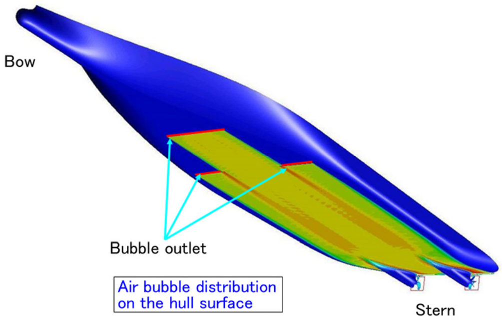
CFD Predictions of Bubbly Flow around an Energy-saving Ship with Mitsubishi Air Lubrication System



MAKOTO KAWABUCHI*1 CHIHARU KAWAKITA*2

SHUJI MIZOKAMI*3 SEIJIRO HIGASA*3

YOICHIRO KODAN*1 SHINICHI TAKANO*1

The Mitsubishi Air Lubrication System (MALS) was the first air lubrication system in the world to be applied to a newly built ship, and resulted in a substantial reduction in the ship's resistance. Therefore, a performance estimation method using computational fluid dynamics (CFD) needs to be established as soon as possible to apply the MALS to general commercial ships. In this study, we predicted the bubble distribution around ships with the MALS using CFD, and developed a method to determine the reduction in flow resistance based on the bubble coverage around the hull. Furthermore, we also predicted the intrusion of bubbles on the area of propeller disks, which could deteriorate the performance, and confirmed that the deterioration in propeller disk performance was negligible.

1. Introduction

The development of energy-saving ships has been greatly anticipated by the shipping industry as a countermeasure against the surging prices of raw materials, including oil, arising from the economic growth of developing countries, and environmental issues such CO_2 emission regulations for international shipping operations. The air lubrication method, which reduces the resistance of the hull by using air bubbles, has been studied by a number of institutes because the method is expected to result in prominent energy-saving effects. Kodama et al.[1] performed tests using a flat-plate model ship with a total length of 50 m and confirmed that the total resistance working on the model ship and the local frictional force working on the ship bottom were reduced by bubbles. Verifications on actual ships have also been conducted; Kodama et al.[2] demonstrated an energy-saving effect of 5% in an actual ship test using a cement carrier.

Research on the use of computational fluid dynamics (CFD) to predict the effect of bubbly flow has also been promoted. Murakami et al.[3] simulated the flow around the cement carrier used by Kodama et al.[2] for experiments, and evaluated the effects of changes in the ship posture and location of the bubble outlets on the resistance reduction ratio for the ship and the void fraction on the propeller disk area. The Nagasaki Shipyard & Machinery Works of Mitsubishi Heavy Industries, Ltd., (MHI) completed YAMATAI, a module carrier belonging to the NYK-Hinode Line, Ltd., in April 2010. An air lubrication system was installed on a ship for the first time on this occasion. The ship achieved an energy-saving effect of more than 10% at sea trials prior to delivery.

While developing the Mitsubishi Air Lubrication System (MALS) installed on YAMATAI, air bubble predictions utilizing CFD technology were performed in addition to tests on model and actual ship. The tests on a model ship confirmed the flow of bubbles along the bottom of the ship and were used to evaluate the effect of the void fraction on the propeller disk area on the propeller characteristics and fluctuating pressure (Takano et al.[4]). We used CFD with the same model-scale ship to predict the distribution of the air bubble void fraction on the hull surface, which is required to predict the reduction of the hull resistance, as well as the distribution of the void fraction on the

*1 Nagasaki Research & Development Center, Technical Headquarters

*2 Research Manager, Nagasaki Research & Development Center, Technical Headquarters

*3 Engineering Manager, Ship & Ocean Engineering Department, Shipbuilding & Ocean Development Headquarters

propeller disk area, which affects the propeller performance. The results of these predictions are reported in this paper.

2. Prediction of the Air Bubble Distribution on the Hull Surface

The air bubble distribution around the hull surface is believed to be an important parameter for reducing the resistance working on the hull, and must therefore be predicted accurately. The predicted void fraction distribution due to air bubbles on the hull surface is reported in this chapter. Here, the void fraction is the ratio of the air volume to the air–fluid mixture.

2.1 Ship specifications and flow conditions

The ship specifications are shown in **Table 1**. The ship was a twin-screw vessel characterized by its wide breadth and shallow draft. The calculation results reported here are based on a model ship navigating in a straight line using double model approximation without considering waves on a free surface. The calculations were performed on the port side only based on the line of symmetry along the hull centerline.

