Journal of
Marine Science
and Engineering

MDPI

Review

The Modelling of the Multiphase Flow Mechanics in Air Lubrication Systems and Their Interaction with Appendages: A Review

David Hitchmough [1,*], Eddie Blanco-Davis [1,*], Andrew Spiteri [1], Mehdi Seddighi [1], Onur Yuksel [1,2], G Viknash Shagar [1] and Jin Wang [1]

[1] Liverpool Logistics, Offshore and Marine Research Institute (LOOM), School of Engineering, Liverpool John Moores University, Byrom Street, Liverpool L3 3AF, UK
[2] Marine Engineering Department, Maritime Faculty, Zonguldak Bülent Ecevit University, Kepez District, Hacı Eyüp Street, No. 1, 67300 Zonguldak, Türkiye
* Correspondence: d.m.hitchmough@ljmu.ac.uk (D.H.); e.e.blancodavis@ljmu.ac.uk (E.B.-D.)

Abstract

This review paper investigates the use of air lubrication to reduce ship hull skin frictional drag, a technology whose fundamental drag-reduction mechanisms and impact on sea-keeping are increasingly being studied through Computational Fluid Dynamics (CFD). Simulating this process is challenging, as the air phase often manifests as dispersed bubbles rather than a continuous film, necessitating high-fidelity models. Traditional simulations treating air and water as distinct phases fall short, and while Direct Numerical Simulation (DNS) captures bubble behaviour, its computational cost is prohibitive for practical application. This paper, therefore, reviews numerical simulation methods for air lubrication systems, evaluating their capabilities and limitations in capturing the system's hydrodynamics and structural interaction, in contrast to traditional towing tank testing. The evaluation reveals a critical trade-off: methods with high computational feasibility (e.g., standard LES with VOF) provide an adequate estimation of overall drag reduction but consistently fail to accurately model the detailed bubble breakup and coalescence dynamics crucial for predicting system performance across different vessel speeds and pressures. Specifically, the review establishes that current mainstream CFD approaches underestimate the pressure-induced stability effects on bubbles. The paper concludes that accurate and practical simulation requires the integration of advanced techniques, such as Population Balance Models or Lagrangian Particle Tracking, to account for these distinct, flow-dependent phenomena, thereby highlighting the path forward for validated numerical models in marine air lubrication.

Keywords: air lubrication; air layer drag reduction; bubble drag reduction; microbubble drag reduction; decarbonisation; drag reduction

check for updates

Academic Editors: Kostas Belibassakis and Md Jahir Rizvi

Received: 30 September 2025
Revised: 11 November 2025
Accepted: 19 November 2025
Published: 24 November 2025

Citation: Hitchmough, D.; Blanco-Davis, E.; Spiteri, A.; Seddighi, M.; Yuksel, O.; Shagar, G.V.; Wang, J. The Modelling of the Multiphase Flow Mechanics in Air Lubrication Systems and Their Interaction with Appendages: A Review. *J. Mar. Sci. Eng.* **2025**, *13*, 2238. https://doi.org/10.3390/jmse13122238

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 decades, extensive research has been conducted to investigate the simulation of bubbly flows. One such critical application of bubbly flows is in air lubrication systems (ALS) to reduce skin frictional drag on a ship's hull. One early recorded instance of the technology being developed and tested was at the Krylov Shipbuilding Institute in 1961 [1], using electrolysis to generate hydrogen and oxygen bubbles directly on the hull surface rather than injecting air. From its development, interest was sustained during the 1970s due to the oil crisis and the desire for methods to reduce fuel consumption in shipping. Initial research into bubble drag reduction was carried out through physical testing, utilising electrolysis to generate hydrogen bubbles rather than air. The production of hydrogen bubbles via electrolysis allows for precise control of the gas production rate by applying an electrical current. Examples of hydrogen hydrolysis are shown in the methods of [1–3] in towing tank tests. Hydrogen electrolysis allowed the operators to determine that gas lubrication can have a positive effect on skin frictional drag. The impact of towing speed was also established, with the authors indicating that the peak levels of drag reduction occurred at lower towing speeds, whereby increasing the towing speed decreased the level of drag reduction observed. The highest efficiency witnessed for the tested scenarios was 76.6% net power saved using this hydrogen-based technique. However, a variation in net power saving was evident, as the net power saved dropped from 65.7% at 2.5 knots to 33.2% at 5.0 knots, despite a constant electric power of 0.1 kW. The drag reduction mechanism was found to differ based on the flow regime in the boundary layer. In the laminar Boundary Layer, the gas acted as a simple lubricant due to its relatively low viscosity. In the Turbulent Boundary Layer, the phenomenon was more complex and in early studies, this was difficult to assess. The minute gas bubbles, which are extremely small compared to the injected gas, initially act to destroy the laminar sublayer, a region of high viscous shear stress. Subsequently, the bubbles enter the wall and outer-regions of the turbulent boundary layer, where they absorb turbulent momentum and alter the characteristics of the drag-producing vortices. The sustained interest in this technology during the 1970s was a direct response to the global energy crisis of the time. However, as the 1980s oil glut began, resulting in a six-year decline in crude oil prices, interest in developing these high-cost, fuel-efficient technologies waned as the immediate, pressing economic need to reduce fuel consumption subsided. The impact of the oil crisis on shipping is shown in the discussion by [4], which provides an operational analysis of changes in demand during this crisis.

Air injection introduces air into the vessel's boundary layer, typically via a compressor. It modifies the boundary layer's density, wetted surface area, viscosity, turbulence viscosity, and wall shear, reducing drag. Ref. [5] divided ALS into stratified and dispersed regimes. A stratified regime is characterised by a continuous air layer and is demonstrated by air layer drag reduction (ALDR). Alternatively, a non-continuous dispersed air regime is referred to as bubble drag reduction (BDR). The stratified regime in ALDR often exhibits a higher drag reduction rate and requires a higher rate of air injection to form and maintain the air layer in the stratified state. A further distinction must be made between bubble drag reduction and microbubble drag reduction. However, the exact definition of what constitutes a microbubble varies between sources [6]. However, there is a general indication that this relates to bubbles of the order of microns up to millimetres in diameter [7–9]. A representation of the air lubrication regimes can be found in [5].

In the early flat plate tests conducted by Madavan and Deutsch [10], the experiments were performed in a water tunnel, yielding drag reduction values of 80%. A diagram of the apparatus used is shown in [10]. The study found that the duration of the frictional drag reduction effect following microbubble injection depends on the plate's gravitational orientation and the freestream velocity. Several recent studies have explored bubble drag reduction (BDR) as a means to mitigate the impact of skin friction on a ship's hull. One such study [11], examined this phenomenon at Reynolds (Re) numbers as high as 210×10^6 showing the effect of bubble drag reduction. The work produced a near-zero pressure gradient turbulent boundary layer (TBL) on a flat plate and recorded drag reduction levels, which agreed with those reported in [8].

Ref. [12] considered Reynolds numbers up to 220×10^6 and referred to how computational and resource restrictions have limited many previous studies of ALDR and BDR to low Reynolds numbers and small scales. The researchers performed a series of experiments using a variable-pressure closed-loop water tunnel, as described in the setup of [13]. All three tests were conducted on the same model, injecting air into a turbulent boundary layer with a near-zero pressure gradient under a flat plate in hydrodynamically smooth and fully rough conditions.

Ref. [14] studied the effect of BDR on a flat plate in a towing tank and a water tunnel at varying speeds (2.45 m/s to 4.2 m/s), in which the bubble introduction flow rates ranged from 12 L/min to 75 L/min. The maximum drag reduction in the water tunnel and towing tank tests was 80% and 22%, respectively.

Operator knowledge about the nature and mechanisms of ALDR and BDR is still incomplete. Comparative studies on critical parameters can sometimes demonstrate discrepancies, leading to uncertainty regarding the significance of the parameters' influence. The mechanism of ALS is yet to be fully defined. Consequently, our understanding of the effects of parameters such as the distribution of bubbles and the persistence of the effect downstream of injection is still developing [15].

Furthermore, it is not merely a case of confirming the ALS drag reduction mechanism. Not all authors are convinced of the benefits of air lubrication in reducing drag and fuel consumption. Ref. [16] pointed to the fact that prior studies on the technology have indicated that air lubrication systems positively impact operation parameters for ships with an air lubrication system in place. On the contrary, this paper analysed the operational effectiveness of a modern passenger ship and stated that its analysis had not shown a positive impact. The authors of this study considered the air lubrication system on the M.T. Amalienborg, which showed that using an ALS system on the boat could result in a 4.5% benefit at 14 knots. However, the study raised doubts about the system's effectiveness at slower speeds, as the savings dropped to 3.2% at 13 knots and 1.7% at 12 knots.

When using an ALS, the operator must consider the power requirement to operate the compressors needed in many cases. Using compressors in this manner results in a substantial power demand. The use of compressors means the net drag reduction effect is reduced and, in some cases, entirely offset due to this increase in power requirement, especially when the ship is slow-steaming [17]. For instance, ref. [18] achieved a drag reduction of 10–12%, but the energy needed to power the compressor represented a 5–10% additional power requirement, resulting in a total net energy reduction of only 0–7%.

Additionally, ref. [19] found no scaling law to predict the net drag reduction in full-size ships, making it difficult to predict the net drag reduction in the model scale. The inability to ascertain the operational effectiveness of air lubrication in the model scale poses a significant hurdle to those investigating air lubrication.

Considering the benefits of ALS is crucial, especially when simulating the technology using CFD. With the advancement of technology and its widespread use, it is essential to have competent CFD simulations of air lubrication mechanisms. A deeper understanding of the optimal stimulation of air lubrication will enhance our knowledge of the system and stimulate further research. Additionally, it will encourage further discussion of the technology's financial and environmental benefits or impact, which is highly desirable considering the technology's increasing popularity and usage due to its implications with the Energy Efficiency eXisting ship Index (EEXI) and Carbon Intensity Indicator (CII).

2. Novelty Statement and Contribution of the Review

Air lubrication has been investigated as a potential decarbonisation technology for several decades. Despite ongoing research, there is a lack of sufficiently holistic reviews of the technology's current state of the art. This review paper addresses part of that gap by presenting a holistic view of physical testing and computer simulation related to air lubrication, alongside a review of the parametric considerations involved and some of the complex phenomena identified in existing work. Air lubrication, as a decarbonisation technology, has the potential to make a significant contribution to global maritime decarbonisation targets. However, for further research on air lubrication to be carried out, quality review papers on the topic are needed that consider a broad range of factors relating to the technology in context. With these elements in mind, this review paper will make a significant contribution to the maritime sector by facilitating future work and research. It will investigate the current state of the art in simulating air lubrication, identify existing limitations in the methods, and assess the applicability of other novel approaches.

There are some existing reviews which are related to the topic of this review, such as [20], which considers CFD-based studies of ship energy-saving technologies in a general sense, and briefly mentions air lubrication. There are also some reviews which consider air lubrication more explicitly, such as the work of [21], which focuses more on the nature of air lubrication and the mechanisms by which it induces drag reduction, as well as classification of the air layer regimes. The work of [7] is particularly notable, as it summarises the current state of research progress in the area of drag reduction using air lubrication on ships. Ultimately, although reviews on air lubrication currently exist, there is a lack of reviews specifically regarding the modelling of air lubrication using computational methods and comparisons with physical testing. In the recent review paper by [22] the researchers considered the ability of CFD to replicate the results of physical testing. The work of [22] does not directly compare the predictions of experimental and numerical approaches. Still, it does indicate that the phenomena, behaviours, and findings of numerical approaches corroborate those of experimental approaches.

3. Laboratory Testing

There is a greater abundance of physical tests for air lubrication; this could be attributed to many factors, such as uncertainty with scaling of the results, limited experience with simulating the mechanism in computational packages, and uncertainty with modelling the effect in the first instance due to the complexity of the multiphase flow, as well as the free surface aspect. This section will give an account of the considerations in physical testing. In recent years, the use of CFD and the level of complexity which these methods can handle have increased. As such, a greater body of work now exists using these computational methods; however, it remains the case that much of the foundational work was conducted through physical testing. There remains some reliance on physical testing data for validation of the computational approaches proposed.

An empirical formula for BDR on a flat plate test is proposed by [23], utilising $\frac{C_f}{C_{f_0}}$, which is the ratio of skin friction with bubbles to that without. $\overline{\alpha}_a$ is an average volume ratio, which is defined in [24], where Q_a is the volumetric flow rate of air in the test section, and Q_w is the volumetric flow rate of water in the whole test section. The flat plate in [24] measured 100 mm $\times$ 15 mm.

$$\frac{c_f}{c_{f_0}} = 0.8^{-4\overline{\alpha}_a} + 0.2 \tag{1}$$

$$\overline{\alpha}_a = \frac{Q_a}{Q_a + Q_w} \tag{2}$$

The equation allows for plotting $\frac{C_f}{C_{f_0}}$ against $\overline{\alpha}_a$ and serves as a comparative tool, representing a critical analysis approach for physical testing.

Many studies utilise water tunnels, refs. [24,25] used a closed circulating water tunnel designed for microbubble experiments. The water tunnel used in these studies was a high-speed circulating water tunnel, with inner dimensions of 100 mm in width, 15 mm in height, and 3000 mm in length. The use of an 'array-of-holes' plate caused the air introduced to be dispersed in the desired microbubble manner. Air was introduced via four injection points, with the first located 1038 mm from the start of the test section, the second point spaced 500 mm downstream from the first, the third 500 mm downstream from the second, and the fourth 500 mm downstream of the third. The work of [24] achieved a maximum of 30% drag reduction, notably lower than the 80% predicted by [23]. This could be attributed to the fact that the experiment in [23]. The obtained data were collected 50–65 mm downstream of the injection location. In contrast, the study in question gathered data from 500 mm to 1500 mm further downstream, which would impact the distribution and topography of the bubbles.

