Indonesian Journal Of Maritime Technology (ISMATECH)
Vol. 1, Issue 1, June 2023
pISSN: 0000-0000, eISSN: 0000-0000)
DOI: https://doi.org/10.35718/ismatech.v1i1

Analysis of Effect of Propeller Rotational Speed Variations on Torque and Power Using Computational Fluid Dynamics (CFD) Method

Dicky Ocza Tama[1*], Daffa Kurnia Pradisty[1], Richard Marbun Hosea[1], Gad Gunawan[1]

[1]Department of Mechanical Engineering, Kalimantan Institute of Technology, Balikpapan, 76127, Indonesia

KEYWORDS
Propeller
Rotational speed
Power
CFD
Torque

ABSTRACT – The speed on the ship is inseparable from a good propeller design in order to get the optimal thrust generated by the propeller motion. The propeller is a very important part in determining the ship's motion. The propeller itself is a tool used to generate thrust that comes from engine power transmitted through the shaft. Computational Fluid Dynamics (CFD) is a very suitable method for analyzing complex systems that are difficult to solve using manual calculations. With these advantages, CFD is often used to analyze a pattern of a system, one of which is the propeller, which in this study uses variations in flow velocity and propeller speed which can affect the value of the torque and power produced by the propeller.

*Corresponding Author | Dicky Ocsa Tama | ✉ 03201027@student.itk.ac.id

INTRODUCTION

Many aspects affect and must be considered in achieving maximum speed on a ship, namely good hull planning and good engineering system planning, and not only that, the propeller is also one of the aspects that must be planned properly in order to achieve the ship's function goals in terms of speed [1]. The speed on the ship is inseparable from a good propeller design in order to get the optimal thrust generated by the propeller motion. The propeller is a very important part in determining the ship's motion [2]. The propeller itself is a tool used to generate thrust that comes from engine power transmitted through the shaft [3]. In other words, the propeller functions to convert engine power into encouragement according to the combination of RPM and speed. Propeller load characteristics can be displayed graphically by several coefficients in the form of sizes. The diagram gives Torque and Thrust as a function of speed [4]. Propellers are widely used in the aviation, maritime, and energy engineering industries. On submarines, propellers have special criteria, namely being able to provide large thrust and low noise levels [5-6].

At this time, a computer system-based method has been found that is capable of simulating and analyzing the flow of a fluid. With this method, it is easy to find out how much torque and power a propeller can produce, because a simulation is carried out and then the results can be analyzed. Computational Fluid Dynamics (CFD) is a very suitable method for analyzing complex systems that are difficult to solve using manual calculations. With these advantages, CFD is often used to analyze a pattern of a system. The development of viscous flow analysis around the propeller using Computational Fluid Dynamics (CFD) is getting better [7-8]. Based on the description above, the researcher wants to examine according to the title of the research entitled "Analysis of Effect of Propeller Rotational Speed Variations on Torque and Power Using Computational Fluid Dynamics (CFD) Method."

METHOD

This research was conducted using a simulation method using CFD software. The location for this research was carried out at the Kalimantan Institute of Technology Campus, Balikpapan City starting from February 2023 to June 2023. The tools used in this research were CAD software which functions to design propeller geometries, and CFD software which functions to carry out simulations and analyzes of the propellers studied. The procedures carried out in this study are as follows:

1. Look for references related to research
2. Designing the propeller using CAD that will be analyzed, inputting the geometry that has been made into the CFD software for the meshing and setup process.
3. Displays propeller results in CFD software.

Computational Fluid Dynamic (CFD)

Computational Fluid Dynamics (CFD) is a systems analysis that includes fluid flow, heat transfer, and related phenomena. Such as chemical reactions using computer-based simulations (numeric). The calculation controls are space division which is called meshing. Later, at each calculation control point, calculations will be carried out by the application with predetermined domain boundaries and boundary conditions. This principle is widely used in the calculation process using computer computing assistance. Computational Fluid Dynamics (CFD) is very suitable for analyzing complex systems that are difficult to solve using manual calculations. With these advantages, CFD is often used to analyze a system pattern[9]. The turbulence model used is k-w SST because this model is good in terms of modeling turbulence in the boundary layer and free stream [10].

