

Article

Numerical Analysis of Air-Injection Drag Reduction for the KVLCC2 Hull Using the VOF Interface-Capturing Method

Xiaojie Zhao ¹, Yanping Hao ¹ and Qi Zhang ^{2,*} ¹ School of Naval Architecture and Ocean Engineering, Jiangsu University of Science and Technology, Zhenjiang 212100, China; xjzhao@just.edu.cn (X.Z.); haoyp_1998@163.com (Y.H.)² State Key Laboratory of Robotics and Intelligent Systems, Shenyang Institute of Automation, Chinese Academy of Sciences, Shenyang 110016, China

* Correspondence: zhangqi_hydy@163.com

Abstract

To investigate the air layer drag reduction and the related flow field characteristics of ships, the gas–liquid two-phase numerical model using the VOF solver in STAR-CCM+ has been established, considering the effects of free surface and surface tension. The numerical model is first validated through experimental results for the drag reduction by air-injection on a simplified ship model. Then, the numerical simulations for the KVLCC2 at varying speeds and air-injection rates are conducted, considering different ship attitudes and air-injection surface configurations. The impacts of flow velocity, air-injection rates, ship attitude and air-injection configurations on air layer drag reduction are analyzed. The distributions of air and pressure around the ship and their influence mechanisms on drag reduction are discussed. The simulation results show that the drag reduction exhibits a positive correlation with air-injection rate until it reaches an optimal peak value. The combined action of the incoming flow and injection velocities causes the vortex recirculation of the air layer under the ship, leading to its disruption and the subsequent formation of air-free zones on the hull bottom. High air-injection rates and the stern trim induce air layer lateral spillage, increasing frictional resistance on the hull side surfaces. An air layer on the stern surface will reduce the viscous pressure resistance by changing the flow separation near the ship stern. Air-layer coverage area is closely correlated with inflow velocity and injection surface configurations. The reasonable configurations of the air-injection surfaces can significantly improve the drag reduction.

Keywords: volume of fluid model; air layer; drag reduction; ship; numerical simulation

Academic Editor: Decheng Wan

Received: 1 November 2025

Revised: 20 November 2025

Accepted: 24 November 2025

Published: 27 November 2025

Citation: Zhao, X.; Hao, Y.; Zhang, Q. Numerical Analysis of Air-Injection Drag Reduction for the KVLCC2 Hull Using the VOF Interface-Capturing Method. *J. Mar. Sci. Eng.* **2025**, *13*, 2253. <https://doi.org/10.3390/jmse13122253>

Copyright: © 2025 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) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction

In recent years, the carbon emission pollution caused by maritime transportation has increasingly attracted attention. The consumption of ship fuel is mainly used to overcome various resistances encountered during ship navigation. Air lubrication technology, as an emerging drag reduction technology, can effectively reduce resistance during navigation and is characterized by its advantages of being environmentally friendly, highly efficient and easy to operate. Based on the gas–liquid interfacial morphology, air lubrication drag reduction systems can be fundamentally categorized into two distinct regimes [1], namely dispersed-bubble drag reduction and continuous air layer drag reduction. The continuous air layer drag reduction achieves significant hydrodynamic resistance mitigation by establishing a stable air film along the ship’s bottom surface, thereby reducing the wetted

surface area of the bodies. Cai's [2] tests on a 1000-ton barge demonstrated that air layer technology reduced drag by approximately 30% in the early stages. Park et al. [3] conducted towing-tank tests on a 50,000 dwt medium-range product-tanker model fitted with an air layer drag reduction system. By optimally splitting the airflow between two injectors, they achieved an 18.1% reduction in model-scale total resistance. Mohamad et al. [4] conducted an experimental study on a flat plate in a towing tank. They found that the turbulence effect facilitates the coexistence of the air layer with the transitional boundary layer, eventually covering a larger wetted surface area and achieving a drag reduction of up to 25%.

The main difficulty of air layer drag reduction (ALDR) technology lies in forming a large and stable air layer under the bottom surface of the hull. Consequently, previous studies have conducted extensive experimental research on issues such as the characteristics of the air layer in ALDR. Dong et al. [5] conducted air layer drag reduction tests on three planing craft and concluded that larger injection slots and pores benefit practical implementation. Mäkiharju [6,7] analyzed the effects of single-orifice and multi-orifice gas injection on air layer topology formation. To mitigate the loss of drag reduction from air layer escape, substantial research has been conducted. Huang et al. [8] experimentally investigated large flat plates under high-Reynolds-number conditions and achieved a 9 m drag reduction persistence length by fitting 50 mm baffles along both edges. Gao et al. [9] used a flat-plate setup to compare injection schemes and found that both stepped streamwise slots and vertical orifices produce a uniform, stable air layer. Mao et al. [10] tested a scaled ship model to evaluate side versus bottom air injection for surface-film drag reduction, finding bottom injection superior at low speeds and side injection more effective at high speeds. Kringel et al. [11] observed side-edge vortex shedding and the influence of varying flow regimes on gas-phase topology and the resulting drag reduction performance in their flat-plate towing experiments.

Many numerical investigations have also been performed on the ALDR. Li et al. [12] conducted the numerical analysis of air curtain drag reduction for deep-V hull forms using a hybrid two-phase flow model. Their findings demonstrated that employing multi-stage air injection can enhance the drag reduction rate by up to 12.07%. An et al. [13] numerically investigated a strut-equipped surface vessel and found that the pressure-drag reduction rate grows with increasing air layer thickness. Subsequently, Zheng et al. [14] conducted a systematic numerical and experimental study on the air layer morphology and drag characteristics of a scaled axisymmetric submarine model (SUBOFF); the results indicated that excessive ventilation causes gas entrainment, leading to air leakage and an increase in pressure drag. Chen et al. [15] summarized the global research advances in air layer drag reduction, concluding that cavity configurations on hull bottoms significantly enhance air layer stability and drag reduction performance, representing a promising research domain. Wu et al. [16,17] experimentally investigated air-injection drag reduction on a cavitating flat-plate model, establishing key dependencies on velocity and airflow rate. Subsequent validation with a 95,000 DWT bulk-carrier model confirmed that under non-injection conditions, resistance would increase for both the plate and the ship model. Furthermore, cavity configurations significantly enhanced the drag reduction when the air was injected. Ye et al. [18] executed air lubrication drag reduction tests on a 95,000 DWT bulk carrier model in a towing tank. The results demonstrated that an optimal groove depth exists ($h/B = 0.024$) to maximize air layer drag reduction. Fang et al. [19] simulated the drag reduction effect and cavity morphology of an air cavity ship (ACS). The results indicated that after establishing a stable cavity configuration, the excessive air injection would lead to the expansion of the stern air outlet in both width and thickness rather than increasing the air coverage area. Zhang et al. [20] performed numerical simulations of air layer wave characteristics within a cavity, revealing that the upstream wave height consistently

surpassed the first downstream wave crest. These elevated crests, which contacted the cavity roof, induced localized air layer rupture, which can explain the experimentally observed central void region phenomenon. While benefiting from the drag reduction effect of an air layer, several researchers have also considered its influence on the propulsor. Dong et al. [21] using viscous-flow theory, investigated the mutual interference between the hull and the propeller of a planing craft equipped with air layer drag reduction; their results showed that the air layer exerts a favorable effect on propeller performance. Shao et al. [22] performed numerical simulations—validated by experimental data—on a pump-jet propulsor of an underwater vehicle that employs rear-end air injection for drag reduction. The results revealed that the motion of air within the blade passages is dominated by the pressure-gradient force; substantial gas accumulates on the rotor suction side and gathers at the trailing edge, altering the local flow structure, introducing additional flow losses and reducing hydraulic efficiency.

According to previous studies, people have paid more attention to the total drag reduction effects of plates or ships in ALDR. However, ships have various drag components, such as frictional resistance and viscous pressure resistance. Existing studies have shown that air layers influence the frictional drag and pressure drag of underwater vehicles through different mechanisms. However, research on air layer drag reduction for surface ships remains limited. Major knowledge gaps still exist in several areas. These include whether underwater air layers remain continuous or exist as discrete air masses, whether the effects of air–liquid surface tension need to be considered, and the underlying mechanisms by which air layers affect drag components such as frictional and pressure resistance. Other unresolved issues are the impact of lateral air leakage on drag, and the lack of sufficient investigation into how air layers influence stern pressure distribution.

Therefore, this paper establishes a numerical model for air layer drag reduction using the VOF solver in STAR-CCM+. The model accounts for free-surface effects and surface tension. The numerical studies are first conducted on a simplified ship model at different speeds and varying air-injection rates, and the feasibility and accuracy of the numerical model are validated by the comparison of experimental measurements and numerical results. Subsequently, the numerical studies are conducted on ALDR for different resistance components of the KVLCC2 ship model. The effects of speed, air-injection rate and trim on drag reduction rates are examined. The mechanisms of the air layer instability and the impacts of the air layer on the frictional drag and viscous pressure drag are elucidated. Finally, a preliminary investigation is conducted into the influences of different air-injection configurations on air layer drag reduction. This study systematically investigates the mechanisms underlying the breakdown of bottom-air-layer continuity, the effects of lateral-air-layer spillage on the ship's side walls, and the mechanism by which the stern air layer influences pressure drag; finally, it preliminarily examines how different air-injection configurations affect air layer stability. The findings of this study can further improve the understanding of air layer drag reduction for the ships and guide the design of practical ALDR techniques.

