IJEAST

INTERNATIONAL JOURNAL
OF ENGINEERING APPLIED SCIENCE
AND TECHNOLOGY

VOLUME : 10 ISSUE : 01 Print / Issue Publication Date: 30-Jun-2025

ISSN : 2455-2143

DOI : 10.33564/IJEAST.2025.v10i01.024

Indexed In

WWW.IJEAST.COM

editor@ijeast.com

International Journal of Engineering Applied Sciences and Technology, 2025
Vol. 10, Issue 01, ISSN No. 2455-2143, Pages 164-168
Published Online May 2025 in IJEAST (http://www.ijeast.com)

NUMERICAL PREDICTION RESISTANCE AND FLOW FIELD FOR SUBMARINE USING CFD METHOD

Le Dinh Dung
Faculty of Marine Engineering
Vietnam Maritime University, Hai Phong, Vietnam

Abstract— **This paper reports on the numerical simulation of a submarine's resistance under submerged conditions using a CFD approach. The study also discusses how the mesh size influences the simulation outcomes. To verify the reliability of these computed results, a comparative analysis is performed against experimental data from towing tank tests. In addition, the article details the characteristics of the flow field around the submarine, including the pressure and skin friction distribution on submarine surface, which are vital for addressing challenges such as optimizing the hull design to reduce resistance. The research employs the DARPA SUBOFF – the US submarine model.**

Keywords— **Submarine, Resistance, CFD, DARPA SUBOFF**

I. INTRODUCTION

Resistance prediction is a typical and indispensable problem generally in ship design and especially in submarin design. Its result is an input data of the propulsion system design, identify the ship main engine power to achive the design speed. However, it also serve other problems such as: optimizing the submarine hull shape with respect to the resistance.

Computational Fluid Dynamics (CFD) has been extensively utilized in numerous studies worldwide for analyzing hydrodynamic behaviors of ships in general [1-4], and for forecasting the performance of submarines in particular [5-8]. In the work of Budak, Gokhan and Beji, Serdar, [9] their have calculated the submarine resistance and studied the affect of different hull form on its resistance to identify an optimal form of submarine. The selected model using in their research was an US DARPA-SUBOFF submarine model. To confirm the reliable of the obtained result, a comparison was made between it and the towing tank test result. Mark Bettle, Serge L. Toxopeus in [10], studied the computational simulation of shallow water on the hydrodynamic chracteristics of Walrus submarine using CFD method. The simulation results in some research cases was compared with the experimental results and was agree well with these. In the work [3], the group of authors Pan Yu-cun, Zhang Huai-xin has studied and calculated hydrodynamic parameters of submarines in deep diving mode using CFD. The US DARPA-SUBOFF submarine model was used as a research object in this research. The obtained computed results were very close to the measured data in the towing tank.

Nowadays, with the strong development of electronic computer as well as CFD computation software, CFD has become an efficency tool for designers in solving many complicated hydrodynamic problems with much more precise results than using the semi- empirical formulas. As a result, our research applied CFD method with the support of Star-CCM+ solver in simulating the flow around the hull and predicting the resistance of US submarine model DARPA-SUBOFF.

II. SIMULATION

A. Submarine model

The submarine used in this paper is the DARPA SUBOFF, which was developed and tested by the Carderock Division of the Naval Surface Warfare Center (CDNSWC) in collaboration with the Hydronautics Ship Model Basin (HSMB). Experimental data obtained from tests on the DARPA SUBOFF model are employed to validate the CFD simulations of hydrodynamic resistance and to investigate the flow characteristics around the submarine hull. The primary geometric specifications and 3D configuration of the DARPA SUBOFF model are presented in Table 1 and Figure 1. The experimental dataset referenced in this study was originally published in [11].

B. Input data

The flow simulation around the submarine hull was conducted under conditions replicating those of the physical model tests in a controlled towing tank environment. The water properties were set with a density of $\rho = 998.67$ kg/m³ and a kinematic viscosity of $\nu = 1.08 \times 10^{-6}$ m²/s, assuming a smooth surface with zero roughness. The simulated speeds of the submarine ranged from 3.05 m/s to 9.15 m/s. [11].

