International Journal of Marine Engineering Innovation and Research, Vol. 8(4), Dec. 2023. 730-734
(pISSN: 2541-5972, eISSN: 2548-1479)

Preliminary Analysis of ROV AF-150114 Movement Using CFD Method (Computational Fluid Dynamics)

S. Manullang[1], A. Setiawan [2], R. Irvana[3], Mercy Patanda[4]

(Received: 07 November 2023 / Revised: 09 November 2023 /Accepted: 23 November 2023)

***Abstract*— This research was carried out with the aim of measuring the effect of the body on the fluid flow that occurs around it and measuring the stress that occurs in the ROV AF-150114 design. The research method uses software with a CFD model approach to analyze the stress that occurs in the designs made. In general, there are three stages that must be passed in a CFD simulation: Pre-processing. Solving and post-processing. What is calculated is the velocity, viscosity and pressure of the water flow around the ROV body. The results obtained show that the balance of the ROV body greatly affects the ability to dive and maneuver during maneuvers. The highest pressure/pressure for fluid flow occurs at the ROV AF-150114 speed of 2.5 m/s with a value 39,825 Pa and the value of viscosity is 10,860 Nm/s^2. ROV AF-150114 movement test results found that the experiment has a faster movement time speed than the calculation.**

Keywords— CFD, Fluid, pressure and body.

I. INTRODUCTION

The limited ability of humans to mapping Indonesia's underwater potential, especially for unexplored deep-sea areas. To explore and utilize this potential, a supporting device is necessary to help the exploration process and one of them is by using an ROV robot.

ROVs are underwater vehicles powered by electricity and controlled through a source, maneuverable according to human command with hydraulic or electric thrusters [1]. Another definition submitted by Christ and Wernli [2] ROV is a camera that is installed in a waterproof enclosure, with thrusters for maneuvering, attached to a cable to the surface where video signals are sent. An ROV receives energy and change information with a control panel located on the surface through a central cable. From the control panel, the operator can plan the job or use a single joystick to maneuver the vehicle directly. Now the application of ROV are widely diverse, such as the oil and gas industry, discovery, agriculture, marine biology dan military purpose [3][4][5][6].

To find out how the strength of the ROV design is approached using CFD software. CFD is a computer-aided method of numerically calculating, predicting and approaching fluid flow. Fluid flow in real life has many types and certain characteristics that are so complex, CFD approaches with numerical methods and uses fluid equations.

This research has been done in several stages at UNSADA, at the beginning this research was only to design the proper design for the ROV , after two studies were carried out, the existing design was considered to be the best [7], so the next step is to investigate how the fluid flow reacts around the ROV body when it is operated, because this flow will affect the strength of the construction of the ROV.

The purpose of this research is to measure the influence of the body (body) in the performance of underwater robots (underwater) and find out where the location of the greatest stress that occurs on the body of the Y-Type ROV design made.

II. METHODOLOGY

The research method uses software with a CFD model approach. The definition of a model is actually choosing which equations to activate in a CFD process. There are many equations used in the general CFD concept because all of them are an approximation of fluid characteristics that will bring it closer to real conditions.

1. The Mass Conservation Equation is the general equation of mass conservation and is valid for any compressible and incompressible flow.
2. Momentum Conservation Equation. The momentum conservation equation is an equation that defines fluid motion when forces are applied to its particles at each fluid element defined in the CFD model.

