

Article

Numerical Investigation of Hydrodynamic Performance of an AUV Moving near the Bottom Wall

Nguyen Dong and Ngo Van He * 

School of Mechanical Engineering, Hanoi University of Science and Technology, Hanoi 100000, Vietnam;
dong.nguyen@hust.edu.vn

* Correspondence: he.ngovan@hust.edu.vn

Abstract

Autonomous underwater vehicles (AUVs) are widely employed in missions conducted near the seabed, including underwater inspection, seabed mapping, and marine resource exploration. During such operating conditions, the interaction between the AUV and the bottom wall can significantly influence the surrounding flow field and the hydrodynamic characteristics of the vehicle. In this study, a numerical investigation is carried out to examine the influence of near-bottom effects on the hydrodynamic performance of an AUV using a commercial Computational Fluid Dynamics (CFD) solver. The seabed is assumed as a flat wall, and two operating conditions are considered, including an open-water case and a near-bottom case with a clearance ratio of $h/L_A = 1.93$. The flow field is investigated through analyses of hydrodynamic force, pressure distribution, and wake structures. The results indicate that the wall bottom noticeably alters the pressure field and wake development around the AUV, leading to changes in total resistance and flow separation. The findings provide useful insights into the hydrodynamic mechanisms associated with near-bottom operation and offer valuable guidance for the design, control, and operation of AUVs performing missions in shallow or seabed-related missions.

Keywords: AUV; hydrodynamic; near-wall bottom; CFD; seabed; resistance

1. Introduction

In recent years, Autonomous Underwater Vehicles (AUVs) have become increasingly important in the field of marine engineering applications, such as seabed topographic surveys, underwater infrastructure inspection, and environmental monitoring [1–3]. In practical applications, AUVs often operate in confined environments or close proximity to physical boundaries such as the seabed, canal walls, pipelines, and offshore structures. Unlike operations in idealized open-water environments, working near these boundaries significantly alters the flow field surrounding the AUVs, resulting in complex hydrodynamic phenomena known as the wall effect or ground effect [2–6]. Although previous studies have investigated AUV hydrodynamics in open water or near confined boundaries, detailed analyses of near-bottom flow behavior considering both propeller and non-propeller configurations remain limited. In particular, the combined influence of bottom proximity on wake evolution, vortex structures, and hydrodynamic loads has not been fully clarified.

Experimental and numerical CFD simulation studies have demonstrated that as an AUV approaches a solid surface, the pressure difference between its sides generates a suction force that pulls the vehicle toward that surface [7]. This effect not only causes deviations



Academic Editor: Decheng Wan

Received: 20 April 2026

Revised: 14 May 2026

Accepted: 15 May 2026

Published: 19 May 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

from the intended trajectory but also significantly increases the hydrodynamic resistance acting on the AUV hull [1,2,4]. For instance, research on axisymmetric AUVs has shown that resistance can increase by as much as 64.3% when operating near cylindrical offshore structures compared to free-field conditions. Furthermore, a critical clearance threshold has been identified, below which resistance increases sharply due to pronounced pressure asymmetry and enhanced flow separation. In confined open channel environments, the presence of a free surface has been found to have a greater impact on hydrodynamic performance than solid walls, although these effects become negligible when the submergence depth exceeds 3.1 times the vehicle's height [3,4,7].

Hydrodynamic interaction effects are also significant in multi-hull configurations, such as during underwater docking, and cooperative formation navigation [8–10]. When an AUV enters a cone-shaped docking structure, the hydrodynamic resistance may increase significantly compared with open-water conditions, thereby requiring precise control strategies to ensure safe and successful docking operations. In tandem formations, the trailing AUV is strongly affected by the wake and pressure distribution generated by the leading AUV. Appropriate spacing between the AUVs can reduce overall energy consumption by approximately 20% [10]. For intervention AUVs operating near the seabed, hydrodynamic characteristics become even more complex due to the motion of onboard manipulator systems. The continuous movement of the manipulator arm modifies the hull's hydrodynamic coefficients and flow field, requiring adaptive control and disturbance compensation techniques to maintain operational stability and positioning accuracy during intervention tasks [11–13].

To investigate the interaction effect on hydrodynamic performance of the AUVs, CFD has become one of the most widely adopted approaches due to its capability to provide detailed results on pressure, velocity distributions, and vortex structures. Among the available turbulent models, the $k-\omega$ SST turbulence model is commonly employed because of its strong performance in predicting near-wall flow behavior, adverse pressure gradients, and flow separation under marine operating conditions [2,4,12]. Recent advancements have further improved numerical hydrodynamic analysis through the integration of artificial intelligence techniques. In particular, physics-informed generative adversarial networks and other data-driven approaches have demonstrated considerable potential in accelerating hydrodynamic prediction while maintaining acceptable accuracy compared with conventional CFD simulations [14–17]. Additionally, smart sensing systems, which combine flexible sensing skins with active vortex generators, offer new possibilities for flow control and thermal management in high-resistance environments [18–20].

Previous studies have provided valuable insights into the hydrodynamic interactions between AUV hulls and surrounding structures. The effects of propulsion and control devices, such as propellers and rudders, have also been investigated. However, the combined influence of multiple boundaries and complex flow phenomena in realistic operating environments remains insufficiently understood. In particular, the hydrodynamic behavior of an AUV operating near the seabed remains a challenging problem. This is mainly due to the complex interactions among the hull, the bottom boundary, the surrounding flow field, and the propulsion and control.

Motivated by these challenges, the present study employs CFD to investigate the hydrodynamic characteristics of an AUV operating in proximity to the seabed. The primary objective is to evaluate the variation in hydrodynamic force and moment coefficients under near-bottom operating conditions. The findings of this study may serve as a useful foundation for the development of more effective control strategies, vehicle design optimization, and safe trajectory planning for AUV operations in confined underwater environments.

2. Methodology

2.1. Governing Equations

The hydrodynamic performances of the AUV were simulated using CFD. The numerical simulations were carried out using the finite volume method within the ANSYS Fluent version 19.1 environment [1–4,8,16]. The flow field was modeled as a three-dimensional incompressible turbulent flow governed by the RANS (Reynolds-Averaged Navier–Stokes) equations.

The mathematical formulation includes the continuity equation together with the momentum conservation equations, which can be written as follows [1–3,5,8,12,14,21]:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho \bar{v}_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \bar{v}_i) + \frac{\partial}{\partial x_i}(\rho \bar{v}_i \bar{v}_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[\mu \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial \bar{v}_l}{\partial x_l} \right) \right] - \frac{\partial}{\partial x_j}(\rho \bar{v}'_i \bar{v}'_j) + F_i \quad (2)$$

where v_i represents the velocity components, p is the pressure, F_i denotes the body force components, μ is the fluid dynamic viscosity, and ρ is the fluid density. In the RANS formulation, the instantaneous velocity field is decomposed into mean and fluctuating components in order to represent the effects of turbulence.

2.2. Geometric Model of the AUV

In this study, the autonomous underwater vehicle adopts an axisymmetric hull, which is a commonly used configuration for unmanned underwater vehicles due to its favorable hydrodynamic characteristics. The vehicle is equipped with four stern control fins with a NACA 0012 airfoil profile [22] and a two-bladed propeller to ensure maneuverability and propulsion performance. The hull shape of the AUV is presented in Figure 1. The main dimensions include a total length $L_A = 1557$ mm, and a hull diameter $D_A = 180$ mm, resulting in a slender body suitable for efficient underwater operation [2,23].

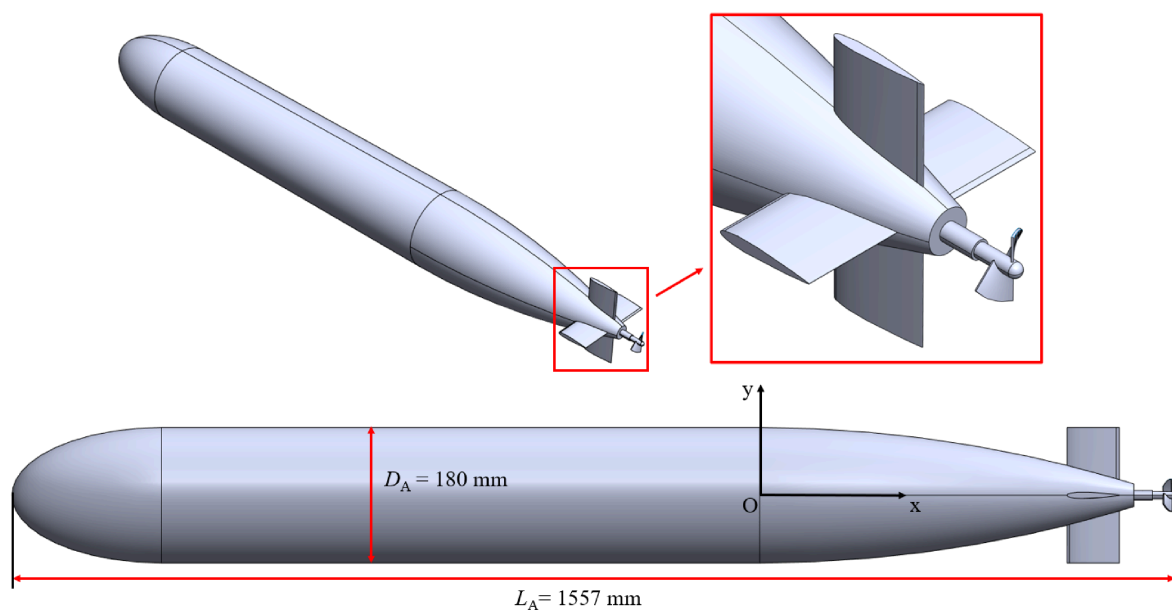


Figure 1. Cont.