The work of [11] used a water tunnel test as the method of investigation and looked at this phenomenon at high Re numbers (210×10^6). The study produced a near-zero pressure gradient turbulent boundary layer on a flat plate. The experiments of [12] were also carried out in the same test model as [11], which is the USA Navy's Large Cavitation Channel (LCC) [13] which incorporates a 6:1 contraction for the flowing water, and a schematic drawing of the LCC can be seen in [11]. Both [11,12] have continued to be referenced to the present day in the design of air lubrication experiments and as a reference for the validity of CFD results.

Ref. [14] studied the effect of BDR on a flat plate in both a towing tank test and in a water tunnel at varying speeds (2.45 m/s to 4.2 m/s) and bubble flow rates (12 L/min to 75 L/min). A maximum drag reduction of 80% was achieved in the water tunnel. A maximum drag reduction of 22% was observed in the towing tank, with drag reduction increasing monotonically as the airflow in the water tunnel increased. The authors noted that an optimal airflow existed for each velocity investigated in the towing tank test. Furthermore, drag reduction was observed solely as a function of air volume fraction in the water tunnel test (i.e., independent of velocity). In contrast, drag reduction in the towing tank test was dependent on velocity.

Parametric Considerations

One crucial aspect of utilising ALS is how speed and air injection rate impact the amount of drag reduction. The work of [15], is an experimental study using an air injector unit with 225 holes of either 1 mm or 2 mm in diameter, indicated that for a particular speed of operation and air injection rate, the percentage savings in energy depend on the percentage reduction in the power required to run a ship and the percentage of power needed to inject the air. The study established that for a 1:23 scaled model of an 8000-ton deadweight bulk carrier, the power required to inject air bubbles is proportional to the air injection rate, which in turn depends on the static and dynamic pressures at the air injection point. These pressures, in turn, rely on the ship's draft and speed. Total resistance was measured in the towing tank, and an airflow metre was used to measure and control the air injection rate. The study observed that maximum savings in energy were achieved at lower air injection rates, specifically from 0.5 to 1.5 CFM. Beyond these rates, the power needed to compress the air was found to be greater than the power saved from drag reduction, especially at slower speeds. For the Froude number of 0.09, the study found that 2 mm diameter injection holes resulted in higher energy reduction compared to 1 mm holes. However, at a Froude number of 0.15, 1 mm holes generally resulted in higher energy reduction.

Ref. [26] Conducted a combined full-scale experimental and numerical study, which sought to investigate the effect of air lubrication on frictional drag. The full-scale experiment was performed on the cement carrier Pacific Seagull, measuring resistance reduction in both ballast (11%) and full-load (6%) conditions. The study then utilised an integral boundary layer computation method to model to compute local skin friction. A key assumption of the model was to apply a constant reduction factor (k = 0.5) to the local skin friction in areas affected by bubbles.

$$\frac{\tau}{\rho U^2} = 0.123 \times 10^{-0.678H} \left(\frac{U\theta}{\nu} \right)^{-0.268} \tag{3}$$

$$\frac{\tau}{\rho U^2} = \alpha(H)(R_\theta)^{n_1} \tag{4}$$

The variables in these equations, which are based on the Ludwieg-Tillmann and Webster-Huang integral boundary layer methods, are as follows: τ is the local skin friction (shear stress), ρ is the fluid density, and U is the fluid velocity. The remaining variables are related to the boundary layer profile: H is the shape factor, θ is the momentum thickness, and ν is the kinematic viscosity. In the Webster-Huang equation, R_θ is the Reynolds number based on the momentum thickness, and α is a coefficient dependent on H, while n_1 is an exponent.

The results of the work agreed with and explained the numerical findings in [26], and demonstrated that boundary layer development is different in the presence and absence of bubbles. This difference was indicated by the observable difference in axial velocity distribution witnessed at the propeller plane.

Many authors agree that increasing the void fraction of air decreases drag. However, [27] stated that increasing the void fraction of air only decreases drag if the fraction of air increases near the wall. The authors performed a direct numerical simulation, concluding that high concentrations of bubbles in the buffer layer (roughly 20 wall units away from the surface) are necessary for drag reduction. The authors state that this is because bubbles actively modulate the flow structure by preventing the formation of large, sheet-like spanwise vortices, which disrupt the turbulence. Refs. [28,29] corroborated the concepts put forward by [27]'s work and suggests that there exists a point at which, once exceeded, the addition of more air conversely begins reducing the effect of drag reduction. Beyond a critical point, further addition of microbubbles can lead to a 'pilling up' effect, causing coalescence into large cavities. The large cavities can then disrupt the turbulent layer, diminishing the drag-reducing effect. This relationship is shown in the diagrams included in [29]. The work of [29] was experimental and conducted on a 70 cm catamaran model. The authors found that the most suitable injection flow rate, with a coefficient between 0.4 and 0.6, resulted in a total drag reduction of 5–8%. This relationship is shown in the diagrams included in [29].

According to [30], the ratio of fluid density to the density of air, also known as the density ratio, plays a significant role in reducing drag by affecting fluid flow. The effectiveness of drag reduction is enhanced with a higher density ratio, corresponding to a lower-density injected gas, as demonstrated in the CFD analysis. Ref. [30] found that helium was the most effective gas, followed by air, argon, and carbon dioxide, in descending order of effectiveness, although the differences between them were minimal. The skin frictional drag reduction effect related to the density ratio can be seen in the diagrams included in [30], and is tied to the non-dimensional flow coefficient Q/UA. Q is the flow rate of the microbubbles, and U and A are the water's velocity and the flat plate's area, respectively. A representation of the relation between drag reduction and air density is shown in [30].

Another impacting factor is that when injecting a known bubble size, bubble dynamics have such a large number of variables that once the bubble is injected, its size will change immediately [31,32]. Capturing the bubbles is also challenging in experimental work and computationally expensive.

Another impacting factor is that when injecting a known bubble size, bubble dynamics dictate that its size will change immediately as it interacts with the turbulent flow. This phenomenon is driven by bubble coalescence and breakup mechanisms, especially at high injection rates where bubbles are closely packed together. The dynamics are so complex that, assuming a constant bubble size—a common simplification in early models—can introduce significant errors in predictions. Capturing these bubble dynamics is also difficult in experimental work, requiring expensive, large-scale facilities like the 400-m-long water tank used in [32], which employed a 36-m-long model ship and an array of specialised sensors and cameras. This complexity also translates to numerical methods, where advanced models like the Multiple-size Group (MUSIG) model are needed to resolve the distribution of bubble sizes mechanistically, and even these models may require extensive calibration to match experimental observations [31]. Common air lubrication variables and dimensionless parameters are presented in Tables 1 and 2.

Table 1. Table of Air Lubrication Variables.

Controlled Variables	
U	Liquid Flow Velocity
Q	Air Injection Rate
$\mathrm{D_i}$	Distribution of Injectors
$\mathrm{I_x}$	Injection Location
$\mathrm{I_A}$	Injection Area
$\mathrm{I_{AN}}$	Injection Angle
$\mathrm{I_M}$	Injection Mode
d	Bubble Diameter
Boundary Layer Characteristics	
δ	Boundary Layer Thickness
TKE	Turbulence Kinetic Energy
μ_t	Turbulent Viscosity
$\varnothing$	Air Layer Thickness
Environmental Parameters	
P	Hydrostatic Pressure
g	Gravity
μ	Liquid Viscosity
ρ	Density Ratio
‰	Salinity
Ra	Surface Roughness of Plate

Table 2. Table of Dimensionless Parameters.

Dimensionless Parameters	
Re	Reynolds Number
Fr	Froude Number
We	Weber Number
Ca	Capillary Number
Ma	Mach Number
$\propto$	Void Fraction

4. The Capability of CFD to Simulate Air Lubrication Behaviour

Despite the added complexities of simulating air lubrication, it is essential to recognise the advancements in CFD in simulating free surface cases. The predictive ability of CFD when considering single-phase flow around a ship has improved dramatically over the last few decades, and this predictive ability serves as a benchmark for studying multiphase flow in air lubrication around a ship.

CFD is a numerical technique widely used to investigate ship design and operational scenarios, and has been increasingly utilised in the simulation of seakeeping and operational parameters. Verifying if CFD has successfully replicated the physical conditions of the test to the degree deemed satisfactory, with acceptable and quantifiable error, is challenging. The commercial software SHIPFLOW (https://www.flowtech.se/), for example, has been validated for full-scale delivered power simulations with an average comparison error of just 1% [33]. However, this benchmark does not fully account for the complexities of air lubrication. AL is a complex mechanism, as the nature of air-water interaction, whether in the medium of bubbles or as a continuous layer of air, gives rise to phenomena such as Wave Pinch-Off and Re-Entrant Jet. In physical testing, these phenomena are inherently present. In computational techniques, due to the limitations of numerical modelling, there is a risk of incorrectly replicating the nature of air-water interaction or omitting it entirely. This is a key challenge for RANS (Reynolds-Averaged Navier-Stokes) methods, which are widely used for ship hydrodynamics. While RANS is computationally efficient, it struggles to capture the dynamic nature of multi-phase flows accurately. For instance, early two-fluid models were often considered inappropriate because they treated the presence of bubbles only as changes in local time-averaged viscosity and density, without considering the exact shapes and sizes of the microbubbles. RANS solvers typically utilise the Finite Volume Method (FVM) to discretise the governing equations (Navier-Stokes and turbulence model equations) in space. The resulting algebraic equations are solved iteratively using pressure-velocity coupling algorithms, such as SIMPLE or PISO, to enforce mass and momentum conservation across the computational domain. Moreover, The VOF method uses a phase fraction equation, an advection equation, that is solved alongside the RANS equations. The value of the phase fraction is tracked, and specialised interface reconstruction schemes (e.g., PLIC) are used to maintain a sharp air-water boundary and prevent numerical diffusion (smearing).

More modern Eulerian-Eulerian RANS approaches, such as the one used by [31], attempt to overcome this by incorporating a population balance model like the Multiple-size Group (MUSIG) model. This approach resolves a wide range of bubble sizes. It mechanistically accounts for bubble breakup and coalescence, which significantly enhances the accuracy of drag reduction predictions at higher air injection rates.

In addition to bubble dynamics, the accuracy of CFD hinges on properly modelling the boundary layer. The k-omega Shear Stress Transport (SST) turbulence model is often preferred because it solves the turbulence scalars up to the wall boundary, thereby eliminating errors associated with empirical wall functions that fail to capture the near-wall flow accurately. Furthermore, the effect of hull roughness is a critical factor for accurate full-scale predictions that RANS models, like the one used by [33], are being developed to address without relying on wall functions. This is a significant challenge, as the measured average hull roughness (AHR) is a single number that does not fully represent the complex topology of the surface, making it difficult to correlate with the equivalent sand grain roughness (kS) used in CFD models.

Regarding the simulation of air lubrication systems (ALS) on ships, the field is evolving beyond simple finite volume Navier-Stokes solvers to more sophisticated numerical methods that can capture complex two-phase phenomena. To address the challenge of accurately modelling bubble formation, ref. [34] developed a 3D two-fluid model that includes a novel sub-grid air entrainment model to determine bubble source locations from air ingestion automatically. This model operates on the principle that the non-linear relationship between a bubble's terminal velocity and its diameter creates a region of high void fraction around the air entrainment zone. This approach represents a significant advancement by eliminating the need for a human expert to manually define injection points, thereby reducing potential sources of error and uncertainty. The model's predictive capability was validated against the experimental data of [35], showing a good correlation between the simulated and measured vertical locations of the 1% void fraction line. This approach automatically determines bubble source locations by identifying regions where the downward liquid velocity satisfies two conditions: First, that the liquid velocity component in the direction of gravity is greater than the critical entrainment velocity.

$$v_c(x_s) \cdot \frac{g}{|g|} > v_{ent} \approx 0.22\,\text{m/s} \tag{5}$$

Secondly, the liquid velocity component normal to the free surface is also greater than the critical entrainment velocity.

$$v_c(x_s) \cdot n > v_{ent} \approx 0.22\,\text{m/s} \tag{6}$$

Other research has focused on enhancing the modelling of bubble dynamics once they are in the flow. Ref. [36] utilised a 3D CFD-Population Balance Model (PBM) to simulate bubbly drag reduction (BDR) on a flat plate. The PBM approach solves the Population Balance Equation (PBE), which tracks the number density of bubbles across multiple size bins. This PBE is coupled to the RANS equations, and the PBE itself is solved using a discrete method to resolve the rates of bubble breakup and coalescence mechanistically at each time step. This approach explicitly models the size distribution of bubbles and accounts for bubble breakup and coalescence, which often change bubble size in high-shear, near-wall regions. A key strength of their model was its full inclusion of both drag and lift forces based on applicable closure models, a notable improvement over prior studies that had often neglected these forces. Their work achieved a 30% drag reduction, which aligned with experimental results, confirming the model's ability to capture physical conditions accurately. Similarly, ref. [37] extensively validated a CFD-PBM model by comparing its computational results with experimental work across various flow speeds and air injection rates, achieving good concordance in both predicted drag reductions and bubble distributions. The PBM approach used discrete size bins, and the conservation equation for the fractional volume of these bins is given by

$$\frac{\partial(\alpha_g f_i)}{\partial t} + \nabla \cdot (\alpha_g U_g f_i) = S_i \tag{7}$$

The breakup frequency model used is the Luo and Svendsen model [38] and is given by

$$\Gamma_i = \frac{1}{2}\int_0^{v_i} \gamma_{br}(v_j : v_i)\,dv \tag{8}$$

The first equation is the conservation equation for the fractional volume of the bubble size bins, where α_g represents the gas void fraction, and f_i is the volume fraction of the i-th bubble size class. U_g is the gas phase velocity. S_i is the source term representing the net rate of bubbles entering or leaving the i-th size class due to processes like breakup and coalescence.