All of the air bubbles were assumed to be of a uniform diameter and remain unchanged by the flow. No consideration was given to bonding of bubbles or division of a bubble into multiple bubbles. The bubble diameters indicated in **Table 2** were used for calculations. The bubble diameter for actual ships was assumed to be 2 to 3 mm. The diameters examined in Cases 2 and 3 were set to 5 and 10 times the value of Case 1, respectively, to provide an approximate indication of the effect of the bubble diameter.

Bubble outlets were created at three locations along the bottom of the hull, symmetrically on both sides of the centerline, as shown in **Figure 1**. The bubble outlets were created at two locations, one near the front and the other near the rear, because the calculations were performed on the port side only in this study. A velocity boundary was created at the bubble outlet; the air was blown at a constant flow rate to form the bubbles.

Table 1 Specifications of YAMATAI

Ship		YAMATAI (Module Carrier)
Length	Loa	162m
Breadth	B	38m
Depth	D	9.0m
Draft	d	4.5m / 6.34m
Deadweight	-	Approximately 19,500t
Design speed	U	13.25kt
Model Scale		1/27.7

Table 2 Air bubble dimensions

	Model Ship (mm)	Actual Ship (mm)
Case 1	0.1	2.77
Case 2	0.5	13.8
Case 3	1.0	27.7

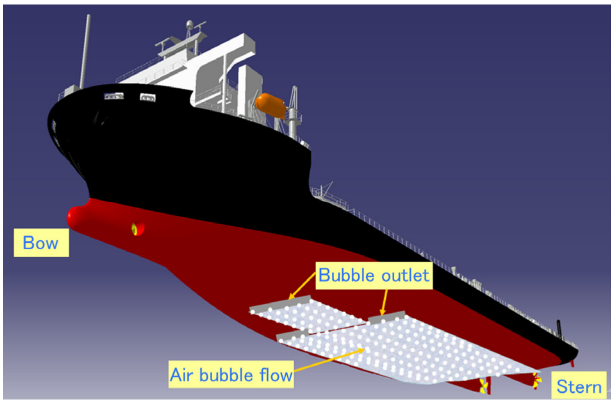


Figure 1 Image of the air lubrication system
The bottom of the ship is covered by air bubbles released from the bubble outlets.

2.2 Comparison of calculations and test results

The calculated void fraction distributions from the air bubbles on the hull surface are shown in **Figure 2** for Cases 1 to 3. The contours indicate the void fraction distribution. A higher void fraction corresponds to a higher ratio of air per unit volume. The calculations indicated that the influence of the air bubble diameter on the air bubble distribution was limited. Therefore, although actual air bubble diameters can vary, the bubble effect can be roughly predicted from calculations based on bubbles with a particular uniform size.

The calculated results using 0.1-mm bubbles are compared to the experimental data of Takano et al.[4] in **Figure 3**. Both sets of data correspond to an ordinary water level and do not take into consideration hull motion other than a straight, forward movement. A quantitative evaluation of the air bubble distribution was difficult; therefore, the predicted void fraction distributions were qualitatively compared with air bubble distribution images obtained from the experiments.

The following trends were observed in both the predictions and the experiments.

- Air bubbles flowed along the hull without escaping from the bottom of the ship.
- A smaller amount of air bubbles were distributed in the area near the hull centerline.
- Air bubbles were concentrated in the area around the skeg that covered the propeller shaft on the stern side.

We believe that an appropriate air bubble distribution can be reproduced based on these results. Consequently, the approximate reduction in ship resistance can be predicted based on the air bubble coverage ratio on the hull surface for an experimental tank model.

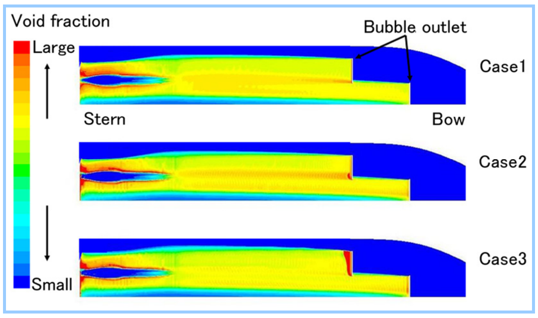


Figure 2 Distribution of the void fraction on the hull surface
The port side of the bottom of the ship as viewed from below. The contours indicate the void fraction.