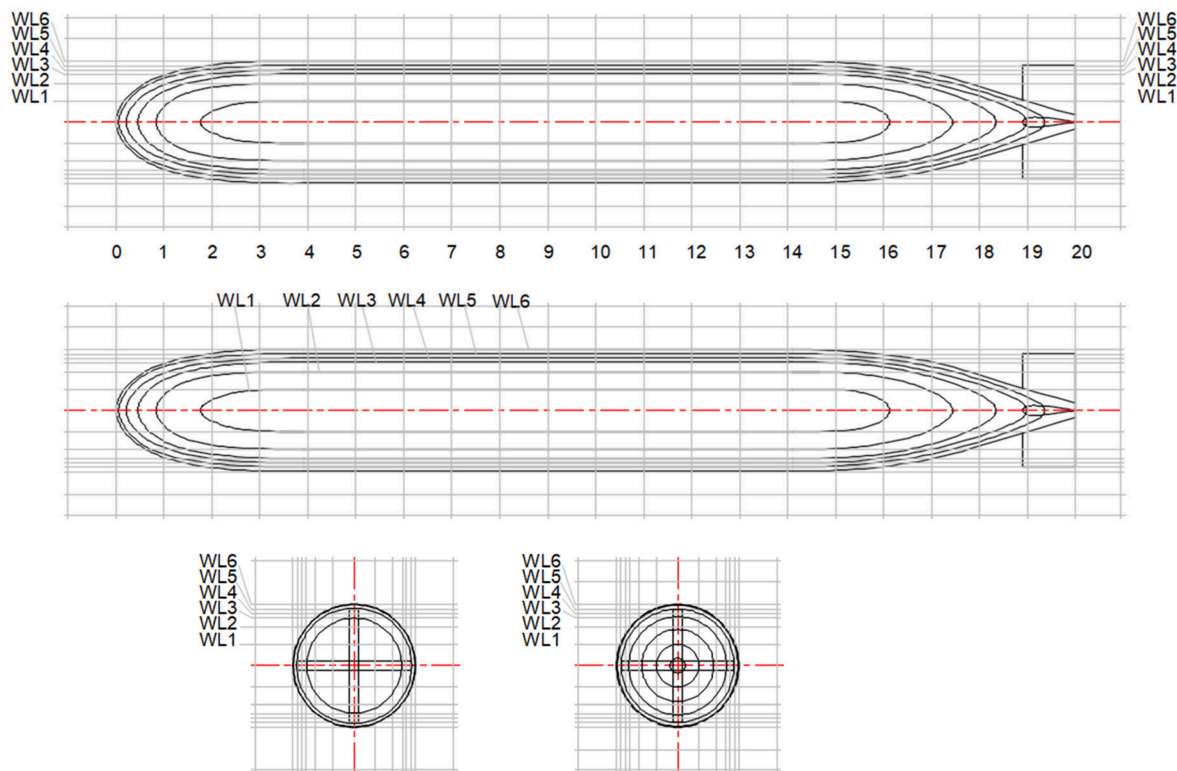


Figure 1. Body plan of the AUV used in computation.

The stern control fins employ a symmetrical NACA 0012 airfoil, providing stable hydrodynamic performance and effective control during maneuvering. The propulsion system consists of a propeller with a 43 mm propeller diameter and a 57 mm of length, operating at a rotational speed of 12,000 rpm.

The geometric characteristics of the AUV are defined: the total surface area is $7.9292 \times 10^5 \text{ mm}^2$, and the displacement volume is $3.1318 \times 10^7 \text{ mm}^3$. The projected areas in the longitudinal and transverse directions are $S_x = 2.5448 \times 10^4 \text{ mm}^2$ and $S_y = 2.4662 \times 10^5 \text{ mm}^2$, respectively. These parameters are employed to establish the computational model and to assess the hydrodynamic characteristics of the AUV through the CFD simulations.

2.3. Computational Fluid Domain and Setup Boundary Conditions for Simulation

In simulation with CFD, the definition of the computational fluid domain and the selection of boundary conditions play a crucial role in determining the accuracy of the numerical results. Numerous studies have demonstrated that the computational domain size, mesh resolution, and boundary condition settings can significantly affect the predicted hydrodynamic performance of an AUV [1–4,7,9,14,23]. Therefore, the design of the computational domain and the specification of boundary conditions should be conducted based on established guidelines and validated practices reported in the literature [24–26].

In addition, experience from similar numerical studies and experimental investigations, such as towing tank tests, is essential for defining an appropriate computational setup. These considerations ensure that the numerical model accurately represents the physical problem under investigation [2–4,23,24]. In this study, the computational domain was designed with a length of $5L_A$, a width of $1.2L_A$, and a height of $1.2L_A$, corresponding to dimensions of $7500 \times 1800 \times 1800 \text{ mm}$. The clearance between the AUV and the bottom wall was set to $h/L_A = 1.93$.

In this study, the boundary conditions were defined by imposing a uniform velocity inlet at the inlet of the computational fluid domain and a pressure outlet at the outlet of the computational fluid domain. A no-slip wall condition was applied at the bottom surface to represent the seabed. The inflow velocities were selected to correspond to Reynolds numbers of $Re = 0.8 \times 10^6$, 2.09×10^6 , and 3.48×10^6 . The numerical simulations are conducted using a pressure-based solver with second-order discretization schemes. The SST $k-\omega$ turbulent model is employed to accurately capture boundary layer behavior and flow separation. Convergence is achieved when the residuals fall below 10^{-5} , and the monitored hydrodynamic coefficients reach stable values. The computational fluid domain and the corresponding boundary conditions are shown in Figure 2.

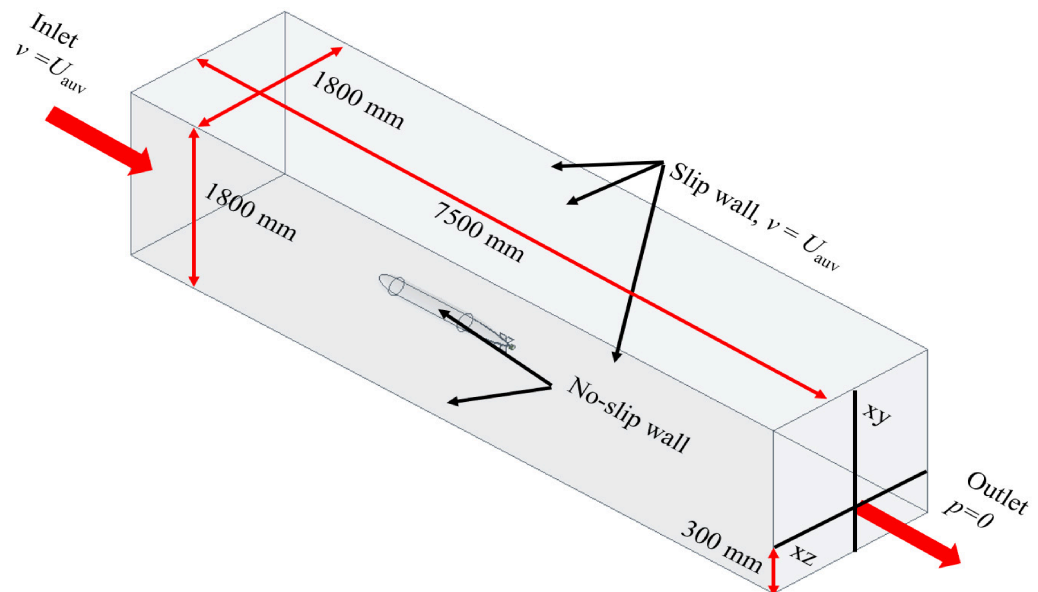


Figure 2. Computed domain and boundary conditions.

In the computed case of the AUV hull with a propeller, the propeller rotation is modeled using the moving reference frame method. The propeller region is defined as a rotating subdomain with a prescribed angular velocity, while the surrounding computational domain remains stationary. The moving reference frame approach provides a steady state representation of the propeller-induced flow field and is computationally efficient for capturing the time-averaged hydrodynamic characteristics of the AUV. This method is considered appropriate for the present study, which mainly focuses on the overall hydrodynamic forces, pressure distribution, wake development, and near-bottom interaction effects rather than detailed transient blade passing phenomena.

The propeller rotational speed is specified as 12,000 rpm, and appropriate interface conditions are applied between the rotating and stationary regions to ensure flow continuity across the domain interfaces. The additional description improves the transparency and reproducibility of the numerical methodology and clarifies how the propeller-induced flow effects are represented in the simulations. In this study, several dimensionless parameters, including the Reynolds number, hydrodynamic force coefficients, and the moment coefficients acting on the model, are defined as follows:

$$Re = \frac{\rho U_{auv} L_A}{\mu} \quad (3)$$

$$C_x = \frac{F_x}{\frac{1}{2} \rho S_A U_{auv}^2} \quad (4)$$

$$C_y = \frac{F_y}{\frac{1}{2}\rho S_A U_{\text{auv}}^2} \quad (5)$$

$$C_{m_{x,y,z}} = \frac{M_{x,y,z}}{\frac{1}{2}\rho S_A U_{\text{auv}}^2 L_A} \quad (6)$$

where F_x and F_y denote the resistance and lift forces in their respective directions, M represents the hydrodynamic moment acting on the AUV, and S_A is the reference frontal projected area of the AUV, in calculated hydrodynamic forces coefficients, S_A has a value as frontal projected area S_x or S_y . The Reynolds number, R_e is a dimensionless quantity representing the ratio between inertial forces and viscous forces in the flow field surrounding the AUV.

In this study, to visualize the vortical structures in the wake, the Q criterion is adopted, which distinguishes rotation-dominated regions from strain-dominated flow. An iso-surface of $Q = 0.1$ is used to identify the main coherent vortices in the computed case of an AUV with a propeller. This threshold is selected based on sensitivity analysis to filter out weak shear-induced structures while retaining the dominant vortical features associated with wake evolution and flow separation. Specifically, the Q criterion is introduced as [27]:

$$Q = \frac{1}{2} (\|\Omega\|^2 - \|S\|^2) \quad (7)$$

where Ω and S denote the antisymmetric (rotation) and symmetric (strain rate) components of the velocity gradient tensor, respectively. The Q criterion quantifies the local balance between rotational and strain effects in the flow field. Regions with positive Q values correspond to rotation-dominated flow and are commonly associated with coherent vortex structures, whereas negative Q values indicate strain-dominated regions.

2.4. Mesh Generation

In this study, the computational grid for all simulations was generated using ANSYS ICEM-CFD, employing an unstructured tetrahedral mesh. The mesh topology is presented in Figure 3, illustrating the overall structure of the computational domain. To accurately resolve the flow behavior in the vicinity of solid boundaries, local mesh refinement was applied around the model, leading to a higher concentration of control volumes in near-wall regions.

To improve the accuracy of near-wall flow prediction, prism-layer refinement was applied along the AUV surface to better resolve the boundary layer region. A total of nine prism layers were generated with a layer growth ratio of 1.2. The overall prism-layer thickness was selected to ensure adequate coverage of the boundary layer throughout the Reynolds number range considered in the present study. This near-wall treatment significantly improves mesh quality and contributes to more accurate predictions of wall-bounded flow characteristics.

The thickness of the first prism layer was determined to achieve appropriate y^+ values for wall-function-based turbulence modeling, following the recommended guidelines provided in ANSYS Fluent. In the present simulations, the y^+ values over the AUV surface were maintained below approximately 63. The corresponding y^+ distribution is presented in Figure 4. To evaluate mesh quality and near-wall resolution, representative y^+ distributions on the AUV surfaces are presented for a selected operating condition. The results indicate that the near-wall mesh is sufficiently refined and satisfies the resolution requirements associated with the selected turbulence viscous model.