The second equation, the Luo and Svendsen breakup frequency model, represents the breakup frequency. Γ_i is the breakup frequency for bubbles in the i-th size class.

v_i is the volume of the i-th size class, and $\gamma_{br}(v_j : v_i)$ is the breakup rate of a bubble of volume v_j into two bubbles of volume v_i.

These studies collectively highlight a shift towards more comprehensive modelling strategies. While traditional RANS models laid the groundwork, their inability to fully resolve phenomena such as bubble dynamics and entrainment has prompted the development of more advanced, physics-based approaches, including PBM and sub-grid models. The continued validation of these models against experimental data is crucial for building confidence in their predictive power for complex, full-scale applications.

Ref. [39] endeavoured to better simulate the nature of bubbly flows by considering a two-fluid model of a turbulent adiabatic bubbly flow implemented in the CFD program CFX4.2, validated against experimental data and the predictions of a model by [40]. Their approach utilised a two-fluid model so that the conservation equations for mass and momentum are handled for each phase. Turbulence in the dispersed phase was neglected, and the liquid turbulence was modelled as an extension of the standard k–ε model. A closure for the terms of interfacial turbulence was proposed based on the assumption that the bubbles would maintain low inertia, while ignoring surface tension, which the author states applies to most bubbly flows. To further enhance accuracy, the same authors developed a two-phase wall law that served as a computationally inexpensive boundary condition [41]. This law modifies the conventional logarithmic law of the wall by introducing a correction based on an additional turbulent viscosity from bubble mixing. Comparisons were made between the numerical predictions and experimental data. The data compared to that of [40], used a two-fluid model implemented into the PHOENICS code to improve the prediction of phase distribution for two-phase bubbly flows in ducts. Critically [39] achieved comparable results to that of [40] without using a computationally expensive algebraic stress model, particularly in predicting bubble-induced turbulence suppression. The model demonstrated a good level of ability in predicting all shear stresses, with better agreement between the model and experimental data than the conventional single-phase wall laws, especially in cases of low Reynolds number flows containing small bubbles. A significant limitation of the model developed in [39] is its inability to consider anisotropy, a prominent characteristic in gravity-dominated bubbly flows where chaotic bubble motion enhances specific velocity fluctuation components. It is for this reason that the model of [39] generally underpredicts the turbulence scale, with the authors concluding that a more advanced Reynolds stress closure would be necessary to achieve a broader range of predictive capabilities.

The prediction of bubbly flow around a ship hull is critical for understanding its hydrodynamic performance as well as its acoustic and optical signatures. A significant challenge in this area is accurately modelling air entrainment. Ref. [42] considered the prediction of bubbly flow around a surface ship's hull to determine the hydrodynamic performance and its acoustic and optical signatures. The work of [42] considers this using a phenomenological sub-grid air entrainment model coupled with a two-fluid RANS model for bubbly flow. The model predicts the location and rate of air entrainment by assuming that turbulence creates a rough air/liquid interface with cavities that are entrained once the downward liquid velocity exceeds the downward velocity of the interface. This approach was validated against at-sea measurements on the research vessel Athena in both straight-ahead and turning motions. The model successfully predicted entrainment at the masker, the hull contact line, and the transom, demonstrating good agreement with experimental void fraction profiles for three different ship speeds. The authors highlight that this model provides a first-of-its-kind quantitative numerical prediction of void fraction distributions around a full-scale ship. A separate 3D numerical investigation by [43] focused on frictional drag reduction on a KRISO container ship model using microbubbles. Their approach utilised a standard RANS solver with a k–ε turbulence model and a Mixture Model for multiphase flow, aiming to understand resistance reduction across different Froude numbers, injection rates, and void fractions. The study successfully achieved a maximum drag reduction of 27.6% at a Froude number of 0.282 with an air volume fraction of 4.8%. This research provided several key findings. Firstly, regarding the optimal injection rate, the model demonstrated a strong relationship between Froude number and the optimal air injection rate. At higher Froude numbers, a higher injection rate was required to counteract the bubbles being pushed out of the boundary layer, whereas at lower Froude numbers, decreasing the injection rate was more effective for achieving drag reduction. Further elucidated by the study, it indicates that the drag reduction effect decreases with increasing distance from the injection point, reconfirming findings from existing experimental studies. Ref. [43] concluded that excessive air injection at higher rates can increase local frictional drag by causing bubbles to escape from the hull sides, increasing turbulence. Both the work of [42,43] showcase different applications of RANS-based solvers in the simulation of air lubrication. The work of [42] stands out because of how its sub-grid model to predict air entrainment is physically grounded, as this is often addressed empirically. Ref. [42]'s validation against full-scale data from experiments and void fraction profiles is a particular strength and supports its predictions. However, the model is limited to a monodispersed bubble assumption and is coupled using a simplified one-way assumption, where the bubbles do not affect the flow themselves, which is a potential weakness. It is essential to look at these two studies in tandem because, ref. [43] focuses on demonstrating the effects of varying flow parameters on drag reduction, using a more conventional approach. Ref. [43]'s finding of an optimal injection rate and downstream decay are consistent with existing literature, whilst using a more simplified mixture model and k-ε turbulence model with enhanced wall functions. In both cases, the studies highlight the ongoing challenge of accurately representing complex, unsteady multiphase flow phenomena using RANS models, which often necessitate simplifications to maintain computational feasibility.

In the pursuit of a better understanding of the fundamental mechanisms of drag reduction, some researchers have employed high-fidelity numerical methods. Unlike RANS approaches, high-fidelity approaches such as Direct Numerical Simulation and Large Eddy Simulation solve the momentum equations in a time-accurate manner (unsteady state). This requires high-order spatial and temporal schemes (e.g., second-order or higher time-marching schemes) to capture the transient turbulent flow structures and bubble dynamics accurately. The authors of [44] utilised Direct Numerical Simulation (DNS), which solves the unsteady Navier-Stokes equations without a turbulence model, capturing the full spectrum of turbulent scales. This Eulerian-Lagrangian approach solves the fluid continuity and momentum equations within a Eulerian framework, while tracking the trajectory of individual bubbles using a separate acceleration equation. The study considered bubbles as rigid, massless spheres, a valid assumption for small bubbles in a non-purified fluid where the interface is less deformable. The key finding from this high-resolution DNS study was detailed physical observations linked to drag reduction. Ref. [44] presents the idea of a positive velocity divergence resulting from the presence of bubbles ($\nabla U > 0$), which creates a mean velocity away from the wall. Furthermore, the authors identify the presence of vortical structure displacement, which displaces the quasi-streamwise longitudinal vortical structures away from the wall. Ultimately in [44] the observation of reduced skin friction is critical and indicates that the displacements witnessed lessen the intensity of wall streaks and moves the peak production of Reynolds stress farther from the wall to a region of smaller mean shear. The overall effect witnessed in [44] is a reduction in skin friction, with the highest drag reduction of 20.2% achieved at a bubble volume fraction of 0.02.

The use of DNS stands in stark contrast to lower-fidelity methods, such as RANS turbulence models. While RANS relies on turbulence models to represent the effects of turbulence, DNS directly resolves these turbulent scales. This is its primary strength, as it provides fundamental physical insights that simpler models cannot capture. However, this high fidelity comes at a computational cost. The small scales involved mean that DNS is often limited to much smaller computational domains and lower Reynolds numbers than those seen in real-world engineering problems. Ref. [45] performed a similar DNS study to [44] on channel flow, highlighting that while DNS provides invaluable insight, models and computational resources still have limitations that prevent the direct simulation of full-scale problems.

To further assess the predictive ability of numerical methods, ref. [46] investigated the performance of air lubrication systems on a planning hull with a step. The study utilised the commercial CFD software STAR-CCM+ (Version 9.0) to solve the unsteady Reynolds-averaged Navier-Stokes equations with the Volume of Fluid method to capture the free surface and air layer. The turbulence model used was the $k - \omega$ Shear Stress Transport (SST) model. All simulations used a time-marching approach to capture the unsteady phenomena. The simulations employed an overset mesh approach to model the hull's motion, and these simulations were validated against towing tank testing results. The results of the CFD simulation compared to the towing tank test results showed a difference of up to 8% for the non-lubricated case at the initial velocity. In the air-lubricated case, the difference was generally below 6%, leading the authors to conclude that the CFD approach was suitable for simulating these complex air lubrication mechanisms. Similarly to [46], the work of [47] compared experimental data with two different CFD models, on an axisymmetric body to investigate the mechanisms of drag reduction. The work made a direct comparison between a Eulerian-Eulerian two-fluid approach and a standard VOF modelling approach, and this comparison provides critical insight into the applicability of each of these methods. The Eulerian-Eulerian approach was found to be better suited for microbubble drag reduction. It demonstrated good resolution of the phenomenon here, while the VOF model more accurately represented the stratified air layer in air layer drag reduction. With this comparison of the two modelling approaches in mind, the authors proposed a data-coupling method to bridge the transition between the two regimes. These data coupling method demonstrated good agreement with the experimental data, indicating that the predicted morphologies of the mixture flow were in qualitative agreement with the observations made in the experimental work. This confirms the success of these CFD models in replicating the nature of air-water interaction. Furthermore, the study also noted a weakness in the Eulerian-Eulerian approach, as it did not accurately capture the rolling of the air layer in the ALDR region.

Overall, it can be stated that the current state of the art in predicting air lubrication on a ship's hull and its corresponding drag reduction enables the operator to replicate the levels of drag reduction that would be expected from physical testing. The methodologies employed are increasingly robust and capable of reproducing several of the expected mechanisms seen in real-world scenarios. For example, computational models can demonstrate bubble breakup and coalescence, which are vital indicators of the simulation's ability to capture the multiphasic interaction intrinsic to air lubrication technology. Aside from these advances, there are inherent limitations associated with CFD in simulating air-water mixtures and high Reynolds numbers. The limitations can be attributed to two primary sources: the first being the difficulty in accurately modelling physical characteristics. An example of this is that simulating hull surface roughness with a single parameter inherently involves oversimplification, and the idealised scenario can cause deviation from experimental data. The second source is computational error, which is a persistent question regarding computational approaches, and this can propagate through the model and cause deviations in the achieved result [30]. This can be especially prominent in multiphasic scenarios and at high velocities. For instance, studies comparing CFD results for planing hulls to experimental data have demonstrated a range of accuracy. The work of [48] used a RANS solver with the Volume of Fluid (VOF) method, and concluded that CFD predictions for planing hulls have an accuracy of around 10% when compared to experimental data, representing a significant improvement over traditional semi-empirical formulations. Ref. [49] corroborated the conclusions of [48] stating that their RANS simulations, also using VOF and the k-omega SST turbulence model, had a resistance prediction error below 10%. Furthermore, the authors of [50] found that resistance error can vary from 4.5% to 9.5%, depending on the mesh strategy and software used. These differences highlight the variability in accuracy for current CFD approaches, which is often a function of the specific numerical methods and mesh generation techniques employed.

The work of [51] discusses how previous work has considered the mechanisms of air bottom cavitation. However, the effects of the interaction of the ventilated cavities with the ship boundary layer have been neglected, even though these effects may be substantial. The work suggests that existing methods for ship hull design and retrofits are based on solving inverse problems of ideal fluid theory; however, it also suggests that a framework of ideal fluid theory cannot accurately predict the drag reduction and power-saving rates for this technology. Ref. [51] suggests that ideal fluid theory can predict the shape of ventilated cavities to a satisfactory degree; however, these rates are not directly proportional to the ratio of the hull surface covered by the cavity to the total wetted area. Furthermore, the friction reduction by the cavity and an air escape from the cavity will subsequently affect the thickness of the ship boundary layers and wakes and the velocity profiles across them. The author mentions a substantial scale effect for the air demand that partially ventilated cavities require, and the power saving subsequently depends on this demand. The work suggests that a numerical analysis of the interaction between air-ventilated cavities and boundary layers requires a unique flow model, as existing methods cannot accurately describe the gas flow within cavities or its escape from them.