2. Numerical Models

2.1. Governing Equations

The numerical simulations of the resistance of the two ship models are conducted using STAR-CCM+ software. The Reynolds-averaged Navier–Stokes (RANS) equations consist of the continuity equation and the momentum equations, and they are the primary governing equations for viscous fluid dynamics. These equations are given by

$$\nabla \cdot \mathbf{U} = 0 \quad (1)$$

$$\frac{\partial(\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = -\nabla p + \nabla \cdot \left[\mu \left(\nabla \mathbf{U} + (\nabla \mathbf{U})^T \right) \right] + \rho \mathbf{g} + \mathbf{f}_\sigma \quad (2)$$

where $\mathbf{U}$ is the velocity vector, μ is the viscous coefficient of fluid motion, $\mathbf{g}$ is the acceleration of gravity, ρ is the fluid density, and p is the pressure. To incorporate the effects of surface tension at the gas–liquid interface, surface tension effects cannot be ignored as the air layer beneath the hull is completely surrounded by water. These effects govern bubble coalescence below the injection surface and dictate the topological shape of the air layer. The following expression is used:

$$\mathbf{f}_\sigma = \sigma \kappa \nabla \alpha \quad (3)$$

where α is the phase volume fraction; $\mathbf{f}_\sigma$ is the surface tension coefficient, and its value is taken as 0.072 N/m in this study. κ is the curvature of the free surface, which is defined as

$$\kappa = -\nabla \cdot \frac{\nabla \alpha}{|\nabla \alpha|} \quad (4)$$

2.2. Turbulence Model

The RANS equations require turbulence models to achieve closure. The k - ω Shear Stress Transport (SST) model is applied here, and this model demonstrates superior accuracy for flows featuring strong adverse pressure gradients and separation. This model integrates the k - ω model for near-wall regions with the k - ε model for outer flow regions through blending functions and shear stress corrections. Consequently, it is widely regarded as the optimal turbulence modeling choice for boundary layer drag reduction applications. For a detailed description of the turbulence model, please refer to the official STAR-CCM+ 2306 User Guide. The relevant equations and parameter settings will also be presented in this paper. The transport equations for the turbulent kinetic energy k and the specific dissipation rate ω are as follows:

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho k \mathbf{U}) = \nabla \cdot [(\mu + \sigma_k \mu_t) \nabla k] + P_k - \beta^* \rho k \omega + S_k \quad (5)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \nabla \cdot (\rho \omega \mathbf{U}) = \nabla \cdot [(\mu + \sigma_\omega \mu_t) \nabla \omega] + P_\omega - \beta \rho \omega^2 + S_\omega \quad (6)$$

where P_k and P_ω are production terms; the detailed calculation formulas can be found in the official help documentation of STAR-CCM+. S_k and S_ω are the user-specified source terms. The turbulent eddy viscosity μ_t is calculated as

$$\mu_t = \rho k \max \left(\frac{\alpha^*}{\omega}, \frac{\alpha_1}{S F_2} \right) \quad (7)$$

where S is the average strain rate tensor. F_2 is the second blending function calculated as

$$F_2 = \tanh \left\{ \left[\max \left(\frac{2\sqrt{k}}{\beta^* \omega y'}, \frac{500\nu}{y^2 \omega} \right) \right]^2 \right\} \quad (8)$$

where α_1 and β^* are the empirically determined correction coefficients with values of 0.31 and 0.09, respectively; σ_k , σ_ω , and α^* are the model coefficients derived through a blending function that interpolates between the corresponding turbulence model coefficient φ_1 (from the k - ω model) and φ_2 (from the k - ε model).

$$\varphi = \varphi_1 F_1 + (1 - F_1) \varphi_2 \quad (9)$$

where φ represents the parameters σ_k , σ_ω , and α^* , respectively, and the blending function F_1 interpolates between the near-wall value and the far-field counterpart of any coefficient, defined as

$$F_1 = \tanh \left\{ \left\{ \min \left[\max \left(\frac{\sqrt{k}}{0.09\omega y}, \frac{500\nu}{y^2\omega} \right), \frac{2k}{y^2 CD_{k\omega}} \right] \right\}^4 \right\} \quad (10)$$

where y is the distance to the wall, ν is the kinematic viscosity, and $CD_{k\omega}$ is the cross-diffusion coefficient, which is defined as

$$CD_{k\omega} = \max(\nabla \kappa \cdot \frac{\nabla \omega}{\omega}, 10^{-20}) \quad (11)$$

2.3. VOF and Interface Capturing Method

To accurately capture the morphological changes in the gas–liquid interface beneath the ship, the VOF method is employed to locate and track the interfacial dynamics. A new mixed fluid is defined, whose density and viscosity are taken as the volume-weighted averages of air and water. The volume fraction α denotes gas occupancy. $\alpha = 0$ indicates pure water, $\alpha = 1$ indicates pure air, and $0 < \alpha < 1$ indicates the gas–liquid interface, where α represents the air volume fraction. The calculations of density and viscosity are

$$\rho = (1 - \alpha)\rho_l + \alpha\rho_a \quad (12)$$

$$\mu = (1 - \alpha)\mu_l + \alpha\mu_a \quad (13)$$

where ρ is the mixture density. ρ_l and ρ_a are the densities of water and air, respectively. μ is the mixture viscosity. μ_l and μ_a are the viscosity of water and air, respectively. The volume fraction transport equation is

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{U}\alpha) + \nabla \cdot (\mathbf{U}_c \alpha(1 - \alpha)) = 0 \quad (14)$$

where $\mathbf{U}_c$ is the compression velocity, which is used for sharpening the gas–liquid interface. This term only acts at the gas–liquid interface, suppressing the diffusion behavior of the interface while not affecting the overall phase field solution. The calculation formula for the compression velocity is $\mathbf{U}_c = C_\alpha |\mathbf{U}| (\nabla \alpha / |\nabla \alpha|)$, where C_α is the sharpening factor for reducing numerical diffusion. This study adopts a first-order discretization scheme with $C_\alpha = 1$ to achieve a sharp interface.

3. Computational Materials and Methods

3.1. Geometric Model

Referring to the model test results of a 2600 DWT bulk carrier by Ding et al. [23], the air layer side overflow will occur if the hull bottom is unmodified. Based on Wu's experimental study of air-injection drag reduction with grooved plates, the baffle installation effectively prevents air layer lateral leakage with a very small increase in resistance [24]. Therefore, in order to prevent the air layer lateral leakage from the ship bottom, the baffles on the hull bottom are installed in this study.

Figure 1 shows the schematic diagram of the KVLCC2 hull at a 1:58 model scale. The main parameters of KVLCC2 are listed in Table 1. The baffles measuring 2000 mm in length, 30 mm in height, and 1 mm in thickness are symmetrically installed along both sides of the hull. Additionally, one air jetting surface is arranged along the centerline of the hull bottom, and it is 600 mm in length and 100 mm in width.

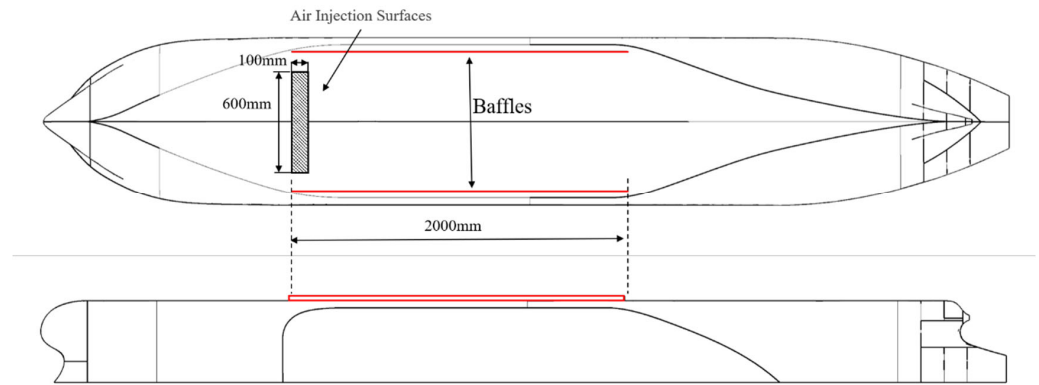


Figure 1. The schematic the KVLCC2 hull with baffles.

Table 1. Main parameters of KVLCC2.

Parameters	Value	
	Ship	Model
scale	1	58
Lpp (m)	320	5.5172
Lwl (m)	325.5	5.6121
Bwl (m)	58	1
D (m)	30	0.5172
T (m)	20.8	0.3586
Displacement (m ³)	312,622	1.6023
S w/o rudder (m ²)	27,194	8.0838
KG(m)	18.6	0.3207

3.2. The Definition of Drag Reduction and Air Injection Rate

To elucidate the air layer's distinct effects on specific resistance components in the KVLCC2 drag reduction calculations, this study employs the defined reduction rates for frictional (η_f), pressure differential (η_p), and total resistance (η_t), and they are calculated by:

$$\eta_i = \frac{R_{0i} - R_i}{R_{0i}} \times 100\% \quad (15)$$

where the drag reduction efficiency is denoted by η_i . The subscripts $i = f, p$ and t denote the frictional drag, pressure drag and total drag, respectively. R_{0i} represents the resistance component without air injection, and R_i corresponds to the resistance with active air injection.

In the simulations on the air-lubrication drag reduction, the dimensionless air-injection flux Q_v introduced by Madavan et al. [25] is adopted here and it is calculated as:

$$Q_v = \frac{Q_a}{Q_a + Q_w} \quad (16)$$

where Q_a denotes the actual air-injection rate. Q_w represents the liquid flow rate within the boundary layer. Based on boundary layer theory, Q_w can be computed with the following expression:

$$Q_w = u(\delta - \delta^*)b \quad (17)$$

$$\delta = \frac{0.37x}{Re_x^{0.2}} \quad (18)$$

$$\delta^* = 0.125\delta \quad (19)$$

where u denotes flow velocity, b represents the injection surface width, δ indicates the boundary layer thickness, δ^* denotes the boundary layer displacement thickness, x is the streamwise coordinate, and Re_x is the Reynolds number based on the characteristic length x .

3.3. Computational Domain and Boundary Conditions

The computational domain and mesh generation are illustrated in Figure 2. To reduce computational resource consumption, the semi-domain is employed to lower the computational cost. The domain extends longitudinally over $4L_{pp}$, with the velocity-inlet boundary $1L_{pp}$ upstream of the bow and the pressure-outlet boundary $2L_{pp}$ downstream of the stern. The height of the computational domain is $1.5L_{pp}$, with velocity-inlet boundaries on the top and bottom surfaces. The velocity-inlet boundaries are placed $1L_{pp}$ from the ship's side, with a symmetry-plane condition applied at the semi-domain centerline. The same boundary conditions above will be used for both the validation and the KVLCC2 cases. To achieve air-injection conditions, the jetting surfaces of both the simplified ship and the KVLCC2 model are assigned mass-flow inlet boundary conditions. A grid refinement was implemented in the vicinity of the vessel, with Figure 3 presenting the local refined grids near the ship. A three-layer prismatic boundary layer mesh is constructed on the hull surface, and the dimensionless wall distance of $y^+ = 35$ is used.

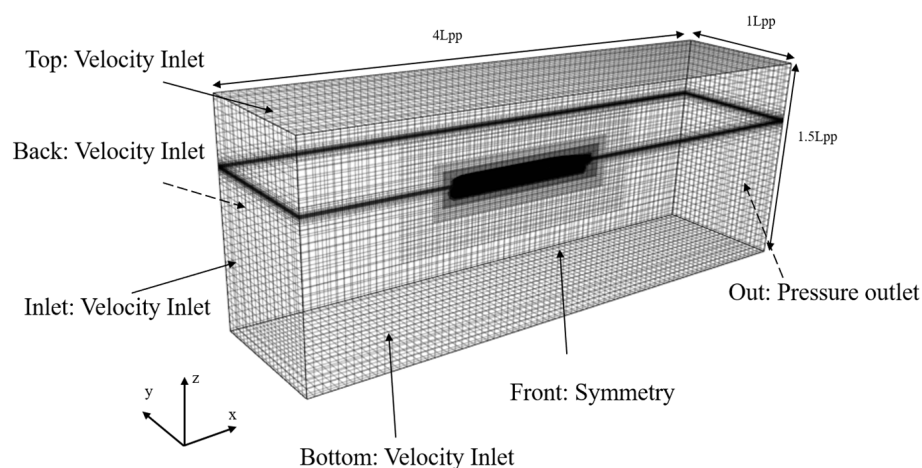


Figure 2. Computational domain and boundary conditions.

