



Design and Hydrodynamic Characteristics of Remotely Operated Vehicle (ROV) Using Computational Fluid Dynamics

Mostafa Ali Taher Allam, Soliman Ahmed Ahmed, Omar Maher Hafez, Mohand Hussien
Department of Ships and Submarine, Military Technical College, Egypt

Supervisor: Col. Dr. Mohamed Ibrahim Bennaya, Commandor. Dr. Mohamed Mostafa.

Abstract. Remotely operated vehicles (ROVs) are being used worldwide for various tasks such as environmental survey, structure inspection, pipeline tracking, mine hunting, hull cleaning, and even in the recovery of wrecks. The analysis of their hydrodynamic characteristics is important. The easiest way to obtain hydrodynamic characteristic is by simulation by using software of Computational Fluid Dynamic (CFD). Hydrodynamic characteristics can be obtained by fluid dynamic visualization and simulation of CFD. In this study, the hydrodynamic design process of the robot body is described in detail. Optimal body design process with compromises among conflicting design requirements is given. The results showed an increase in the hydrodynamic characteristics of the optimal ROV design indicated by the value of drag, pressure and velocity contours and calculation of stability. Analysis of hydrodynamics characteristic with the consideration of velocity and pressure contour was done by using software of ANSYS.

Key words-- CFD, ROV, Drag force, ANSYS.

I. INTRODUCTION

There are many types of underwater robots including remotely operated vehicles, unmanned underwater vehicle, nowadays, solar-powered autonomous underwater vehicles (SAUVs). Their usage covers exploration of underwater resources as hydrocarbons exploration and military situations as spying. The underwater robots are usually called remotely operated vehicles. The most important techniques of ROV is cleaning hull of the ship. The body of large ships in service need to continuously cleaned to make safety inspection and to reduce the total resistance

and so the total drag force. A vehicle that runs with a clean underwater body surface can save up to 6% of fuel consumption and so the total cost [2]. Cleaning of the hull also saves effectiveness of antifouling paint and saves the life span of the paint on the hull. The most effective way to clean hulls with large body is to use remotely operated vehicles and to reduce the drag of the ship and so reduce fuel consumption. Using ROVs also reduce labour cost. Computational Fluid Dynamics (CFD) was used for prediction and verification of flow effects on ROV movement, such as drag force resistance and lift force values and distribution around the hull of ROV. Simulations performed make us to realise water flow upon ROV, reduce the cost of several prototypes, eliminate rework and delays, and save the time and the costs of development. By knowing the thrust force, the maximum velocity of our ROV could be obtained. Most of the CFD calculations is solved with the standard $k-\epsilon$ model [3], the design of ROV body begins from a simple preliminary model. There is a comparative study on the fluid dynamic behavior is done with many models. In this study, the optimum shape of the body that satisfies the design requirements, from hydrodynamics point of view, is determined. Analysis of the flow field around the body is made with computational fluid dynamic (CFD) techniques using turbulence models,

standard and RANS $k-\omega$ SST models using ANSYS fluent software calculations.

II. NUMERICAL METHOD

The physical laws that govern flow of fluid: Conservation of momentum, Conservation of Mass, and Newton's Second Law. However, these equations can hardly solve analytically for all kinds of problems. One alternative is to solve them numerically using Computational Fluid Dynamics (CFD). In real life, fluids flow is turbulent and it can be modelled by RANS technique. As the number of equations are greater than the number of unknowns, RANS needs some extra terms (model) to solve the equations.

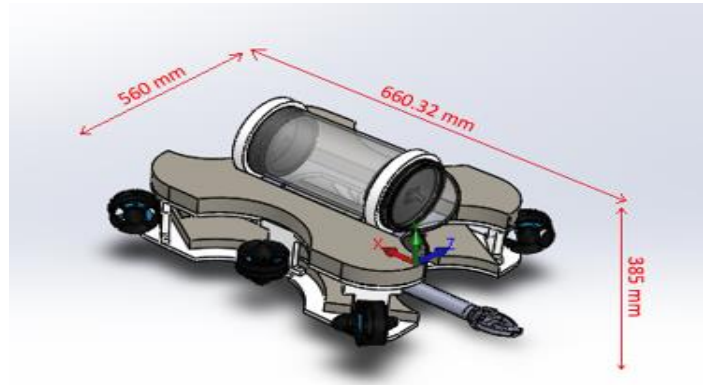
Generally, a $k-\omega$ sst model, developed by ANSYS fluent, is added in this study. This model is greatly used in industrial applications and provides a good compromise between robustness, computational cost and accuracy for such cases.

Using ANSYS fluent to estimate the drag resistance of the ROV and the steps will be as following:

- A. Geometry and computational domain.
- B. Meshing.
- C. Results.

A. Geometry and computational domain

Fig.1. represents the main dimensions of the remotely operated vehicle geometry. the hull geometry of the whole remotely operated vehicle presented into Ansys design modeler. The dimensions of the domain are also effective it will determine the mesh size and therefore the number of nodes, having a direct impact in the total computational time.