The paper uses a multi-zone flow model to analyse the interaction between the ship's bottom air cavity and the boundary layer, rather than existing widely used RANS approaches. The model divides the flow into three distinct zones, each with its own set of governing equations: the inviscid outer flow zone, the air cavity zone and the boundary layer zone. Each of these zones is said to be governed by its own set of governing equations. Zone 1 describes the incompressible, curl-free water flow outside the boundary layer and is therefore governed by the Laplace equation.

$$\Delta\Phi = 0 \tag{9}$$

Zone 2 models the incompressible air flow inside the cavity. Therefore, the air mass conservation equation is integrated across the cavity to describe the air demand and escape.

$$u_0 \frac{d\delta_a^*}{dx} - (H - \delta_a^*)\frac{du_0}{dx} - \frac{dQ}{dx} = 0 \tag{10}$$

Zone 3 models the boundary layer on the cavity and hull's wetted surface, treating it as a compressible two-phase flow. This region is governed by a set of integral equations which include a mass conservation law.

$$\frac{d(\delta\beta)}{dx} + \frac{dQ}{u_0 dx} = -\beta \tag{11}$$

Momentum equation

$$\frac{d\delta^{**}}{dx} + \frac{\epsilon W}{U}\frac{dQ}{dx} = \frac{\left(M^2 - 2\right)\delta^{**} - \delta^* + \delta}{U}\frac{dU}{dx} \tag{12}$$

And a differential momentum equation for water derived from the Reynolds equation.

$$u_0\frac{du_0}{dx} = -\frac{\partial\langle u'v'\rangle}{\partial N} - \frac{\partial\langle u'u'\rangle}{\partial x} \tag{13}$$

The equations govern the three flow zones. In the air mass conservation equation (Zone 2): u_0 is the velocity outside the boundary layer, δ_a^* is the displacement thickness of the boundary layer on the cavity surface, H is the boundary layer thickness at the cavity closure, and Q is the mass flow rate of air escaping from the cavity. In the boundary layer mass conservation equation (Zone 3): $\delta\beta$ is the boundary layer thickness of the gas phase, β is the friction coefficient, and u_0 and Q are as defined above. In the momentum equation (Zone 3): δ^{**} is the momentum thickness, M is the Mach number, and ϵW is a coefficient related to the velocity ratio, with U being the liquid flow velocity. The final differential momentum equation $u'v'$ which represent fluctuating velocity components in the x (streamwise) and N (normal to the body surface) directions.

The work of [51], uses a combination of numerical techniques to solve the equations for the zones. In Zone 1, for example, the method uses a quasi-linearisation of the boundary conditions and iterates to find the unknown boundary shape. However, the paper notes that other methods, such as the boundary element method, could also have been employed. For the boundary layers in Zone 3, an integral computational method was utilised to solve the coupled equations across all three zones. The integral method approximates the velocity profiles across the boundary layers, thereby simplifying the problem. The strength of this paper's multi-zone approach lies in its ability to overcome the weaknesses of other methods by explicitly modelling the interaction between the air cavity and the ship's boundary layer. The method also accounts for the escape of air and its effect on the downstream boundary layer. The method by which the approach of [51] can be applied, depends on the context of the problem being solved. For preliminary Hull design and cavity shape prediction, the ideal fluid theory, which [51] is critical of would be suitable and efficient as s starting point as it can quickly provide a satisfactory prediction of the ventilated cavity's shape. However, where detailed performance metrics are required, the novel multi-zone model may be most suited.

Eulerian-Lagrangian Approach

An alternative approach to the Eulerian-Eulerian approach in simulating multiphase flows is the Eulerian-Lagrangian approach. The Eulerian-Lagrangian approach is beneficial in low-void-fraction scenarios. The Eulerian-Lagrangian approach treats the fluid as a continuum governed by Eulerian principles, while simultaneously representing individual bubbles or bubble clouds as discrete Lagrangian particles. The hybrid approach offered by the Eulerian-Lagrangian method has unique advantages and challenges. Ref. [52] demonstrated the use of this method in representing bubble trajectories in turbulent flow. The approach of [52] solves the continuous phase using RANS equations coupled with a high-fidelity Reynolds Stress Transport (RST) model developed by [53]. The RST model solves transport equations for each component of the Reynolds stress tensor, enabling it to better capture turbulence anisotropy, a key limitation of simpler turbulence models. The pressure-velocity coupling was handled using the Semi-Implicit Method for Pressure-Linked Equations-Consistent (SIMPLEC) algorithm. The core of the Eulerian-Lagrangian scheme lies in the fact that bubble motion can be solved simultaneously with the Rayleigh-Plesset equation, which governs bubble growth and collapse, thereby accurately determining bubble trajectories. This allows for the precise tracking of bubbles and the identification of flow structures around propellers and behind hulls. The work builds on that of [54], who used computer code to model the bubble trajectory by solving the Rayleigh-Plesset equation numerically, combined with a set of trajectory equations to model the bubble motion. A key strength of this method is that it can provide a detailed and high-fidelity view of individual bubble dynamics, making it valuable when considering specific problems such as cavitation and flow near appendages. The Eulerian-Lagrangian approach also faces some significant issues and limitations; however, this computational approach scales dramatically with increasing bubble numbers and increasing void fractions. The overall impact is that such an approach is often impractical for simulating high-injection-rate scenarios, where bubble-bubble interaction becomes more dominant. The concept of tracking a "Lagrangian bubble cloud" was developed as a compromise to reduce this computational burden. This contrasts with the Eulerian-Eulerian approach, which is better suited for high-void-fraction flows but often requires a population balance model to account for the dynamic evolution of bubble sizes.

The motion of the spherical particle within the fluid can be modelled using the formulation of [55], who formulated a general equation of motion for a small rigid sphere in a turbulent flow. This equation accounts for a variety of forces, including added mass, drag, buoyancy, and lift. To handle the frequent interaction between particles in dense multiphase flows, researchers often integrate a collision model. Such a model was developed by [56] and assumes that the movement is two-dimensional, bubbles are spherical and quasi-rigid, collisions are binary and instantaneous with a point contact, interaction forces are impulsive, and all other forces are negligible during a collision. The effects of bubble rotation can be neglected. This approach, however, has key limitations, as identified by Researchers such as [57] who pointed to some issues with treating bubbles as Lagrangian particles, stating that as bubbles increase in size, the one-fluid model can lose its validity since the bubbles become more dynamic, losing their Lagrangian particle nature. When bubbles become non-spherical, their shapes distort in response to pressure variations. Even in cases where the bubble remains spherical, it can change volume due to the compressible nature of gases. The change in bubble properties may pose a critical issue in the case of BDR simulation, as it can distort the prediction of the drag-reducing effect and the bubble's interaction with other entities. Further work is needed on this method and its effectiveness in stimulating multiphasic interaction and potential interaction with different structures and appendages, particularly in a marine setting, regarding BDR.

These changes pose a critical issue for BDR simulations, as they can distort predictions of drag reduction and the bubble's interaction with other structures. While computationally efficient for sparse bubble fields, the simplifying assumptions of this approach render it less suitable for high-void-fraction flows, suggesting that further work is needed to adapt this method for the complex multiphase interactions found in marine applications.

Beyond RANS approaches, higher-fidelity methods, such as Large Eddy Simulation (LES), have been employed to investigate the mechanisms of air lubrication. LES directly resolves the large-scale turbulent structures whilst modelling the smaller, sub-grid scales. LES can be a powerful tool for studying the fundamental physics of microbubble drag reduction, and the authors of [58] suggest that RANS approaches lack the necessary detailed turbulence information to study the microscopic mechanisms present in MBDR. The authors of [58], used an Eulerian-Lagrangian approach on a flat plate model and found that it was possible to successfully replicate the experimental results for both mean velocity distribution and the distribution of fluctuation intensities. The separate work of [59], further investigated the application of this method to simulating air lubrication and bubble drag reduction. The authors sought to emphasise that accurately modelling systems that incorporate significant bubble volume changes, such as in cavitating and two-phase flows, requires a comprehensive understanding of the bubbles' nature and interactions.

The authors state that for simulating air or bubble drag reduction, it is vital that the operator captures the complex physical mechanisms present and may require an approach beyond the limits of a standard Eulerian-Eulerian approach.

LES simulations of ALS have also used the Eulerian-Lagrangian method. The work of [58] investigated the use of LES by considering a flat plate. They found that the LES method agrees with experimental results for velocity distribution and the distribution of fluctuation intensities. Ref. [58] justified the choice of LES when considering microbubble drag reduction, as the authors stated that existing RANS approaches are unable to study the microscopic mechanisms underlying microbubble drag reduction. The reasoning is that the microscopic mechanisms require information regarding turbulence, which RANS cannot provide in sufficient detail. Ref. [59] further examined the Eulerian-Lagrangian approach for analysing bubble drag reduction. The authors indicated that for applications involving significant changes in bubble volume, such as in cavitating or two-phase flows, it is essential to understand the nature and interactions of bubbles to produce an accurate model. If one attempts to use the method for simulation in air or bubble drag reduction approaches, it would be vital to replicate the complex physics involved, and one must perhaps consider more physical mechanisms than those reproduced in a standard Eulerian-Eulerian approach. Ref. [60] considered the simulation of cavitating turbulent flow around a marine propeller behind a ship's hull using vorticity transport equations and particle trajectories. The work analysed the vorticity distribution and particle tracks of the interactions in this scenario explicitly. Lagrangian methods used in the work allowed for the tracking of particles and the identification of flow structures around propellers, especially behind hull operating conditions. The work employed a three-dimensional Lagrangian approach to predict particle trajectories, enabling the study of local cavitating forces from a Lagrangian perspective (i.e., the discrete cavitating effect of a bubble in isolation). The particle tracking study demonstrated that the propeller rotation and geometry result in a much more complex and distorted flow structure than that without propeller consideration (again, due to the ability to track a single discrete bubble). The predicted cavity patterns agree with the experimental results, although there is some over-prediction of cavity size, compared to the numerical results. The results show that sheet cavity shedding from the blade surface is caused by the side entrant jet, with predicted pressure fluctuations, which agrees with the experimental results.

Regarding application and future work, the comparisons made between RANS and LES, as well as the potential insufficiencies identified by the authors, suggest that for a holistic view of the nature of air lubrication, both LES and RANS have their applications under different circumstances. RANS may be indicated for larger-scale, hull-level performance predictions, where computational cost is restrictive. In contrast, LES may be seen as more applicable to fundamental research aimed at developing and validating sub-grid models.

5. Interaction of the Air Layer on Appendages

The bubbles introduced by an air lubrication system behave separately and distinctly from the steam bubbles, which are the cause of cavitation on a ship's propeller; it is nevertheless essential to consider the effect that introducing air bubbles into the propeller plane may have on the propagation of cavitation on the propeller as well as the noise generated, given that propellers are already established as one of the most significant sources of underwater noise generation on a ship. The ability of CFD to accurately simulate single-phase flow is an essential precursor to modelling the effects of air lubrication systems on downstream components. It is necessary to establish whether a consistent and meaningful prediction of cavitation on the propeller can be achieved using a CFD approach and how that prediction changes when air bubbles from an upstream system are introduced into the propeller's slipstream. The work of [61] was key in this area, as the authors conducted work on modelling propeller cavitation and estimating performance under cavitating conditions. The work of [61], particularly focused on the development of tip vortex cavitation (TVC). The model used in [61] was based on the Rayleigh-Plesset equation to govern cavitation dynamics, and a Volume of Fluid approach was used to capture the vapour-liquid interface. This study successfully predicted the propulsion coefficients and cavity patterns that would be expected for the benchmark propeller considered under conventional flow conditions. Building on [61], the work of [62] examined propeller cavitation further by using a mesh adaptation and refinement approach to cavitation simulation (MARCS), with the intention of improving accuracy in capturing TVC in the propeller slipstream. A subsequent study by [63] then considered the broader challenge of hull-propeller interaction, and highlighted a key limitation in current models which is the interaction between TVC and the rudder which could not be simulated to the desired accuracy that the authors wished to achieve. The findings of these bodies of work are key in the context of future research regarding air lubrication, as they demonstrate the difficulty in accurately predicting how the air-water mixture from an upstream air lubrication system would behave if it were passing through the propeller plane and interacting with other components. These studies collectively show that, while CFD can accurately model single-phase propeller cavitation, significant challenges remain in simulating complex, multi-phase interactions that are a direct consequence of operating an air lubrication system.

Ref. [64] looked at the effect on thrust and torque for the propeller to assess the propulsive performance, and uses a RANS-based solver and a Volume of Fluid (VOF) two-phase flow model. The model relies on the two fluids being immiscible in each control cell. The $k - \omega$ SST model simulates the scenario using the commercial solver STAR CCM+ as the software package. The results of [64] indicated that the injected air does not directly enter the propeller working area for the designed bottom cavity and the model considered in the paper. The work suggests that the average axial speed at the propeller disk surface would increase, and the average wake fraction would decrease as air is injected. The work compares the propeller thrust and torque in calculations with and without air. With the same inflow velocity and rotation, air injection decreases the thrust and torque coefficients. This decrease occurs because the air injection increases the axial velocity at the disk surface, and the average wake fraction decreases; hence, the propeller's thrust and torque coefficients also decrease.

The work of [65] examines the effect of the air layer on propeller characteristics, and it explicitly considers the Mitsubishi Air Lubrication System (MALS). The work examines the void fraction in the propeller region/disk area, and its impact on propeller characteristics and fluctuating pressure. The work utilises CFD to predict the air bubble void fraction distribution on the hull surface, which is necessary to determine the reduction in hull resistance. Additionally, it predicts the void fraction distribution on the propeller disk area, influencing the distribution of propeller performance. All bubbles in the work of [65] were uniform in diameter and remained unchanged in the flow; no consideration was given to the bonding of bubbles or division into multiple smaller bubbles. The bubble diameter for ships is stated as 2 to 3 mm in the case of actual ships, and the work scales this to the model ship, examining bubble size ranges for this ship of 0.1 mm to 1 mm.

The ability to scale bubble sizes along with the model is one advantage of using CFD over physical testing, which is more challenging. There were three cases considered by [65]. In cases 2 and 3, the bubble sizes were set to 5 and 10 times that of case 1, respectively. The work suggests that the influence of air bubble diameter on air bubble distribution is limited; consequently, despite actual air bubble diameters varying, the bubble effect can be roughly predicted from a calculation based on bubbles of a particular uniform size. The work does not provide a quantitative evaluation of the bubble distribution and instead presents the void fraction distributions for comparison in a qualitative manner, alongside the air bubble distribution images obtained from experiments. A visual representation of the flow fields is included in [65].