Table -1 Main parameters of submarine model DARPA SUBOFF

Main parameters	Symbol	Value
The maximum length of the model	Lmax (m)	4.356
Hull diameter	D (m)	0.508
Length of sail	L_{sail} (m)	0.368

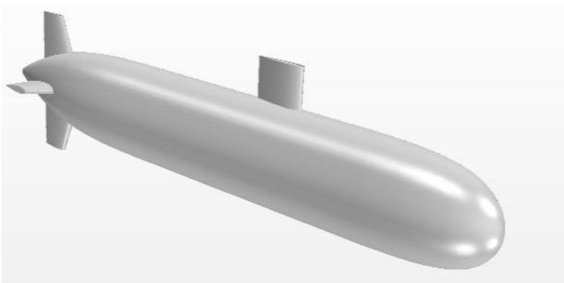


Fig. 1.3D hull form of submarine model DARPA SUBOFF

C. Setting computation

When the submarine dives in deep water (the distance from the bottom of the submarine to the seabed is larger than a haft of the submarine length and the distance from the highest point on the sail to the open surface is larger than a third of the submarine's length [12]), there is no effect of the open surface and the depth on its resistance [12]. In this case, the computational fluid domain will have only one phase, the liquid phase (water) [13]. Then, the size of the calculated fluid domain is determined as shown in Figure 2. The length of the virtual test tank is 4.5 times larger than the vessel length. In which, the longitudinal distance in front of the model of the test tank is 1.5L from the bow of the ship; the longitudinal distance behind the vessel of the virtual test tank is 3.0 L from the stern. The width of the virtual test tank is 2.5 times the length of the submarine measured in the center plane of the submarine. The top and bottom of the virtual test tank are located at a distance from the vessel equal to 9 times the submarine's depth D.

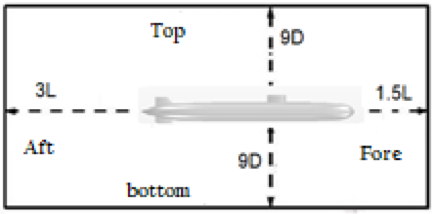


Fig.2. Dimension of the virtual tank

The boundary conditions applied in the resistance simulation for the submarine operating in submerged mode are defined as follows for the computational domain representing the virtual towing tank, the boundary condition is the velocity inlet in the front, pressure outlet in the aft end, top, bottom, side – symmetry plane. For submarines, the boundary condition used is a No-slip wall. After creating a virtual test tank. The next step is to create the mesh. In this research, the hexagonal mesh type is used to divide the liquid domain into finite volumes, the prismatic mesh is used to solve the boundary layer surrounding the submarine, and the surface mesh is used to divide the submarine hull surface into finite surfaces. The number of prismatic mesh layers used is 6 with the thickness of the first prismatic mesh layer 0.0025m from the wall so that the average y+ value is 80. The mesh will be smoothed at important locations such as: (the area near the hull, the bow, aft and bridge areas). The resulted mesh is presented in Figure 3. The physical model used in the simulation of the flow a round the submarine hull is a real fluid model RANSE (Navier-Stokes mean Reynolds equation) with uniform flow. because the flow to the submarine when at diving mode is a steady flow. The turbulent model used in this paper is a realizable k–ε two layer turbulent model. This is one of the turbulent fluid models that give reliable results in calculating the ship resistance in general and the submarine resistance in particular [14].

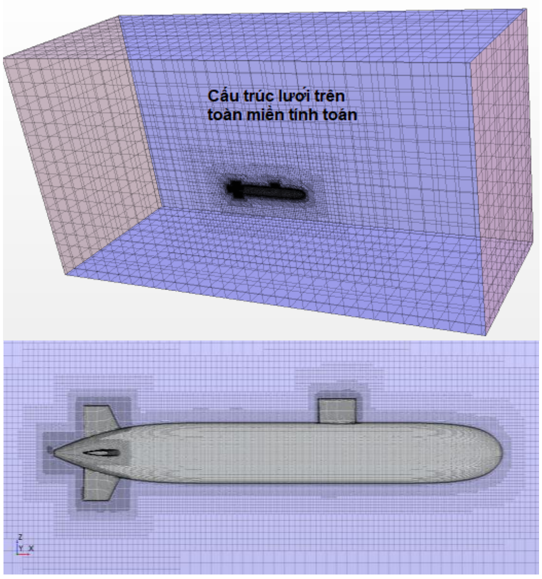


Fig.3. Dimension of the virtual tank

III. COMPUTATE AND ANALYSES THE RESULT

A. Convergence of grid

Defining the convergence of the mesh is the first step in CFD computation to eliminate the result error due to meshing. The grid convergence was perform for three different sizes of mesh at ship speed v=3.05 m/s with the transforming ratio $r_G = \sqrt{2}$ (which was recommended by ITTC related to study the mesh convergence [15]). The grid configurations used for the grid convergence study in the CFD simulation of the DARPA SUBOFF model are presented in Table 2.