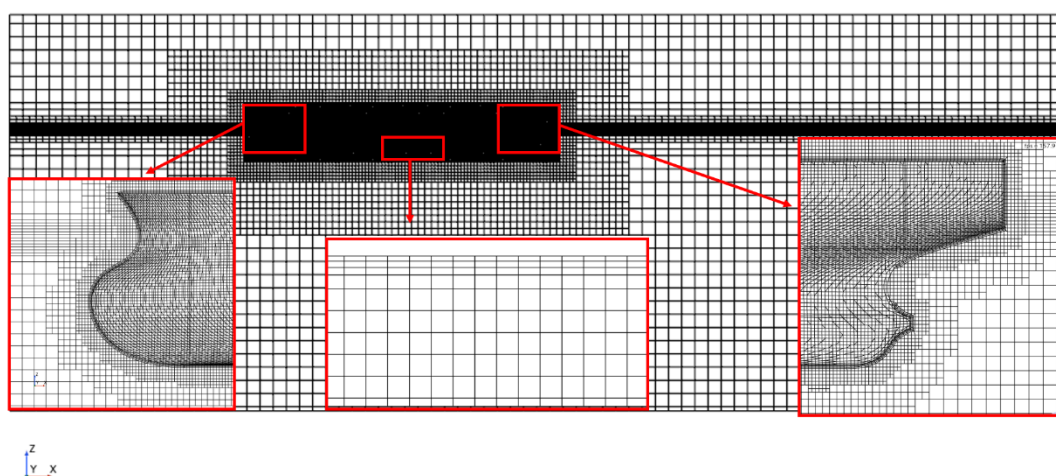


Figure 3. Local refined grids near the ship.

3.4. Mesh Independence and Parameter Settings

Three mesh configurations are employed to verify mesh independence for the model, in order to determine the optimal element size and reduce computational errors due to discretization. Both the simplified hull and the KVLCC2 are evaluated for hydrodynamic resistance at three distinct grid resolutions. Detailed numerical results are provided in Table 2. The computational results from the three grid resolutions exhibited strong convergence. Considering the accuracy requirements for subsequent air lubrication drag reduction simulations and computational resource constraints, grid configurations with 1.59 million cells for the simplified vessel and 1.48 million cells for the KVLCC2 model are selected for the final implementation.

Table 2. Effect of mesh size on model resistance.

Case	Test Ship			KVLCC2		
	Number (Million)	Total Resistance (N)	Change Rate (%)	Number (Million)	Total Resistance (N)	Change Rate (%)
EXP	-	5.154	0	-	-	-
Grid1	0.86	5.183	0.56	1.04	9.08	0
Grid2	1.45	5.130	−0.47	1.48	9.00	−0.88
Grid3	2.22	5.062	−1.78	2.21	9.02	−0.66

Table 3 presents the resistance coefficients of the bare-hull KVLCC2 measured by the Maritime and Ocean Engineering Research Institute, together with the numerically predicted values for the standard KVLCC2 and the baffled KVLCC2 using the second grid resolution. The comparative analysis reveals strong agreement between numerical simulations and experimental data, with discrepancies limited to approximately 0.4%. The resistance coefficient C_t calculation methodology is explicitly defined by

$$C_t = 0.5\rho U^2 S_{\text{water}} / R_t \quad (20)$$

Table 3. KVLCC2 and KVLCC2 (With baffles).

Model	Total Resistance (N)	Total Resistance Rate (%)	Change Rate (%)
EFD	-	0.4056	-
KVLCC2	9.00	0.4072	0.39
KVLCC2 (With Baffle)	9.18	0.4136	1.98

The hydrodynamic evaluation shows that the baffles can introduce a slight additional resistance due to their own frictional resistance. However, this drag increase is very small, with a value of less than 2%. Compared to the drag reduction effect produced by the air layer, this increase in resistance can be entirely offset. Furthermore, under air injection, the friction resistance of the baffle will be further reduced due to the decrease in wet surface. Since this study mainly investigates the drag reduction by the air layer on the ships, the baffles are installed to prevent the air from escaping. The air injection will ultimately achieve the reduction in ship resistance, so the effect of the baffles on the ship drag will be neglected in the study.

3.5. Numerical Setup and Calculation Conditions

The governing equations are discretized and solved using the finite-volume method. The transient term is discretized with a first-order implicit scheme, the convective term with a second-order upwind scheme, and the diffusive term with a central-difference scheme. The Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) is adopted to handle pressure–velocity coupling. An implicit unsteady solver is employed, with the time step

constrained by a local Courant number (CFL) condition of $CFL < 1.0$. To further illustrate the SIMPLE algorithm, Figure 4 presents its flowchart.

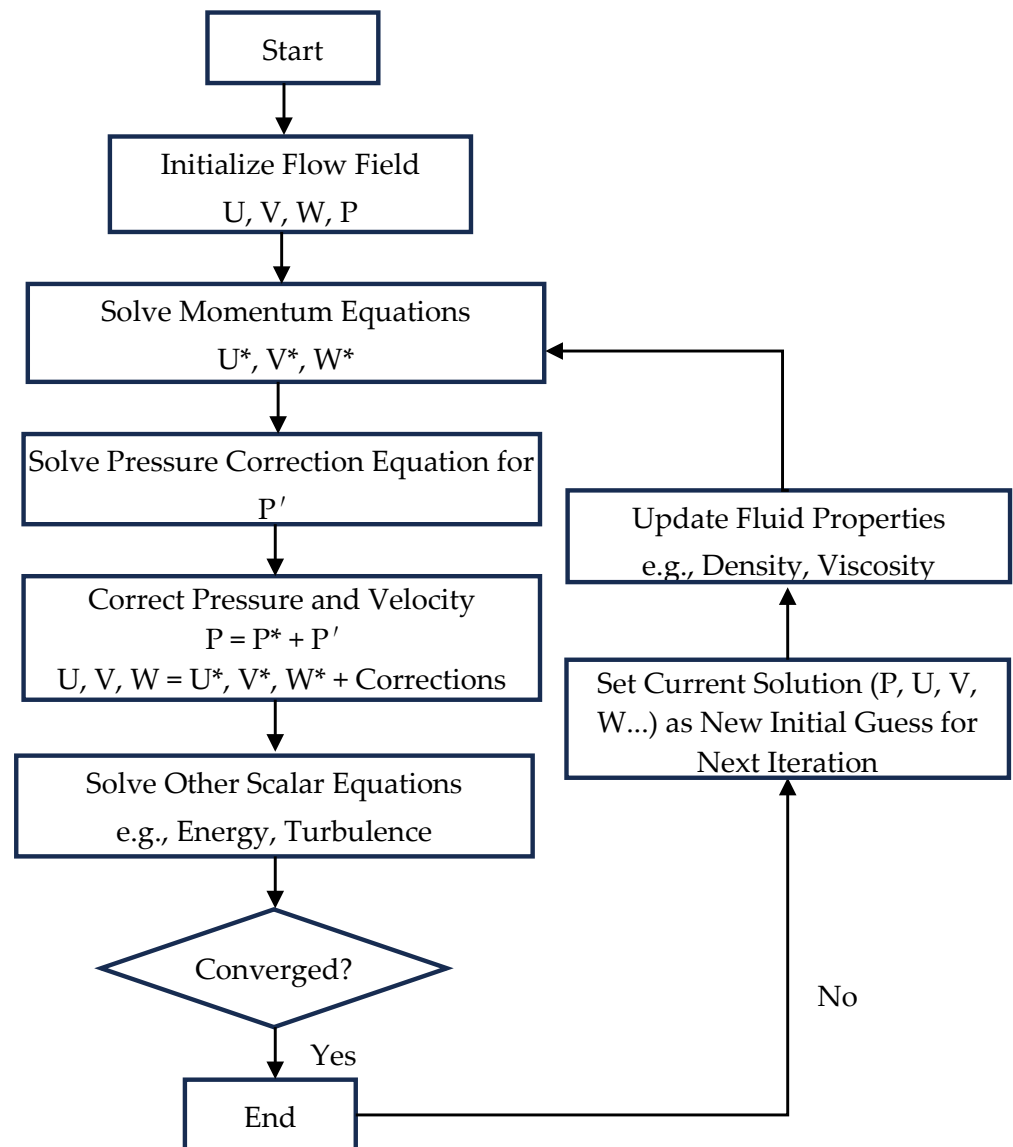


Figure 4. SIMPLE algorithm flowchart.

For the KVLCC2 ship model, the relevant parameters, including air-injection rate, ship speed, navigation attitude, and the number of air-injection surfaces, are taken into account in our numerical investigation on the drag reduction characteristics and air layer morphology variations of the KVLCC2 model. Two flow speeds with the Froude numbers of 0.142 and 0.156 are considered. The dimensionless air-injection flux is between 0 and 0.1 with an interval of 0.02. For the Froude number of 0.142, other trim angles and the number of air-injection surfaces are considered. The detailed calculation conditions are listed in Table 4.

Table 4. Computational conditions.

Fr	Q_v	Trim (deg)	Number of Air Injection Surfaces
0.142	0 0.02 0.04 0.06 0.08 0.10	0, 0.1, 0.15	one, two
0.156	0 0.02 0.04 0.06 0.08 0.10	0	one

4. Results and Discussion

4.1. Validation of Numerical Model

Due to the lack of experimental data on ALDR, the present numerical model for ALDR on the ships is validated using the experiments of ALDR on the simplified ship by Zhao and Zong [26]. The simplified ship is shown in Figure 5. This ship model is 3 m in length, 0.6 m in width and 0.3 m in depth. The baffles are also installed on the bottom of the ship to prevent air escape. Here, the draft of this ship model is 0.048 m. Three water speeds, including 0.651 m/s, 0.868 m/s and 1.084 m/s are considered. The range of air-injection rates Q_a is from 0 to 10 m³/h. Detailed information on the ship model and experimental conditions for this ship model can be found in reference [26].