Ultimately, the work of [65] looks at the effect of bubbles flowing into the propeller region because it can alter the propeller characteristics, reduce the propeller operation's efficiency, and influence the noise generated by the propeller. The work of [65] indicates that the influence of bubble characteristics can be assessed by changing the void fraction of the air bubbles flowing into the propeller disk area. Suppose the void fraction of the air bubbles flowing into the propeller region can be predicted accurately by the method. In that case, their influence on the propeller performance can be confirmed using numerical data.

Both the study of [64,65] use RANS approaches, simplifying the flow behaviour. Ref. [64] make use of the SST $k-\omega$ turbulence model, which is widely used in the study of air lubrication more generally in CFD, and this could be assumed to be a reasonable choice, given the model's ability to predict fluid behaviour in both the near-wall and outer flow; however, the extent to which these conventional turbulence models can represent the behaviour in the propeller region may require further study and would be a possible avenue for further research. Comparing the two papers, of [64,65], both have their specific strengths and weaknesses. In [64], the use of a large continuous air layer provides a substantial reduction in frictional resistance. The study also explicitly accounts for the propeller's effect on the air layer and vice versa. However, concurrently with [64], the specific design of the bottom cavity, with its side slopes, may not be representative of all air lubrication systems. The paper notes that the results are particular to this design and that the air does not directly enter the propeller working area, which may not be the case for all systems. The work also concludes that more research is needed to analyse the effect on propeller performance at the new self-propulsion point. Ref. [65] investigates one specific ALS, the Mitsubishi Air Lubrication System (MALS), which uses a system of bubble outlets to inject air. The study has a particular strength in that it considered the effect of a range of bubble diameters, which is a key parameter that can be difficult to control in physical experiments. The finding that bubble diameter had only a limited influence on the overall air distribution enables simplification of the modelling and possibly indicates a greater applicability of the result. Furthermore, the work shows that the loss of propulsive efficiency due to the bubbles would be negligible. However, the work has some apparent weaknesses, as the study's assumption of a uniform, unchanging bubble diameter is not supported by observations when considering a compressible fluid that is also subject to phenomena such as coalescence and breakup. Furthermore, assuming a single system (the MALS system) and a single case of application is a further indication that the study's conclusions may not be fully applicable in a general case.

5.1. Assessing the Effect on Efficiency and Propulsive Performance

Regarding the MALS system, several findings examine the application of MALS technology in depth [66–68]. These bodies of work point out that propulsive power for a typical ship would often be considered by looking at the results of a towing tank test using a model ship; however, it is difficult to estimate the propulsive power in this manner for a vessel equipped with an air lubrication system due to the need to scale the bubble size with the size of a ship. The development of the Mitsubishi Air Lubrication System (MALS) relies on CFD to predict the behaviour of air bubbles around a ship's hull. This is essential because physical model tests in a towing tank cannot accurately replicate the air bubble flow due to scaling issues. Air bubbles of millimetre order on a real ship would need to be of micrometre order on a model, which is difficult to achieve by simply blowing air from the model. This leads to an underestimation of frictional drag reduction and incorrect air bubble distribution around the propeller in tank tests. In the case of a model test, the operator is faced with no choice but to use bubbles of magnitudes greater than would be correct if the bubbles were scaled to the size of the ship. Failure to scale the bubbles will not only result in an underestimation of the frictional drag effect but will also result in incorrect bubble distribution around the propeller; for this reason, it is suggested that there is not only a desire for CFD technology to be developed that can correctly produce bubbles which are scaled in size with the ship, but it is also necessary. Furthermore, the work of [66–68] draws on the findings of towing tank tests to say that bubbles flowing into the propeller plane will reduce efficiency and increase pressure fluctuation. The work found that in cases where the air layer exists, the propeller pressure fluctuations at the bottom of the hull decrease due to the pressure layer's reflection and the pressure-dampening effect of the air bubble layer. Moreover, the authors found that the air bubble layer has a higher void fraction in the vicinity of the hull's bottom when the bubble layer thickness is reduced. The work of these authors highlights the added complexity involved in attempting to assess the impact of air ingress using conventional methods, both within experimental testing and within standard CFD methods such as RANS.

RANS-based approaches have been used to analyse the complex interaction between a ship's hull and propeller in many studies. These methods are crucial to understanding how the addition of air lubrication may further complicate an already unsteady and non-uniform flow in the stern region. Three such papers [69–71] look at the interaction of the hull on propulsive characteristics. Adding air lubrication will mean the already complex and unsteady interactions within the transom/propeller region may be further complicated. It will be essential to consider how this will affect the region's behaviour, further complicating the modelling of the scenario.

The work of [69] considers the viscous flow around a self-propelled hull using a RANSE solver with a model-scale boat. The solver used the realisable k–ε turbulence model. The main objective of the work is to consider the inherently unsteady hull-propeller interaction resulting from operation in the unsteady, non-uniform wake. The propeller's performance depends on the incoming flow velocity; this dependence indicates that investigation of this interaction requires careful modelling, as its inherent unsteadiness means the interaction is complex. The use of a sliding mesh technique in this paper is a common approach for propeller CFD simulations. The proposed models accurately predicted the augmented drag, thrust, and torque coefficients, as well as the thrust deduction fraction, within a 5% margin. This accurate prediction suggests that it may be an auspicious starting point for simulating an air lubrication case and its interaction with the propeller and transom regions, in addition to the typical hull-propeller interactions observed in a non-air lubrication case.

Ref. [70] considered the hydrodynamic performance of a ship hull, including a propeller. The work considers the performance by using a RANSE solver, similar to [69], but investigated the applicability of various turbulence models, including Standard k-ε, Realisable k-ε, and SST k-ω, for predicting hull and propeller performance. The work investigates the flow features around a ship hull with a rotating propeller in open-water tests, resistance tests, and self-propulsion tests. The paper then examines the applicability of various turbulence models in predicting the hydrodynamic performance of the propeller and the hull-propeller interaction. The work employs a hybrid meshing approach, utilising an unstructured mesh near the complex geometry and a structured mesh in the simpler parts of the geometry. This approach was adopted to incorporate more flow field features and enhance calculation accuracy. A moving mesh was used at the location of the propeller disk, with the moving volume itself rotating, and non-conformal interfaces placed between the rotational and stationary subdomains. Overall, the work found that the Standard k-ε turbulence model was more accurate in the case of the resistance test. In contrast, Realisable k-ε is more suitable for predicting self-propulsion performances. As a result, the CFD method was verified, and the computational results were validated.

Ref. [71] looked at the potential to use a RANS approach combined with a potential flow approach to analyse propeller performance and load for a self-propelled condition. The work presents a combination of unsteady RANS simulations of the flow around the ship in free-surface conditions, utilising the volume of fluid method, and the lifting line model (LLM) for propellers operating behind a boat. This study uses the SST k-ω turbulence model and the Volume of Fluid (VOF) method to account for the free surface. The propeller's effect on the wake is represented by an actuator disk that applies body forces to the flow field, which the LLM iteratively updates based on the calculated effective wake. The simulation results demonstrate that the coupling method can accurately estimate the propeller loading and its impact on the average flow field. Compared to established results, unsteady propeller loading on the hull is predicted via the flow field for both the full RANS approach and the coupling method. The work explains that due to the complex and non-linear features of the flow field structures around the propeller, accurately predicting the ship's powering performance is more complicated than predicting hull resistance or propeller open-water performance.

5.2. Effect on Underwater Noise from Air Ingress

The bubbles introduced by an air lubrication system behave separately and distinctly from the steam bubbles, which cause cavitation on a ship's propeller. It is nevertheless essential to consider the effect that introducing air bubbles into the propeller plane may have on the propagation of cavitation on the propeller and the noise generated. This is vital, given that propellers are established as one of a ship's most significant sources of underwater noise generation [72,73].

The possibility of using air as a means of reducing underwater noise in various applications has been proposed by a range of researchers. One proposed use of air in this manner is presented in [74], and this is in the context of reducing underwater noise in percussive piling. The researchers in [74] state that the use of bubbles can reduce sound transmission, and this is caused by the difference in density between the air medium and the water medium. Within the EU Project SONIC, there is a focus on suppressing underwater noise induced by cavitation [75]. Air injection has been presented within SONIC as a noise-reducing approach, with the partners involved suggesting that air injection can have a significant effect on noise generation even at low flow rates [76].

In the 1960s and 1970s, the US Navy developed systems to reduce underwater noise on warships. These were termed 'Prairie' and 'Masker' systems, with the Prairie system being fitted near to or on the ship propeller itself and the Masker system being designed to silence a vessel's engine noise, typically being fitted on the sides of the hull.

Information and quantitative data on the original Prairie and Masker systems are limited; however, these systems appear to have been used on some US warships. More recently, 'Masker' and 'Prairie-like' systems have been investigated under the EU Horizon 2020 Project SATURN, which sought to reduce ship underwater noise [77,78]. A representation of the system shows the location of the injection point in each system, respectively [78]. The study of these systems under the SATURN project managed to demonstrate a notable reduction in underwater radiated noise from the systems. The 'Prairie-like' system is of particular interest as this system involves injecting air through holes in the leading edges of ship propeller blades in an effort to dampen the collapse of cavitation bubbles. The work under the SATURN project performed tests at four different combinations of propeller thrust coefficient and cavitation number; however, the researchers in [77]. The study does not explicitly consider the impact of air entrainment on the thrust coefficient, suggesting that future work should address this aspect.

The researchers in [79] point to injecting air through the propeller blades to reduce noise and cavitation but do not provide discussion of the effects of introducing air on propeller propulsive characteristics. There is a reference to other systems that introduce air to the propeller for the purpose of reducing noise, referred to as propeller air ingestion and emission systems. However, information on such systems is limited, and there is a lack of evidence and research regarding their use.

One such system is presented by a company called Hanwha [80], who state that air injection into the propeller can reduce noise generation by producing an 'air curtain'; however, there is a lack of corresponding academic or industry research regarding the potential effect on propeller propulsive characteristics. Ultimately, when considering the available literature on the use of air to reduce underwater noise as a whole, it appears that there is precedent for its use for this purpose. Still, there is limited discussion and understanding of its effect on propulsive characteristics.

6. Difficulties with CFD Simulation of Air Lubrication and Complex Mechanisms

6.1. Re-Entrant Jet and Wave Pinch-Off

The work of [5] uses a viscous-flow CFD code, ReFRESCO. It is complemented by turbulence models, cavitation models and volume-fraction transport equations for different phases. The paper utilises CFD to investigate the re-entrant jet and Wave Pinch-off effect, which is observed in air lubrication systems. The work concluded that the existing commonly used industry-standard k-ε and $k - \omega$ turbulence models are insufficient to model the turbulence characteristics for air lubrication accurately. Notably, ref. [5] point to the work of [81], who suspected that the overprediction of the eddy-viscosity dampens the unsteadiness of sheet cavitation, and thus, poor modelling could incorrectly mask the effect of the Re-Entrant Jet by preventing the simulated occurrence of shedding. Ref. [5] State that additional artificial correction models are available to lower the predicted eddy-viscosity; one such model is the Reboud-correction model, which introduces a limiter to the computation of turbulent viscosity [82,83].

Ref. [84] witnessed the mechanism of the re-entrant jet in their work on the capability of RANS solvers to predict the behaviour of marine propellers and their induced pulses. The work identified a re-entrant flow forming due to the convex cavity closure line, blade geometry and wake; the mentioned re-entrant flow was created beneath the sheet cavity surface and travelled towards the blade tip. Suffice it to say that the mechanism of the re-entrant jet is not an Air-Lubrication exclusive mechanism, and the operator must be aware of the mechanism and possibility of a Re-Entrant jet occurring in standard free surface simulations.

Regarding wave pinch-off, standard turbulence models struggle to accurately model the effect, as they only account for the impact of turbulence on the general flow. To identify the scale of these structures for wave pinch-off, a larger number of turbulence scales in the flow must be resolved. It is suggested that scale-resolving simulation models, such as those described by [85] could help resolve large-scale turbulence in the flow. These models have been developed to bridge the gap between RANS and LES models, as resolving the full range of turbulence in DNS remains far too computationally expensive for practical use. Scale-resolving simulations will resolve the larger scales of turbulence and model the smaller scales. Ref. [86], however, state that it is known a priori that a pure RANS approach cannot compute the turbulence structures believed to be responsible for disturbing the cavity surface and entirely disregards the notion of using a pure RANS model to model the wave pinch-off mechanism.

Moreover, ref. [86] further reiterated that RANS methods might be insufficient to model the Re-Entrant Jet and Wave Pinch Off. Standard $k-\omega$ and k-ε models consistently fail to predict large separation areas well, and the incorrect prediction of Eddy Viscosity is apparent when these are used. The work highlights that the standard turbulence models mentioned were initially developed to predict smooth, attached, and single-phase turbulent flows. Consequently, in the case of air lubrication simulation, a distinctly two-phase scenario, the models are ill-equipped to simulate unsteady vapour sheet cavity dynamics. Ref. [86] Attempted to better reproduce the phenomenon of the Re-Entrant Jet by using the KSKL turbulence model described initially by [87], coupled with two eddy viscosity correction functions, but this was unsuccessful in reproducing the effect. The approach failed to produce a re-entrant jet under the conditions they believed to be most conducive to creating the phenomenon, as indicated in their physical testing cases. The work suggests that more factors must cause the re-entrant jet phenomenon than merely eddy viscosity.