International Journal of Engineering Applied Sciences and Technology, 2025
Vol. 10, Issue 01, ISSN No. 2455-2143, Pages 164-168
Published Online May 2025 in IJEAST (http://www.ijeast.com)

Table -2 The parameters of grid employed in studying convergence

Mesh density	Mesh size, m	Mesh quantity, mil. elements
Coarse	0.06	0.62
Medium	0.0425	1.53
Fine	0.03	2.78

The change in calculation result at different mesh sizes is estimated by the following expression:

$$\varepsilon_{12} = (S_1 - S_2)/S_1; \varepsilon_{23} = (S_2 - S_3)/S_2 \quad (1)$$

where: S1, S2, S3 – is the submarine resistance estimated at different mesh sizes as fine mesh, medium mesh and coarse mesh, respectively.

The convergence of simulation results is evaluated based on expression (2). Depending on the sign and value of Rk, three possible cases happened including:

- Monotonous convergence $0<R_k<1$;
- Convergence divergence $R_k<0$;
- Not convergent $R_k>1$.

$$R_k = \frac{\varepsilon_{12}}{\varepsilon_{23}} \quad (2)$$

Figure 4 illustrates the effect of mesh size on the results of resistance calculation for DARPA SUBOFF submarine at V=3.05m/s. As can be seen from the results of the mesh convergence study in Figure 4, the resulting difference between the coarse mesh and the medium mesh is 0.4%, while the difference between the medium and the fine mesh is only 0.15%. That is, there is a monotonous convergence of the grid here and the difference is relatively small. Therefore, in the next calculations, the authors will use a medium-sized grid to calculate DARPA SUBOFF submarine resistance.

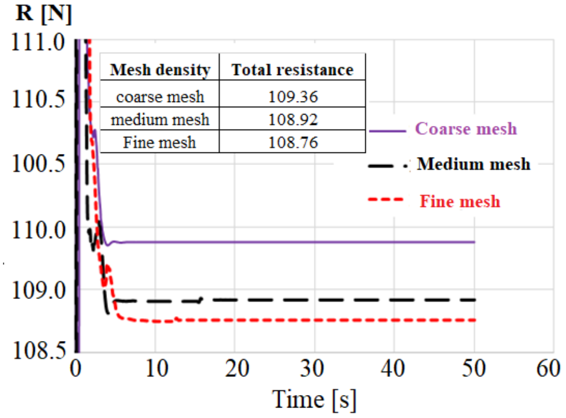


Fig.4. The results of the mesh convergence study at speed V=3.05 m/s

B. The result of DARPA SUBOFF submarine resistance calculation

Employing the medium mesh density in the calculation of DARPA SUBOFF submarine resistance at different speed ranges, the authors obtained the calculation results as shown in Table 3. As can be observed in Table 3, the predicted results of submarine resistance using CFD are quite close to the model test results. The error between the calculated results and the test results ranges from 0.96% to 6.47%.

Table -3 Result of submarine resistance prediction at different speeds compared with the experiment results

No.	V [m/s]	Simulation result R [N]	Experiment result R [N]	% error
1	3.050	108.92	102.3	**-6.47**
2	5.144	288.00	283.8	**-1.48**
3	6.100	395.74	389.2	**-1.68**
4	7.160	531.68	526.6	**-0.96**
5	8.230	690.48	675.6	**-2.20**
6	9.151	843.40	821.1	**-2.72**

