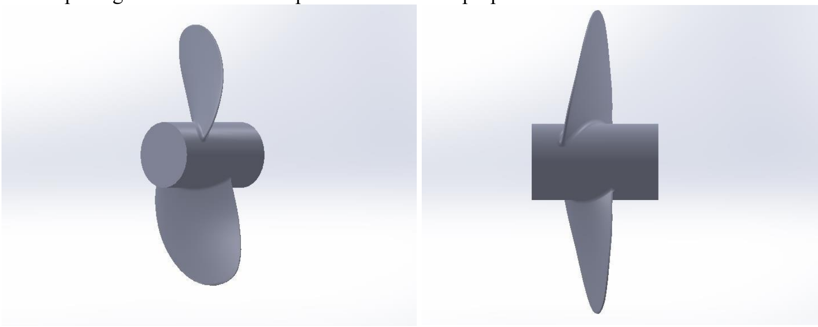

**Fig 1.** Propeller 3D Model

### 2.3 Domain Setting

CFD simulation needs some parameter and initial condition setting to represent the real condition of operation. In these study the initial condition were propeller submerge in the water and rotating with certain rpm. From these initial condition we must made setting in the software such as domain setting. Domain setting were important to determine how far the area that affected when the propeller rotating or operating. The domains setting for these study can be seen at Fig. 2.

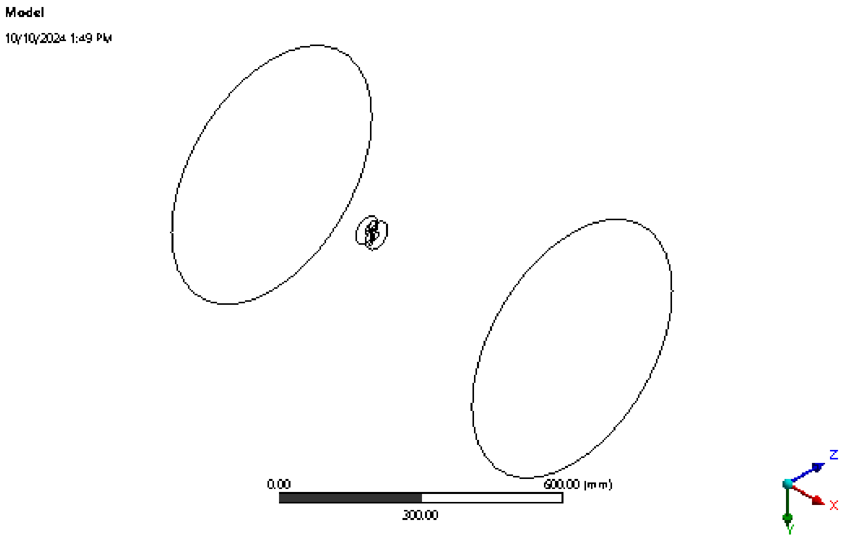


**Fig 2**. Domain Setting in CFD

### 2.4 Mesh Setting

After the domain were set we need the determined the nodes and the elements from the 3d model. This process were called meshing. In meshing we must determine the number and the size of mesh. The size of mesh would change the result. More refined the mesh the result would be more accurate. Not only the size but also the number of mesh would make the result more accurate.

In these study the mesh sizing can be seen at Fig. 3. The nodes were used are shown in Fig. 3. The mesh that shown were the mesh that has been refined. Some of the refined mesh are located at the edges of the 3D model of propeller. At the edges some mesh were more refined then the others area. At the edges needs more refined to accommodate the extreme dimension at the edges.

| Statistics | |
|---|---|
| Nodes | 26895 |
| Elements | 144036 |

**Fig 3**. Nodes and Elements

In the setting we must check and input the setting below. The setting below were used to determined general setting of mesh that used. The setting are shown in the Fig. 4.

The result of mesh setting are shown in the Fig. 5. The result in wire mesh view are shown below. Based on the figure we can see the difference in size of the elements that used. In the domain the elements were bigger than in the propeller. These shown the sizing of mesh that has been set in the setting. The propeller has more refined mesh because the propeller have the smaller area than the domain area. To acquire more accurate result the small area were mesh with smaller size mesh.

| Sizing | |
|---|---|
| Size Function | Curvature |
| Relevance Center | Coarse |
| Transition | Slow |
| Span Angle Center | Fine |
| Curvature Nor... | Default (18.0 °) |
| Min Size | Default (0.616530 mm) |
| Max Face Size | Default (61.6530 mm) |
| Max Tet Size | Default (123.310 mm) |
| Growth Rate | Default (1.20 ) |
| Automatic Mesh ... | On |
| Defeature Size | Default (0.308270 mm) |
| Minimum Edge L... | 4.8501e-005 mm |