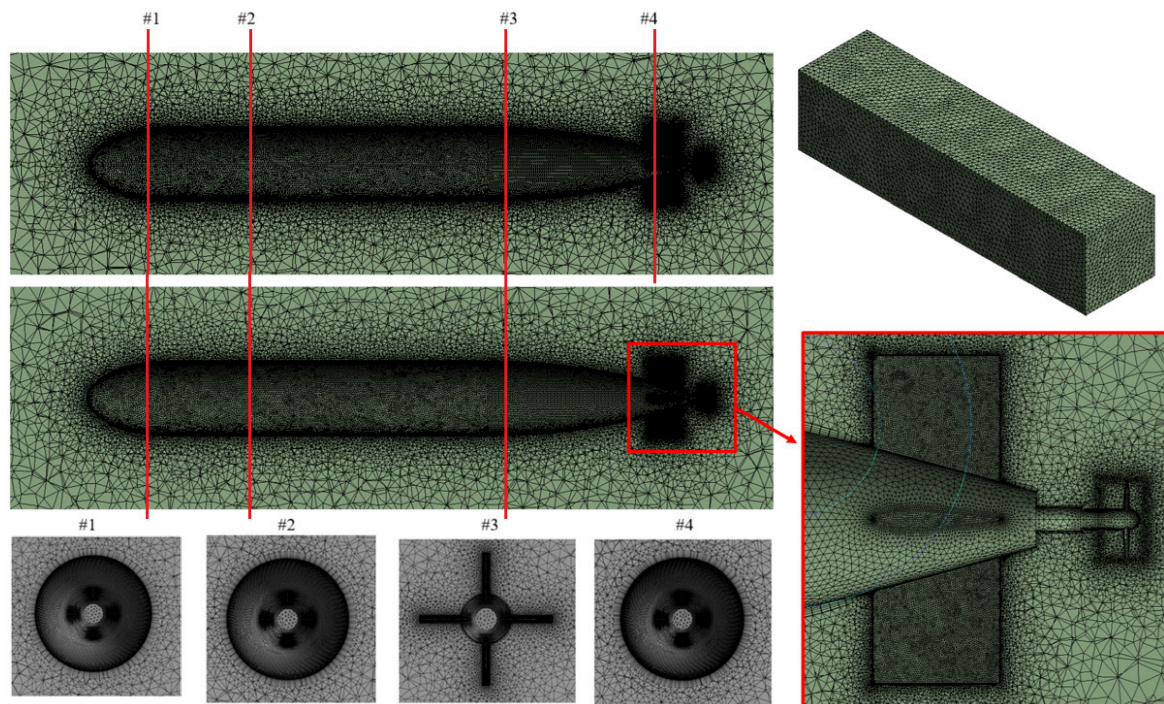


Figure 3. Mesh generation in the computational domain.

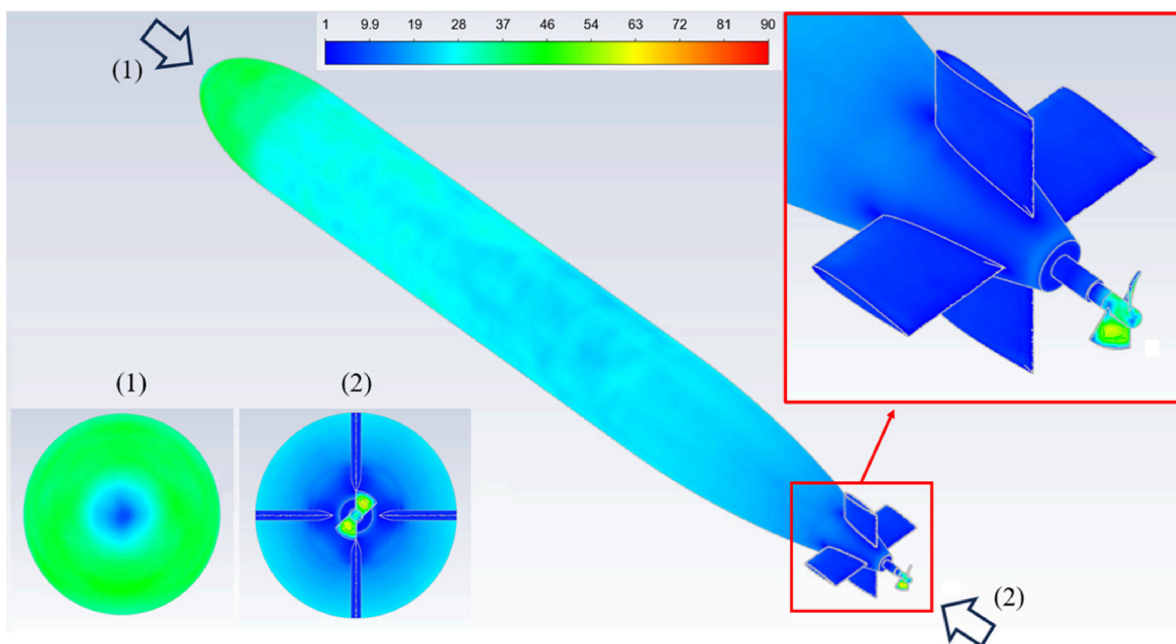


Figure 4. Contour of the y^+ distribution on the AUV hull surface.

In CFD simulations, the mesh resolution is a key factor influencing the accuracy of the numerical results. Therefore, the reliability of the computed results must be ensured through grid-independent verification. In addition, comparisons with available experimental data can further support the validity of the numerical approach. In the present study, a grid independence analysis was performed to evaluate the influence of mesh density on the predicted hydrodynamic resistance of the AUV.

Six computational meshes with progressively refined resolutions were generated for the grid independence study. The total number of cells varies from approximately 0.59 million to 11.83 million, as listed in Table 1. This range of mesh densities enables a

systematic assessment of numerical convergence and solution sensitivity to grid refinement. All simulations were performed at a constant inflow velocity with a Reynolds number of 2.09×10^6 to ensure consistency in the comparison.

Table 1. Results of mesh convergence.

Model	M1	M2	M3	M4	M5	M6
Number of elements ($\times 10^6$)	0.59	1.57	3.31	4.16	7.21	11.83
y+	426.12	367.28	38.426	12.768	1.562	0.623
C_x	0.00633	0.00631	0.00511	0.00498	0.00496	0.00495
Deviation (%)	27.88	27.47	3.23	0.61	0.20	0

The computed results for different mesh densities are presented in Table 1, while the variation in the resistance coefficient is illustrated in Figure 5. As the mesh is progressively refined, the predicted resistance coefficient exhibits a clear convergence trend. When the total cell count exceeds approximately 3.31 million, the variation in resistance becomes marginal, with differences remaining within 3.23%. This behavior indicates that the solution becomes increasingly insensitive to further mesh refinement.

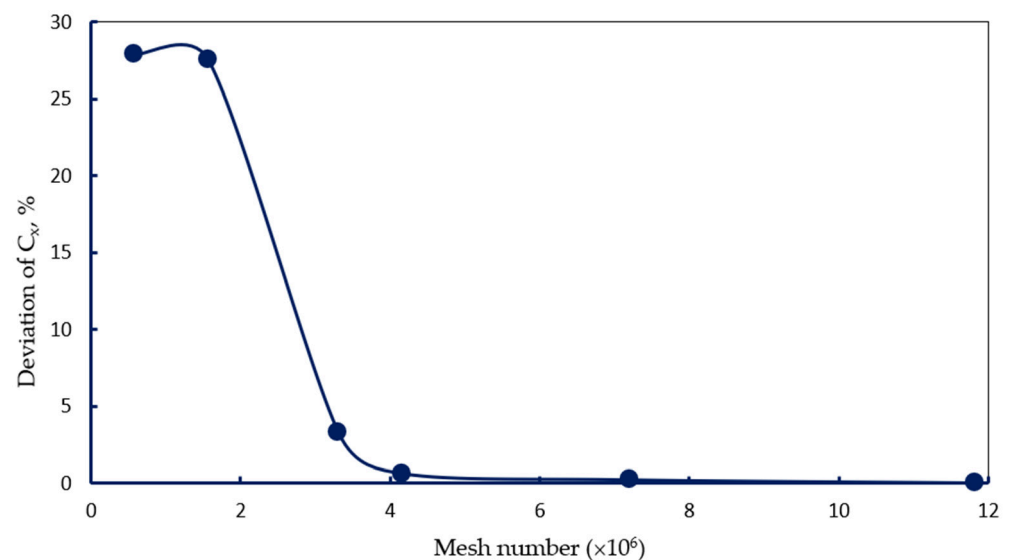


Figure 5. Mesh convergence of the predicted resistance acting on the bare hull.

Based on the obtained results, mesh configuration M4, containing approximately 4.16 million computational cells, was chosen for the subsequent simulations. This mesh offers a suitable balance between computational cost and numerical accuracy.

2.5. Validation of the Simulation Results

To evaluate the reliability of the CFD methodology employed in the present study, the numerical results were validated through comparisons with available experimental measurements and previously published numerical data. The validation case was based on the DARPA SUBOFF model experimentally investigated by Liu and Huang [28–30]. Similar validation studies have also been conducted by the present research group and reported in several of our previous publications related to the AUV research project [2,23,31,32]. The SUBOFF configuration consists of an axisymmetric AUV hull with a length of 4.356 m and a maximum diameter of 0.508 m, corresponding to a Reynolds number of 29.8×10^6 .

In addition, numerical results reported by Yu et al. for the SUBOFF model under similar flow conditions were also considered for comparison [14,28–30]. The resistance

values predicted in the present study agree well with both experimental measurements and previously published CFD results. As presented in Figure 6 and Table 2, the discrepancies between the present results and the reference data remain within an acceptable range, thereby confirming the validity and reliability of the numerical approach adopted in this study.