BOUNDARY CONDITIONS

The computational region and boundary conditions are clearly shown in Fig.2. The length of ROV in the direction of flow be X. Then the length from the inlet to the aft of the body is $4X$, the length from the rear of

Fig.1. main dimensions of ROV.

the body to the outlet is $8X$, and both the height and width of the computation region are $4X$ [4]. The condition of inlet is determined by the speed of flow. The sides, bottom and top are symmetry to avoid effect of walls on flow.

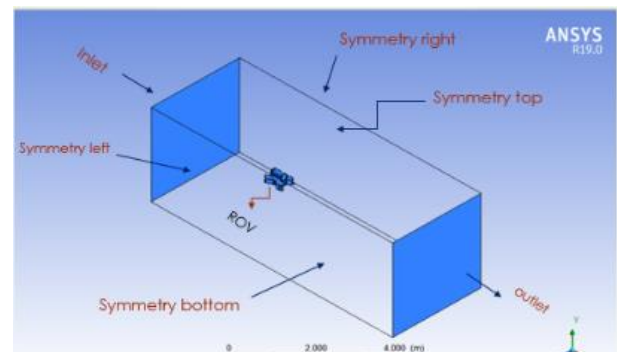


Fig. 2. computation region and boundary conditions.

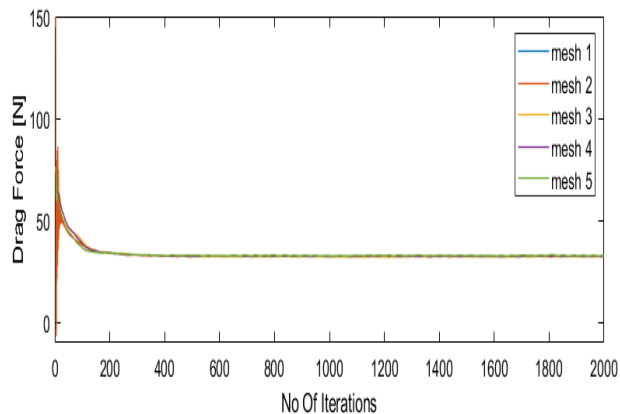
B. Meshing

Mesh verification is done to get the best quality of results, using refining factor $1/\sqrt{2}$ with 2000 iterations. The simulation of flow was presented by using the average equation of the Navier-Stokes. the results estimated do not change. the force on the x-axis has the lowest convergence the results are:

Element size (mm)	Pressure force (N)	Viscous force (N)	Total force (N)	No. of elements
7.07106	30.8499	1.95491	32.809	905849
5	30.527536	1.9535117	32.352	1315233
3.5355	30.3122	2.0405	32.352	1840532
2.5	30.3885	2.109	32.498	2509437
1.25	30.354	2.035	32.389	3504857

The best mesh quality is taken at element size = 3.535 mm with maximum skewness = 0.83

Mesh no.	1	2	3	4	5
(MM)	7.07106	5	3.5355339	2.5	1.25



C. Results

For the preliminary design shown in fig.4. After simulation it is found that there is a stagnation point refers to high pressure drag and so there is increase in total resistance.

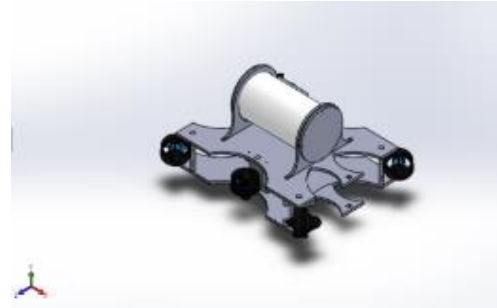


Fig. 4. preliminary design.

The resulting force is at 1 m/s:

Pressure force (N)	35.5496
Viscous force (N)	1.27084
Total force (N)	36.790464

fig.5. represent the pressure distribution at 1m/s at midsection of ROV

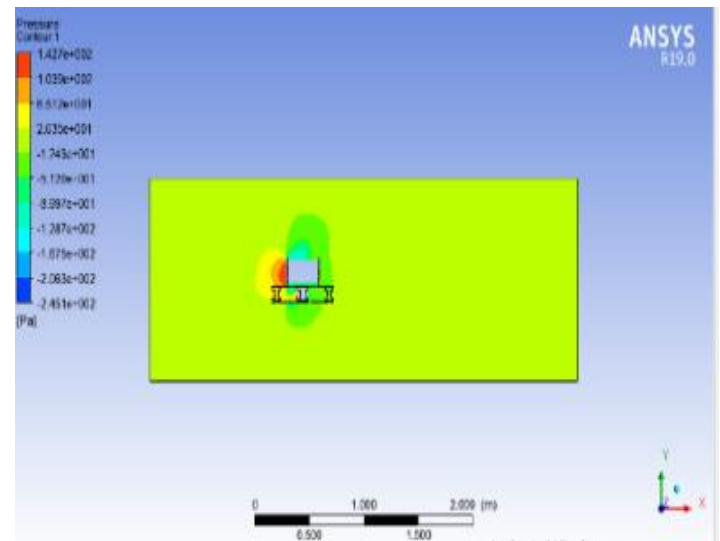


Fig. 5. Pressure distribution at 1 m/s.

fig.6. represent the velocity distribution at 1 m/s.