**Fig 4.** Mesh Setting

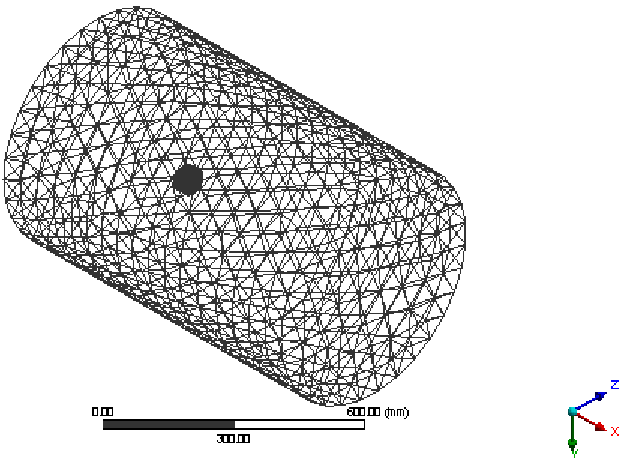


**Fig 5**. Result of Mesh

### 2.5 Governing Equation

The simulation were carried out from application of Navier-Stokes equation using finite element method. In these simulation the propeller is fixed and the fluid is moving.

2 Blades propeller model are usually tested in open water test to determine the performance of propeller. From the CFD the test result usually are Thrust (T), Torque (Q) and Efficiency (□). For the performance data usually are Thrust Coefficient (KT), Torque Coefficient (KQ) and Advance Ra- tio (J) that can be acquired from Thrust Coefficient (KT) and Torque Coefficient (KQ). The equation are shown below.

$$K_T = \frac{T}{\rho n^2 D^2} \tag{1}$$

$$K_Q = \frac{Q}{\rho n^2 D^2} \tag{2}$$

$$J = \frac{V_A}{nD} \tag{3}$$

$$\eta = \frac{J}{2\pi}\frac{K_T}{K_Q} \tag{4}$$

## 3 Results and Discussion

The analysis of 2 blades propeller performance in various rotational speed was performed by CFD simulation. The result are shown below. The simulation were performed in steady state. Fig. 6. Below shown the vector that through the propeller. When the propeller are rotating the fluids around will moved as the influence of the propeller. The movement of the fluids that will resulting in thrust. Where the thrust were needed to make the boat advanced. Every fluids that moved as the influenced by the propeller will make the movement as shown in the Fig. 6. Various colors were made to identified the speed of the fluids movement. With the red color were the highest speed and the blue color were the lowest speed.

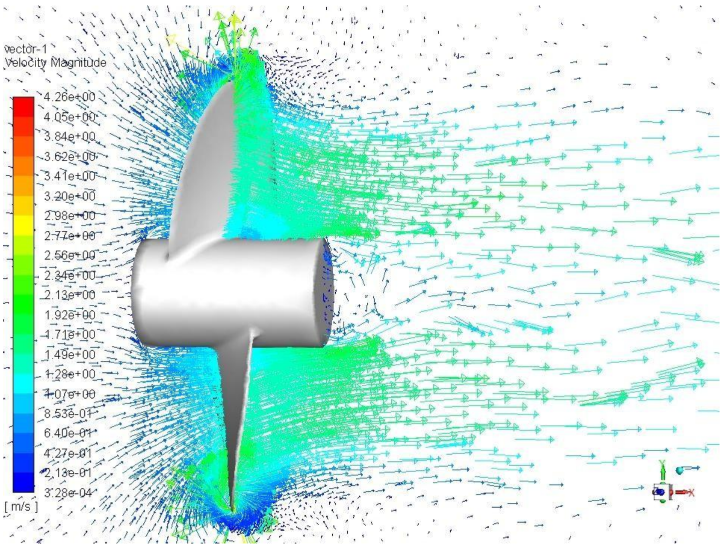


**Fig 6**. Vector Movement

The result of the simulation shown the domain that influenced by the rotation of the propeller. From Fig. 6 below we can see the area that influenced by propeller. The area that influenced by the propeller mainly area behind the propeller and some at the side of the propeller. The area were formed from the fluids that moved as the effect of the rotation of propeller. From the Fig. 6 we can see the area that mostly affected from rotation of propeller. The mostly affected area were marked with the red color. The blue color marking the area that less influenced by the rotation of propeller.