Propeller

The propeller is a mechanical ship propulsion device that transfers force by changing rotational motion to thrust, the propeller consists of several blades and the propeller operates like a screw rotation. In its development, the propeller has undergone several modifications in form with the aim of obtaining good efficiency. The propeller produces a thrust that comes from the engine power which is transmitted through the shaft, in other words the propeller functions to convert engine power into encouragement according to the combination of RPM and speed [11].

Fluids

Fluids or liquids differ from solids because of their ability to flow. Fluid is a substance that continuously deforms when acted upon by a shear stress. Due to the bonding, the liquid flows more easily. Because the molecular bonds in fluids are smaller than the molecular bonds in solids, fluids do not experience significant resistance to deformation due to friction. There are two types of fluids, namely compressible flow and incompressible flow. Compressible flow is a fluid that can vary in density, for example a gaseous substance. incompressible flow is a fluid whose density level does not change or the change is very small and is considered non-existent, for example; liquid substance [12].

External Flow

Internal flows or channels are flows that are completely confined by a solid surface. External flow is the flow that occurs over an object immersed in an infinite fluid. Both internal and external flows can be compressed or uncompressed. Internal flow is fluid flow which is completely limited by the surface of a solid substance, while external flow is flow over objects that are immersed in an infinite fluid, but like the surface of objects that are limited by fluid flow. One example of internal flow is flow out of the pipe to the facet, while an example of external flow is the propeller [13].

RESULTS AND DISCUSSION

In this experiment using a flow velocity of 2 m/s and variations in propeller rotational speed of 10 rad/s, 12 rad/s, 13 rad/s, and 14 rad/s. This experiment was carried out using CFD software. The value of the propeller rotation speed is determined based on the propeller rotation reference that we got. This is done to see the results of the influence of variations in flow velocity and propeller speed on torque and power values. Experiments using variations of the propeller rotation speed of 10 rad/s, 11 rad/s, 12 rad/s, 13 rad/s and 14 rad/s obtained the following pressure contour results :