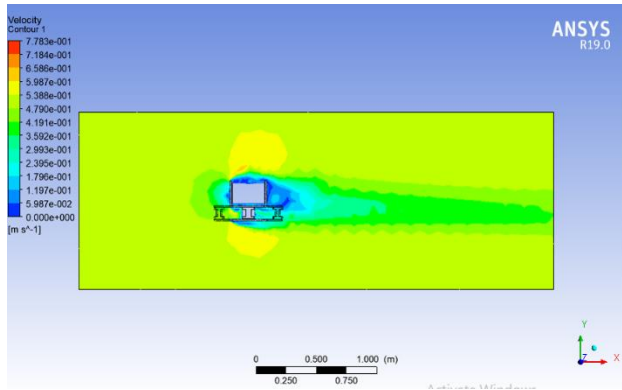


Fig.6. velocity distribution at 1m/s.

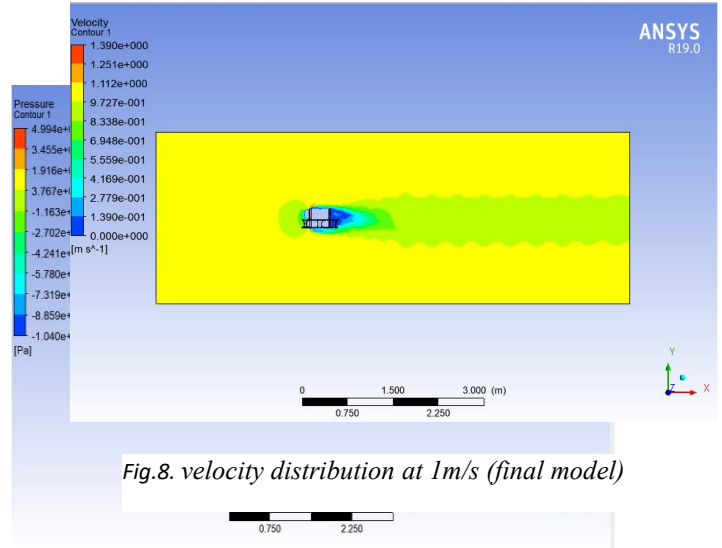


Fig.8. velocity distribution at 1m/s (final model)

for the final model after adding a dome it is found that the total resistance is reduced:

Pressure force (N)	31.202
Viscous force (N)	1.01
Total force (N)	32.212

Fig.7. represents pressure distribution at 1m/s and the stagnation point is reduced

Fig.8. represents the velocity distribution at 1m/s for the final model.

The results obtained from the simulations for different speeds:

Velocity (m/s)	.5	.7	1	1.5	2
Drag force (N)	8.166	15.786	32.212	72.021	127.912

Fig.8. velocity distribution at 1 m/s (final model).

Fig.7. pressure distribution at 1 m/s (final model)

[5] H. Zhang, "Using CFD Software to Calculate Hydrodynamic Coefficients," *Mar. Sci. Appl.*, vol. 9, pp. 149–155, 2010.

III. CONCLUSION

This paper presents design and numerical simulation of the underwater ROV in ANSYS Fluent. Hydrodynamic design is done and showed the comparison between the preliminary design and the final model, the design of ROV is modified to reduce the total drag force and there are no clear stagnation points, the drag force resistance is reduced by 12.5 %. This reduction will improve the performance of ROV and so reduce the power needed to operate the ROV. A set of experimental measurements will be performing to validate the model.

IV. REFERENCES

[1] Satria et al. "Hydrodynamic analysis of Remotely Operated Vehicle (ROV)" *Observa tion* *Fig.9. drag force at different velocities.*

Class using CFD" IOP Conf. Series: Materials Science and Engineering 645 (2019) 012014 IOP Publishing doi:10.1088/1757-899X/645/1/012014

[2] Larson, Scott, "Design and Construction of Unmanned Surface Vehicles" (2015). Theses and Dissertations. 2675. <http://preserve.lehigh.edu/etd/2675>

[3] K. Amjith, V. Arundas, M. Harikrishnan, J. Sebastian, and V. Vineeth, "Underwater Search and Rescue Device," *Int. J. Innov. Res. Sci. Technol.*, vol. 1, no. 11, pp. 481–488, 2015.

[4] Man Hyung Lee et al. "Hydrodynamic design of an underwater hull cleaning robot and its evaluation". *Inter J Nav Archit Oc Engng* (2012) 4:335~352 <http://dx.doi.org/10.2478/IJNAOE-2013-0101>