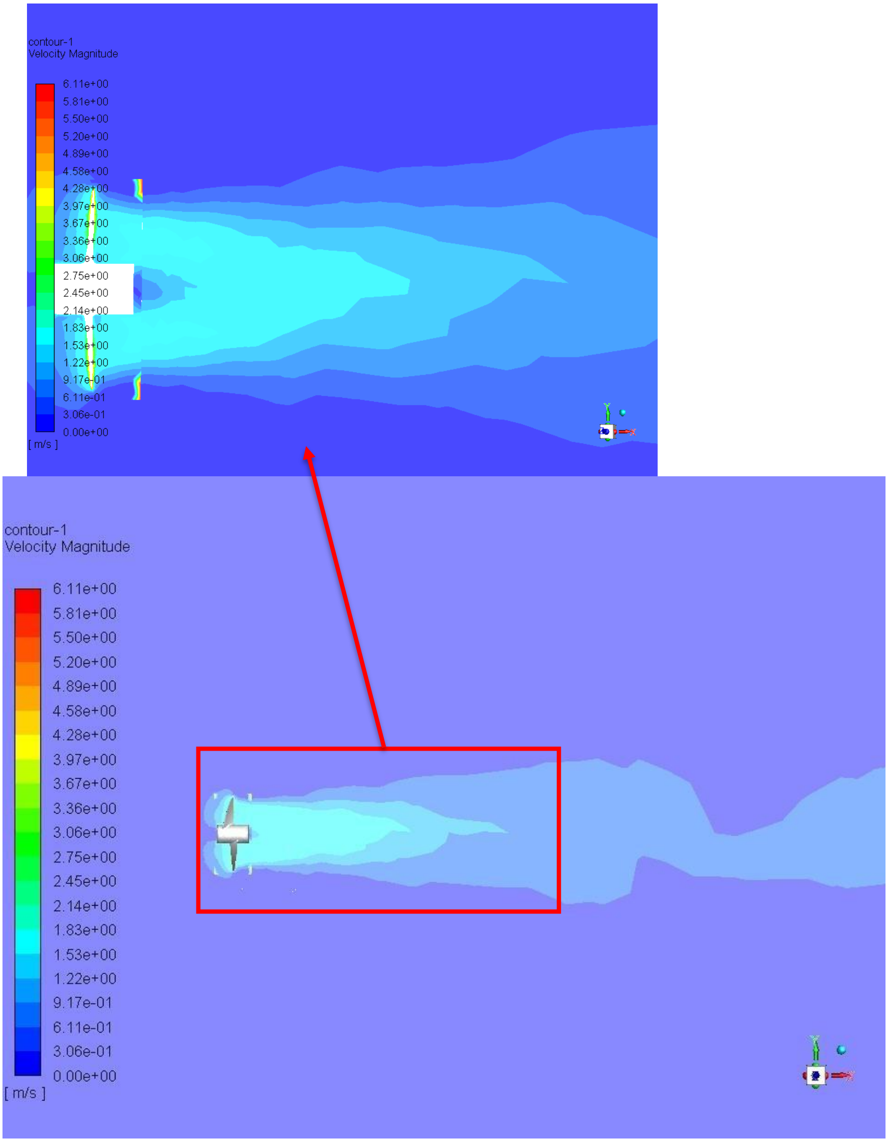


**Fig 7.** Area of Domain

The simulation has generated the result as shown in the Fig. 8 below. From the Fig. 8 we can see the thrust of the propeller are increasing. Pitch that used in these study shown an increasing thrust following the increase of the pitch value. From these simulation the increase of thrust still not reaching the maximum value.


**Fig 8.** Thrust

Power that can be generated from the propeller design were shown at the Fig. 9 below. Fig. 9 shown us the power that can be produced from a 6 cm diameter propeller. The power showing an increasing value following the increasing value of pitch. From the simulation we know that these propeller design can produced more power. With increasing the pitch the power generated increasing too. These phenomena is shown in the equation (1).


**Fig 9.** Power

Propeller performance can be seen from thrust, power and F/P. in these simulation to acquire the value of F/P we need to do calculation from equation (4). The result from the equation are shown in Fig.10. From the Fig 10 we can see the efficiency of the propeller are reducing with the increasing rpm. These shown that the propeller mostly have high efficient at low rpm. From these result we can see the propeller performance are more efficient at low rpm. The reducing efficient are showed at a minimal value. Although there is a decreasing trend, but the value of decrease are not much. The value can be seen at the table 2 below.