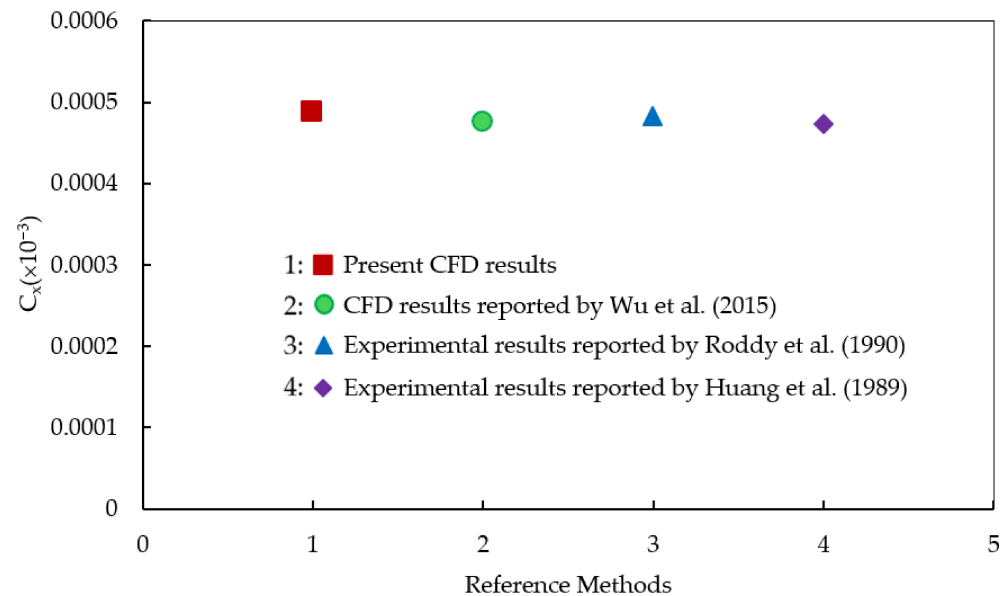


Figure 6. Validation of CFD results with the SubOff model.

Table 2. Verification of the CFD results for the SUBOFF model.

Name	F_x, N	$C_x (\times 10^{-3})$	Deviation (%)
Present CFD result	372.26	0.488	2.75
CFD result by Wu et al. (2015)	363.89	0.476	0.63
Experimental result by Roddy et al. (1990)	368.22	0.482	1.90
Experimental result by Huang et al. (1989)	361.80	0.473	0

3. Results and Discussion

In this study, the hydrodynamic performance of the AUV is investigated with consideration of both bottom wall proximity effects and propeller influence. CFD computations are conducted to evaluate the hydrodynamic performance of the AUV under both open water and near-bottom wall conditions. A comparative analysis is performed for two configurations: an AUV equipped with a propeller and an AUV without a rear propeller.

3.1. CFD Result of the Hydrodynamic Performance Acting on the AUV

In this section, the hydrodynamic performance of the AUV operating in open-water conditions without a rear propeller is presented based on CFD simulations. Figure 7 presents the dynamic pressure distribution within the computational domain for the investigated range of flow velocities, showing the variation in the pressure fields around the AUV under different operating conditions. The results provide insight into the pressure distribution along the hull surface and the surrounding flow field, which are essential for understanding the resistance characteristics and overall hydrodynamic behavior of the AUV.

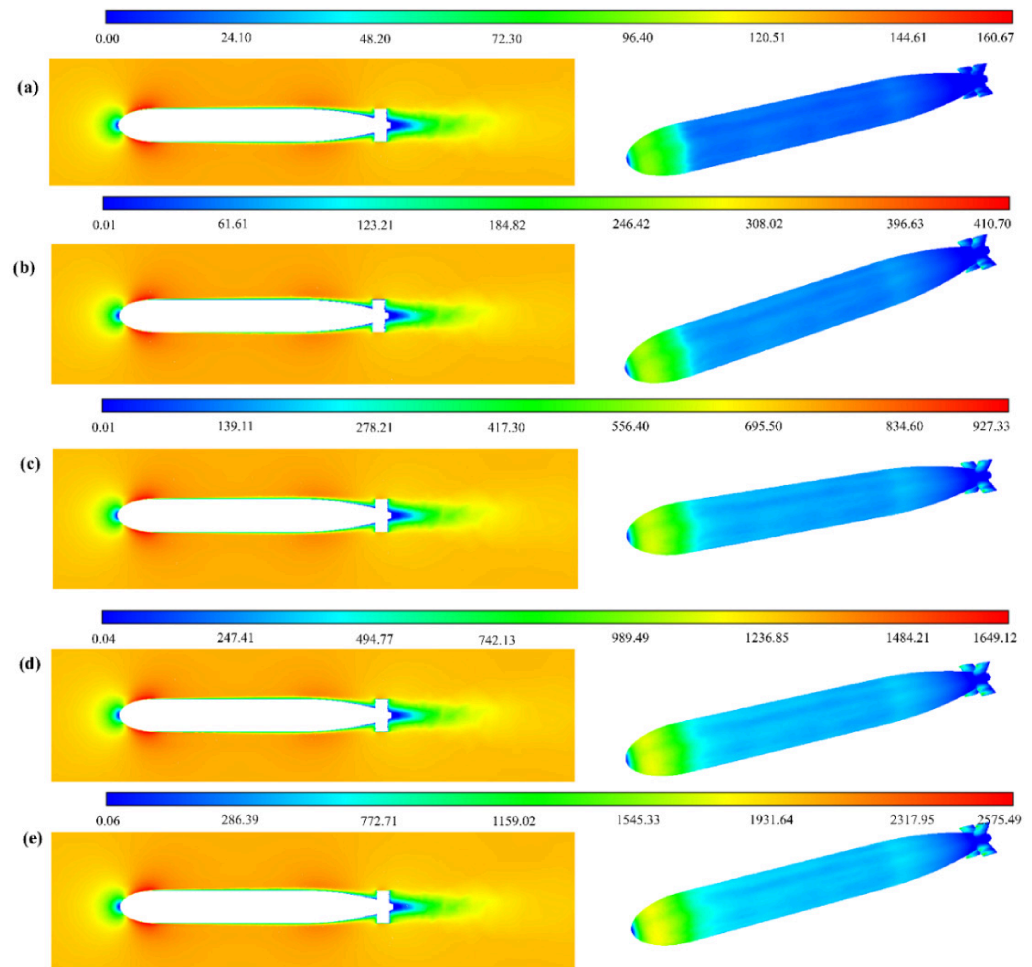


Figure 7. Dynamic pressure distribution around the bare hull in water (P_a): (a) $Re = 0.8 \times 10^6$; (b) $Re = 1.39 \times 10^6$; (c) $Re = 2.09 \times 10^6$; (d) $Re = 2.78 \times 10^6$; (e) $Re = 3.48 \times 10^6$.

The results, as shown in Figure 7, illustrate that the distribution of the dynamic pressure around the AUV over the investigated Reynolds number range from 0.8×10^6 to 3.48×10^6 . Regions shown in red and yellow indicate areas of high dynamic pressure, predominantly located around the forward and mid-body sections of the AUV, while blue regions correspond to low-pressure zones in the surrounding flow field. It can be observed that, as the inflow velocity increases, the magnitude of the dynamic pressure around the AUV changes significantly. However, the overall pattern of pressure distribution remains generally consistent across the investigated velocity range.

Figure 8 presents the variation in hydrodynamic performances as the forces and moments coefficients acting on the AUV as a function of the inflow velocity.

The results as shown in Figure 8 and Table 3 present the hydrodynamic performance as the forces and moments acting on the hull of the AUV in free stream conditions across the investigated velocity range. It is observed that the resistance increases with velocity, while the resistance coefficient, C_x , asymptotically approaches approximately 0.00426. This behavior is in good agreement with the corresponding pressure distribution around the AUV.

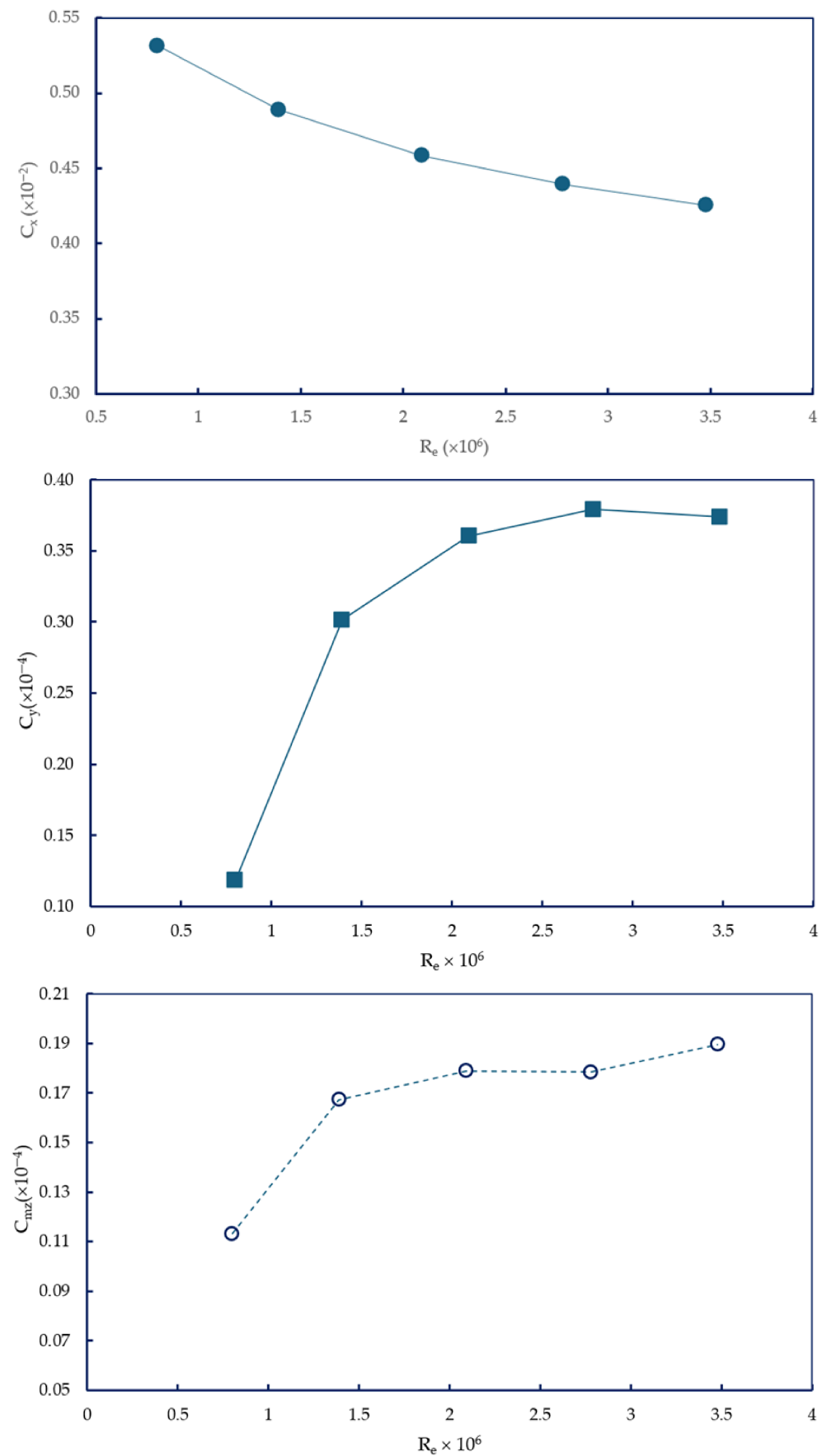


Figure 8. The hydrodynamic force and moment of the bare hull.