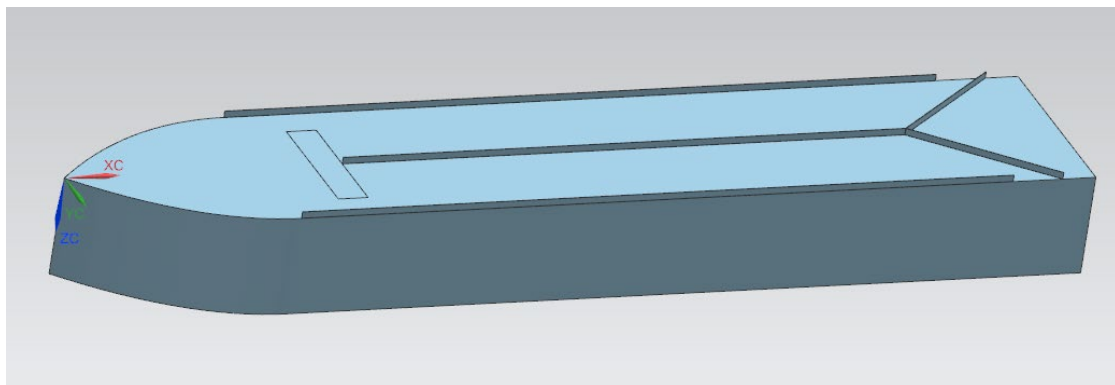
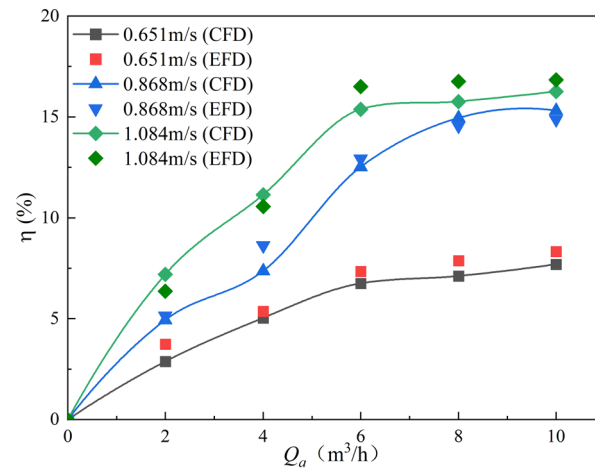
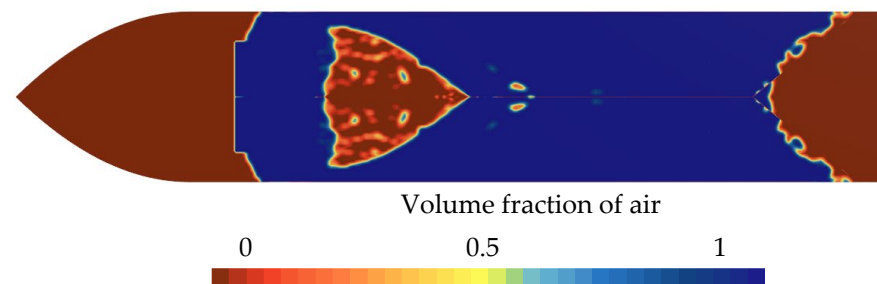
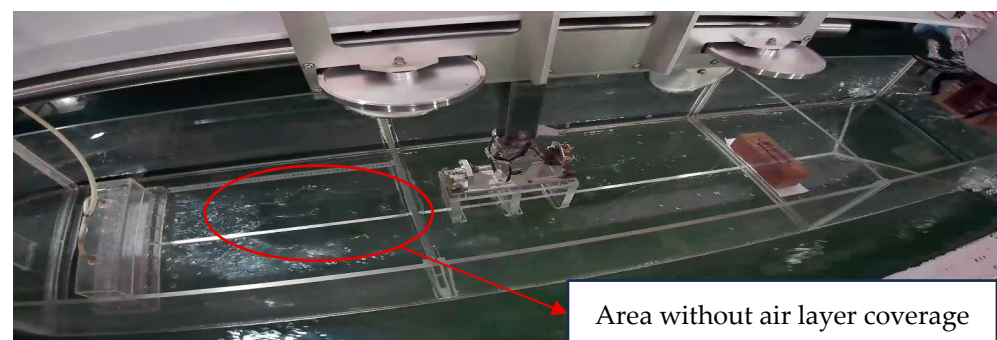


Figure 5. The simplified ship for validation.

To validate the model accuracy, the drag data at zero air injection and an air-injection rate of 8 m³/h were selected as the basic verification dataset. Table 5 presents the comparison of drag values between experiments and numerical simulations under three different navigation speeds. Figure 6 shows the air layer drag reduction curves obtained from both experimental ship tests and numerical simulations. The results depicted in the figure demonstrate that the numerical model accurately predicts the total resistance of the test vessel across three distinct speed conditions, with a maximum computational error of approximately 2.4%. Figure 6 shows that the higher the ship speed, the better the drag reduction effect, indicating that the air layer drag reduction technology achieves optimal drag reduction performance at high speeds. Furthermore, as the air-injection rate increases, the drag reduction rate initially increases substantially. When the air-injection rate exceeds 0.6 m³/h, the growth rate begins to slow down and gradually stabilizes. This indicates that the air layer under the hull reaches the saturation state, and further increases in the air-injection rate mainly affect the thickness of the air layer rather than its coverage area. To further analyze the distribution of the air layer, Figure 7 shows the morphology distribution of the air layer under the hull at a ship speed of 0.868 m/s and an air-injection rate of 8 m³/h. The distribution of the experimental air layer under the same conditions in the experiments by Zhao and Zong is shown in Figure 8. It is found that the air layer patterns observed in the numerical results generally align with the experimental results. Meanwhile, the uncovered zones behind the air-injection holes are consistent with the findings in experiments on air layer drag reduction over flat plates by Wu et al. [27]. Both Figures 7 and 8 validate the feasibility and accuracy of the numerical model, demonstrating its applicability for numerical simulations related to air layer drag reduction in marine engineering.

Table 5. Comparison of drag values between experiments and numerical simulations.

Velocity (m/s)	Total Drag (EFD)		Total Drag (CFD)	
	$Q_a = 0 \text{ m}^3/\text{h}$	$Q_a = 8 \text{ m}^3/\text{h}$	$Q_a = 0 \text{ m}^3/\text{h}$	$Q_a = 8 \text{ m}^3/\text{h}$
0.651	2.86 N	2.65 N	2.69 N	2.50 N
0.868	5.09 N	4.33 N	5.10 N	4.35 N
1.084	8.23 N	7.00 N	8.39 N	6.98 N

**Figure 6.** Results of experiments and numerical simulation.**Figure 7.** Numerical simulation of air layer morphology ($U = 0.868 \text{ m/s}$, $Q_a = 8 \text{ m}^3/\text{h}$).**Figure 8.** Experiment of air layer morphology ($U = 0.868 \text{ m/s}$, $Q_a = 8 \text{ m}^3/\text{h}$).

4.2. Drag Reduction at Different Speeds and Air Flow Rates

Figure 9 shows the variation in the drag reduction rate with air-injection rate at different Fr numbers. The curves for the frictional drag reduction rate are illustrated in Figure 9a. A general increasing trend is evident as the air-injection rate rises. The frictional drag reduction rate at $Fr = 0.156$ is consistently slightly higher than that at $Fr = 0.142$. This is because the air layer drag reduction technique functions by modifying the contact

medium on the hull surface, and at higher speeds, the air layer extends more efficiently along the ship's bottom, thereby enhancing the reduction in frictional resistance. It can also be observed that, for the cases at $Fr = 0.142$, the frictional drag reduction rate will decrease when the dimensionless air-injection flux is 0.10. To investigate the cause of the decrease in frictional drag reduction, Figure 10 presents the time-history curve of frictional resistance at a dimensionless air-injection flux of 0.10. The air layer reaches the stern after 5 s, suggesting that a stable air layer has essentially formed on the hull bottom. Compared to the case at $Fr = 0.156$, the frictional resistance at $Fr = 0.142$ increases after the air layer forms and then gradually stabilizes. This is attributed to the air layer leakage resulting from excessive injection, which reduces the drag reduction effectiveness. Figure 11 shows the air volume fraction distribution on the hull bottom for the two speed conditions with a dimensionless air-injection flux of 0.10. The images show that the overly high injection rate will cause the air layer to flow over the baffle, resulting in air leakage and thereby impacting the reduction in frictional resistance.

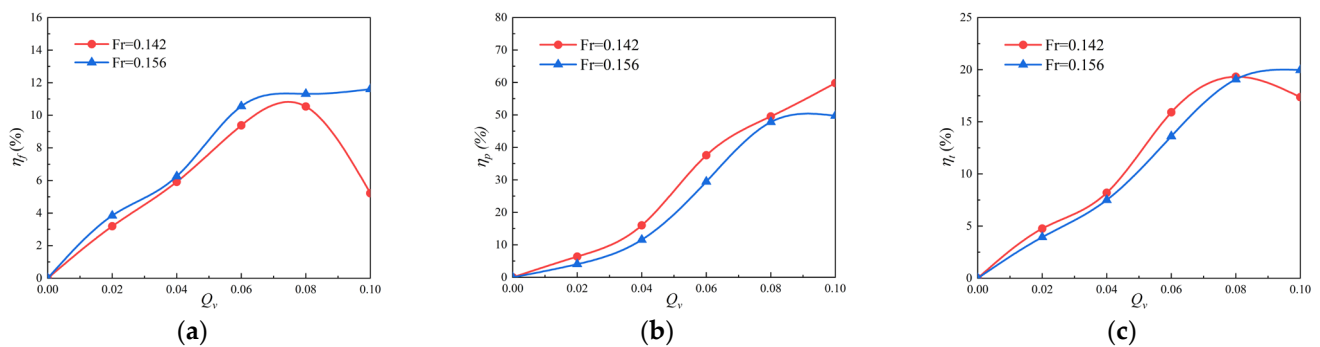


Figure 9. Variation in drag reduction rate with air-injection rate at different Fr number. (a) Variation in frictional reduction rate; (b) variation in viscous pressure reduction rate; (c) variation in total drag reduction rate.

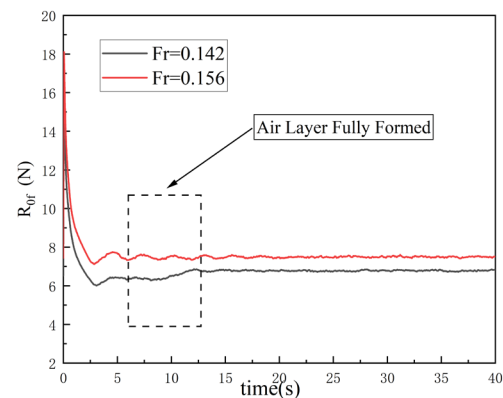


Figure 10. Variation in frictional resistance with time ($Q_v = 0.10$).

