CFD Modeling of Air Lubrication Systems in Ships

Roya Shademani[1], Tommi Mikkola[1], Teemu Manderbacka[1],
Sasan Tavakoli[2], Heikki Remes[1]
[1]Department of Mechanical engineering, Aalto University,[2] University of Melbourne, Australia
Roya.shademani@aalto.fi

1 Introduction

Maritime transportation dominates global trade, with oceangoing ships, including container vessels, bulk carriers, and tankers, handling most of the freight. However, these large, human-made vessels require substantial energy, primarily generated by diesel engines. This reliance on diesel engines has contributed significantly to CO2 emissions, prompting the International Maritime Organization (IMO) to mandate emission reductions across the shipping industry. To address this, two primary approaches are being explored: the adoption of alternative fuels, such as ammonia, and hydrodynamic retrofitting of ships. The latter includes innovations like Air Lubrication Systems (ALS) and wind-assisted propulsion technologies.

Hydrodynamic retrofitting solutions have garnered significant interest, driving companies and industries to develop new methods for implementing these technologies on ships. However, engineering the setup for each vessel must be tailored in advance, as each ship exhibits unique hydrodynamic behavior. Therefore, before installing retrofitting technologies, a thorough understanding of the related fluid dynamics is essential. This requires a hydrodynamic study of ships equipped with such systems, which can be performed either via numerical modeling of the air-water (+bubble) flow around the vessel or via physical experiments, such as towing tank tests. Unfortunately, our understanding of the flow dynamics around ships with air-lubrication systems—one of the most promising retrofitting solutions—remains limited, as research in this area has only gained momentum in recent years.

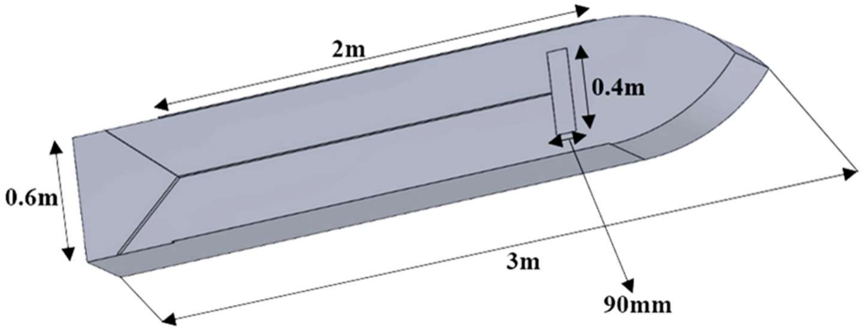
To better understand the problem and move towards numerical modeling, it is essential to first review the foundational literature, beginning with early studies from the 1970s. However, the Air Lubrication System was proposed a century ago, in 1907 Frederick W. Lanchester [1] explored how aerodynamics and fluid mechanics could be applied to watercraft to minimize resistance, laying the groundwork for later research on air lubrication systems. Most of these studies introduced the concept of drag reduction on wet surfaces through bubble generation. Initial experiments were conducted in tunnels or tanks, where symmetric submerged bodies, such as submarines, were tested to observe frictional drag reduction. This early research sparked more practical studies, where scholars tested ships equipped with air-bubble generators on their hulls in towing tank experiments. While experimental research in this field has provided valuable insights, it is limited in scope, and conducting new experiments for each design can be prohibitively expensive. This is why numerical simulations done using Computational Fluid Dynamics (CFD), have become favored for conceptualizing, and engineering such systems in the early stages. Although CFD modeling of fluid flow around ships without air lubrication has been extensively studied over the past decades, research on ships equipped with air lubrication systems (ALS) remains limited. Replicating this problem in a CFD environment is particularly challenging due to the complexity of the system and the various methods needed to solve it. Furthermore, few studies have transparently reported CFD simulations of ALS-equipped ships.

To address this gap, this article presents a CFD-based study on ships equipped with ALS, offering new findings that can assist other researchers in modeling this problem. The remainder of this paper is structured as follows: Section 2 introduces the ship model and provides a general overview of the numerical schemes used to solve the problem. Section 3 presents the results and compares CFD predictions against towing tank data. Section 4 presents concluding remarks.