Table 3. Hydrodynamic Forces and Moments Acting on the AUV.

$R_e \times 10^6$	0.80	1.39	2.09	2.78	3.48
F_x (N)	0.524978	1.236707	2.609123	4.445651	6.724654
F_y (N)	0.001169	0.007622	0.020488	0.038345	0.059106
F_z (N)	0.002311	0.000106	−0.00261	−0.01418	−0.03394
M_x (Nm)	0.000172	0.000476	0.001051	0.002436	0.003786
M_y (Nm)	0.005973	0.016573	0.034797	0.06416	0.10084
M_z (Nm)	0.001733	0.006555	0.01577	0.027961	0.046438
C_x	0.00532	0.00489	0.00459	0.00440	0.00426
C_y	0.00001	0.00003	0.00004	0.00004	0.00004
C_z	0.00002	0.00000	0.00000	−0.00001	−0.00002
C_{mx}	0.000001	0.000001	0.000001	0.000002	0.000002
C_{my}	0.000039	0.000042	0.000039	0.000041	0.000041
C_{mz}	0.000011	0.000017	0.000018	0.000018	0.000019

In the present study, the AUV geometry and boundary conditions are defined to be symmetric with respect to the longitudinal plane. Therefore, the transverse force and moment components are expected to be theoretically zero. The small non-zero values reported in Table 3 are mainly attributed to numerical effects rather than physical asymmetry. Possible sources include slight mesh asymmetry, discretization errors, and convergence tolerances associated with the iterative solution process. In addition, for cases involving the propeller, a slight numerical imbalance associated with the rotating region and interface treatment may also contribute to small off-axis loads.

3.2. Effects of the Bottom Wall on the CFD Results of the AUV's Hydrodynamic Performance

In this section, the influence of bottom wall proximity on the AUV's hydrodynamic characteristics is investigated. A comparative analysis is conducted for two configurations: the AUV equipped with a propeller and the AUV without a propeller, operating in near-bottom conditions. This comparison aims to clarify the effects of the bottom boundary on the AUV's hydrodynamic performance.

Figure 9 presents a comparison of the velocity field distributions within the computational domain surrounding the AUV as it moves close to the bottom wall, for both configurations. In the present study, the AUV is equipped with a two-bladed propeller operating at a rotational speed of 12,000 rpm.

The results presented in Figure 9 clearly demonstrate the influence of the propeller on the velocity field around the AUV, as well as the effect of bottom-wall proximity on the flow distribution. The contour plots represent the velocity magnitude, where red and yellow regions correspond to high-velocity zones associated with elevated dynamic pressure, while blue regions indicate low-velocity areas around the AUV.

It can be observed that, for the propeller-driven configuration, high-velocity regions are predominantly concentrated in the wake downstream of the propeller. In contrast, for the case without a propeller, a distinct low-velocity wake region is formed behind the AUV. This difference is mainly caused by the propeller action, which produces a propulsive jet that accelerates the downstream flow and increases the velocity magnitude in the wake region. As the operating speed increases, the propeller-induced jet becomes stronger and extends further downstream.

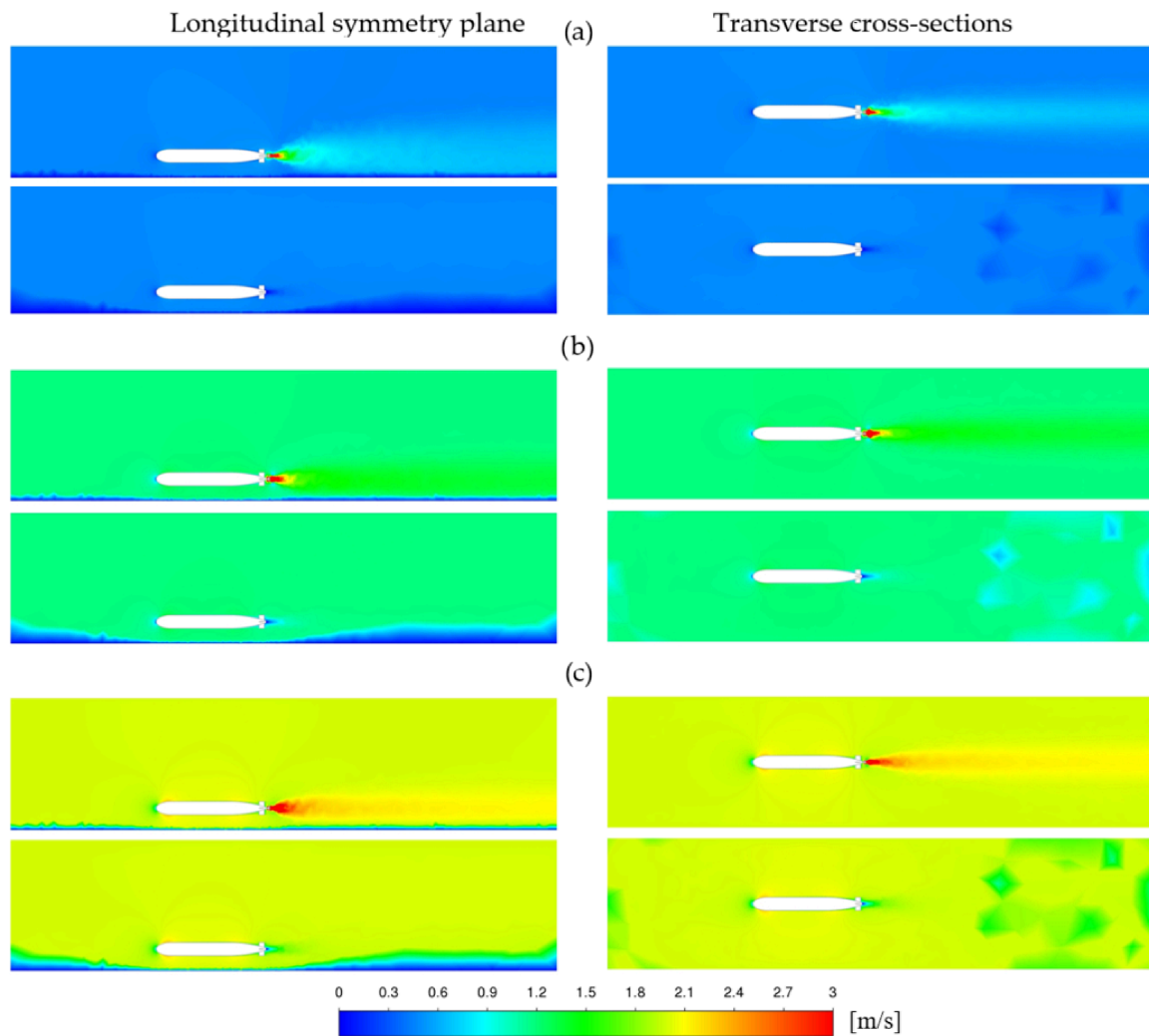


Figure 9. Velocity distribution around the AUV with and without a propeller moving near the bottom wall: (a) $Re = 0.80 \times 10^6$; (b) $Re = 2.09 \times 10^6$; (c) $Re = 3.48 \times 10^6$.

In the absence of a propeller, the flow separates from the hull and control fins, leading to the formation of a low-velocity region in the aft part of the AUV. Moreover, due to the confinement effect between the AUV and the bottom wall, the flow is restricted in this region, resulting in low-velocity zones along the bottom surface. When the propeller is present, these low-velocity regions are significantly reduced and confined closer to the bottom boundary. Additionally, the stern control fins contribute to stabilizing the flow by mitigating disturbances and suppressing vortex shedding in the wake, thereby improving the overall flow uniformity and reducing potential impacts on the surrounding fluid.

To further investigate the vortex structures around the AUV, the vorticity magnitude distributions on both longitudinal and transverse planes passing through the domain center are presented in Figure 10. The results indicate that strong vortical structures are primarily concentrated near the nose and stern regions of the AUV. Furthermore, vortices tend to intensify in the downstream direction and in the vicinity of the bottom wall. In the propeller-driven case, a pronounced vortex region appears immediately downstream of the propeller, corresponding to the initial region of the propeller slipstream. Additional strong vortices are also observed in the wake behind the AUV, highlighting the significant influence of the propeller-induced flow on the downstream hydrodynamic field.

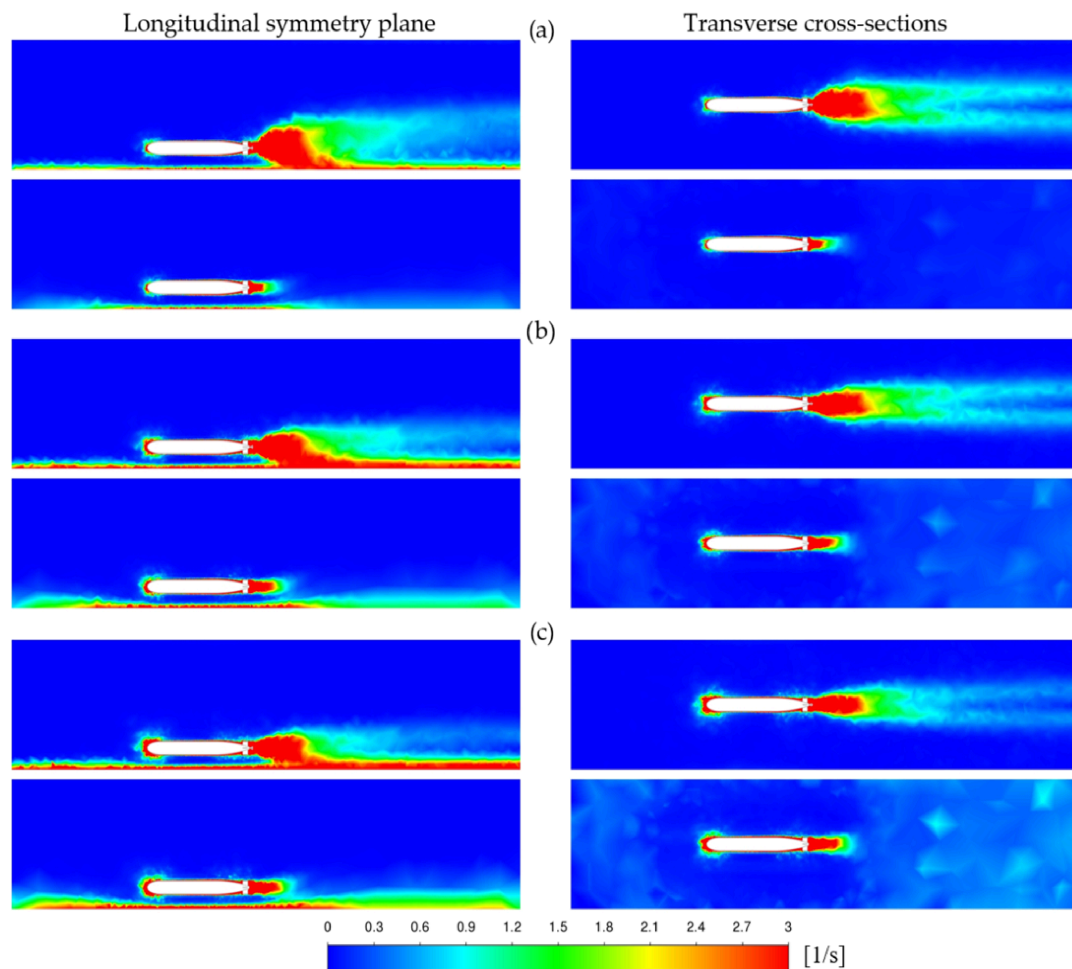


Figure 10. Vorticity magnitude distribution in the computational domain around the AUV with and without a propeller moving near the bottom wall: (a) $Re = 0.80 \times 10^6$; (b) $Re = 2.09 \times 10^6$; (c) $Re = 3.48 \times 10^6$.