Air layer drag reduction primarily decreases the frictional resistance by reducing the wetted surface area of the hull bottom. However, ships exhibit other resistance components, such as viscous pressure resistance, which can also be influenced by the air layer. The related issues have rarely been discussed in previous studies. Figure 9b presents the pressure drag reduction rate. It should be noted that the ship model in the study is a low-speed vessel, and the wave-making resistance constitutes a minor proportion of the total resistance. Thus, the air layer's influence on pressure resistance primarily stems from viscous pressure resistance. The results indicate that the air layer will contribute to the reduction in viscous pressure resistance. The mechanisms of the reduction in viscous pressure resistance by air will be discussed in the following sections. It can also be seen that

the reduction in pressure resistance at $Fr = 0.142$ is slightly larger than that at $Fr = 0.156$. This may be attributed to the fact that viscous pressure drag arises from the gas altering the local pressure field at the stern. At lower speeds, the flow at the stern exhibits relatively less turbulence and the flow separation is also weaker, allowing gas to accumulate more efficiently at the stern, which is conducive to the reduction in viscous pressure drag.

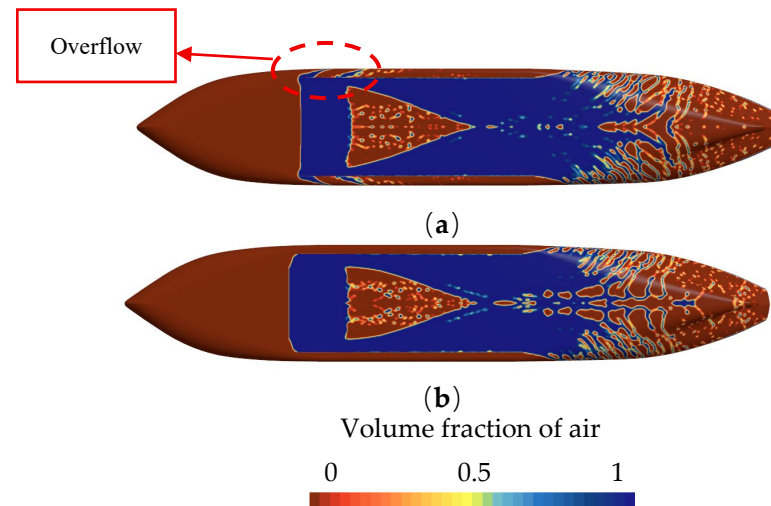


Figure 11. Air volume fraction distribution on the hull bottom. (a) $Q_v = 0.10$, $Fr = 0.142$; (b) $Q_v = 0.10$, $Fr = 0.156$.

Figure 9c illustrates the variation in total drag reduction rate at two different ship speeds. This figure reflects the combined effect of frictional drag reduction and pressure. It is observed that the total drag reduction rate at $Fr = 0.142$ is generally higher than that at $Fr = 0.156$. This result stems from the combined effect of frictional drag reduction and pressure drag reduction. This is because the extent of reduction in frictional drag differs from that in pressure drag between the two Froude numbers. Table 6 lists the frictional and pressure drag values at the two speeds for different jet-flow rates. Under the computational conditions considered in the study, the pressure drag accounts for a minor fraction of the total resistance. Nevertheless, the absolute reduction in pressure drag exceeds the reduction in frictional drag. Consequently, the ultimate decrease in total resistance at $Fr = 0.142$ exceeds that at $Fr = 0.156$. In summary, the air layer contributes to the reduction in both frictional drag and pressure drag.

Table 6. Frictional and pressure drag at two Fr numbers under different airflow rates.

Q_v	$Fr = 0.142$			$Fr = 0.156$		
	R_f (N)	R_p (N)	R_t (N)	R_f (N)	R_p (N)	R_t (N)
0	7.15	2.03	9.18	8.48	2.36	10.85
0.02	6.92	1.89	8.81	8.16	2.26	10.42
0.04	6.73	1.70	8.43	7.95	2.09	10.04
0.06	6.48	1.26	7.72	7.59	1.67	9.37
0.08	6.40	1.02	7.41	7.52	1.24	8.78
0.10	6.78	0.81	7.59	7.49	1.19	8.68

4.3. Mechanism Analysis of Air-Free Zone Formation

According to Figures 7 and 11, the air-free zone will be formed downstream of this injection surface, which disrupts the integrity of the air layer and adversely affects its frictional drag reduction performance on the ship hull. To visually capture the flow state of the air layer at the hull bottom, Figure 12 shows airflow streamlines originating from the

injection surface. Under the combined influence of the free-stream velocity and the vertical injection flow, the air layer forms a recirculation zone downstream of the air-injection surface. The air then migrates laterally under surface tension and continues to move towards the ship stern, with its velocity increasing along the ship sides.

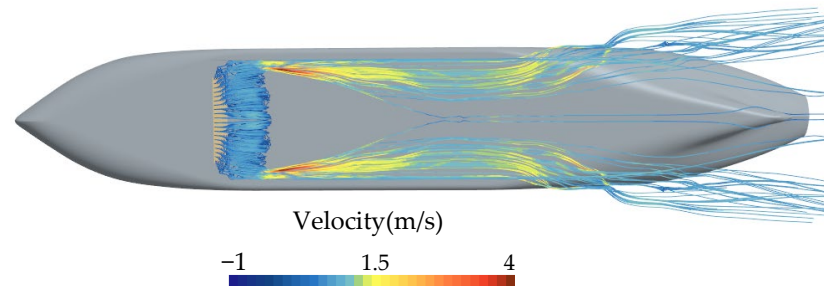


Figure 12. The airflow streamlines over the hull bottom ($Fr_{0.142}$, $Q_v = 0.08$).

To further investigate the mechanisms of air recirculation, the velocity vector plot on the longitudinal section through the air-free region is presented in Figure 13. Figure 13 demonstrates that the fluid velocity decreases toward the air-free zone, and the counter-clockwise backflow will emerge near the hull bottom. It is observed that the significant velocity deflections will occur at both ends of the air-free region, resulting in the formation of a complete recirculation structure. Moreover, as the navigation speed increases, a comparison of Figure 13a,b reveals that this velocity-recirculation zone becomes markedly longer in the streamwise direction. Figure 14 demonstrates that the flow beneath the air layer will generate negative (counterclockwise) vorticity, and the boundary at the hull surface will produce positive (clockwise) vorticity. The interaction of these opposing vortical structures drives a shear-induced recirculation. Compared to the velocity field, a comparison of the vorticity field extent at the two speeds in Figure 14 readily leads to the conclusion that the magnitude of the oncoming flow is the primary factor controlling the size of this recirculation. The combined effects of velocity recirculation and counter-rotating vortices cause the flow destabilization, and this instability drives the air–water interface against the hull surface, resulting in the air-free zone.

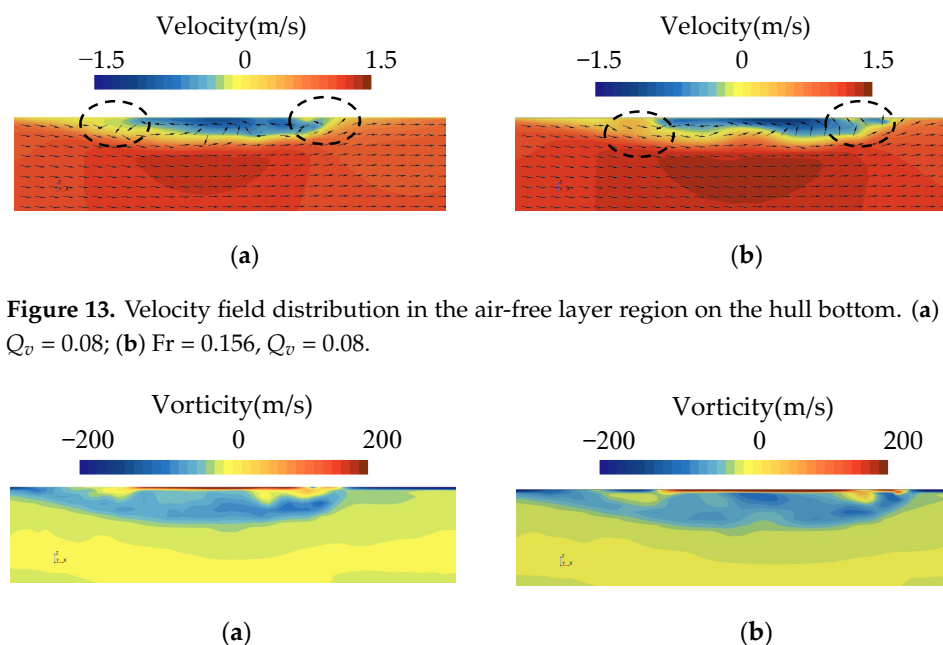


Figure 13. Velocity field distribution in the air-free layer region on the hull bottom. (a) $Fr = 0.142$, $Q_v = 0.08$; (b) $Fr = 0.156$, $Q_v = 0.08$.

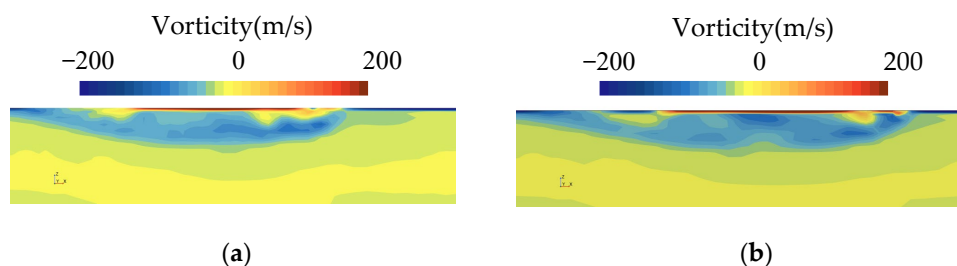


Figure 14. Vorticity field distribution in the air-free layer region on the hull bottom. (a) $Fr = 0.142$, $Q_v = 0.08$; (b) $Fr = 0.156$, $Q_v = 0.08$.

4.4. Mechanism of the Decrease in Frictional Reduction Rate for the Fr Number of 0.142

As mentioned earlier, the excessive air-injection rate will lead to a decline in frictional drag reduction. To further investigate the mechanisms of this phenomenon, Figure 15 shows the distributions of air volume fraction on the hull side at $Fr = 0.142$ with different airflow rates. Figure 15a illustrates the air volume fraction distribution at an air-injection rate of 0.08. It is observed that the air layer flows along the streamlined hull surface from the bottom, forming reticulated air streaks that propagate toward the free surface. Figure 15b shows the distribution at an air-injection rate of 0.1. Here, the thickness of the air layer exceeds the height of the deflector, leading to the lateral overflow. The escaped air then travels along the sidewall towards the free surface. To further investigate the variation in frictional resistance on the ship's side hull, Figure 16 shows the streamwise shear stress distribution on the side hull. The midship side hull of the KVLCC2 is the longitudinal flat surface. As shown in Figure 16a, the air layer only influences the wall shear stress in the stern region. In contrast, Figure 16b demonstrates that air leakage additionally increases the shear stress on the midship side hull. Comparing Figure 16a,b, it can be seen the spilled air layer attached to the side hull can still reduce drag. The mixed flow behind the spilled air layer will increase the shear stress on the side hull. This turbulent air-water mixture will have negative effects on the side hull. Therefore, ship air lubrication systems should avoid forming such mixed flows on the side hull.