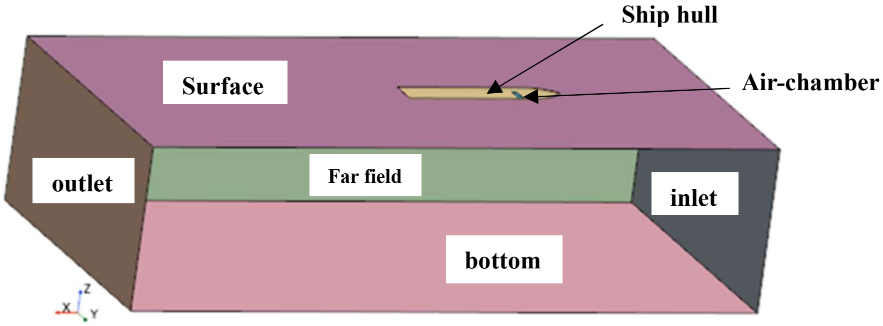
2 Methodology

2.1. Ship Model

In the present study, a three-dimensional ship model equipped with an air injection chamber at its bottom is considered, as shown in Fig. 1(a). Experimental results for the same case were reported in [2]. The fluid flow around the ship is simulated in a virtual towing tank via computational fluid dynamics. To do so, a computational domain with prescribed boundary conditions (BCs) shown in Fig. 1(b) is generated. A symmetry BC is prescribed for the upper surface, which cancels out effects of the free surface and wave making resistance. This helps us to single out the effects of air injection on the frictional drag. Inlet and outlet BCs are respectively set for front and back surfaces. These two surfaces are set to be L and $2L$ far from the ship to avoid the influence of these boundaries on the results, where L represents the length of the ship model.



(a)



(b)

Fig.1 (a) Geometry of ship model, (b) Numerical computational domain.

2.2 Numerical scheme

The fluid is assumed to be viscous and incompressible. We formulate the fluid motion equation using a Eulerian approach. In addition, we expect the fluid flow to be turbulent due to the relatively large Reynolds number and the expected fluid regime around the ship. In addition, we use a two-phase (air and water) fluid flow approach to model the problem. Hence, the flow field around the ship should satisfy the continuity and RANS (Reynolds-Averaged Navier Stokes Equation) equations as follows.

$$\nabla . \boldsymbol{V} = 0 \tag{1}$$

$$\frac{\partial(\rho_{eff}\boldsymbol{V})}{\partial t} + \nabla \cdot \left(\rho_{eff}\boldsymbol{V}\boldsymbol{V}\right) = -\nabla p + [\mu_{eff}(\nabla\boldsymbol{V} + (\nabla \cdot \boldsymbol{V})] + \rho_{eff}\boldsymbol{g} \tag{2}$$

Here, $\boldsymbol{V}$ is the averaged speed, and p is the averaged pressure, and $\boldsymbol{g}$ is the gravity acceleration. ρ_{eff} and μ_{eff} are the effective density and dynamic viscosity, respectively (as in each cell a mixture of air and water would exist). The air-water mixture flow is solved using a volume fraction method that introduces an extra conservative equation to the system of equations, which is not presented in this paper for the sake of brevity. A realizable k-epsilon two-layer turbulence model is used as a turbulence closure.

The fluid flow around the ship is simulated using the commercial package of Star-CCM+ (Version 2402).

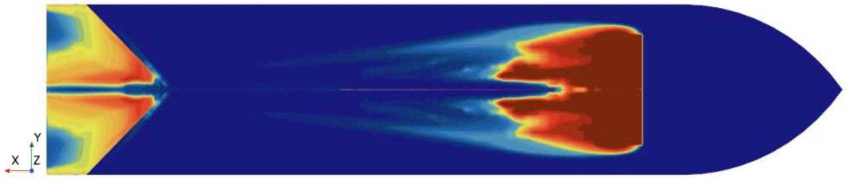
3 Results and Discussion

In this paper, we present three different sets of results. First, we illustrate the pattern of air distribution on the bottom of the ship hull to demonstrate how a realistic pattern can be achieved in CFD simulations. Next, we present the drag reduction observed during numerical simulations and compare the results against experimental data to validate the model. Finally, we show how the addition of appendages to the hull affects the CFD results.