Figures 11 and 12 illustrate the CFD-predicted dynamic pressure distributions in the flow field surrounding the AUV and on the hull surface, respectively. The results show that the dynamic pressure magnitude increases with increasing inflow velocity, consistent with the expected dependence of dynamic pressure on flow speed.

High-pressure regions are primarily observed in the downstream area of the propeller, where the flow velocity is significantly elevated. The flow exiting the propeller exhibits a non-uniform pressure distribution, characterized by alternating regions of high and low pressure. This behavior suggests that, as the AUV operates at higher speeds, a pronounced pressure wake is formed downstream, potentially influencing the surrounding flow field and nearby structures.

Further examination of the dynamic pressure distribution on the AUV surface (Figure 11) shows that low-pressure regions are mainly located at the leading edges of the hull and control fins, where local flow velocities are relatively low. In contrast, high-pressure regions correspond to areas of increased flow velocity, particularly in the vicinity of the propeller. These findings provide important insights into the structural loading conditions of the AUV and can be used to identify critical regions requiring reinforcement during high-speed operation.

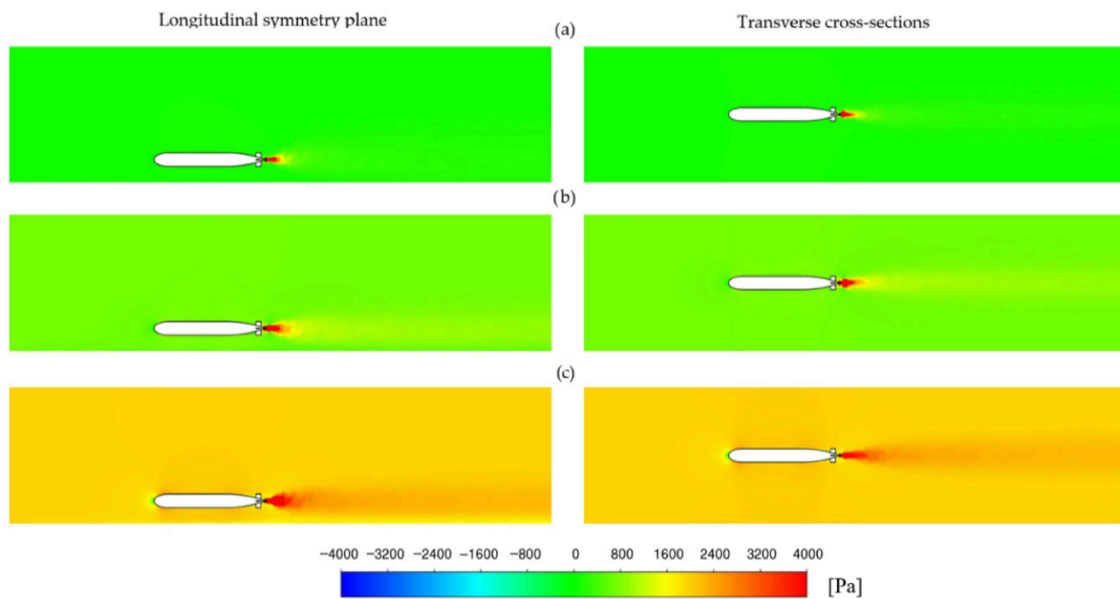


Figure 11. Dynamic pressure distribution around AUV with a propeller in the computational domain: (a) $Re = 0.80 \times 10^6$; (b) $Re = 2.09 \times 10^6$; (c) $Re = 3.48 \times 10^6$.

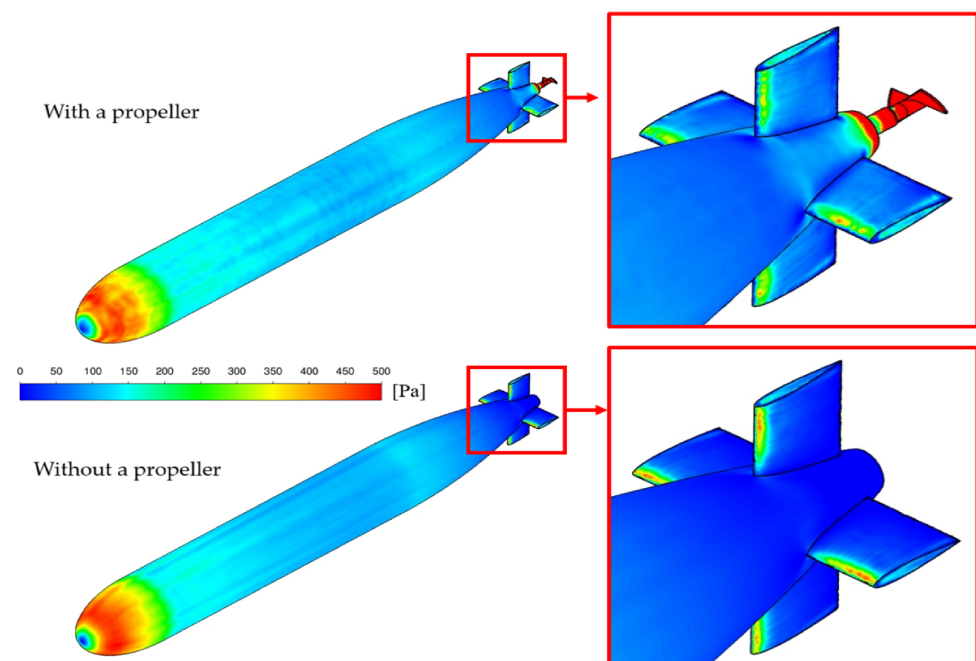


Figure 12. Dynamic pressure distribution on the AUV hull surface with and without a propeller at $Re = 2.09 \times 10^6$.

Furthermore, areas exhibiting elevated dynamic pressure are generally accompanied by a reduction in local static pressure, which may increase cavitation susceptibility and lead to greater hydrodynamic resistance. Consequently, the present analysis also helps identify potential cavitation-sensitive regions on the AUV surface, providing useful insights for the design and optimization of AUV configurations in practical engineering applications.

To further investigate vortex structures, the vorticity distribution based on the Q-criterion iso-surface ($Q = 0.1$) is presented in Figure 12 at $Re = 2.09 \times 10^6$. As shown in Figure 13a, surface vortices are clearly observed along the AUV hull, predominantly forming near the bow region downstream of the leading edge, as well as around the control fins. In addition, distinct tip vortices are identified at the fin tips, mainly concentrated near

the trailing edges. These regions are considered highly susceptible to cavitation, as the local pressure may drop below the vapor pressure of water.

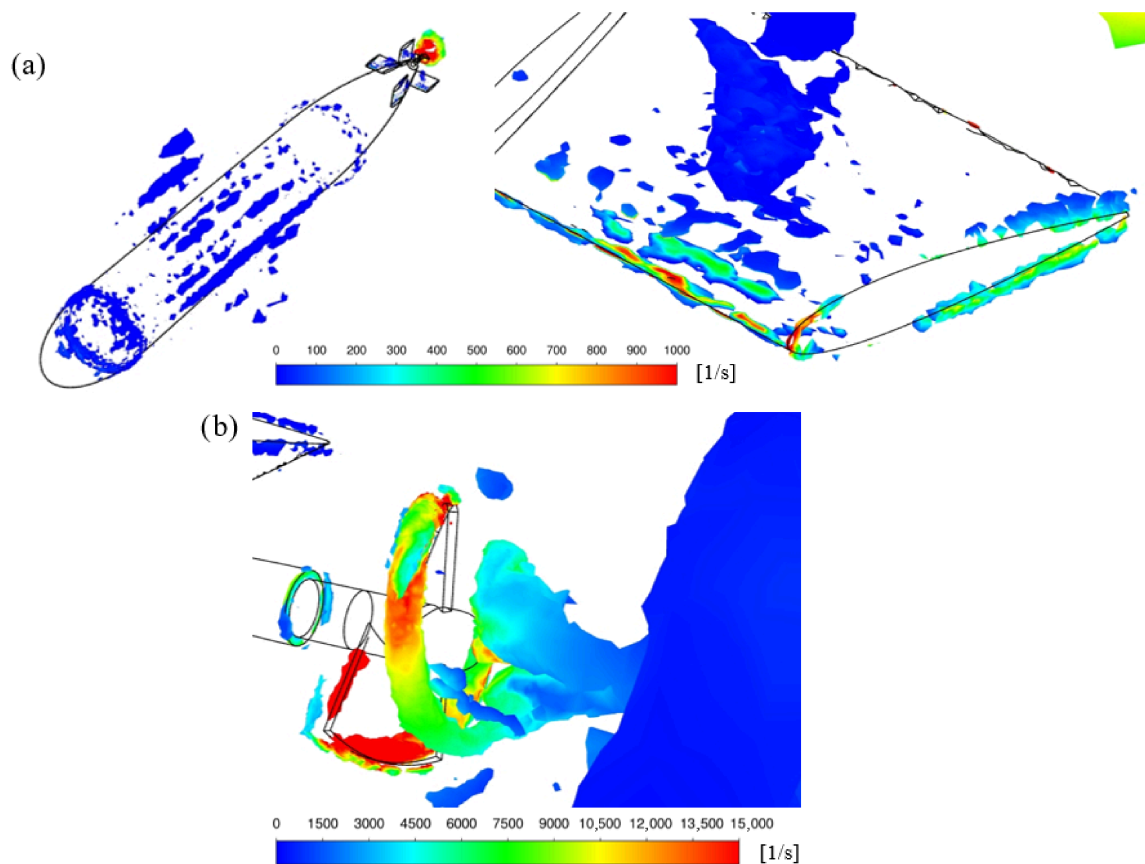


Figure 13. Vortex structures visualized using the Q-criterion ($Q = 0.1$) at $Re = 2.09 \times 10^6$: (a) Around the AUV hull; (b) In the propeller region.