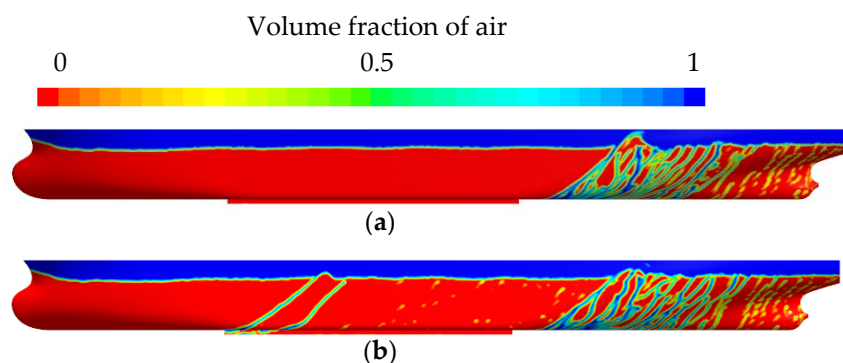


Figure 15. Distribution of air volume fraction on the hull side. (a) $Q_v = 0.08$, $Fr = 0.142$; (b) $Q_v = 0.10$, $Fr = 0.142$.

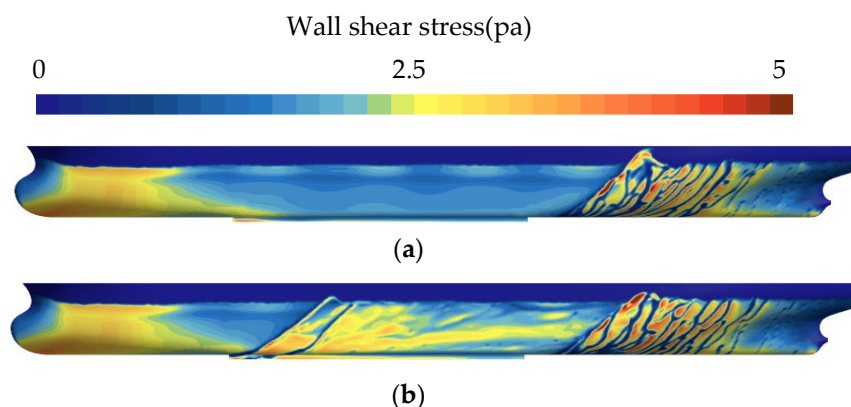


Figure 16. Distribution of wall shear stress on the hull side. (a) $Q_v = 0.08$, $Fr = 0.142$; (b) $Q_v = 0.10$, $Fr = 0.142$.

4.5. Mechanisms of the Reduction in Viscous Pressure Drag Reduction

While air lubrication primarily reduces the body's resistance through the reduction in surface frictional drag, it is revealed that the air layer will have a significant impact on the viscous pressure resistance. As previously noted by Ye et al. [28], the presence of the air

layer can affect the pressure distribution surrounding the hull. However, the mechanisms by which the air layer affects both the pressure distribution and viscous pressure resistance of the ship have not been fully elucidated. To analyze the characteristics of the flow field at the stern of the ship, an oblique section near the stern is extracted, and the plane is oriented at 45° to the primary flow direction, as shown in Figure 17. Figure 18 presents the velocity vector field and air volume fraction distribution on this plane. Figure 18a,b compare the near-wall velocity profiles for cases without and with the air layer, respectively. As shown in Figure 18a, the flow field near the stern wall is smoothly detaching from the surface. In the detailed diagram, the velocity vectors are not aligned with the wall's tangential direction and the flow separation can be observed at the stern wall, which is the cause of the ship's viscous pressure resistance. As shown in Figure 18b, due to the density difference between air and water, the air layer will adhere closely to the hull bottom surface. In the detailed diagram, the air layer remains attached to the stern wall. The air layer at the stern is subjected to an upward buoyancy force, causing the velocity vector to align almost tangentially with the wall surface. Under the combined action of buoyancy and surface tension, the air layer maintains attachment to the stern surface until it reaches the free surface. This demonstrates that the air layer at the stern effectively delays the flow separation, thereby reducing viscous pressure drag. Therefore, the presence of an air layer can suppress the formation of viscous pressure drag.

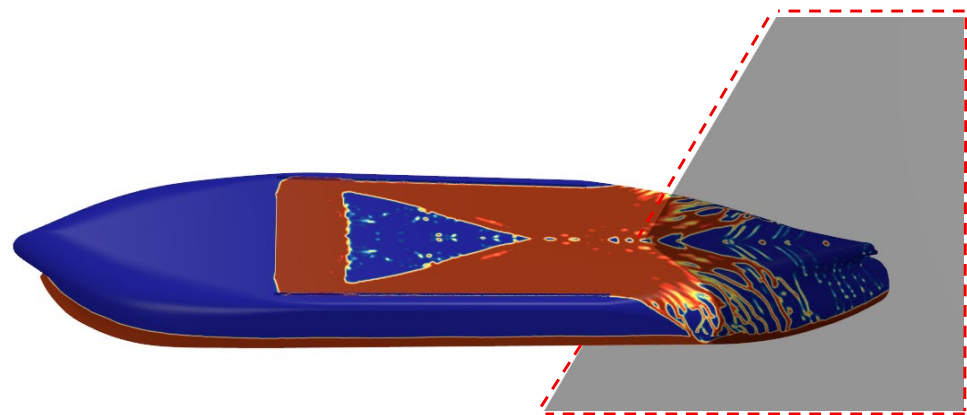


Figure 17. Oblique plane.

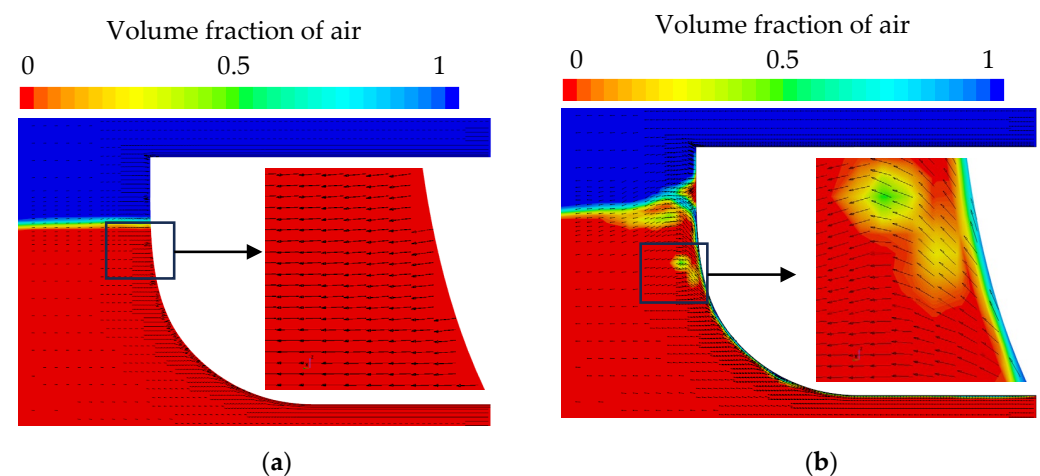


Figure 18. Velocity vector field and air volume fraction distribution on the oblique plane. (a) $Q_v = 0$, $Fr = 0.142$; (b) $Q_v = 0.08$, $Fr = 0.142$.

4.6. Effect of Trim Angle on Drag Reduction Rate

During navigation, the variations in ship attitude are inevitable. Although the ship trim is the most common and desired condition for a vessel compared to the even-keel state, it will readily disrupt the air layer, resulting in a decrease in drag reduction performance. Figure 19a shows the curves of the frictional drag reduction rate under different trim angles. The curves for all three conditions exhibit an initial increase followed by a decrease. Notably, as the angle of stern trim increases, the peak value will appear earlier. In the case of $Q_v = 0.04$, an increase in the trim angle leads to a corresponding increase in the frictional drag reduction rate.

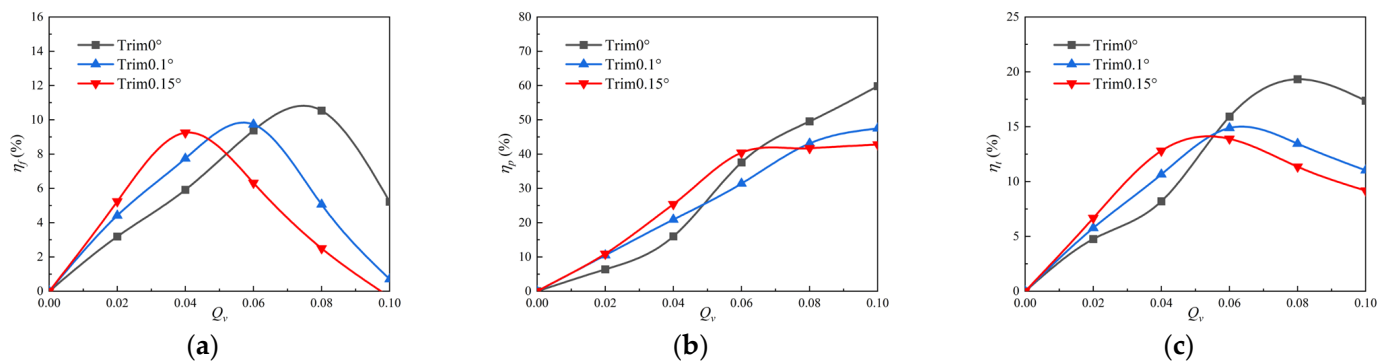


Figure 19. Variation in drag reduction rate with air-injection rate at different trims. (a) Variation in frictional reduction rate; (b) variation in viscous pressure reduction rate; (c) variation in total drag reduction rate.

In order to elucidate the mechanisms of change in frictional drag reduction, the volume fraction distributions of the air layer on the hull bottom are analyzed. Figure 20 shows the volume fraction distribution of the air layer on the hull bottom for $Fr = 0.142$ with different airflow rates. Figure 20a,c,e present the distribution of air volume fraction under the hull at the injection rate of 0.04. The increase in the stern trim angle enhances the adverse pressure gradient of the flow under the hull, and the velocity of the airflow is reduced. Under the effects of surface tension, the distribution area of the air layer will expand, thereby reducing the frictional resistance. As the stern trim angle increases, the reverse pressure gradient is further strengthened, which can promote the accumulation of the air layer downstream of the injection surface and increase the drag reduction. Figure 20b,d,f present the distribution of air volume fraction under the hull at the injection rate of 0.08. It is seen that the intensified adverse pressure gradient due to stern trim causes significant air accumulation upstream of the air-free zone, leading to the continuous increase in the thickness of the air layer above the injection surface until it exceeds the height of the baffle, ultimately inducing the side overflow of the air layer from the front edge of the baffle.