3.1 Air Distribution Pattern

The distribution pattern of air emerging on the bottom of a ship following injection is a critical indicator that provides valuable physical insights into the problem, helping us understand how air injection contributes to drag reduction. Hence, air distribution on the bottom of the ship needs to be predicted accurately. The pattern of air distribution on the bottom of the ship is expected to form a continuous air layer, which can effectively reduce the drag. One question regarding the modeling of fluid flow around a ship under the effects of air lubrication is whether gravity needs to be considered to achieve a realistic pattern of air distribution on the bottom of the ship. So, we modeled the problem using two different settings: in one, gravity was turned off, and in the other, gravity was turned on. Fig. 2 shows the results. As seen, an incorrect and unrealistic air distribution pattern is obtained when gravity is turned off, while a realistic distribution is achieved after activating the gravity model.

The results also closely match those from the numerical simulations in [2]. This phenomenon occurs because, in the absence of gravity, there is nothing to drive the air towards the hull surface, so the air bubbles tend to collect near the air chamber, but the only exception is the low pressure area aft of the inclined arranged strips close to the stern of the model, and eventually the air injected disperses throughout the fluid domain rather than forming an air carpet. However, under the influence of gravity, the air bubbles generated beneath the ship are more likely to rise towards the bottom of the hull, partially drying the ship's hull. The strips close to the stern of the model are higher than the thickness of the air layer and block the transport of air towards the transom.



(a) No gravity model

(b) With gravity model

(c) Validation case [2]

Fig.2 Air distribution at the air flow rate of 8 m^3/h and ship speed of 0.542 m/s.

3.2 The calculated drag reduction

The effect of the air lubrication system on drag reduction is investigated at various air flow rates (ranging from 0 m^3/h to 10 m^3/h). The drag reduction is quantified as follows:

$$\Delta R = \frac{\Delta R_f}{R_0} \times 100\% \tag{3}$$

Here ΔR_f is the relative friction drag reduction in comparison to the case with no air-injection. R_0 is the total drag of the ship model without air injection.

The numerical and experimental results of the drag reduction rate at Froude number of 0.16 are compared in Fig. 3. Here, the drag reduction is plotted versus air flow rate for a complete ship model equipped with all strips configured identically to the experimental control case. As seen, the frictional drag would reduce up to 15% at very high injection rates, though it converges as the injection rate becomes relatively high. The results indicate that having a very large injection rate would just waste the energy and does not lead to further drag reduction.

We now turn our attention to the accuracy of the numerical model in calculating drag reduction. The numerical results seem to follow the experimental data, though they may slightly overpredict drag reduction at low injection rates. This can be explained by the nature of the numerical model. While the two-phase flow approach helps model the problem at these low rates, it does not fully capture the real physics. In the simulations, bubbles are not generated, and a homogenous mixture of air and water exits the chamber, whereas in real physical tests, bubbles do form. Consequently, the two-phase flow modeling results in a thicker air layer under the ship's hull, leading to a larger drag reduction than observed in the tank tests.

At higher injection rates, the numerical model does not appear to overpredict the results, which is a very promising observation. In these cases, the two-phase model yields a more realistic air injection. During physical tests, higher air injection rates resemble the air layer more than bubble formation. Therefore, the numerical model produces a similar air layer thickness and drag reduction to what is observed in real experiments.

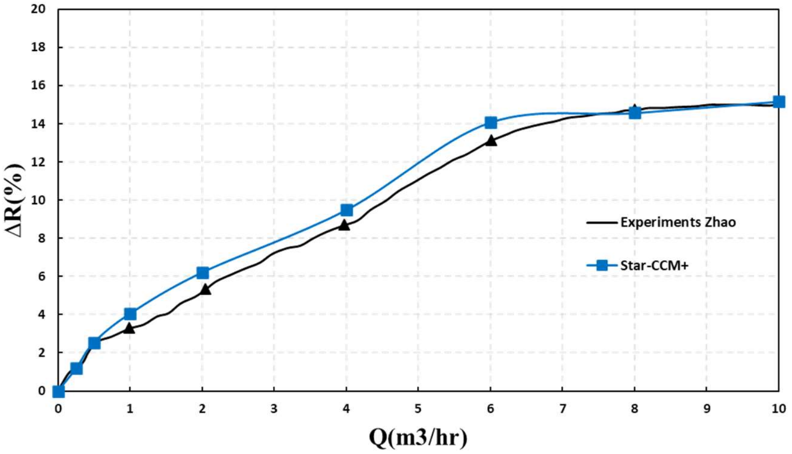


Fig.3 The comparison of drag reduction rates at different air flow rates.