The work of [88] further confronts the issues of Wave Pinch-Off and Re-Entrant Jet. The work of [88] is in agreement with the work of [5], the author concludes that the main difficulty in the numerical modelling of air lubrication techniques is the inherent instability that arises from the modelling of the two-phase flow, further compounded by the relatively small thickness of the air layer to the plate or boat model, which is being considered. The work concludes that the model is the best standard turbulence model for predicting the system's overall behaviour in the $k-\omega$ model. Specifically, the model is preferred due to its stability of the air layer across the plate. The authors found that other models would erroneously predict that as air flux increases, the stability of the air layer would decrease, which is in opposition to established experimental results, such as in the widely cited work of [11,12].

6.2. Kelvin-Helmholtz Instability

Kelvin-Helmholtz instability (KHI), arising from the velocity difference across the air-water interface in air lubrication, is a phenomenon captured in physical experimentation by several authors. Ref. [89] carried out a DNS study of air layer drag reduction over a backward step to confirm established results for air lubrication and, in the process, reported the presence of KHI near the site of air injection, with the particular presence of this instability when the air injection rate was decreased. The work of [88] indicates that the characteristic instability seen in DNS and physical testing is absent in RANS approaches, where, incorrectly, the RANS approaches will present the air-water interface as more stable than reality. The author introduces an artificial perturbation in their RANS approach to imitate the expected initial instability at the injection point. The reasoning for this is that, unlike LES simulations, where one can introduce the initial perturbation by setting differing viscosity values of the sub-grid scale in different cells, RANS models do not vary the turbulent viscosity spatially to a great extent, tending to predict a smooth transition in expected flows. The author introduces a sinusoidal fluctuation to the airflow rate to artificially introduce the instability at the air injection point and finds that this agrees with experimental results.

6.3. Numerical Ventilation

In CFD, the VOF model is often used to model and track the free surface of two immiscible fluids and is particularly relevant for tracking the position of the interface between the two phases. Numerical ventilation (N.V.) is a problem typically encountered when a vessel creates an acute angle with the free surface. The work of [90] states that this is one of the primary sources of error witnessed in numerical simulations of planning hulls and warrants further analysis. The same body of work suggests that interface smearing, caused by an inability to track the free surface, is the primary source of N.V. The additional complexities of air lubrication mean that the difficulties in monitoring the interface between air and water further encourage the propagation of the N.V. effect.

7. Novel Simulation Approaches

Much of this review has focused on conventional approaches to simulating air lubrication, primarily RANS-based methods, with additional discussion of LES and DNS applications. However, there are currently some promising novel approaches to simulating air lubrication and bubble dynamics, which should be discussed explicitly beyond the comparative discussion made earlier in this review. The review has already covered hybrid methods, such as the Eulerian-Lagrangian method. Still, there is scope for the use of entirely separate approaches, including particle-based, front-tracking, and interface-capturing methods.

7.1. Lattice-Boltzmann Method (LBM)

Although LBM has been proposed for bubbly flows, there is limited precedent for its application regarding air lubrication. The work of [91], indicates that the LBM can be applied to fluid-film lubrication, and compared to traditional approaches based on the Reynolds equations, the implementation of LBM is less demanding than that of RANS solvers. LBM does not neglect the inertia forces; LBM models the fluid consisting of fictive particles, and such particles perform consecutive propagation and collision processes over a discrete lattice. However, LBM is likely not suitable for simulating air lubrication, as it is derived from kinetic theory and is inherently a low-Mach-number method. It can be adapted for incompressible flows; however, accurately simulating multiphase flows with a large density ratio, such as between air and water in air lubrication, would be a significant challenge. Moreover, LBM models struggle with numerical stability and accuracy at such high-density ratios. While LBM is highly parallelisable and can be efficient for specific applications, it can be computationally expensive for large-scale simulations. This is due to its reliance on a uniform, Cartesian grid. To resolve a boundary layer near the hull of a ship, a very fine grid would be required. In traditional finite volume methods, you can use high-aspect ratio cells in the boundary layer to save computational resources, but this is not possible with LBM. As a result, the entire computational domain would need a high resolution to capture the small-scale phenomena, leading to an immense number of cells and, consequently, very high memory requirements.

7.2. Smoothed Particle Hydrodynamics (SPH)

SPH is a computational method that can be used for simulating fluid flows and other continuum media. It is a Lagrangian, mesh-free method and tracks the movement of individual particles, rather than solving the equations on a fixed grid. This approach offers several advantages, particularly in situations involving complex fluid dynamics. The work of [92], focuses on 'hydrodynamic lubrication,' however the fundamental principles and the successful application of SPH to a two-phase fluid problem with a free surface are directly transferable to the context of air lubrication, and indicates that this method could be applicable. The work of [92] also highlights the benefit of SPH over other methods, namely that it does not require pre-imposed boundary conditions at the inlet and outlet of the lubricated area. In the context of air lubrication, this is particularly relevant, as the air-water interface and leading edges of the bubble layer can be complex and dynamic. SPH additionally allows for the pressure and velocity profiles to emerge naturally from the fluid flow itself, which could be a significant improvement over traditional methods that require assumptions to be made at the boundaries. A particularly relevant part of the work of [92], is the method proposed for calculating surface tension using a continuum surface force approach. The CSF approach could be a crucial element in the accurate simulation of air lubrication, as the stability and shape of the air layer are highly dependent on surface tension. Despite the possible advantages of applying this method for air lubrication, there is again limited precedent for its use, and many of the most relevant papers focus on oil and traditional lubricant-based applications [93–96]. The reasoning for the lack of available air lubrication-based applications for SPH may be due to several reasons. Firstly, there can be a high computational cost associated with SPH, which can be prohibitive for large-scale engineering problems. Due to the thin boundary layer in air lubrication, this would likely require millions, or even hundreds of millions, of particles. As the SPH approach is not grid-based, it cannot utilise techniques like adaptive meshing to concentrate resolution where it is needed. Secondarily, air lubrication involves a high-density ratio between air and water. Although SPH can be adapted for multiphase flows, it faces specific challenges with numerical stability at high density ratios.

7.3. Discrete Element Method (DEM)

DEM is a numerical method which can simulate the motion and effects of a large number of particles, such as granular materials or rocks. It tracks each particle individually and calculates the forces experienced by the particle resulting from contact with other particles and boundaries, as well as other body forces. DEM, by itself, would not typically be considered for use in air lubrication because it is a method for discrete solid particles, whereas air lubrication deals with fluid phases. However, in theory, it could be combined with other methods in a Multiphysics approach to model such bubbly flows. One of the best examples of this theoretical approach is shown in [97], which describes CFD-DEM coupling. The work of [97] describes a Multiphysics approach used for simulating bubbly flows, which is applicable for considering air lubrication. Although this paper focuses on solid particles, it combines a CFD fluid element that acts upon the particles. For air lubrication, it would be possible to apply the same framework to represent discrete air bubbles within the continuous water phase. Furthermore, the work of [97] specifically uses an immersed boundary method variant, which achieves a 'fully resolved' simulation, and calculates hydrodynamic forces directly without relying on drag law approximations. This is a more accurate way to model the fluid-particle interaction than using simplified drag laws, which is especially important for the complex geometries and high Reynolds numbers in air lubrication. In summary regarding DEM, there are some theoretical and preliminary indications that such an approach could be adapted for use with air lubrication, and the work of [97]. In particular, it presents some elements that would theoretically apply to air lubrication; however, there exist limited contemporary current applications of DEM for this purpose.

7.4. Level Set Method (LSM)

The level set method can be a powerful method for simulating multiphase flows, such as those seen in air-water interactions. It is a fixed-grid, interface-capturing technique and a Eulerian approach, where the interface is tracked implicitly by embedding it as the zero contour of a higher-dimensional function, termed the Level Set Function. The motion of the interface is governed by an advection equation that tracks the evolution of the level set function in relation to the fluid's velocity field. This method is naturally adept at handling complex topological changes, such as bubble merging and splitting, because the interface position is defined by the zero-level set, which can change topology without explicit intervention. However, there is an immediate and considerable drawback to its use with air lubrication, as air lubrication involves a large density ratio and complex interfaces; the lack of mass conservation in a standard LSM approach limits its applicability in air lubrication. To address this issue with mass conservation, most applications of an LSM use a Coupled Level Set and Volume of Fluid (CLSVOF) solver. The work of [98] uses a CLSVOF approach in the simulation of air lubrication on a flat plate, and so is a very pertinent body of work to consider. In [98], the limitations discussed here are directly addressed, and the paper provides strong validation for the CLSVOF method's capability in two-phase flow and its accuracy in predicting drag reduction. The CLSVOF solver was first verified by successfully simulating the Rayleigh-Taylor instability, a benchmark problem for complex multiphase flows, and the authors showed quantitative accuracy for a specific test case (Cq = 0.149), the CLSVOF results showed excellent agreement with experimental data, with a difference of less than 1% for both total drag reduction and frictional drag reduction. The results clearly show that at low injected airflow rates (Cq = 0.037), where the air layer is discontinuous (BDR/TALDR regimes), the CLSVOF solver matches experimental data better than the VOF solver. It is also possible that the physical observations recorded in the work could be used to compare with other simulation methods, such as SPH or LBM, regarding recirculation and cavity formation, as well as air layer wave dynamics and flow regimes.

8. Conclusions

The modelling of the air lubrication mechanism, both physically and computationally, presents several challenges. In physical testing, the operator can hold more certainty that the physical mechanisms that arise from air-water interaction, such as re-entrant jet and wave pinch-off, will be replicated in the physical testing. However, simultaneously, the operator must consider that it is difficult to assess the scaling requirements when the ship is itself scaled down to model size, especially in the case of bubble drag reduction; bubbles may not necessarily scale with the size of the boat, and even if this was the case it is not an easy task to scale the bubbles down to this size. The operator is therefore faced with two scenarios: either securing a testing facility that is large enough to accommodate a full-size model or testing the technology on a scaled-down model and accepting that the bubble/air interaction with the water and its hydrodynamic implications may not be fully represented.

Questions are raised about CFD's ability to consistently replicate the air's effect on appendages, including cavitation patterns and its impact on operational parameters such as propeller efficiency and noise generation. In CFD, the issue of scaling can somewhat be addressed by the technology's ability to allow the operator to control parameters such as bubble size more quickly; however, confidence in the CFD approach has not yet reached a point where we can have confidence enough to move away from physical testing and onto computational simulation entirely. More broadly, the ability of computational models to handle the multiphase interaction of air and water in this scenario requires further research and development to prove and improve this method, given that the industry-standard turbulence, cavitation, and aeroacoustics models may not be fully fit for purpose in simulating these multiphase characteristics.

Physical testing relies on empirical formulas and the use of towing tanks and water tunnel tests; such methods have formed the basis of research into air lubrication over the decades that followed the renewed interest in the technology after the oil crisis of the 1970s. Using high-speed cameras and visual confirmation, physical testing has allowed operators to understand how the bubbles interact with the boundary layer. Furthermore, some of the more influential works that have employed physical testing have served as the comparative basis for evaluating the accuracy of CFD results.

Physical testing has been capable of replicating the results of laboratory tests on flat plates, and as such, has achieved considerable skin frictional drag reductions, up to 80%. Furthermore, physical testing has been capable of investigating and carrying out parametric tests on the effect of speed and air injection on the drag-reducing effect. Physical testing has managed to establish that energy savings are dependent on the percentage reduction in power to run a ship, ultimately meaning that vessels travelling at higher speeds will see that the power required to inject air represents a smaller overall increase in power expenditure (which is offset and improved in theory by the reduction in drag from the technology).

There have been several questions raised about the ability of CFD to competently replicate the mechanisms of air lubrication and the results of physical testing; CFD has shown good capability in the replication of full-scale ship experiments in single-phase conditions, and the technology has been used for this purpose successful for several decades now, consistently achieving results which are below 10% error and in some cases below 5%.

CFD methods have replicated physical testing results for air lubrication, and their percentage drag reduction values are broadly comparable; moreover, bubble distributions have been predicted in concordance with experimental results, suggesting that the approach can resolve the location and interaction of the stratified air competently. Several studies using CFD have considered the simulation of air lubrication on a ship model, and standard $k - \omega$ and k-ε turbulence models have been used frequently throughout such studies in three-dimensional approaches.

Alternative approaches, such as considering bubbles as Lagrangian particles, have been explored within CFD simulations to more accurately predict the tracking of bubbles compared to a standard Eulerian-Eulerian scheme. It is hoped that such schemes will be helpful when considering the interaction of the injected air on the propeller and other appendages; however, this approach has not been used as broadly for this application as traditional Eulerian-Eulerian schemes; hence, further research is required into the method.

Lagrangian particle methods have seen increasing research in recent times, and they present an exciting way to simulate better the effect of non-stratified air in the form of bubbles and their impact on operational parameters; however, the method has its limitations, and special consideration much be made in regards to whether bubbles can be simulated in this way to achieve meaningful results about their effect, given that bubbles often deviate from the expected behaviour of ideal Lagrangian particles.

CFD approaches have shown their ability to predict the operation of a propeller in single-phase conditions, including the prediction of cavitation patterns as well as thrust and torque, and an appreciation of the current approaches and their ability to predict the operation of a propeller in single-phase condition is an essential step to considering the ability of such solvers to predict the behaviour of propellers in multiphase conditions. Predicting propeller behaviour in multiphase conditions, such as those present with air lubrication, presents an additional challenge, and the question raised is whether current methods can predict this scenario robustly and realistically. The bubbles induced in cavitation, being steam bubbles, are distinct from those introduced in air lubrication. Still, it is nevertheless essential to consider how the introduction of air into the propeller plane will affect the propagation of cavitation.