Shanty Manullang Department of Naval Architecs, Faculty of Ocean Engineering, Darma Persada University, Jakarta, 13450, Indonesia. E-mail: shantymanullang77@gmail.com

Agus Setiawan Department of Naval Architecs, Faculty of Ocean Engineering, Darma Persada University, Jakarta, 13450, Indonesia. E-mail: laborashanty@yahoo.com

Rizky Irvana Department of Naval Architecs, Faculty of Ocean Engineering, Darma Persada University, Jakarta, 13450, Indonesia. E-mail: rizky4568@gmail.com

Mercy Patanda Department of Fisheries Resources Utilization, Faculty of Fisheries and Marine Science, Satya Negara Indonesia University, Jakarta, 12240, Indonesia E-mail: mercypatanda@usni.ac.id

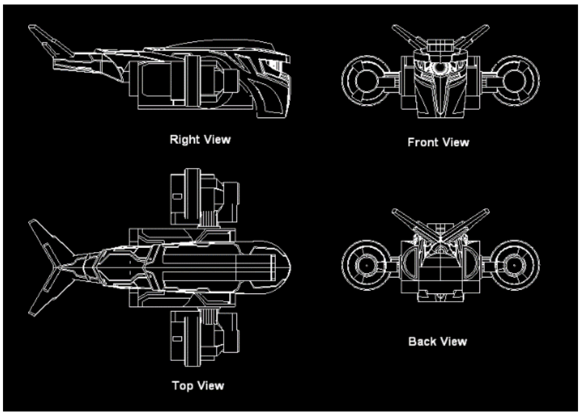

Figure 1. 2 D ROV View

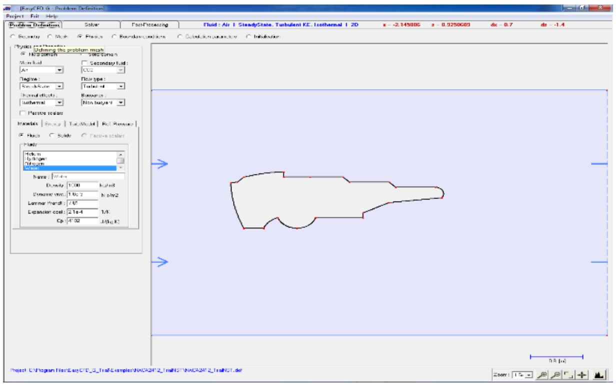

Figure 2. Results of data input in CFD-G

Simulation Stages with CFD

In general, there are three stages to go through in a CFD simulation [8]:

Pre-processing. At this stage, a geometry model is built with a CAD (Computer Aided Design) package, an appropriate mesh or grid is constructed, and boundary conditions and fluid and solid properties are applied.

Solving. This is the core stage of CFD, at this stage, the solution is calculated based on the conditions applied in the pre-processing stage.

Post-processing. The last stage in CFD. At this stage, interpretation of the simulated data is done which can be in the form of visualization of contours, vectors, curves, histograms, and so on.

There are three common benefits of CFDs that are widely recognized: insight, foresight and efficiency.

Insight - Deep Understanding. There are many systems that we want to study, but are difficult to prototype or difficult to test, such as the respiratory organs and arterial system. With CFD, virtual prototypes can be created which can enhance our understanding of events involving fluid flow.

Foresight - Prediction. Basically, CFD is used for prediction, with CFD we can easily answer typical 'what if' questions by changing the parameters, geometry, and boundary conditions to get an optimal design.

Efficiency. With the help of CFD, the process of designing a system becomes more cost-effective and efficient, in terms of cost, effort, and time. With CFD, research time can be shortened and research costs can be cut.

Result Analys

In this research, the process of drawing the ROV is through AutoCAD. Then the image is exported in the form of an ACIS file in "sat" form, so that it can be read in the CFD-G software (Figure 1)

Figure 2 is the result of data input after performing CFD simulation. There are several speeds that will be carried out in the simulation process which will later be known the value of Thrush, drag and CD on the ROV Body.