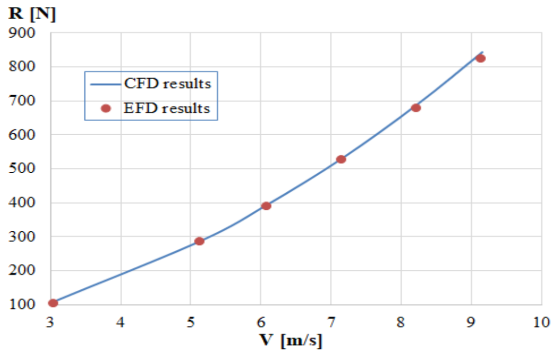
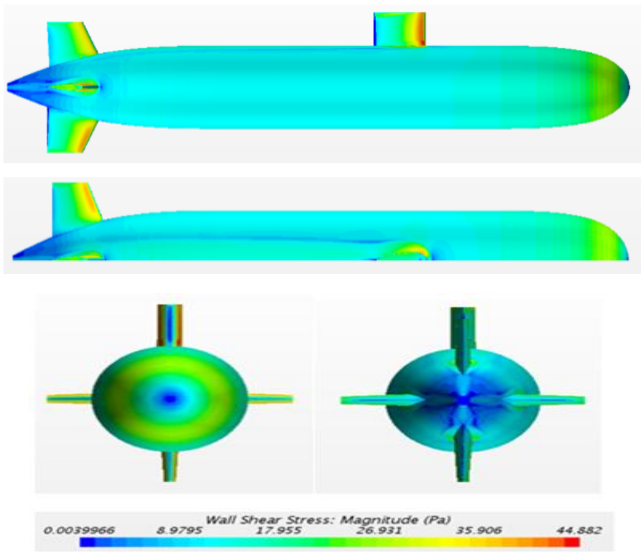


Fig.5. Comparison of calculated results and experiment results at different speeds



Fig,6. Shear stress distribution along the hull at speed V=3,05m/s

The images of pressure distribution, shear stress, and the flows along the submarine hull at speed of 3.05m/s are shown in Figures 6, 7 and 8.

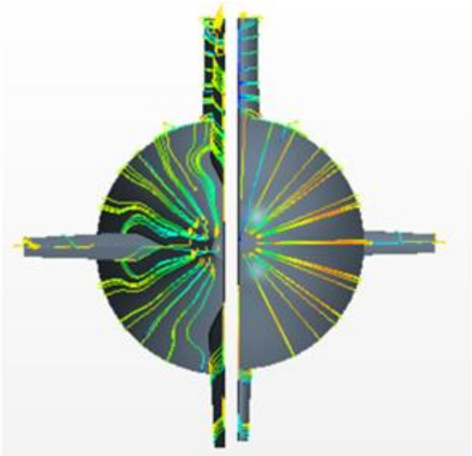


Fig.7. Streamline and flow rate along the hull at speed V=3.05m/s

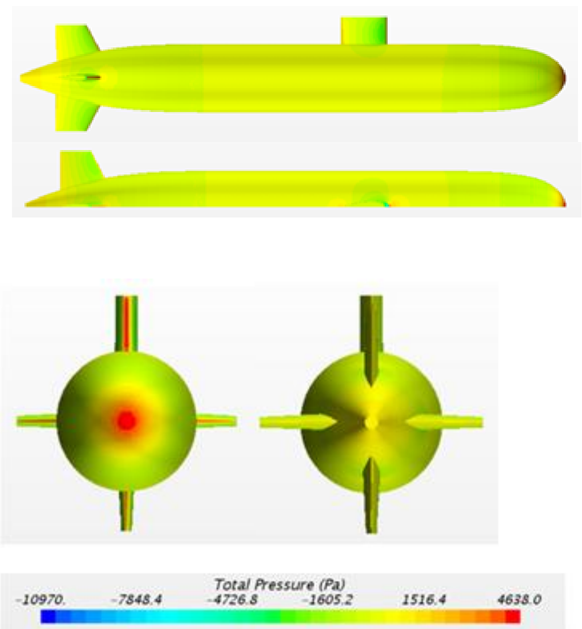


Fig.8. Pressure distribution along the hull at speed V=3,05m/s

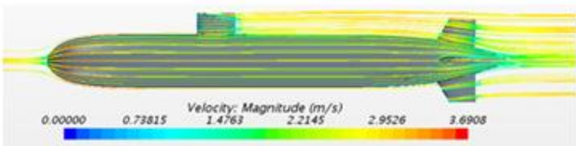


Fig.9. Streamline at V=3.05m/s

IV. CONCLUSION

This study has effectively applied CFD to predict resistance and flow aroud submarine. The numerical results show strong agreement with experimental towing tank data, with discrepancies ranging from 0.96% to 6.47%. Otherwise, it also presents the influence of mesh size on the obtained simulation results, providing the images of pressure and stress distribution on the hull surface, the streamline surrounding the hull when it is moving. These are very important images in streamline analysis for other problems in submarine hull design optimization.