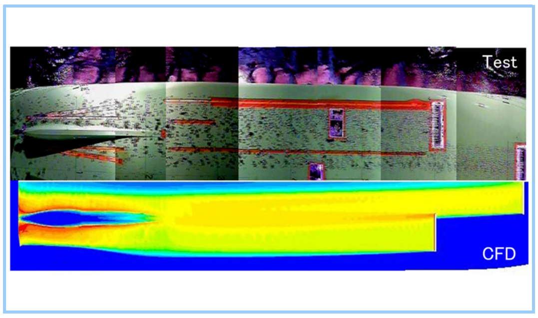


Figure 3 Comparison of the void fraction distribution
Top: Experimental model test results
Bottom: CFD results

3. Predicted Air Bubble Distribution Flowing into the Propeller Disk Area

The air lubrication method, which reduced a ship's resistance by using air bubbles, may raise concerns about the efficiency, vibration, and noise of the propellers caused by bubbles flowing into the propeller disk. Takano et al.[4] evaluated the influence of air bubbles on the characteristics and fluctuating pressures of a propeller by changing the void fraction of air bubbles flowing into the propeller disk area. If the void fraction of air bubbles flowing into the propeller can be predicted precisely by CFD, their influence on the propeller performance can be confirmed by using numerical data. In this chapter, the effects of bubbles on propeller performance were evaluated by targeting module carriers.

3.1 Flow field prediction on the propeller disk area

The module carrier had reaction fins (RFs) in front of the propeller. Since the flow field was anticipated to be complex, the flow on the propeller disk area was predicted first before examining the amount of air bubbles flowing into the propeller disk area.

Three blades of RFs were modeled on the outer side of the skeg and angled at 45, 90, and 135 degrees, respectively, as seen vertically from above.

(1) Propeller disk area flow field prediction results

The wake contour / velocity vector distributions on the propeller disk are compared with test results in **Figure 4**.

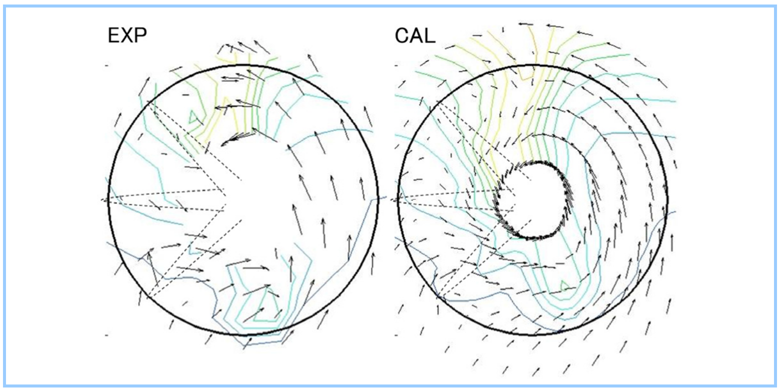


Figure 4 Comparison of the flow fields on the propeller disk area
The contours indicate the flow velocity distribution in the longitudinal direction, and the vectors indicate the width and height velocity vectors. The dotted lines indicate the reaction fins located on the upstream side.

(2) Evaluation of the flow field prediction results

The RFs positions are indicated by dotted lines in Figure 4. The contour distributions and vectors were nearly identical. The vector was smaller on the left side of the propeller shaft and larger on the right side in both the predictions and the experiment, and the flow directions were similar.

As shown in Figure 4, the predicted velocity distribution of flow toward the propeller disk area was approximately the same as that in the experimental results. For this reason, we believe that the flow of air bubbles out of the bubble outlets, which were located on the bottom of the bow of the ship, toward the propeller disk area can also be accurately predicted by CFD.

3.2 Prediction of the void fraction distribution on the propeller disk area

Calculations were performed for bubbles flowing along the ship bottom to predict the amount of bubbles flowing into the propeller disk area.

(1) Calculation conditions

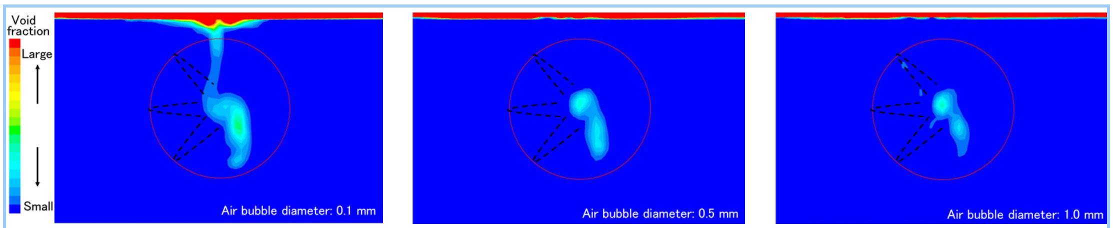
The basic flow conditions for the calculations were the same as those used in Section 2.1. The calculations were performed for each case indicated in **Table 2** to evaluate the effect of the air bubble diameter.