Object simulation

CFD simulation is carried out to determine the state of the body when the fluid flow with a predetermined speed. Some of the speeds taken in the simulation process are as follows:

- 0.5 m/s, with ROV weight 2.289 kg
- 1 m/s, with ROV weight 2.289 kg
- 1.5 m/s, with ROV weight 2.289 kg
- 2 m/s, with ROV weight 2.289 kg
- 2.5 m/s, with ROV weight 2.289 kg

The above will be used to simulate the fluid flow as shown below:

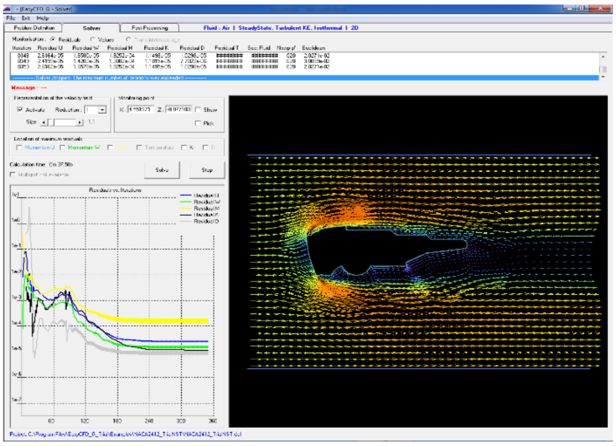

Figure 3. Simulation process in CFD-G

III. RESULT AND DISCUSSION

1. **Velocity**. Speed or in English is velocity is a quantity that states the trajectory (direction of displacement) per unit time.

The maximum velocity at the exit from the thrusters, the backward fluid flow behind the underwater ROV main body and indicate the magnitude and orientation of the free stream velocity that corresponds to the ROV motion velocity. The contour line appeared caused by fluid mass flow rate through the model. The colors gradation shows difference velocity when fluid flow across the body [9]

Figure 3 shows the color difference in fluid flow velocity and various areas of flow at the front located on the upper and lower sides of the ROV AF-150114 body which can be found with a value of 15.335 to 16.277 m/s, and the maximum velocity are in the side of the body. This velocity contour displayed actual fluid flow velocity. It meant the velocity of fluid flow travelled significant distance per time. Even though has same unit with superficial velocity (m/s), water velocity has detailed in small region. The figure shown on rear body mini-ROV has significant contour, fluid velocity fell dropped compared to other areas. Rear body has transom formed, where body has streamlined which lead the fluid particle fully transmitted far away behind body hence suddenly suffered small turbulence [10]. The ROV AF-150114 motion velocity is here as the input parameter to the CFD simulation program through the velocity components in X- and Z-axis directions.

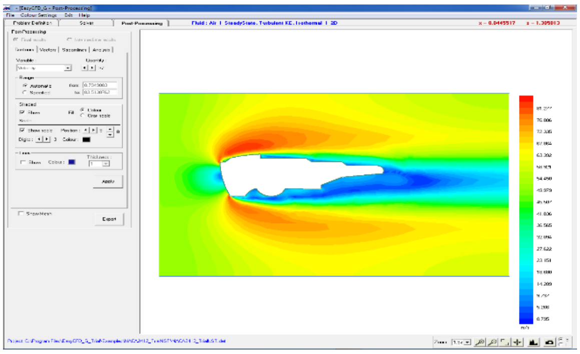

Figure 4.

2. **Pressure**. Pressure is a term for the amount of force applied to a given surface area. The engineering unit of measure for pressure is a combination of the force unit quantity and the area quantity. For example, kilograms per square centimeter = Kg/cm^2 or Pounds per Area (pa).

Figure 4 shows the color difference Pressure or pressure on the front of the ROV body which can be found with a value of 31.97 x 10^3 Pa to 39.82 x 10^3 Pa. As expected, the highest pressure is on the leading side of the ROV. The thrusters also substantially affect the pressure distribution.