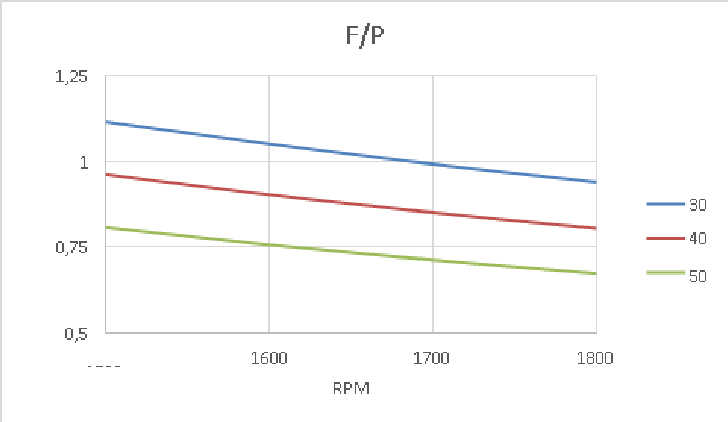


**Fig 10.** Efficiency

**Table 2**. Thrust, Momen, Power , P/F for various rotational speed

| Pitch | rpm | F(N) | M(N) | P(W) | F/P |
|---|---|---|---|---|---|
| | 1500 | 0,992404 | 0,005665 | 0,889816 | 1,115291 |
| **30** | 1600 | 1,137125 | 0,006457 | 1,081869 | 1,051074 |
| | 1700 | 1,286186 | 0,007281 | 1,29614 | 0,99232 |
| | 1800 | 1,445072 | 0,008154 | 1,53703 | 0,940172 |
| | 1500 | 1,453306 | 0,009615 | 1,510273 | 0,96228 |
| **40** | 1600 | 1,65727 | 0,010952 | 1,835107 | 0,903091 |
| | 1700 | 1,875849 | 0,012374 | 2,202873 | 0,851547 |
| | 1800 | 2,107111 | 0,013876 | 2,615542 | 0,805612 |
| | 1500 | 1,827231 | 0,014392 | 2,260651 | 0,808277 |
| **50** | 1600 | 2,082016 | 0,016406 | 2,748847 | 0,757414 |
| | 1700 | 2,354629 | 0,018537 | 3,300004 | 0,713523 |
| | 1800 | 2,643612 | 0,020793 | 3,919428 | 0,674489 |

The various value as result from the simulation are shown at table 2. From table 2 we can see the exact result from simulation. There are increasing or decreasing value but the change are not more than 2%. These condition shown that not much performance different between the model that simulated in this study. From the table 2. We can see there are reducing value in efficiency with the increasing value of thrust and increasing value or rpm. These condition show us the thrust and rpm give a lot of influence to the propeller efficiency.

## 4 Conclusion

The result of this study shown an increasing Thrust with increasing pitch value. This condition are compatible with the equation (1). From equation (1) we can see the Thrust are influenced by the rpm. And this condition more influenced by the increased of the pitch value. The highest thrust can be seen at pitch 50 and 1800 rpm with the value 2.643612.

The highest power can be produced by the propeller is 3.919428 at the pitch of 50 and the rpm at 1800.

Beside that the efficiency of the propeller shown a reducing trend line. From each increasing and reducing shown the value changes are not more than 2 %. From the efficiency we can see the more power generated the more thrust generated the more efficiency will reduce. This condition shown that more power and more thrust will needs more energy consumption. Which is lead to more energy supply needed.

## References

1. Ahmad Fitriadhy, Nur Amira Adam, W. S. Kong, Faisal Mahmuddin, C. J. Quah, Prediction of Propeller Performance using Computational Fluid Dynamics Approach, EPI International Jour- nal of Engineering, Volume 2, Number 2, pp. 185-193, 2019.
2. Mizzi, K., et al., Design optimization of Propeller Boss Cap Fins for enhanced propeller perfor- mance. Applied Ocean Research. 62: p. 210-222. 2017.
3. Rahman, A., M.R. Ullah, and M.M. Karim, Marine Propeller Design Method based on Lifting Line Theory and Lifting Surface Correction Factors. Procedia engineerin7. 194: p. 174-181. 2017.
4. Fitriadhy, A., N. Razali, and N. AqilahMansor, Seakeeping performance of a rounded hull cata- maran in waves using CFD approach. Journal of Mechanical Engineering and Sciences. 11(2): p. 2601-2614. 2017.
5. Kamal, I.M. and T.M.A.T.M. Yusof, A CFD RANS Cavitation Prediction for Propellers. 2017.
6. Fitriadhy, A., et al., Computational fluid dynamics analysis on the course stability of a towed ship. Journal of Mechanical Engineering and Sciences, 11(3):p. 2919-2929. 2017.
7. Khameneifar F, Feng H-Y Extracting sectional contours from scanned point clouds via adaptive surface projection. Int J Prod Res 55(15):4466–4480.2017.
8. Arapakopoulos A, Polichshuk R, Segizbayev Z, Ospanov S, Ginnis A, Kostas K .Parametric models for marine propellers.Ocean Eng 192:106595. 2019.
9. Lothar Birk, Fundamentals of Ship Hydrodynamics: Fluid Mechanics, Ship Resistance and Pro- pulsion, 2019.
10. Ashna Khose. Design parameter and Manufacturing methids of Marine Propellers for Increased Efficiency in Ship Propulsion. International journal of Engineering Research and Applications. Vol 10 issue 9 (series-II).pp 56-61. 2020.