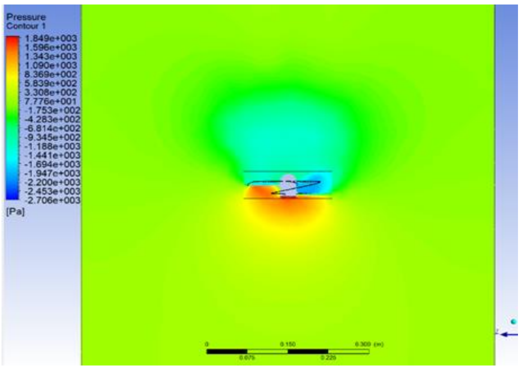

Figure 1 Contour pressure results of 10 rad/s

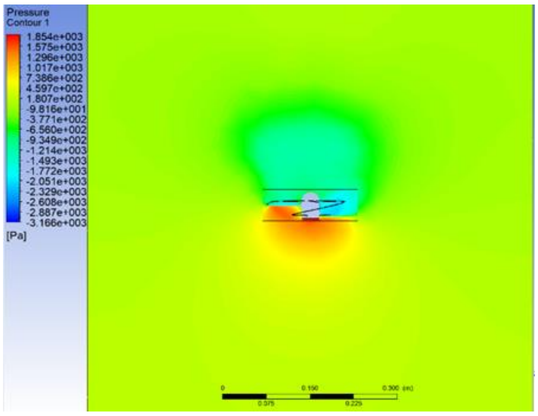

Figure 2 Contour pressure results of 11 rad/s

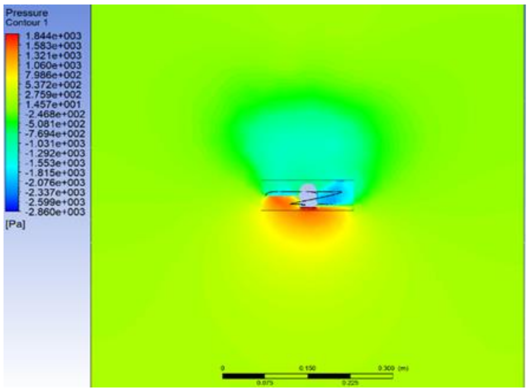

Figure 3 Contour pressure results of 12 rad/s

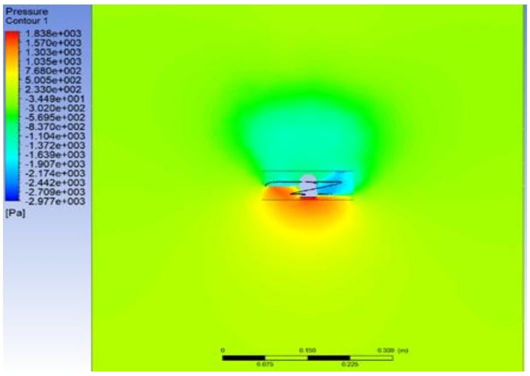

Figure 4 Contour pressure results of 13 rad/s

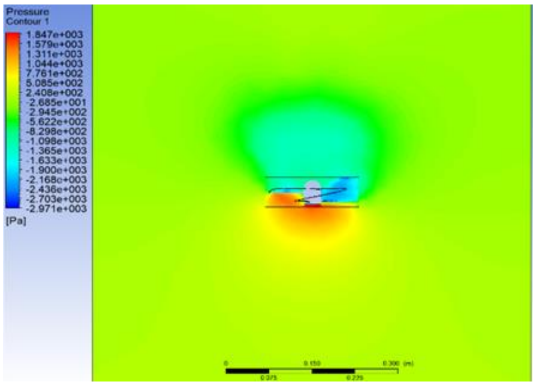

Figure 5 Contour pressure results of 14 rad/s

From Figure 1-5 it can be seen that the area of pressure distribution that occurs in the propeller area is marked in red indicating that this area is the area with the greatest pressure experienced by the propeller and followed by the yellow color because in that area the fluid velocity slows down due to the collisions that occur between the fluids. with the surface of the propeller and causes the pressure to increase, then the green color shows low pressure and blue is the lowest pressure area because in that area the fluid velocity flows quickly so the pressure drops, in accordance with the theory that if the velocity value is large then the pressure will little value.

Experiments using variations of the propeller rotation speed of 10 rad/s, 11 rad/s, 12 rad/s, 13 rad/s and 14 rad/s obtained velocity contour results as follows:

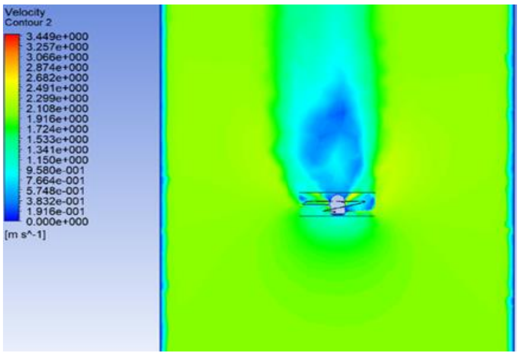

Figure 6 The result of the Velocity Contour 10 rad/s

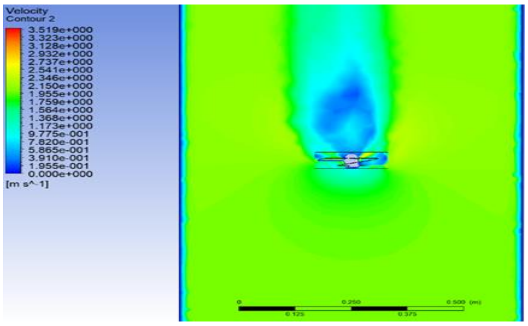

Figure 7 The result of the Velocity Contour 11 rad/s

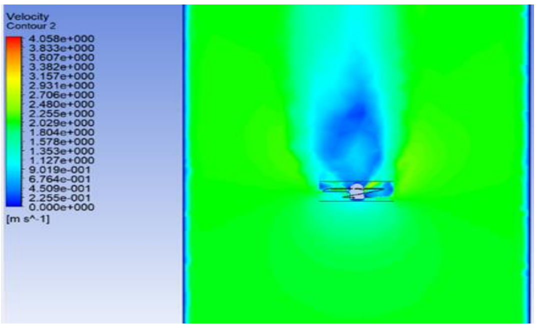

Figure 8 The result of the Velocity Contour 12 rad/s

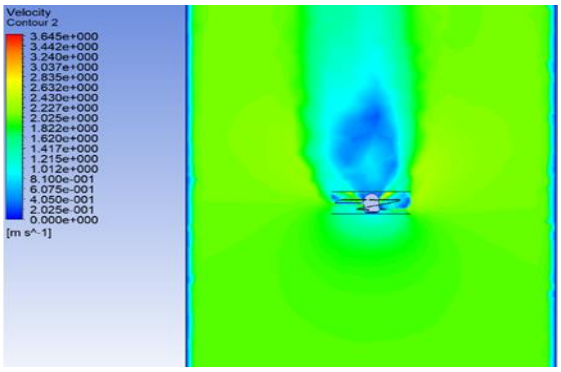

Figure 9 The result of the Velocity Contour 13 rad/s

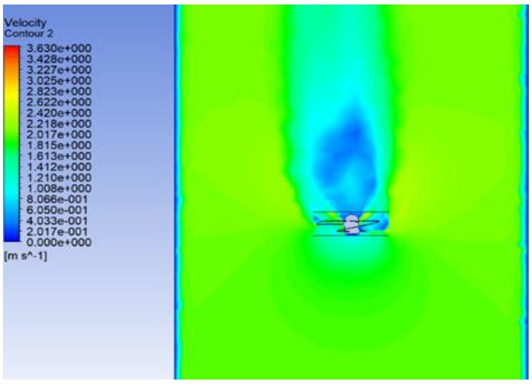

Figure 10 The result of the Velocity Contour 14 rad/s

In Figure 6-10 there is an area that is blue, the blue color shows the lowest speed, this is because in that area the fluid flows slowly. Then in the inward area there are green and yellow colors which indicate that the speed is starting to increase, indicating that the speed in the propeller area is the highest pressure, this is because the fluid that previously was flowing was being pushed due to the propeller.

Experiments using variations of the propeller rotation speed of 10 rad/s, 11 rad/s, 12 rad/s, 13 rad/s and 14 rad/s obtained the velocity Streamline results as follows:

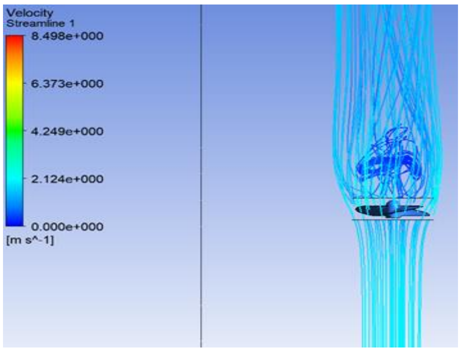

Figure 11 The result of the *Velocity Streamline* 10 rad/s

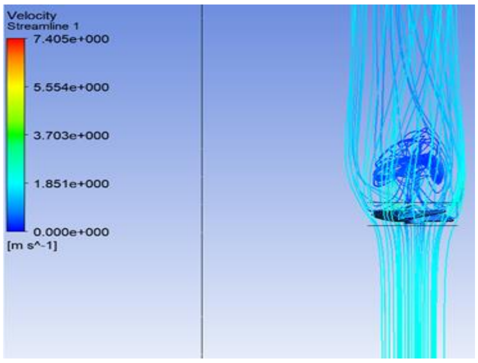

Figure 12 The result of the *Velocity Streamline* 11 rad/s

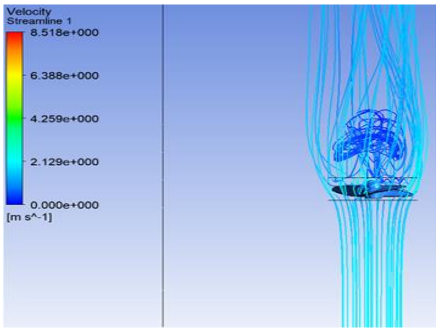

Figure 13 The result of the *Velocity Streamline* 12 rad/s

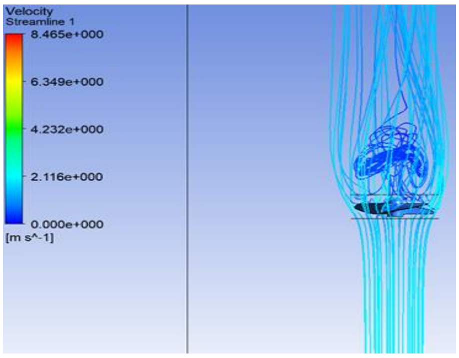

Figure 14 The result of the *Velocity Streamline* 13 rad/s

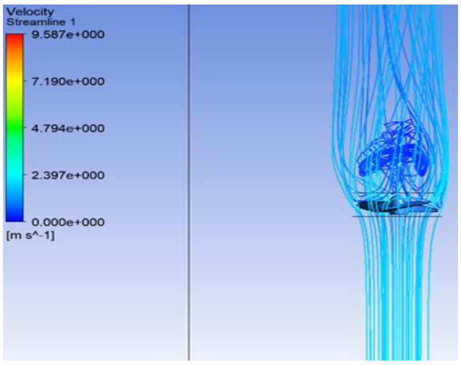

Figure 15 The result of the *Velocity* Streamline 14 rad/s

In Figure 11-15 it can be seen that the lines are made in such a way in the velocity field, so that at any time these lines will be in the same direction as the flow at every point in the flow field.

In this study, there were several variations of the propeller rotation speed of 10 rad/s, 11 rad/s, 12 rad/s, 13 rad/s, 14 rad/s with a flow rate of 2 m/s, the following data results were obtained:

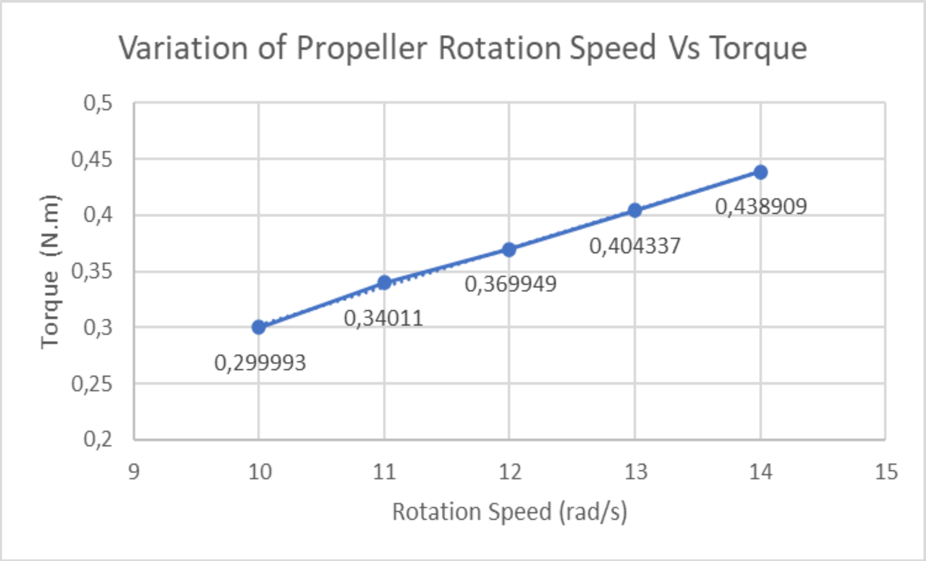

Figure 16 Variation of Propeller Rotation Speed Vs Torque

After carrying out the simulation, it can be seen in Figure 16 that the propeller rotation speed can affect the torque value where when the propeller rotation speed increases, the torque generated will also be greater, and vice versa if the propeller rotation speed decreases, the torque generated will also be smaller.

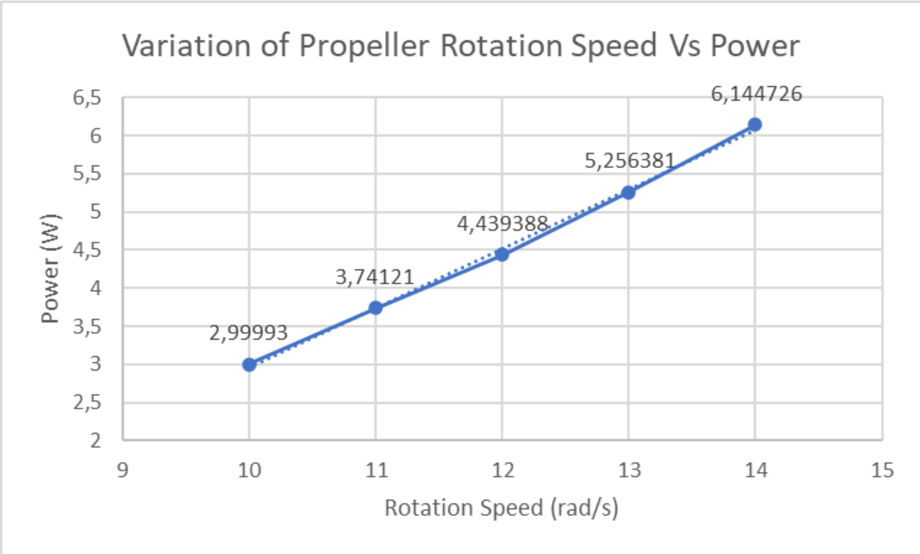

Figure 17 Variation of Propeller Rotation Speed Vs Power

After carrying out the simulation, it can be seen in Figure 17 that the propeller rotation speed can affect the power value, where when the propeller rotation speed increases, the power generated will also be greater, and vice versa if the propeller rotation speed decreases, the power generated will also increase small.

CONCLUSION

From the research it can be concluded that the propeller rotation speed has an influence on the value of torque and power. The flow speed used is 2 m/s and the variations in propeller rotation speed are 10 rad/s, 11 rad/s, 12 rad/s, 13 rad/s and 14 rad/s. when the propeller rotation speed increases, the torque and power generated will also be greater, and vice versa if the propeller rotation speed decreases, the torque and power generated will also be smaller.

ACKNOWLEDGEMENT

We would like to thank the Integrated Laboratory of the Kalimantan Institute of Technology for its support during this research process.

REFERENCES

[1] A. H. Muhammad, Syarifuddin, D. Paroka, S. Rahman, Wisyono and A. A. Pratama, "Performa Maneuvering Kapal Perikanan 30 GT dengan Konfigurasi Propeler Asimetrik," *Jurnal Ilmu dan Teknologi Kelautan Tropis* , vol. 9, no. 2, pp. 491-498, 2017. http://dx.doi.org/10.29244/jitkt.v9i2.19314 .

[2] A. R. Z. Abdillah, I. Malik and S. Effendi, "PERANCANGAN BERBASIS CAD-CAE PADA PROPELLER TIGA DAUN UNTUK PERAHU KECIL," *MACHINERY,* vol. 3, no. 1, pp. 1-8, 2022. https://doi.org/10.5281/zenodo.6413283 .

