1st International Symposium on Air Lubrication Technologies (SALT'25),
Trondheim June 16th- 17th, Norway

CFD simulation of full-scale ship with active and passive air lubrication systems

Fengjian Jiang[1], Anders Östman[1], Kourosh Koushan[1]

[1] Ship and Ocean Structures, SINTEC Ocean AS, Trondheim, Norway NO-7052

INTRODUCTION

Energy efficiency in the maritime sector is receiving increasing attentions, with the dual challenges of high fuel costs and the need to reduce greenhouse gas emissions. Alternative fuels have been a hot topic, but their limited availability and higher cost still make them impractical in the near term. Therefore, measures that lower energy consumption, such as reducing the skin friction over a ship's hull, have become critical. Since skin friction can account for more than 70% of the total resistance on large seagoing vessels, innovative solutions that can reduce friction on ship's wetted surface are needed. One promising approach is air lubrication, where air is introduced into the boundary layer to either form a dispersed bubble field (bubble drag reduction) or a continuous film (air layer drag reduction) that effectively lowers the viscous force on the hull.

Over the last few decades, a significant number of studies have focused on the underlying physics governing the air lubrication techniques. Early studies revealed that injecting microbubbles into the turbulent boundary layer can reduce the effective density near the wall and mitigate vortex interactions, thereby lowering friction. Similarly, creating a continuous air film fundamentally alters the stress distribution on the hull by providing a low-shear interface between the ship hull and water. Many studies further emphasized that some key parameters, such as air flux rate, injection angle, and orifice size, play important roles in achieving optimal drag reduction. Experiments have been widely used to explore the air lubrication technique, while recent literature also shows stronger interest in CFD modelling of these multiphase interactions (see reviews by Ceccio (2010) and detailed study by Mäkiharju et al. (2017)). These studies offered valuable insights into the interplay between bubble dynamics and boundary layer behaviour. Building upon these works, the present study employs full-scale CFD simulations of a high-speed catamaran to investigate air lubrication effects, with both an active air lubrication system (feeding air actively) and a passive system (leveraging suction via an interceptor design).

CFD MODEL AND METHOD

The target full scale vessel for this study is MS Fjordøy, a high-speed catamaran that runs a route in the fjord of Norway's west coast. MS Fjordøy is a fully electrical vessel, with an overall length of 24m and 7m beam, allowing maximum 89 passengers. MS Fjordøy has an air lubrication system installed. Due to its small size and high operating speed, a passive air lubrication system was chosen over the active type. For the CFD study, however, both active- and passive air lubrication setups are considered.

Full scale CFD simulations were carried out using the commercial code StarCCM+. For the present study, calm water resistance simulations were conducted. The simulations are multiphase simulations, with the volume of fraction (VOF) method handling both the free surface and the air lubrication under water. To model turbulence, the simulations used two equation Reynolds Averaged Navier Stokes (RANS) model, k-ω SST. Gamma-ReTheta transition model in the k-ω SST formulation is included. Wall function was used, targeting at surface Y-plus of about 50. The simulation includes only half of the catamaran hull, assuming that the flow is symmetric around the mid-ship. The ship has 2 degrees of freedom, i.e. heave and pitch. The DFBI model in StarCCM+ using mesh morphing handles the dynamic motions during the simulation until the equilibrium position is achieved. The simulations are unsteady and used 1st order implicit scheme for time marching. The mesh is hexahedra trimmer mesh. The mesh parameters and refinements, computational domain setup, boundary conditions, VOF parameters and time step choice all follow SINTEF Ocean's best practices that were validated through hundreds of different ships through years' of experiences in projects. and the total number of cells for a typical calm water resistance simulation is around 10 million.

[1] Ceccio, S.L. Annu. Rev. Fluid Mech. 2010,42, 183–203.
[2] Mäkiharju, S.A.; Lee, I.-H.R.; Filip, G.P.; Maki, K.J.; Ceccio, S.L. J. Fluid Mech.2017,818, 141–183.

ACTIVE AIR LUBRICATION SYSTEM

The active air lubrication system is particularly suited for large sea-going vessels since it relies on a dedicated infrastructure that pressurizes and feeds air beneath the hull. Although such systems are more complex in real-world applications, their CFD implementation in calm water scenarios is straightforward: by simply prescribing air-phase inlet boundary conditions at designated hull locations, we can simulate active air injection at specific flow rates.