3.3 Strips effect

We now present some results that provide physical insights into the setup used to increase the effects of air lubrication. When air is injected, our goal is to prevent its leakage from the sides and trap the air beneath the ship. From an engineering perspective, this is achieved by adding strips to the hull to control airflow and stop it from escaping. In the experimental tests conducted in [2], two side strips, one middle strip, and two inclined strips were installed on the ship model. We have run simulations both for a bare ship with no strips and for a ship equipped with these strips.

When these strips were added to the numerical model in the CFD tank, the drag reduction was found to be more significant. The installation of inclined strips, by facilitating the escape of air along the sides of the ship's hull, helps to make the friction drag more dominant and mitigate the effect of air on the pressure drag at the stern.

Figure 4 shows examples of the time history of shear drag developed on the hull surface of the ship under air injection, both for the bare hull and the hull equipped with strips. The results correspond to the Froude Number of 0.10 and the air flow rate of 10 m^3/h. As seen, the case with the ship equipped with strips exhibits lower shear drag.

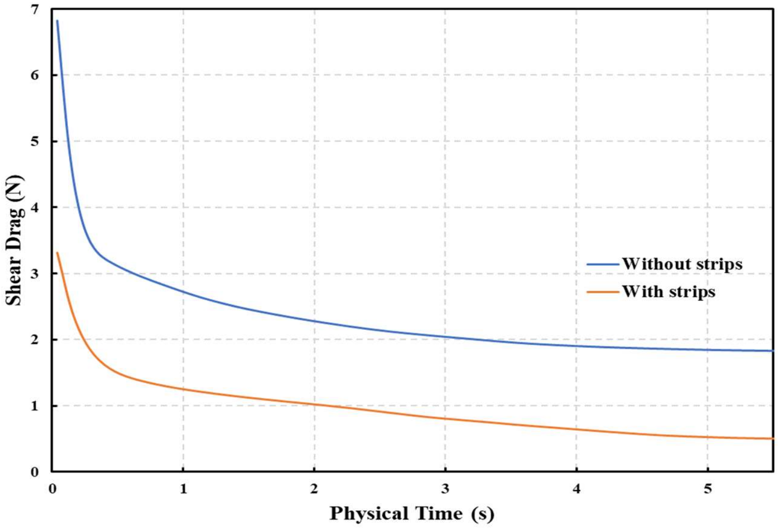


Fig.4 Impact of strip installation on shear drag at a flow rate of Q = 10 m³/h

4 Concluding remarks

This study presents a detailed CFD analysis of air layer drag reduction based on early CFD simulations of drag reduction for a flat-bottomed ship hull equipped with an air-lubrication system, conducted using Star-CCM. The fluid flow is modeled using a volume fraction method. The results indicated that the CFD model can predict the drag reduction up to 15% at the highest considered airflow, which is like what was measured at experimental tank tests. The simulation results closely aligned with experimental data, yet the model was seen to slightly over-predict drag at low injection rates.

Additionally, it was shown that activating gravity in CFD modeling of the problem allows for prediction of a more realistic pattern of air distribution on the bottom of a ship equipped with an air lubrication system. Additionally, it was demonstrated that when the problem is solved using the CFD model, strips need to be added to the ship hull to effectively address the influences of friction drag on the numerical results.

Future work will focus on using the Eulerian Multiphase (EMP) model for simulating the problem, providing a significant opportunity to compare the capabilities of EMP with those of the Volume of Fluid (VOF) method. This will also involve a numerical evaluation of the wave effects on the performance of air lubrication systems (ALS).

Acknowledgements

The authors acknowledge research funding received from the Horizons Europe project” RETROFIT solutions to achieve 55% GHG reduction by 2030 (RETROFIT55) – Project No.: 101096068”. The views set out in this paper are those of the authors and do not necessarily reflect the views of their sponsors.

References

[1] F. W. Lanchester, Aerodynamics, Constituting the First Volume of a Complete Work on Aerial Flight. London: A. Constable & Co., Ltd., 1907.
[2] Xiaojie Zhao, Zhi Zong, Experimental and numerical studies on the air-injection drag reduction of the ship model.