V. REFERENCE

[1] Luu, D.D., et al. Numerical Study on the Influence of Longitudinal Position of Centre of Buoyancy on Ship Resistance Using RANSE Method. Naval Engineers Journal, Vol. 132 No.4: pp. 151-160, 2020.

[2] Tu, T., et al., Numerical Prediction of Ship Resistance in Calm Water by Using RANS Method. Journal of Engineering and Applied Sciences, Vol. 13 No.17: pp. 7210-7214, 2018.

[3] Le, T.-H., et al., (2023). Numerical Investigation of Length to Beam Ratio Effects on Ship Resistance Using RANSE Method. Polish Maritime Research, Vol. 30 No.1: pp. 13-24.

[4] Tu, T.N., et al. (2021), Effects of Turbulence Models On RANSE Computation Of Flow Around DTMB 5415 Vessel. Naval Engineers Journal, Vol. 133 No.3: pp. 137-151, 2021.

[5] Tu, T.N., et al., (2022). Numerical Study on the Effect of Turbulence Models on RANSE Computation of Flow Around Submarine. in 2022 9th NAFOSTED Conference on Information and Computer Science (NICS). IEEE.

[6] Uzun, D., et al., A CFD study: Influence of biofouling on a full-scale submarine. Applied Ocean Research, Vol. 109: pp. 102561, 2021.

[7] Moonesun, M., Y. Korol, and H. Dalayeli, CFD analysis on the bare hull form of submarines for minimizing the resistance. International Journal of Maritime Technology, Vol. 3: pp. 1-16, 2015.

[8] Liu, L., et al., Full-scale simulation of self-propulsion for a free-running submarine. Physics of Fluids, Vol. 33, No. 4: pp. 047103, 2021.

[9] Budak, G. and S. Beji, Computational resistance analyses of a generic submarine hull form and its geometric variants. Journal of Ocean Technology, Vol. 11, No.2, 2016.

[10] Bettle, M., S.L. Toxopeus, and A. Gerber, Calculation of bottom clearance effects on Walrus submarine hydrodynamics. International Shipbuilding Progress, Vol. 57(3-4): pp. 101-125, 2010.

[11] Summary of DARPA Suboff Experimental Program Data. Naval Surface Warfare Center, Carderock Division (NSWCCD).
[12] Zhang, N., Shen, Hong-Cui, Yao, Hui-zhi, Numerical simulation of flow around submarine operating close to the bottom or near surface. Journal of Ship Mechanics, Vol. 11, No.4: pp. 498-507, 2007.
[13] Pan, Y.-c., H.-x. Zhang, and Q.-d. Zhou, (2012). Numerical prediction of submarine hydrodynamic coefficients using CFD simulation. Journal of Hydrodynamics, Vol. 24, No. 6: p. 840-847, 2012.
[14] Yong, Z., et al., Turbulence model investigations on the boundary layer flow with adverse pressure gradients. Journal of Marine Science and Engineering, Vol. 14, No. 2: pp. 170-174, 2015.
[15] ITTC-Quality Manual 7.5-03-01-01, 2008 (https://www.ittc.info/media/8153/75-03-01-01.pdf).

IJEAST

INTERNATIONAL JOURNAL
OF ENGINEERING APPLIED SCIENCE AND TECHNOLOGY

ABOUT IJEAST

International Journal of Engineering Applied Science and Technology (IJEAST) is a peer-reviewed, open access journal that publishes high-quality research papers in the field of Engineering, Applied Science and Technology.

IJEAST aims to provide a platform for researchers, academicians, and professionals to share their innovative ideas, research findings, and practical experiences with the global scientific community.

FOCUS AREAS

- Engineering
- Applied Science
- Technology
- Innovation & Development
- Interdisciplinary Studies

PEER REVIEWED

All submissions are rigorously peer reviewed to ensure quality.

OPEN ACCESS

Free and unrestricted access to research for all.

GLOBAL REACH

Connecting researchers and professionals worldwide.

TIMELY PUBLICATION

We ensure a swift and efficient publication process.

For more information, visit our website
www.ijeast.com

INTERNATIONAL JOURNAL
OF ENGINEERING APPLIED SCIENCE
AND TECHNOLOGY

editor@ijeast.com
www.ijeast.com
India

2455-2143