The hydrodynamic pressure forces acting upon the underwater ROV are then obtained by the integration of the pressure over the whole ROV surface. Components of these forces in the axial and cross flow directions determine the axial and crossflow hydrodynamic coefficients respectively used for calculating the quadratic damping coefficients in the Fortran ROV motion model.[11]

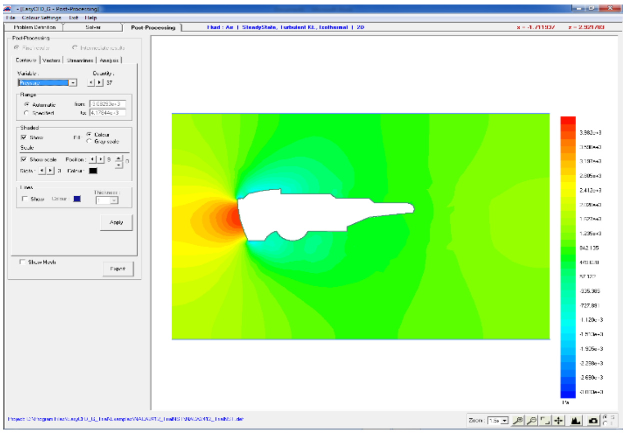

Figure 5. Pressure simulation results in CFD-G

3.**Viscosity**. Viscosity is the viscosity of the resistance to water to flow at a certain temperature. If a liquid flows easily, it means it has a low viscosity, a liquid that does not flow easily means it has a high viscosity. Figure 5 shows the difference in color Viscosity or viscosity on the back of the ROV body after passing the fluid viscosity in the ROV body area can be known with a value of 9.66 N m / s^2 to 10.86 N m / s^2.

From the pictures of the simulation model above, we get the value at a speed of 1 m/s so that we can know the data previously presented in the ROV simulation that has been simulated before. This data will facilitate the motion analysis. The following data is obtained in the CFD-G simulation.

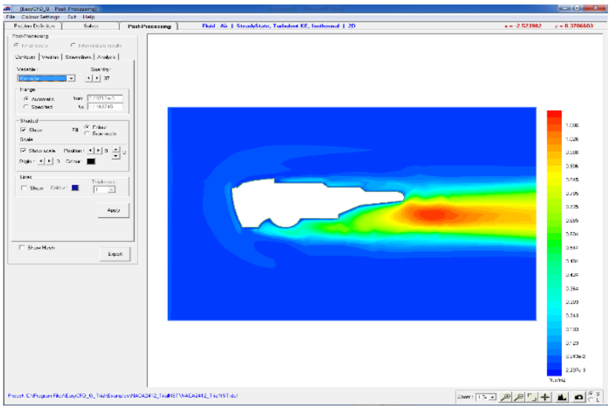

Figure 6. Results of viscosity simulation process in CFD-G

TABLE 1.
TESTING ROV FORWARD MOVEMENT IN CFD-G

NO	Speed (m)	Velocity (m/s)	Preassure (Pa)	Viscocity (Nm/s^2)
1	0.5	6.318	7.234	1.932
2	1	8.277	10.275	3.864
3	1.5	10.273	14.684	5.620
4	2	13.122	21.913	8.430
5	2.5	16.277	39.825	10.860

TABLE 2.
ROV MOVEMENT TESTING

No	Command Name	Result	Distance	Time (s)
1.	Forward	Works	1 m	**9,2**
2.	Reverse	Works	1 m	**13,4**
3.	Left	Works	180˚	**10,8**
4.	Right	Works	180˚	**10,3**
5.	Rotate	Works	360˚	**18,7**
6.	Pump Inlet (Down)	Works	1 m	**7,6**
7.	Pump Outlet (Up)	Works	1 m	**8,4**

The motion performance of underwater robots to move as desired as maneuvering horizontally to move forward and move backward and maneuver vertically to move up or move down is because of the thruster. Thruster is one of the main components as a driving force in the operation of the underwater robot [12]. From the underwater trial tests, the ROV AF-150114 was able to perform all maneuvering tests (table 2) like moving forward, reverse, left, right and submerge. The ROV AF-150114 move backward is slower than move forward at the same distance of 1 meter. The problems in these maneuvering tests were large turning radius and slow reverse speed. The distance between the two thrusters quite is close to each other, resulting in a large turning radius.