This review sought to elucidate the current state of the art in investigating air lubrication and its mechanisms. In pursuit of this, the review has considered both physical testing and numerical simulation. This review has focused on simulation and experimental approaches. With this in mind, further reviews could be proposed to investigate other pertinent aspects of the technology which will impact its efficacy. Any further advancements in replicating air lubrication should be considered in future work or reviews of the technology. It is especially pertinent to consider advancements in numerical simulation as computational resources continue to improve alongside advancements in CFD software capabilities. Ultimately, work remains to be carried out regarding the scaling effect of bubbles used in BDR. The operator has a choice when they wish to conduct testing on the model scale. If one uses a scaled model, the bubbles will not necessarily scale the same as the model. As such, this can lead to bubble sizes incompatible with the model, resulting in an artificially over- or under-predicted void fraction under the ship's hull and subsequent poor prediction of drag reduction values and bubble behaviours. Additionally, further research into the use of Lagrangian particle tracking in a Lagrangian-Eulerian simulation would be a fortuitous avenue of research, as it could provide an excellent method of modelling the nature of bubbles in BDR, provided that the operator can ensure the physical qualities and interactions of the bubbles can be replicated using this approach.

Of the approaches considered in the novel simulation approaches section, the Coupled level-set and volume of fluid (CLSVOF) method appears, in the opinion of the authors, to be the most promising alternative method. However, this particular method is again based on the framework of a VOF method coupled with a level set method. This fact is likely an indication of the continuing prominence and applicability of VOF methods above other, less conventional approaches. The different approaches discussed in this section incorporated significant drawbacks, which ultimately limit their applicability regarding air lubrication.

The uptake of air lubrication technology will ultimately be influenced by the holistic benefits of using air lubrication. The drag-reducing benefit of using air lubrication will be offset either partially or wholly by the pumping requirement to inject the air, as well as any potential detrimental effects of the air's presence on seakeeping parameters and the propeller. Future work should consider the possible detrimental impacts of utilising the technology and subsequent consideration of the overall holistic effects. Regarding the impact of air ingress into the propeller, there is limited existing research, although some authors have acknowledged the potential detrimental effects of air ingress on propulsive characteristics. Concurrently, there is an existing basis for using air to reduce underwater radiated noise, and future investigations of air ingress should consider the purpose of the air injected in a given scenario, whether for drag reduction or noise reduction. Regardless of the motivation for introducing air, the effect of air ingress on the propulsive characteristics of the propeller is likely a significant driving factor for the adoption of the technology and warrants further in-depth research.

Author Contributions: Conceptualisation, D.H. and A.S.; methodology, D.H. and A.S.; software, D.H. and A.S.; investigation, D.H., A.S. and E.B.-D.; resources, E.B.-D., J.W. and M.S.; writing—original draft preparation, D.H. and A.S.; writing—review and editing, D.H., A.S., E.B.-D., J.W., M.S., O.Y. and G.V.S.; visualisation, D.H., A.S., E.B.-D., J.W., M.S., O.Y. and G.V.S.; supervision, E.B.-D., J.W. and M.S.; project administration, E.B.-D., J.W. and M.S.; funding acquisition, E.B.-D. and J.W. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: No new data was created or analysed in this study.

Acknowledgments: We would like to acknowledge Milad Armin for his contributions to this research work.

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

References

1. Bogdevich, V.; Evseev, A.; Malyuga, A.; Migirenko, G. Gas-saturation effect on near-wall turbulence characteristics. In Proceedings of the 2nd International BHRA Fluid Drag Reduction Conference (United States), BHRA, Cambridge, MA, USA, 31 December 1976.
2. McCormick, M.; Bhattacharyya, R. Drag reduction of a deep submergence vehicle by electrolysis. In Proceedings of the Ocean 73-IEEE International Conference on Engineering in the Ocean Environment, Seattle, WA, USA, 25–28 September 1973; pp. 406–410.
3. McCormick, M.E.; Bhattacharyya, R. Drag reduction of a submersible hull by electrolysis. *Nav. Eng. J.* **1973**, *85*, 11–16. [CrossRef]
4. Isaak, D.T. World Oil Shipping Demand: An Operational Analysis. Master's Thesis, University of Hawai'i at Manoa, Honolulu, HI, USA, 1981.
5. Rotte, G.; Zverkhovskyi, O.; Kerkvliet, M.; van Terwisga, T. On the physical mechanisms for the numerical modelling of flows around air lubricated ships. In Proceedings of the International Conference on Hydrodynamics (ICMF), Egmond aan Zee, The Netherlands, 18–23 September 2016.
6. Foeth, E.-J. The efficacy of air-bubble lubrication for decreasing friction resistance. In *The Naval Architect*; RINA: London, UK, 2011.
7. An, H.; Pan, H.; Yang, P. Research Progress of Air Lubrication Drag Reduction Technology for Ships. *Fluids* **2022**, *7*, 319. [CrossRef]
8. Madavan, N.; Deutsch, S.; Merkle, C. Reduction of turbulent skin friction by microbubbles. *Phys. Fluids* **1984**, *27*, 356–363. [CrossRef]
9. Feng, Y.-Y.; Hu, H.; Peng, G.-Y.; Zhou, Y. Microbubble effect on friction drag reduction in a turbulent boundary layer. *Ocean Eng.* **2020**, *211*, 107583. [CrossRef]
10. Madavan, N.K.; Deutsch, S.; Merkle, C.L. Measurements of local skin friction in a microbubble-modified turbulent boundary layer. *J. Fluid Mech.* **1985**, *156*, 237–256. [CrossRef]
11. Sanders, W.C.; Winkel, E.S.; Dowling, D.R.; Perlin, M.; Ceccio, S.L. Bubble friction drag reduction in a high-Reynolds-number flat-plate turbulent boundary layer. *J. Fluid Mech.* **2006**, *552*, 353–380. [CrossRef]
12. Elbing, B.R.; Winkel, E.S.; Lay, K.A.; Ceccio, S.L.; Dowling, D.R.; Perlin, M. Bubble-induced skin-friction drag reduction and the abrupt transition to air-layer drag reduction. *J. Fluid Mech.* **2008**, *612*, 201–236. [CrossRef]
13. Etter, R.J.; Cutbirth, J.M.; Ceccio, S.L.; Dowling, D.R.; Perlin, M. High Reynolds number experimentation in the US Navy's William B Morgan large cavitation channel. *Meas. Sci. Technol.* **2005**, *16*, 1701. [CrossRef]
14. Tsai, J.-F.; Chen, C.-C. Boundary Layer Mixture Model for a Microbubble Drag Reduction Technique. *Int. Sch. Res. Not.* **2011**, *2011*, 405701. [CrossRef]
15. Sindagi, S.; Vijayakumar, R.; Saxena, B.K. Experimental investigation on ship's model in carrying out energy economics of BDR/ALS methodology. *Ships Offshore Struct.* **2021**, *17*, 1437–1446. [CrossRef]
16. Giernalczyk, M.; Kaminski, P. Assessment of the propulsion system operation of the ships equipped with the air lubrication system. *Sensors* **2021**, *21*, 1357. [CrossRef]
17. Vijayan, S.; Kiran Babu, K.; Sendhilkumar, S.; Duraimurugan, G.K.; Deepak, P. CFD analysis of frictional drag reduction on the underneath of ship's hull using air lubrication system. *Int. J. Mech. Eng. Technol.* **2018**, *9*, 408–416.
18. Mizokami, S.; Kawakita, C.; Kodan, Y.; Takano, S.; Higasa, S.; Shigenaga, R. Experimental study of air lubrication method and verification of effects on actual hull by means of sea trial. *Mitsubishi Heavy Ind. Tech. Rev.* **2010**, *47*, 41–47.
19. Watanabe, O.; Masuko, A.; Shirose, Y. Measurements of drag reduction by microbubbles using very long ship models. *J. Soc. Nav. Archit. Jpn.* **1998**, *1998*, 53–63. [CrossRef]
20. Wang, K.; Li, Z.; Zhang, R.; Ma, R.; Huang, L.; Wang, Z.; Jiang, X. Computational fluid dynamics-based ship energy-saving technologies: A comprehensive review. *Renew. Sustain. Energy Rev.* **2025**, *207*, 114896. [CrossRef]
21. Ceccio, S.L. Friction Drag Reduction of External Flows with Bubble and Gas Injection. *Annu. Rev. Fluid Mech.* **2010**, *42*, 183–203. [CrossRef]
22. Chillemi, M.; Raffaele, M.; Sfravara, F. A Review of Advanced Air Lubrication Strategies for Resistance Reduction in the Naval Sector. *Appl. Sci.* **2024**, *14*, 5888. [CrossRef]
23. Merkle, C.L.; Deutsch, S. Drag reduction in liquid boundary layers by gas injection. *Prog. Astronaut. Aeronaut.* **1990**, *123*, 351–412.
24. Kodama, Y.; Kakugawa, A.; Takahashi, T.; Kawashima, H. Experimental study on microbubbles and their applicability to ships for skin friction reduction. *Int. J. Heat Fluid Flow* **2000**, *21*, 582–588. [CrossRef]

25. Takahashi, T.; Kakugawa, A.; Nagaya, S.; Yanagihara, T.; Kodama, Y. Mechanisms and scale effects of skin friction reduction by microbubbles. In Proceedings of the 2nd Symposium on Smart Control of Turbulence, University of Tokyo, Tokyo, Japan, 4–6 March 2001; pp. 1–9.
26. Liem, H.C.; Toda, Y.; Sanada, Y. A consideration on drag reduction by air lubrication using integral type boundary layer computation. *J. Jpn. Soc. Nav. Archit. Ocean. Eng.* **2011**, *13*, 59–65. [CrossRef]
27. Kanai, A.; Miyata, H. Direct numerical simulation of wall turbulent flows with microbubbles. *Int. J. Numer. Methods Fluids* **2001**, *35*, 593–615. [CrossRef]
28. Deutsch, S.; Moeny, M.; Fontaine, A.; Petrie, H. Microbubble drag reduction in rough walled turbulent boundary layers. In Proceedings of the Fluids Engineering Division Summer Meeting, Honolulu, HI, USA, 6–10 July 2003; pp. 665–673.
29. Sayyaadi, H.; Nematollahi, M. Determination of optimum injection flow rate to achieve maximum micro bubble drag reduction in ships; an experimental approach. *Sci. Iran.* **2013**, *20*, 535–541.
30. Goolcharan, J.D. Computational Fluid Dynamic Analysis of Microbubble Drag Reduction Systems at High Reynolds Number. Master's Thesis, Florida International University, Miami, FL, USA, 2016.
31. Mohanarangam, K.; Cheung, S.; Tu, J.; Chen, L. Numerical simulation of micro-bubble drag reduction using population balance model. *Ocean Eng.* **2009**, *36*, 863–872. [CrossRef]
32. Tanaka, T.; Oishi, Y.; Park, H.J.; Tasaka, Y.; Murai, Y.; Kawakita, C. Frictional drag reduction caused by bubble injection in a turbulent boundary layer beneath a 36-m-long flat-bottom model ship. *Ocean Eng.* **2022**, *252*, 111224. [CrossRef]
33. Orych, M.; Werner, S.; Larsson, L. Validation of full-scale delivered power CFD simulations. *Ocean Eng.* **2021**, *238*, 109654. [CrossRef]
34. Moraga, F.; Carrica, P.; Drew, D.; Lahey, R., Jr. A sub-grid air entrainment model for breaking bow waves and naval surface ships. *Comput. Fluids* **2008**, *37*, 281–298. [CrossRef]
35. Waniewski, T.; Brennen, C.; Raichlen, F. Bow wave dynamics. *J. Ship Res.* **2002**, *46*, 1–15. [CrossRef]
36. Qin, S.; Wu, D. Experimental and Numerical Study of Bubble Drag Reduction on a Flat Plate. In Proceedings of the Fluids Engineering Division Summer Meeting, Waikoloa, HI, USA, 30 July–3 August 2017; p. V01CT16A004.
37. Qin, S.; Chu, N.; Yao, Y.; Liu, J.; Huang, B.; Wu, D. Stream-wise distribution of skin-friction drag reduction on a flat plate with bubble injection. *Phys. Fluids* **2017**, *29*, 037103. [CrossRef]
38. Luo, H.; Svendsen, H.F. Theoretical model for drop and bubble breakup in turbulent dispersions. *AIChE J.* **1996**, *42*, 1225–1233. [CrossRef]
39. Troshko, A.; Hassan, Y. A two-equation turbulence model of turbulent bubbly flows. *Int. J. Multiph. Flow* **2001**, *27*, 1965–2000. [CrossRef]
40. de Bertodano, M.A.L. *Turbulent Bubbly Two-Phase Flow in a Triangular Duct*; Rensselaer Polytechnic Institute: Troy, NY, USA, 1992.
41. Troshko, A.; Hassan, Y. Law of the wall for two-phase turbulent boundary layers. *Int. J. Heat Mass Transf.* **2001**, *44*, 871–875. [CrossRef]
42. Ma, J.; Oberai, A.A.; Hyman, M.C.; Drew, D.A.; Lahey, R.T., Jr. Two-fluid modeling of bubbly flows around surface ships using a phenomenological subgrid air entrainment model. *Comput. Fluids* **2011**, *52*, 50–57. [CrossRef]
43. Gamal, M.; Kotb, M.; Naguib, A.; Elsherbiny, K. Numerical investigations of micro bubble drag reduction effect for container ships. *Mar. Syst. Ocean. Technol.* **2021**, *16*, 199–212. [CrossRef]
44. Ferrante, A.; Elghobashi, S. On the physical mechanisms of drag reduction in a spatially developing turbulent boundary layer laden with microbubbles. *J. Fluid Mech.* **2004**, *503*, 345–355. [CrossRef]
45. Xu, J.; Maxey, M.R.; Karniadakis, G.E. Numerical simulation of turbulent drag reduction using micro-bubbles. *J. Fluid Mech.* **2002**, *468*, 271–281. [CrossRef]
46. Cucinotta, F.; Guglielmino, E.; Sfravara, F.; Strasser, C. Numerical and experimental investigation of a planing Air Cavity Ship and its air layer evolution. *Ocean Eng.* **2018**, *152*, 130–144. [CrossRef]
47. Zhao, X.; Zong, Z.; Jiang, Y.; Sun, T. A numerical investigation of the mechanism of air-injection drag reduction. *Appl. Ocean Res.* **2020**, *94*, 101978. [CrossRef]
48. Brizzolara, S.; Serra, F. Accuracy of CFD codes in the prediction of planing surfaces hydrodynamic characteristics. In Proceedings of the 2nd International Conference on Marine Research and Transportation, Ischia, Italy, 28–30 June 2007; pp. 147–159.
49. Frisk, D.; Tegehall, L. Prediction of High-Speed Planing Hull Resistance and Running Attitude. A Numerical Study Using Computational Fluid Dynamics. Master's Thesis, Department of Shipping and Marine Technology, Chalmers University of Technology, Gothenburg, Sweden, 2015.
50. De Luca, F.; Mancini, S.; Miranda, S.; Pensa, C. An extended verification and validation study of CFD simulations for planing hulls. *J. Ship Res.* **2016**, *60*, 101–118. [CrossRef]
51. Amromin, E.L. Analysis of interaction between ship bottom air cavity and boundary layer. *Appl. Ocean Res.* **2016**, *59*, 451–458. [CrossRef]
52. Raoufi, A.; Shams, M.; Ebrahimi, R. A Novel CFD Scheme for Collision of Micro-bubbles in Turbulent Flow. *Eng. Lett.* **2008**, *16*, 280.