[3] S. C. Godge, A. M. Pandya, S. P. Shet and D. K. Amin, "Performance Analysis and Enhancement of Marine Propeller," *International Journal of Engineering Research & Technology (IJERT),* vol. 9, no. 2, pp. 104-108, 2020. DOI : 10.17577/IJERTV9IS020020.

[4] M. I. Nikmatullah, A. H. Muhammad, Haryanto and Zulkifli, "Pengaruh Jarak Rudder dan Propeller terhadap Kemampuan Thrust Kapal Perikanan 30 GT," *Jurnal Riset & Teknologi Terapan Kemaritiman (JRTK),* vol. 1, no. 1, pp. 22-25, 2022. DOI: 10.25042/jrt2k.062022.05.

[5] P. B. S. Budi, D. Chrismianto and G. Rindo, "ANALISA PERFORMANCE PROPELLER TIPE B-SERIES PADA KAPAL SELAM MIDGET 150M DENGAN VARIASI SKEW ANGLE DAN BLADE AREA RATIO (AE/AO) MENGGUNAKAN METODE CFD," *Jurnal Teknik Perkapalan,* vol. 4, no. 4, pp. 725-737, 2016.

[6] F. Nuriman, A. F. Zakki and H. Yudo, "Perancangan Propeller Kapal Selam Komersial Tipe B series X-80 Dengan Variasi Sudut Skew Dan Jumlah Daun Untuk Mengoptimalkan Nilai Thurst," *Jurnal Teknik Perkapalan,*

vol. 8, no. 3, pp. 414-425, 2020.

[7] N. Prakash, A. Muthuvel and D. Roychowdhury, "Numerical Simulation of a Three Bladed Marine Propeller in Steady and Unsteady State," *Applied Mechanics and Materials,* Vols. 592-594, pp. 1136-1141, 2014. https://doi.org/10.4028/www.scientific.net/AMM.592-594.1136.

[8] S. P. M. N and D. R. Nath, "A Computational Method for Determination of Open Water Performance of a Marine Propeller," *International Journal of Computer Applications,* vol. 58, no. 12, pp. 1-5, 2012. DOI : 10.5120/9331-3636.

[9] U. S. Dharma and G. Prasetyo, "PENGARUH PERUBAHAN LAJU ALIRAN TERHADAP TEKANAN DAN JENIS ALIRAN YANG TERJADI PADA ALAT UJI PRAKTIIKUM MEKANIKA FLUIDA," *Turbo : Jurnal Program Studi Teknik Mesin,* vol. 1, no. 2, pp. 106-115, 2012. http://dx.doi.org/10.24127/trb.v1i2.653.

[10] F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA Journal,* vol. 32, no. 8, pp. 1598-1605, 1994. https://doi.org/10.2514/3.12149.

[11] J. Jalaluddin, S. Akmal, N. ZA and I. Ishak, "ANALISA PROFIL ALIRAN FLUIDA CAIR DAN PRESSURE DROP PADA PIPA L MENGGUNAKAN METODE SIMULASI COMPUTATIONAL FLUID DYNAMIC (CFD)," *Jurnal Teknologi Kimia Unimal,* vol. 8, no. 1, pp. 97-108, 2019. https://doi.org/10.29103/jtku.v8i1.3396.

[12] P. B. Setyabudi, D. Chrismianto and G. Rindo, "ANALISA NILAI THRUST DAN TORQUE PROPELLER TIPE B-SERIES PADA KAPAL SELAM MIDGET 150M DENGAN VARIASI SKEW ANGLE DAN BLADE AREA RATIO (AE/AO) MENGGUNAKAN METODE CFD," *Kapal: Jurnal Ilmu Pengetahuan dan Teknologi Kelautan,* vol. 13, no. 3, pp. 109-118, 2016, https://doi.org/10.14710/kpl.v13i3.12352.

[13] P. J. Pritchard, Fox and McDonald's Introduction to Fluid Mechanics, New York: Wiley, 2011.