TABLE 3.
ROV FORWARD MOVEMENT TESTING

No	Distance (m)	Calculation (s)	Trial (s)	Error (%)
1	0.5	5	5.6	0.280
2	1	10	9.2	0.920
3	1.5	15	12.8	1.920
4	2	20	16.24	3.248
5	2.5	25	20.33	5.083

From the above figures and tables in the table 3 Shows the difference in ROV movement test values from calculations and experiments. The conclusions are as follows: the experiment has a faster movement time speed than the calculation.

IV. CONCLUSION

1. The fluid flow velocity and various areas of flow at the front located on the upper and lower sides of the ROV AF-150114 body which can be found with a value of 15.335 m/s to 16.277 m/s, and the maximum velocity are in the side of the body.
2. The highest pressure is on the leading side of the ROV AF-150115, and the pressure value is 31.97 x 10^3 Pa to 39.82 x 10^3 Pa
3. Viscosity or viscosity on the back of the ROV AF-150114 body after passing the fluid viscosity in the ROV body area can be found with a value of 9.66 N m/s^2 to 10.86 N /s^2.
4. ROV AF-150114 movement test results found that the experiment has a faster movement time speed than the calculation.

REFERENCES

[1] C. Joochim, R. Phadungthin and S. Srikitsuwan, "Design and development of a Remotely Operated Underwater Vehicle," 2015 16th International Conference on Research and Education in Mechatronics (REM), Bochum, Germany, 2015, pp. 148-153, doi: 10.1109/REM.2015.7380385.
[2] R. L. Wernli and R. D. Christ, “Observation Class ROVs Come of Age,” 2009.
[3] V. Raoult et al., “Remotely operated vehicles as alternatives to snorkellers for video-based marine research,” J. Exp. Mar. Bio. Ecol., vol. 522, 2020
[4] Rahimuddin, H. Hasan, H. A. Rivai, Y. Iskandar, and P. Claudio, “Design of Omni Directional Remotely Operated Vehicle (ROV),” 2018.
[5] R. Jones and J. Drabble, “U.K.’s submarine rescue efforts,” Sea Technol., 1993
[6] L. Bezanson, S. Reed, E. M. Martin, J. Vasquez, and C. Barbalata, “Coupled control of a lightweight ROV and manipulator arm for intervention tasks,” 2017.
[7] S Manullang, A.Pusaka and A.Setiawan, The preliminary of Design and movement of Remptly Operated Vehicle (ROV), IOP Confrence Series : Eart and Enviromental Science, 2020
[8] Jiyun Tu, Guan Heng Yeoh and Chaoqun Liu, “Computational Fluid Dynamic”2008
[9] M.S. Baital, F. Mangkusasmito, M.A. Rahmawaty, “Specification Design and Performances Using Computational Fluid Dynamics for Mini-Remotely Operated Underwater Vehicle”, Jurnal system computer, Vol 14 no 1 June 2022
[10] F. D. Kurniawati and I. K. A. Pria Utama, “An Investigation into the Use of Ducktail at Transom Stern to Reduce Total Ship Resistance,” IPTEK J. Proc. Ser., 2017
[11] Thai Nguyen, D., V. Horák, H. Thu Tran, L. The Nguyen And C. Quang Hoang. A Motion Model for a Complex-Shaped Remotely Operated Underwater Vehicle. Advances in Military Technology, 2020, 15(2), pp. 343-353. DOI 10.3849/aimt.01403.
[12] Abu Sharkh S., Harris M.R., Crowder R.M., Chappel P.H. & Stoll R. L. & Sykulski, J. K., Design consideration for electric drives for the thruster of unmanned underwater vehicles, 6th European Conference on peer Electronics and Application, Sevill