53. Launder, B.E.; Reece, G.J.; Rodi, W. Progress in the development of a Reynolds-stress turbulence closure. *J. Fluid Mech.* **1975**, *68*, 537–566. [CrossRef]
54. Meyer, R.; Billet, M.; Holl, J. Freestream nuclei and traveling-bubble cavitation. *J. Fluids Eng.* **1992**, *114*, 672–679. [CrossRef]
55. Maxey, M.R. The gravitational settling of aerosol particles in homogeneous turbulence and random flow fields. *J. Fluid Mech.* **1987**, *174*, 441–465. [CrossRef]
56. Hoomans, B.; Kuipers, J.; Briels, W.J.; van Swaaij, W.P.M. Discrete particle simulation of bubble and slug formation in a two-dimensional gas-fluidised bed: A hard-sphere approach. *Chem. Eng. Sci.* **1996**, *51*, 99–118. [CrossRef]
57. Lo, T.; L'vov, V.S.; Procaccia, I. Drag reduction by compressible bubbles. *Phys. Rev. E* **2006**, *73*, 036308. [CrossRef]
58. Wang, T.; Sun, T.; Wang, C.; Xu, C.; Wei, Y. Large Eddy simulation of microbubble drag reduction in fully developed turbulent boundary layers. *J. Mar. Sci. Eng.* **2020**, *8*, 524. [CrossRef]
59. Ma, J.; Chahine, G.L.; Hsiao, C.-T. Spherical bubble dynamics in a bubbly medium using an Euler–Lagrange model. *Chem. Eng. Sci.* **2015**, *128*, 64–81. [CrossRef]
60. Long, Y.; Long, X.; Ji, B.; Huang, H. Numerical simulations of cavitating turbulent flow around a marine propeller behind the hull with analyses of the vorticity distribution and particle tracks. *Ocean Eng.* **2019**, *189*, 106310. [CrossRef]
61. Yilmaz, N.; Atlar, M.; Khorasanchi, M. An improved Mesh Adaption and Refinement approach to Cavitation Simulation (MARCS) of propellers. *Ocean Eng.* **2019**, *171*, 139–150. [CrossRef]
62. Yilmaz, N.; Dong, X.; Aktas, B.; Yang, C.; Atlar, M.; Fitzsimmons, P.A. Experimental and numerical investigations of tip vortex cavitation for the propeller of a research vessel, "The Princess Royal". *Ocean Eng.* **2020**, *215*, 107881. [CrossRef]
63. Yilmaz, N.; Aktas, B.; Atlar, M.; Fitzsimmons, P.A.; Felli, M. An experimental and numerical investigation of propeller-rudder-hull interaction in the presence of tip vortex cavitation (TVC). *Ocean Eng.* **2020**, *216*, 108024. [CrossRef]
64. Wu, H.; Ou, Y.; Ye, Q. Numerical study on the influence of air layer for propeller performance of large ships. *Ocean Eng.* **2020**, *195*, 106681. [CrossRef]
65. Kawabuchi, M.; Kawakita, C.; Mizokami, S.; Higasa, S.; Kodan, Y.; Takano, S. CFD Predictions of Bubbly Flow around an Energy-saving Ship with Mitsubishi Air Lubrication System. *Mitsubishi Heavy Ind. Tech. Rev.* **2011**, *48*, 53–57.
66. Kawakita, C. Mechanism about change of pressure fluctuation of marine propeller running in bubbly flow. In Proceedings of the Fourth International Symposium on Marine Propulsors SMP'15, Austin, TX, USA, 30 May 2015.
67. Kawakita, C. Study on marine propeller running in bubbly flow. In Proceedings of the Third International Symposium on Marine Propulsors, SMP, Austin, TX, USA, 31 May–4 June 2013; pp. 405–411.
68. Kawakita, C.; Sato, S.; Okimoto, T. Application of simulation technology to Mitsubishi air lubrication system. *Mitsubishi Heavy Ind. Tech. Rev.* **2015**, *52*, 50–56.
69. Dhinesh, G.; Murali, K.; Subramanian, V.A. Estimation of hull-propeller interaction of a self-propelling model hull using a RANSE solver. *Ships Offshore Struct.* **2010**, *5*, 125–139. [CrossRef]
70. Shen, H.; Abdelhak, G.; Chen, Q.; Su, Y. The Hydrodynamic Performance Prediction of Ship Hull with Propeller. *Appl. Mech. Mater.* **2011**, *117-119*, 598–601. [CrossRef]
71. Sun, W.; Yang, L.; Wei, J.; Chen, J.; Huang, G. Numerical analysis of propeller loading with a coupling RANS and potential approach. *Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci.* **2019**, *233*, 6383–6396. [CrossRef]
72. Cianferra, M.; Petronio, A.; Armenio, V. Non-linear noise from a ship propeller in open sea condition. *Ocean Eng.* **2019**, *191*, 106474. [CrossRef]
73. Collier, R.D. Ship and platform noise, propeller noise. In *Handbook of Acoustics*; Wiley: Hoboken, NJ, USA, 1998; pp. 407–415.
74. Würsig, B.; Greene, C.R.; Jefferson, T.A. Development of an air bubble curtain to reduce underwater noise of percussive piling. *Mar. Environ. Res.* **2000**, *49*, 79–93. [CrossRef]
75. Sampson, R.; Turkmen, S.; Aktas, B.; Shi, W.; Fitzsimmons, P.; Atlar, M. On the full scale and model scale cavitation comparisons of a Deep-V catamaran research vessel. In Proceedings of the Second Workshop on Cavitation and Propeller Performance, Austin, TX, USA, 31 May–3 June 2015.
76. Prins, H.; Flikkema, M.; Bosschers, J.; Koldenhof, Y.; De Jong, C.; Pestelli, C.; Mumm, H.; Bretschneider, H.; Humphrey, V.; Hyensjö, M. Suppression of underwater noise induced by cavitation: SONIC. *Transp. Res. Procedia* **2016**, *14*, 2668–2677. [CrossRef]
77. Lloyd, T.; Lafeber, F.H.; Bosschers, J. Ship URN mitigation by air injection: Model-scale experiments and application to full-scale measurement data. In Proceedings of the 8th International Symposium on Marine Propulsors, Smp 2024, Berlin, Germany, 17–20 March 2024; Department of Marine Technology, University of Science and Technology: Trondheim, Norway, 2024; pp. 237–247.
78. Lloyd, T.; Lafeber, F.H.; Bosschers, J.; Kaydihan, L.; Boerrigter, B. Scale model measurements of ship machinery noise mitigation by air injection. In Proceedings of the 7th International Conference on Advanced Model Measurement Technology for the Maritime Industry, Istanbul, Turkey, 24–26 October 2023.
79. Merchant, N.D. Underwater noise abatement: Economic factors and policy options. *Environ. Sci. Policy* **2019**, *92*, 116–123. [CrossRef]

80. Hanwha. *Seeking Tranquil Waters: How to Improve Marine Ecosystems by Reducing Underwater Noise*; Hanwha: Seoul, Republic of Korea, 2024.
81. Li, Z.R.; Pourquie, M.; Van Terwisga, T.J. A numerical study of steady and unsteady cavitation on a 2d hydrofoil. *J. Hydrodyn.* **2010**, *22*, 728–735. [CrossRef]
82. Coutier-Delgosha, O.; Fortes-Patella, R.; Reboud, J.-L. Evaluation of the turbulence model influence on the numerical simulations of unsteady cavitation. *J. Fluids Eng.* **2003**, *125*, 38–45. [CrossRef]
83. Reboud, J.-L.; Stutz, B.; Coutier, O. Two phase flow structure of cavitation: Experiment and modeling of unsteady effects. In Proceedings of the 3rd International Symposium on Cavitation CAV1998, Grenoble, France, 7–10 April 1998.
84. Ge, M.; Svennberg, U.; Bensow, R.E. Investigation on RANS prediction of propeller induced pressure pulses and sheet-tip cavitation interactions in behind hull condition. *Ocean Eng.* **2020**, *209*, 107503. [CrossRef]
85. Klapwijk, M.; Lloyd, T.; Vaz, G.; van Terwisga, T. Evaluation of scale-resolving simulations for a turbulent channel flow. *Comput. Fluids* **2020**, *209*, 104636. [CrossRef]
86. Rotte, G.; Kerkvliet, M.; van Terwisga, T. Exploring the limits of RANS-VoF modelling for air cavity flows. *Int. Shipbuild. Prog.* **2019**, *66*, 273–293. [CrossRef]
87. Menter, F.R.; Egorov, Y.; Rusch, D. Steady and unsteady flow modelling using the $k-\sqrt{}$ kL model. In Proceedings of the Turbulence Heat and Mass Transfer 5. Proceedings of the International Symposium on Turbulence Heat and Mass Transfer, Dubrovnik, Croatia, 25–29 September 2006.
88. Montazeri, M.; Alishahi, M. An efficient method for numerical modeling of thin air layer drag reduction on flat plate and prediction of flow instabilities. *Ocean Eng.* **2019**, *179*, 22–37. [CrossRef]
89. Kim, D.; Moin, P. Direct numerical study of air layer drag reduction phenomenon over a backward-facing step. In *Center for Turbulence Research, Annual Research Briefs*; Center for Turbulence Research: Stanford, CA, USA, 2010; pp. 351–363.
90. Gray-Stephens, A.; Tezdogan, T.; Day, S. Strategies to Minimise Numerical Ventilation in CFD Simulations of High-Speed Planing Hulls. In Proceedings of the ASME 2019 38th International Conference on Ocean, Offshore and Arctic Engineering, Glasgow, UK, 9–14 June 2019.
91. Kucinschi, B.R.; Afjeh, A.A. A lattice-Boltzmann approach to fluid film lubrication. *J. Tribol.* **2010**, *132*, 021705. [CrossRef]
92. Tanaka, K.; Fujino, T.; Fillot, N.; Vergne, P.; Iwamoto, K. Smooth Particle Hydrodynamics Analysis of Hydrodynamic Lubrication with Free Surface Flow. *Tribol. Lett.* **2025**, *73*, 101. [CrossRef]
93. Keller, M.C.; Braun, S.; Wieth, L.; Chaussonnet, G.; Dauch, T.; Koch, R.; Höfler, C.; Bauer, H.-J. Numerical modeling of oil-jet lubrication for spur gears using smoothed particle hydrodynamics. In Proceedings of the 11th International SPHERIC Workshop, Munich, Germany, 13–16 June 2016; pp. 14–16.
94. Keller, M.C.; Braun, S.; Wieth, L.; Chaussonnet, G.; Dauch, T.F.; Koch, R.; Schwitzke, C.; Bauer, H.-J. Smoothed particle hydrodynamics simulation of oil-jet gear interaction1. *J. Tribol.* **2019**, *141*, 071703. [CrossRef]
95. Schnabel, D.; Özkaya, E.; Biermann, D.; Eberhard, P. Modeling the motion of the cooling lubricant in drilling processes using the finite volume and the smoothed particle hydrodynamics methods. *Comput. Methods Appl. Mech. Eng.* **2018**, *329*, 369–395. [CrossRef]
96. Ji, Z.; Stanic, M.; Hartono, E.A.; Chernoray, V. Numerical simulations of oil flow inside a gearbox by Smoothed Particle Hydrodynamics (SPH) method. *Tribol. Int.* **2018**, *127*, 47–58. [CrossRef]
97. Hassanzadeh Saraei, S.; Peters, B. Immersed boundary method for considering lubrication effects in the CFD-DEM simulations. *Powder Technol.* **2023**, *426*, 118603. [CrossRef]
98. Kim, H.; Park, S. Coupled Level-Set and Volume of Fluid (CLSVOF) Solver for Air Lubrication Method of a Flat Plate. *J. Mar. Sci. Eng.* **2021**, *9*, 231. [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.