Furthermore, Figure 13b reveals the formation of propeller-induced tip vortices. Intense vortex structures originate near the blade tips and evolve into coherent vortex trails propagating downstream with the flow. Owing to the high rotational motion of the propeller, the pressure inside these vortex cores may decrease substantially, which can increase the susceptibility to cavitation inception.

These findings provide valuable insights for the design and manufacturing of AUV systems, particularly in identifying critical regions that are prone to cavitation-induced damage and thus require careful consideration during structural reinforcement and hydrodynamic optimization. Figure 14 presents a comparison of the AUV's hydrodynamic force and moment with and without a propeller, moving near the bottom wall.

Figure 14 presents a comparative analysis of the force coefficients C_x , C_y , and the moment coefficient C_{Mz} for AUV configurations operating near the bottom wall, both with and without a propeller, in comparison with the free-stream condition. The results clearly indicate that bottom wall proximity has a pronounced effect on the AUV's hydrodynamic performance. In particular, the hydrodynamic force coefficients increase significantly as the AUV operates closer to the bottom boundary. The corresponding quantitative values of the force and moment components are provided in Table 4.

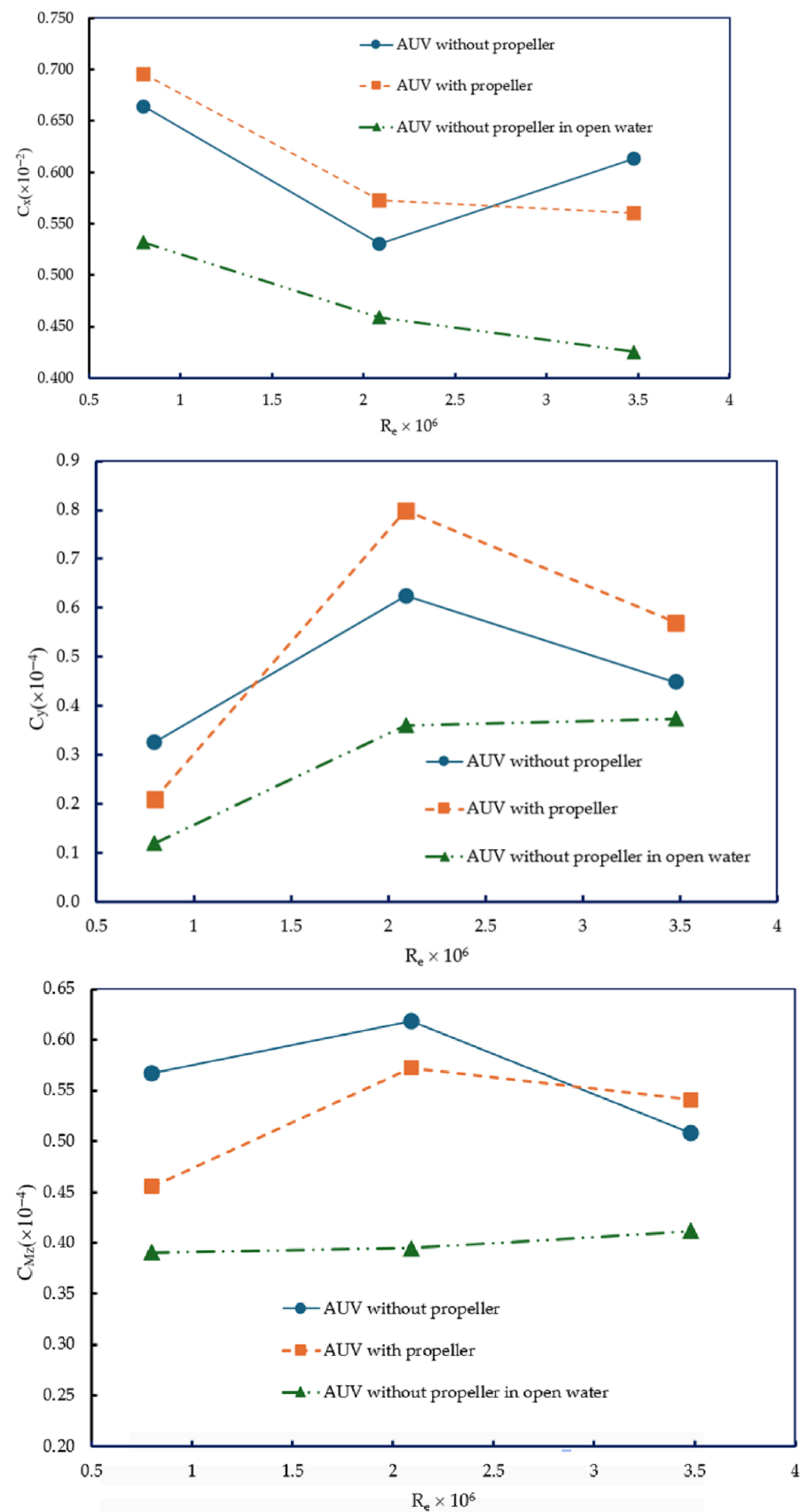


Figure 14. The AUV's hydrodynamic force and moment coefficient corresponding to the Reynolds number.

Table 4. AUV's hydrodynamic forces and moments in the computed case hull without a propeller.

$R_e \times 10^6$	F_x , (N)	F_y , (N)	F_z , (N)	M_x , (Nm)	M_y , (Nm)	M_z , (Nm)
0.80	0.6562	0.0032	0.0032	0.00019	0.00757	0.00868
2.09	3.0174	0.0355	0.0043	0.00112	0.01958	0.05451
3.48	9.6977	0.0708	0.0459	0.00479	0.06325	0.12452
$R_e \times 10^6$	C_x	$C_y (\times 10^4)$	$C_z (\times 10^4)$	$C_{mx} (\times 10^4)$	$C_{my} (\times 10^4)$	$C_{mz} (\times 10^4)$
0.80	0.00664	0.3257	0.3276	0.0127	0.4942	0.5668
2.09	0.00530	0.6238	0.0760	0.0127	0.2221	0.6182
3.48	0.00614	0.4480	0.2905	0.0196	0.2583	0.5084

The results reported in Tables 4 and 5 quantify the AUV's hydrodynamic forces and moments in the computed case of the AUV operating near the bottom wall, both with and without a propeller. Compared to the free-stream condition, significant variations in both force and moment components are observed. The hydrodynamic loads increase markedly with inflow velocity in all cases. Moreover, the interaction between the propeller-induced flow and bottom wall proximity leads to a further amplification of these loads. The combined influence of these effects is detailed in Table 6.

Table 5. AUV's hydrodynamic forces and moments in the computed case hull with a propeller.

$R_e \times 10^6$	F_x , (N)	F_y , (N)	F_z , (N)	M_x , (Nm)	M_y , (Nm)	M_z , (Nm)
0.80	0.6868	0.0021	0.0026	0.00022	0.00840	0.00698
2.09	3.2593	0.0455	0.0305	0.00085	0.01470	0.05048
3.48	8.8518	0.0305	0.0819	0.00546	0.08145	0.13253
$R_e \times 10^6$	C_x	$C_y (\times 10^4)$	$C_z (\times 10^4)$	$C_{mx} (\times 10^4)$	$C_{my} (\times 10^4)$	$C_{mz} (\times 10^4)$
0.80	0.00695	0.2089	0.2633	0.0142	0.5487	0.4559
2.09	0.00573	0.7996	0.5362	0.0096	0.1667	0.5725
3.48	0.00560	0.5703	0.5183	0.0223	0.3326	0.5411

Table 6. Effect on the AUV's hydrodynamic force and moment in the computed case of the AUV hull with and without a propeller moving near the bottom wall.

$R_e \times 10^6$	The AUV Without a Propeller, Moving Near the Bottom Wall					
	(%) C_x	(%) C_y	(%) C_z	(%) C_{mx}	(%) C_{my}	(%) C_{mz}
0.80	20.00	63.66	28.58	11.45	77.09	31.16
2.09	13.53	42.26	160.34	6.21	19.48	36.16
3.48	30.66	16.51	173.95	20.95	26.59	19.02
$R_e \times 10^6$	The AUV with a propeller, moving near the bottom wall					
	(%) C_x	(%) C_y	(%) C_z	(%) C_{mx}	(%) C_{my}	(%) C_{mz}
0.80	23.56	43.35	11.12	21.10	79.37	14.41
2.09	19.95	54.96	108.56	24.29	7.28	31.07
3.48	24.03	34.42	141.44	30.61	42.99	23.91

Table 6 summarizes the comparative hydrodynamic loads for AUV configurations operating near the bottom wall, with and without a propeller, at $h/L_A = 1.93$. The percentage deviations are calculated relative to the free stream, propeller-free reference case. The results indicate a pronounced bottom-wall effect, with the resistance force increasing by up to

30.66%. Additionally, all force and moment components are amplified under near-wall conditions. Notably, the yaw moment coefficient C_{MZ} increases by up to 36.16%, highlighting the significant impact of wall proximity on AUV stability and hydrodynamic performance.

The observed increase in resistance and yaw moment under near-bottom conditions can be attributed to the restriction of flow beneath the hull, which leads to flow acceleration and pressure redistribution. This effect enhances both pressure and viscous contributions to the hydrodynamic forces. In addition, modifications to the wake structure and vortex dynamics contribute to increased flow asymmetry, thereby amplifying the yaw moment. It should be noted that the present study is limited to a single clearance ratio and thus does not allow for the determination of a critical clearance threshold. However, existing studies suggest that bottom effects become increasingly pronounced as the clearance decreases below a certain range, highlighting the need for further parametric investigations.

4. Conclusions and Future Work

In this study, the hydrodynamic performances of an AUV were systematically investigated using CFD, with particular emphasis on the influence of bottom wall proximity under both propeller-assisted and bare-hull configurations. The results clearly demonstrate that the presence of the bottom wall significantly alters the hydrodynamic performances of the AUV. At a relative clearance of $h/L_A = 1.93$, the hydrodynamic resistance increases by up to approximately 30.66% compared to the free-stream condition. Moreover, the presence of a propeller further amplifies the wall-induced effects on the AUV. Compared to the bare-hull configuration, the propeller-assisted AUV experiences a more pronounced increase in hydrodynamic loads. In particular, when operating near the bottom wall, the resistance can increase by up to 24.03% relative to the case without a propeller. These findings highlight the strong coupling between propeller-induced flow and wall effects in modifying the overall hydrodynamic performance.