In this study, three ship speeds (16, 20, and 24 knots) were investigated. For each speed, four to five different air flow rates, reflected in the CFD model as the air inlet velocity (Va), were examined to understand how the ship speed and air flux influences drag reduction. At 16 knots, the simulations indicate an approximate 3% reduction in drag, largely independent of Va values in the range of 1–4 m/s. However, at higher speeds the dependency on air flow rate becomes clearer. At 20 knots, a low injection speed of 1 m/s yields about 2.8% drag reduction, which increases to approximately 6.2% at 4 m/s. At 24 knots, drag reduction reaches a promising 8% when Va is increased to 5 m/s. It is also noteworthy that the implementation of the active system exerts minimal influence on the position (heave and trim) of the vessel.

PASSIVE AIR LUBRICATION SYSTEM

Passive air lubrication is less complex in real-world applications yet poses additional challenges in CFD modelling due to the necessity of accurately representing the internal air path responsible for suction. In a passive system, the volume of air drawn beneath the hull depends on the shape and location of the interceptor and the ship's operational speed. An effective CFD implementation must reasonably represent the actual internal system installed on ship, while also ensuring a clear pathway for air to be drawn from above the deck to the underwater region. Certain parts may not be the same as in the real ship, such as the air intake and container sizes, they are sized to avoid bottlenecks in air delivery. However, the lower part of the implementation replicates as closely the real system as possible to ensure the air is drawn in a similar manner as in real operation.

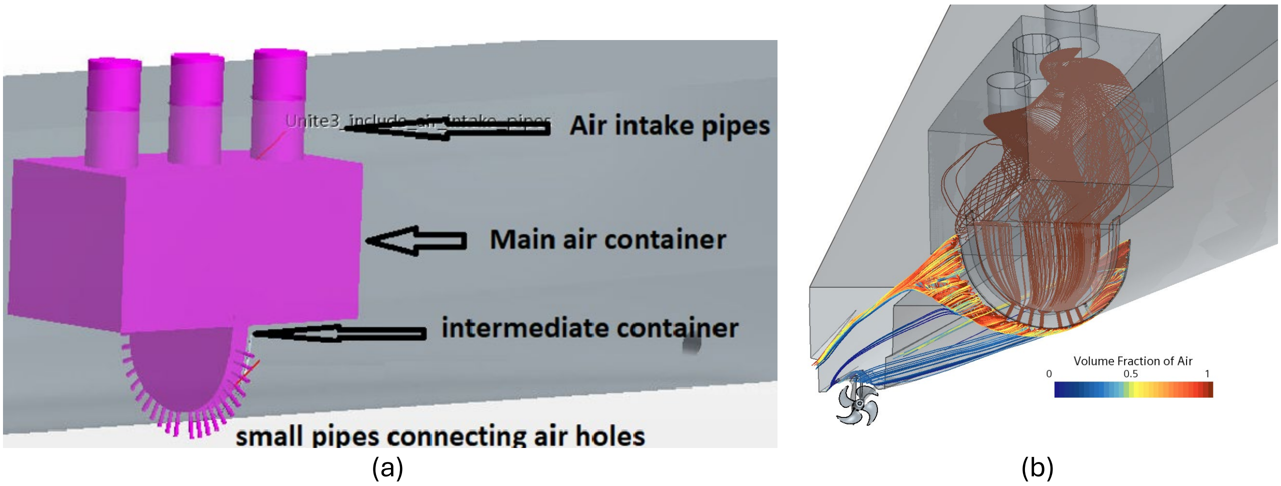

Figure 1 CFD setup for passive air lubrication system. (a): Sketch of the internal system. (b) Illustrative results of how the air is sucked in through the internal system and form an air layer beneath the ship hull. (Note that the propeller in (b) is only for illustrative purposes, the calm water simulation results discussed in this paper does not include propeller).

Figure 1 illustrates the detailed CFD implementation of the passive system used in this study. Special attention has been paid to mitigating any unintended effects on the vessel's dynamic heave and trim during simulation, particularly addressing numerical challenges associated with the transient motion as air is drawn through the openings. Simulation results demonstrate a promising drag reduction of about 8% at a ship speed of 24 knots, compared to a baseline configuration with the interceptor installed but without active air lubrication.

Acknowledgement: The research was performed under the project OnAir (317869) supported by the Norwegian Research Council and industrial partners Jotun, GS Marine Produksjon, Rødne, Norled, Kumera and Servogear.

[1] Ceccio, S.L. Annu. Rev. Fluid Mech. 2010,42, 183–203.
[2] Mäkiharju, S.A.; Lee, I.-H.R.; Filip, G.P.; Maki, K.J.; Ceccio, S.L. J. Fluid Mech.2017,818, 141–183.