Figure 19b shows the variation curve of the pressure differential drag reduction rate. It can be seen that the influence of the trim angle on the reduction in pressure resistance is complex. At the lower air inflow rate (such as 0.04), a larger trim angle will obtain a greater reduction in pressure resistance. This is because when the airflow rate is small, most of the gas accumulates at the front of the hull, while less gas accumulates at the ship's stern. According to Figure 20a,c,e, under a larger trim angle, the gas content at the ship stern is relatively higher, which is conducive to the reduction in pressure resistance. At the larger airflow rate (such as 0.08), the reduction in pressure resistance decreases as the trim angle increases. This is because large airflow rates combined with the ship trim make it easy for gas to escape from the front of the ship, as shown in Figure 20d,f. This will result in less gas accumulation at the ship stern and affect the reduction in pressure resistance. The most obvious phenomenon is that when the trim angle is 0.15 degrees, for the large airflow rates

(larger than 0.06), the decrease in pressure resistance does not increase with the increase in airflow rates and remains almost constant.

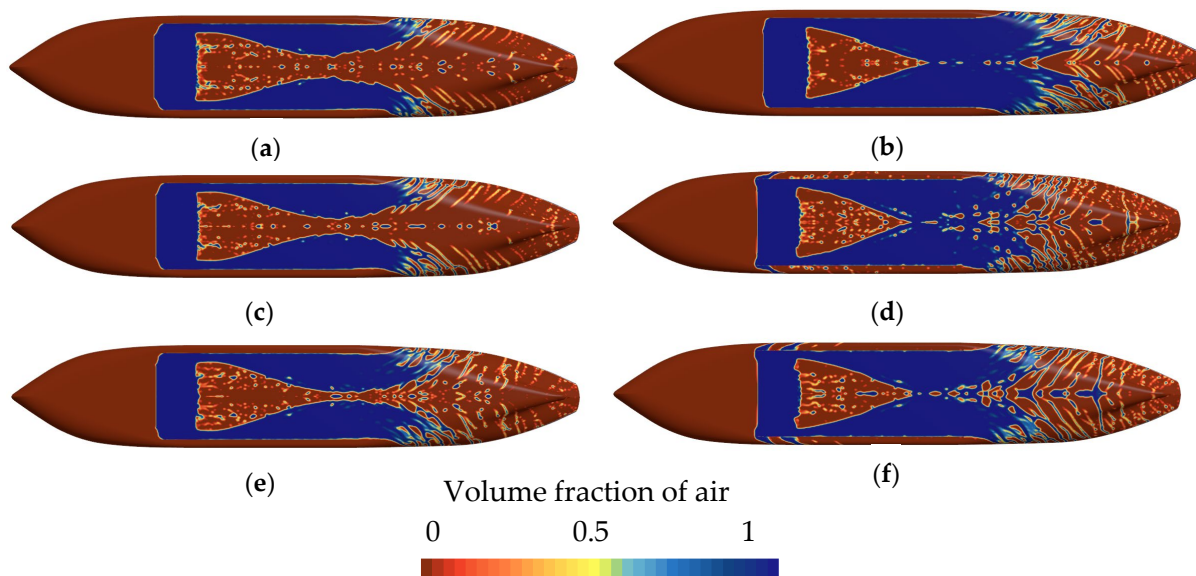


Figure 20. Volume fraction distribution of air on the hull bottom ($Fr = 0.142$). (a) $Q_v = 0.04$, $\text{trim} = 0^\circ$; (b) $Q_v = 0.08$, $\text{trim} = 0^\circ$; (c) $Q_v = 0.04$, $\text{trim} = 0.1^\circ$; (d) $Q_v = 0.08$, $\text{Trim} = 0.1^\circ$; (e) $Q_v = 0.04$, $\text{trim} = 0.15^\circ$; (f) $Q_v = 0.08$, $\text{trim} = 0.15^\circ$.

Figure 19c shows the variation in the total drag reduction rate. Stern trim provides additional reduction in both frictional and pressure resistance when it does not cause air layer spillage. However, if stern trim leads to air layer spillage, it significantly adversely affects both frictional and pressure drag reduction. These findings offer theoretical insights for the practical application of air layer drag reduction in actual ships. This observation demonstrates that the ship trim will induce the flow modifications, and this will cause air layer spillage downstream of the injection surfaces for both the air cavity and equipped baffles. The large airflow rates are prone to causing gas overflow, which affects the comprehensive drag reduction effect of the ships.

4.7. Effect of Injection Surface Configuration

Previous calculations indicated that a single jet surface would result in air-free areas. Previous studies demonstrate that the configuration of the injection surface is critical for the formation of the air layer topology [9,10]. Chen et al. [29] took a very large crude carrier as the test case to evaluate the air layer drag reduction piping system and to devise a scheme for the balanced distribution of airflow among branch lines; by adjusting the valve openings of the branches, a uniform air supply can be maintained, thereby enhancing the energy-saving performance of the air layer drag reduction system. To achieve greater gas layer coverage, this study adopts a dual-injection-surface configuration for hull-bottom air layer analysis, as shown in Figure 21. The total air-injection rate remains identical to that of the single-surface configuration, with each dual-surface injector operating at 50% of the total mass flow. Figure 22 presents the drag reduction rate curves for the two air-injection surface configurations. For the constant total air-injection rates, the dual-injection-surface configuration will improve the reduction in both frictional resistance and viscous pressure resistance.

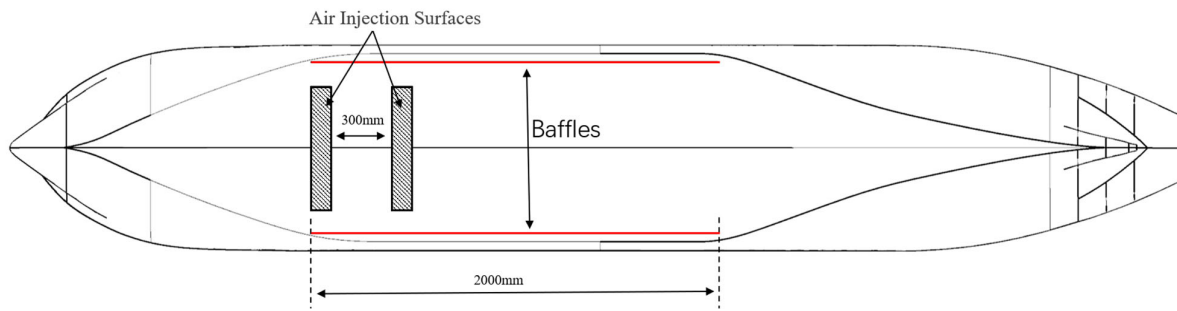


Figure 21. Schematic of dual-injection-surface configuration at hull bottom.

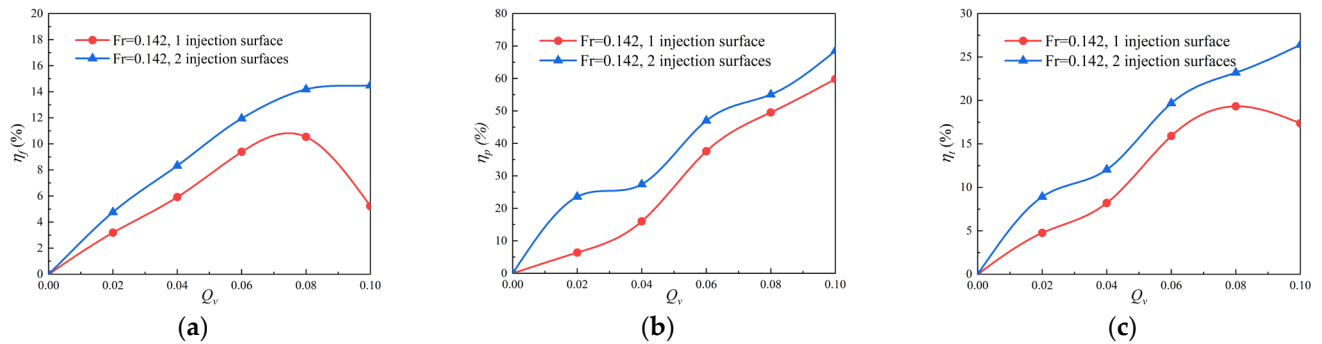


Figure 22. Variation in drag reduction rate with air-injection rate at different injection surface configuration. (a) Variation in frictional reduction rate; (b) variation in viscous pressure reduction rate; (c) variation in total drag reduction rate.

Figure 23 shows the air layer distribution on the hull bottom for the two configurations with $Q_v = 0.10$. It can be seen that the dual-injection-surface configuration significantly reduces the air-free zone, resulting in a more uniform air layer distribution across the hull bottom. Meanwhile, the dual-injection-surface configuration facilitates the development of the continuous air layer, thereby preventing the fragmentation of the gas layer, as depicted in Figure 23. This demonstrates that the appropriate injection surface configuration can effectively enhance the drag reduction performance of the gas layer with respect to both frictional drag and viscous pressure drag.

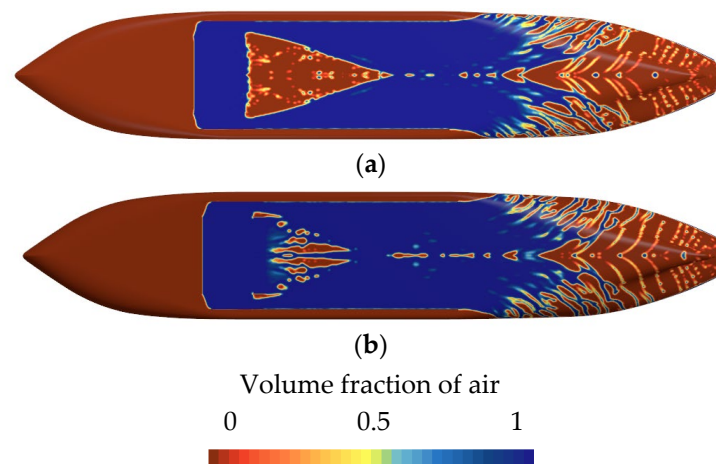


Figure 23. Air volume fraction contour map at KVLCC2 bottom. (a) One injection surface; (b) two injection surfaces.