The present study provides important insights into the hydrodynamic performances of AUVs operating in near-wall environments, where both confinement effects and propulsion-induced flow interactions play a critical role. Such understanding is particularly relevant for applications involving close-proximity operations, such as structural inspection, crack detection, and underwater pipeline monitoring.

Future research will further investigate the hydrodynamic optimization of AUV hull configurations intended for operations in near-wall and near-bottom environments. Particular attention will be given to the influence of hull shape on flow behavior, wake development, and hydrodynamic resistance in order to identify configurations capable of improving overall performance and reducing resistance. The findings are expected to support the development of high-performance AUV platforms for practical underwater applications such as structural inspection, fault diagnosis, and exploration missions in complex operating environments.

It should also be emphasized that the present study considers a representative clearance ratio of $h/L_A = 1.93$ corresponding to a typical near-bottom operating condition. Although the importance of clearance effects is recognized, a comprehensive parametric analysis covering a wider range of clearance ratios is beyond the scope of the current work and is planned for future investigations.

Author Contributions: Conceptualization, N.V.H., methodology, N.V.H. and N.D.; software, N.D. and N.V.H.; validation, N.D. and N.V.H.; writing—original draft preparation, N.V.H.; writing—review and editing N.V.H., supervision, N.V.H.; project administration, N.V.H. All authors have read and agreed to the published version of the manuscript.

Funding: This research is funded by Hanoi University of Science and Technology (HUST) under project number T2025-PC-038.

Data Availability Statement: The original contributions presented in this study are included in the article. Further inquiries can be directed to the corresponding author.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Hong, L.; Wang, X.; Zhang, D.-S. CFD-based hydrodynamic performance investigation of autonomous underwater vehicles: A survey. *Ocean Eng.* **2024**, *305*, 117911. [\[CrossRef\]](#)
2. Van He, N.; Danh, B.T.; Tuan, P.A.; Van Duc, N. Investigated effects of cylinder proximity on the hydrodynamic performance of axisymmetric AUV by using CFD. *Ocean Eng.* **2026**, *353*, 124637. [\[CrossRef\]](#)
3. Chen, C.-W.; Chen, Y.; Cai, Q.-W. Hydrodynamic-interaction analysis of an autonomous underwater hovering vehicle and ship with wave effects. *Symmetry* **2019**, *11*, 1213. [\[CrossRef\]](#)
4. Cao, J.; Feng, Y.; Li, Y.; Li, H.; Han, C.; Sun, Y.; Liu, Z. Research on the hydrodynamic performance of AUVs in small size open channel area. *Ocean Eng.* **2023**, *288*, 116190. [\[CrossRef\]](#)
5. Jiang, M.; Chen, C.; Dong, Y.; Zhang, L.; Tan, L. Research on hydrodynamic characteristics and dynamics model of an open-frame ROV moving near the wall. *Ocean Eng.* **2025**, *320*, 120358. [\[CrossRef\]](#)
6. Tian, W.; Song, B.; Ding, H. Numerical research on the influence of surface waves on the hydrodynamic performance of an AUV. *Ocean Eng.* **2019**, *183*, 40–56. [\[CrossRef\]](#)
7. Wang, X.; Zhang, G.; Sun, Y.; Cao, J.; Wan, L.; Sheng, M.; Liu, Y. AUV near-wall-following control based on adaptive disturbance observer. *Ocean Eng.* **2019**, *190*, 106429. [\[CrossRef\]](#)
8. Wu, L.; Li, Y.; Su, S.; Yan, P.; Qin, Y. Hydrodynamic analysis of AUV underwater docking with a cone-shaped dock under ocean currents. *Ocean Eng.* **2014**, *85*, 110–126. [\[CrossRef\]](#)
9. Li, G.; Chen, J.; Kasaya, T. Hydrodynamic interactions of two AUVs cruising in close proximity: The potential for enhanced energetic efficiency amid significant stability challenges. *Ocean Eng.* **2025**, *323*, 120612. [\[CrossRef\]](#)
10. Kang, S.; Yu, J.; Zhang, J.; Hu, F.; Jin, Q. Research on accurate modeling of hydrodynamic interaction forces on AUVs operating in tandem. *Ocean Eng.* **2022**, *251*, 111125. [\[CrossRef\]](#)
11. Zhang, W.; Jia, G.; Wu, P.; Yang, S.; Huang, B.; Wu, D. Study on hydrodynamic characteristics of AUV launch process from a launch tube. *Ocean Eng.* **2021**, *232*, 109171. [\[CrossRef\]](#)
12. Huang, H.; Bian, X.; Jiang, T.; Tang, Q.; Qin, H.; Wang, Z. Hydrodynamic analysis and manipulation control on a streamlined I-AUV. *ISA Trans.* **2024**, *153*, 453–466. [\[CrossRef\]](#)
13. Mei, B.; Li, C.; Liu, D.; Zhang, J.; Wang, H. AUV maneuvering modeling using system identification methods with adaptive VMD, improved CNN, and free-running model test. *Ocean Eng.* **2025**, *324*, 120645. [\[CrossRef\]](#)
14. Liu, J.; Yu, F.; Yan, T.; He, B.; Soares, C.G. CFD-driven physics-informed generative adversarial networks for predicting AUV hydrodynamic performance. *Ocean Eng.* **2024**, *313*, 119638. [\[CrossRef\]](#)
15. Hou, Y.; Duan, F.; Zhu, W.; Li, H.; Shen, S.; Shen, X.; Wang, J.; Huang, Y.; Wei, W.; Liu, X. Time-sequenced hydrodynamics prediction system for underwater vehicles based on AI edge computing. *Ocean Eng.* **2024**, *294*, 116797. [\[CrossRef\]](#)
16. Zhao, Y.; Hu, Z.; Du, W.; Geng, L.; Yang, Y. Research on modeling method of autonomous underwater vehicle based on a physics-informed neural network. *J. Mar. Sci. Eng.* **2024**, *12*, 801. [\[CrossRef\]](#)
17. Zhang, Q.; Geng, C.; Wang, Z.; Sheng, Z.; Qiao, J.; Huang, Y.; Yu, J. The coupling effect between lateral thruster and uniform incoming flow on the mechanical characteristics of an AUV. *Ocean Eng.* **2026**, *345*, 123663. [\[CrossRef\]](#)
18. Hao, M.; Zhang, F.; Li, N.; Wang, Z.; Liu, Y.; Rao, Z.; Ling, J.; Chen, K.; Wang, R.; Guo, D. Conformable sensing and adaptive vortex generating for active flow control of AUV propulsion systems. *Appl. Therm. Eng.* **2026**, *289*, 129752. [\[CrossRef\]](#)
19. Siau, W.L.; Bonnet, J.-P.; Tensi, J.; Cordier, L.; Noack, B.R.; Cattafesta, L. Transient dynamics of the flow around a NACA 0015 airfoil using fluidic vortex generators. *Int. J. Heat Fluid Flow* **2010**, *31*, 450–459. [\[CrossRef\]](#)
20. Dang, F.; Nasreen, S.; Zhang, F. Background flow sensing for autonomous underwater vehicles using model reduction with dynamic mode decomposition. In *2020 IEEE/ASME International Conference on Advanced Intelligent Mechatronics (AIM)*; IEEE: New York, NY, USA, 2020.
21. Liu, X.; Hu, Y.; Mao, Z.; Tian, W. Numerical Simulation of the Hydrodynamic Performance and Self-Propulsion of a UUV near the Seabed. *Appl. Sci.* **2022**, *12*, 6975. [\[CrossRef\]](#)
22. Hamdamov, M.; Fayziev, R.; Aminov, K.; Muzaffarov, S. A Numerical Study of Turbulent Flows Around a NASA 0012 Profile for Wind Turbines Using the KE Model. In *Proceedings of the 8th International Conference on Future Networks & Distributed Systems, Marrakech, Morocco, 11–12 December 2024*.

23. Van He, N.; Van Hien, N. Viscous Resistance Acting on a Symmetrical Hull with Different Turbulent Viscous Model. In *2022 9th NAFOSTED Conference on Information and Computer Science (NICS)*; IEEE: New York, NY, USA, 2022.
24. Alterskjær, S.; Eggers, R.; Gao, Y.; Kim, Y.; Kume, K.; Trodden, D.; Werner, S.; Zhang, X. *Procedures and Guidelines*; International Towing Tank Conference: Hobart, Australia, 2024.
25. Sunarsih; Nanda, M.I.; Purnamasari, D.; Baidowi, A.; Syarif, I.S. Verification and validation of ITTC benchmark ship CFD performance towards development of CFD and EFD combined methods. *IOP Conf. Ser. Earth Environ. Sci.* **2023**, *1198*, 012027.
26. Zhang, L.; Chen, W.; Chen, J.; Zhou, C. Verification and validation of CFD uncertainty analysis based on SST k- ω model. In *International Conference on Offshore Mechanics and Arctic Engineering*; American Society of Mechanical Engineers: New York, NY, USA, 2020.
27. Zhao, W.-W.; Wang, J.-H.; Wan, D.-C. Vortex identification methods in marine hydrodynamics. *J. Hydrodyn.* **2020**, *32*, 286–295. [[CrossRef](#)]
28. Roddy, R.F. *Investigation of the Stability and Control Characteristics of Several Configurations of the DARPA SUBOFF Model (DTRC Model 5470) from Captive-Model Experiments*; David Taylor Research Center (DTRC): West Bethesda, MD, USA, 1990.
29. Liu, H.-L.; Huang, T.T. *Summary of DARPA SUBOFF Experimental Program Data*; Defense Technical Information Center (DTIC): Fort Belvoir, VA, USA, 1998.
30. Lin, Y.-H.; Tseng, S.-H.; Chen, Y.-H. The experimental study on maneuvering derivatives of a submerged body SUBOFF by implementing the Planar Motion Mechanism tests. *Ocean Eng.* **2018**, *170*, 120–135. [[CrossRef](#)]
31. Van Hien, N.; Van He, N.; Diem, P.G. A model-driven implementation to realize controllers for Autonomous Underwater Vehicles. *Appl. Ocean Res.* **2018**, *78*, 307–319. [[CrossRef](#)]
32. Van He, N.; Ikeda, Y. Optimization of bow shape for a non ballast water ship. *J. Mar. Sci. Appl.* **2013**, *12*, 251–260. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.