5. Conclusions

This study employs a Eulerian multiphase flow model that incorporates surface tension and free-surface effects to simulate the air layer drag reduction. A high-resolution interface-capturing method captures air layer morphology, combined with the SST $k-\omega$ turbulence model. Experimental data validate the numerical approach. Following numerical validation against experimental data, this study investigated air layer drag reduction for the KVLCC2 hull equipped with baffles. The distribution characteristics of the air layer on the hull bottom and its drag reduction performance have been investigated. The mechanisms of the formation of air-free zones and the influence mechanisms of the air layer on both frictional resistance and pressure drag are analyzed. The effects of stern trim and air-injection configurations on air layer drag reduction are also examined. The conclusions are summarized as follows:

1. The computational model in this study comprises 1.48 million grids, employing the SST turbulence model and first-order discrete time formulation. Taking into account the free surface and surface tension, the morphology of the air layer under the hull and the variation in resistance can be accurately predicted based on the RANS equations and the VOF two-phase flow model. The simulation of 40 s of physical time required an actual computation time of 8 h, demonstrating high computational efficiency and result accuracy. The results indicate that the air layer exhibits varying degrees of drag reduction effects on resistance components such as frictional and pressure drag. In this paper, the maximum total drag reduction rate achieved for a KVLCC2 ship model equipped with baffles is up to 19.3%.
2. Subjected to the combined effects of inflow velocity and jet injection velocity, the air downstream of the injection site develops a recirculating velocity vector and two counter-rotating vortices. This flow pattern causes the gas-liquid interface to contact the hull bottom, resulting in an air-free region.
3. The ship stern trim directly influences the effects of air layer drag reduction. At low airflow rates, the ship trim can expand the gas coverage area beneath the ship, enhancing the reduction in frictional drag and pressure drag. However, at high airflow rates, the trimmed ship is prone to gas overflow from the bow, thereby diminishing the drag reduction effects on both frictional drag and pressure drag.
4. The excessive air injection or ship stern trim can increase the air layer thickness, causing it to overflow the baffles. The escaping air changes the flow field along the hull sides and increases frictional stress on the hull-side surface. Preventing overflow of the bottom air layer is essential to maintain drag reduction effectiveness.
5. The air layer at the stern can effectively reduce the viscous pressure drag. Due to buoyancy and surface tension, the air layer remains attached to the stern wall. The velocity vector is nearly parallel to the tangential direction of the wall surface. Therefore, the flow separation will be reduced, thereby reducing the viscous pressure resistance.
6. Adopting the dual-injection-surface configuration effectively reduces the air-free zones, resulting in more uniform hull-bottom air layer coverage and a significant increase in drag reduction in frictional resistance and viscous pressure resistance. The appropriate injection surface configuration can effectively improve the performance of air layer drag reduction.

This study primarily employs numerical simulations to investigate air layer drag reduction for a KVLCC2 model with a scale ratio of 58, yet several limitations remain to be addressed. Currently, there is a lack of actual ship model drag reduction data for reference. Future work could involve full-scale ship numerical simulations to mitigate the effects of scale. The formation of air layer coverage could be further optimized by adjusting

the position of air-injection holes and the air flux. The issue of lateral air leakage may be improved through hull form optimization to enhance air layer stability. Additionally, the influence of the air layer on pressure drag at the stern warrants further investigation into the mechanisms involving buoyancy and surface tension.

Author Contributions: The authors confirm their contribution to the paper as follows: Conceptualization, X.Z. and Q.Z.; methodology, Y.H.; software, X.Z.; validation, X.Z., Q.Z. and Y.H.; formal analysis, X.Z.; investigation, Y.H.; resources, X.Z.; data curation, Y.H.; writing—original draft preparation, Y.H.; writing—review and editing, X.Z.; visualization, Y.H.; supervision, X.Z.; project administration, Q.Z.; funding acquisition, Q.Z. and X.Z.; All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the NSFC Project (Grant Nos. 52401380 and 12302322), the Natural Science Foundation of Jiangsu Province (Grant No. BK20220652), the Natural Science Foundation of Liaoning Province (Grant No. 2025-BS-0199) and the China Post-doctoral Science Foundation (Grant No. 2024M753414).

Data Availability Statement: Data is contained within the article.

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

References

1. Qin, S.J.; Ji, S.; Sun, S.; Wu, R.; Wu, D.Z. Current state and prospects on applications of ship drag reduction using air lubrication. *Chin. J. Ship Res.* **2023**, *18*, 1–10.
2. Cai, J.Q. Ship Resistance Reduction by using Thin Layer of Air Film. *Shipbuild. China* **2000**, *41*, 11–15.
3. Park, S.H.; Lee, I. Optimization of drag reduction effect of air lubrication for a tanker model. *Int. J. Nav. Arch. Ocean Eng.* **2018**, *10*, 427–438. [[CrossRef](#)]
4. Mohamad, S.A.; Youssef, M.; Bennaya, M.; Elsherbiny, K. Experimental and Numerical Investigation of Air Layer Drag Reduction on a Flat Plate. *Alex. Eng. J.* **2025**, *121*, 436–451. [[CrossRef](#)]
5. Dong, W.C.; Guo, R.X.; Cheng, X.L.; Lu, Y.S. Experimental study on resistance reduction of planing craft by air injection. *Shipbuild. China* **2002**, *43*, 13–18.
6. Mäkiharju, S.A.; Lee, I.-H.R.; Filip, G.P.; Maki, K.J.; Ceccio, S.L. The topology of gas jets injected beneath a surface and subject to liquid cross-flow. *J. Fluid Mech.* **2017**, *818*, 141–183. [[CrossRef](#)]
7. Mäkiharju, S.; Ceccio, S. On multi-point gas injection to form an air layer for frictional drag reduction. *Ocean Eng.* **2018**, *147*, 206–214. [[CrossRef](#)]
8. Huang, H.B.; He, S.L.; Gao, L.J.; Shi, X.Y.; Zhang, C.H.; Xue, Q.Y. Reduction of friction drag by gas injection in a high-Reynolds-number flat-plate turbulent boundary layer. *Shipbuild. China* **2018**, *59*, 1–15.
9. Gao, L.J.; Chen, S.F.; Yun, Q.Q.; He, S.L.; Wang, L.Y.; Lu, F. Research on influence factors in air layer drag reduction technology. *Shipbuild. China* **2018**, *59*, 1–13.
10. Mao, L.; Wu, R.; Liu, H.; Ji, S. Blowing out of Ship Bottom or Ship Sides—A Study on Drag Reduction of Air Film. *J. Shanghai Ship Res.* **2020**, *43*, 1–6.
11. Kringel, C.; Simonsen, A.S.; Tziaros, F.; Bielicki, S.; Hærvig, J.; Papaioannou, G. Experimental Study of Air Lubrication with Fluidic Oscillators for Flat Plate Drag Reduction. *Ocean Eng.* **2025**, *334*, 121632. [[CrossRef](#)]
12. Li, Q.Y.; Dong, W.C. Numerical analysis of resistance reduction by bubbly layer for deep-Veeship. *J. Nav. Univ. Eng.* **2011**, *23*, 31–35.
13. An, H.; Liu, X.Q.; Zhang, H.Y.; Cui, Z.H. Numerical study on pressure drag reduction for a surface vessel with struts with the drag reduction mode of ventilated super cavitating combined with air layer. *Ocean Eng.* **2025**, *324*, 120727. [[CrossRef](#)]
14. Zheng, X.; Zhang, Z.; Tu, G.; Xiong, C.; Wang, M.; Wang, S. On the Effects of Ventilation Rate and Froude Number on Air-Layer Drag Reduction over an Axisymmetric Underwater Vehicle. *Int. J. Multiph. Flow* **2026**, *195*, 105527. [[CrossRef](#)]
15. Chen, Y.; Zou, X. Review of air layer and air cavity drag reduction on ships. *Ship Eng.* **2021**, *43*, 44–52.
16. Hao, W. Numerical study of the effect of ship attitude on the perform of ship with air injection in bottom cavity. *Ocean Eng.* **2019**, *186*, 106119. [[CrossRef](#)]
17. Wu, H.; Ou, Y.P. Numerical Study of Influence of Trim and Heel on Stability of Ship Bottom Air Layer. *Ship Eng.* **2015**, *37*, 21–25.
18. Ye, Q.; Dong, W.C. Model experimental study on resistance reduction of a 95,000 DWT bulk carrier by air lubrication. *J. Harbin Eng. Univ.* **2015**, *36*, 68–72.

19. Fang, H.Z.; Chen, Z.C.; Chen, C.; Sun, S.; Wang, X.Y.; Wu, D.Z. On the drag reduction and heeling stability of a heeled planing air cavity hull. *Ocean Eng.* **2023**, *283*, 115087. [[CrossRef](#)]
20. Zhang, X.S.; Ji, X.Q.; Chen, L.Q.; Huang, H.B.; Hong, F.W. Numerical study on the wave pattern characteristic of air layer in cavity and the effects of multiple influence factors. *J. Hydrodyn.* **2023**, *35*, 923–941. [[CrossRef](#)]
21. Dong, M.Y.; Tong, Z.; Che, L.Y.; Xing, L. Analysis of mutual interference between hull and propeller of air-layer drag reduction planning ship. *Ship Eng.* **2025**, *47*, 17–24, 82.
22. Shao, X.; Zhang, J.; Cai, H.; Chen, P.; Liu, H.; Cao, L.; Wu, D. Study on the Characteristics of an Underwater Vehicle Pump-Jet Propeller under the Influence of Drag-Reduction Air. *Ocean Eng.* **2025**, *328*, 121071. [[CrossRef](#)]
23. Ding, Z.Q.; Dai, Y.; Chen, Z.G.; Feng, Y.K.; Wu, Y.F. Model experimental research of the air injection drag reduction system without air maintenance devices for a 2600 DWT bulk carrier. *Ocean Eng.* **2025**, *323*, 120661. [[CrossRef](#)]
24. Wu, H.; Dong, W.C.; Ou, Y.P. Experimental study for the influence of groove configuration on plate air layer drag reduction. *J. Natl. Univ. Def. Technol.* **2017**, *39*, 158–163.
25. Madavan, N.K.; Deutsch, S.; Merkle, C.L. Reduction of turbulent skin friction by microbubbles. *Phys. Fluids* **1984**, *27*, 356–363. [[CrossRef](#)]
26. Zhao, X.J.; Zong, Z. Experimental and numerical studies on the air-injection drag reduction of the ship model. *Ocean Eng.* **2022**, *251*, 111032. [[CrossRef](#)]
27. Wu, H.; Dong, W.C.; Ou, Y.P. Experimental study of flow field characteristics and air distribution strategies on a flat plate with air injection. *J. Harbin Eng. Univ.* **2016**, *37*, 1220–1226.
28. Ye, Q.; Ou, Y.P.; Xiang, G.; Chen, J.J. Numerical Study on the Influence of Water Depth on Air Layer Drag Reduction. *Appl. Sci.* **2024**, *14*, 431. [[CrossRef](#)]
29. Chen, S.F.; Wang, X.; Gao, L.J.; Zhang, X.S.; Yan, Z.G.; Huang, H.B. Study on piping design and evaluation of air layer drag reduction system. *Shipbuild. China* **2025**, *66*, 198–208. [[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.