(2) Calculation results

The air bubble void fraction distribution on the propeller disk area predicted using 0.1-, 0.5-, and 1.0-mm air bubbles are shown in **Figure 5**, respectively. In each figure, the propeller disk area is indicated by a red circle. The top end of the figure is the still water surface. The area mean of the void fraction of air bubbles flowing into the propeller disk area is indicated in **Table 3**.

Table 3 Average void fraction of air bubbles flowing into the propeller disk area

Bubble Diameter (mm)	Mean Void Fraction (%)
0.1	0.030
0.5	0.017
1.0	0.014



(a) Air bubble diameter: 0.1 mm (b) Air bubble diameter: 0.5 mm (c) Air bubble diameter: 1.0 mm

Figure 5 Void fraction distribution prediction on the propeller disk area
The red circle indicates the propeller disk. RFs are on the upstream side of the dotted line.

(3) Influence of air bubble on propeller performance

The amount of air bubbles flowing into the propeller disk area decreased as the air bubble diameter increased, as indicated by Figures 5. This may have been due to the buoyancy force, which increased with the air bubble diameter. However, the peak position of the void fraction did not vary substantially with the air bubble diameter.

Figure 6 shows the measured propeller characteristics with air bubbles near the operation point (Slip = 0.4) obtained during an experimental test performed by Takano et al.[4] In this test, different amounts of air bubbles were released from the area in front of the propeller, and circular bubble clusters dispersed and traveled into the center of the propeller and the area above the propeller (approximately 0.7 R, where R is the propeller radius). These results illustrate influence of changes in the void fraction on the propeller disk in terms of the propulsion force, torque, and propeller efficiency under these conditions. Judging from the average void fraction distribution on the propeller disk obtained by calculations and the results shown in Figure 6, the propeller efficiency of this ship remained nearly unchanged by the bubbles.

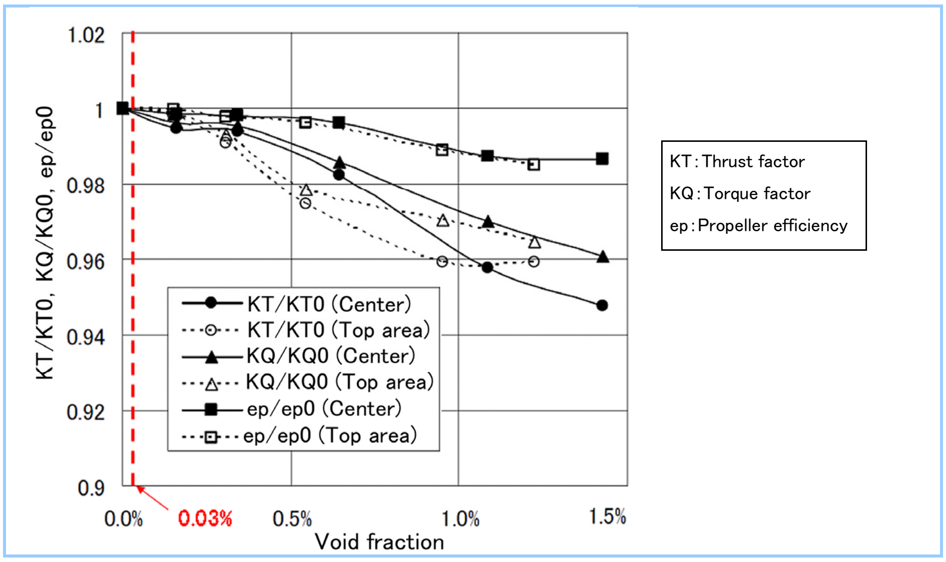


Figure 6 Relationship between the average void fraction on the propeller disk area and KT, KQ, and ep (Slip = 0.4)
The graph shows the amount of air flowing into the propeller and the decrease in propeller performance.

4. Conclusion

The air bubble distribution on the hull surface of a ship with the Mitsubishi Air Lubrication System (MALS) and the amount of air bubbles flowing into the propeller were roughly predicted using a model-scale analysis. The results confirmed that the air bubble distribution on the ship bottom surface varied little in response to changes in the air bubble diameter. The experimental results were qualitatively similar to the air bubble distribution predicted by CFD. We confirmed that changes in the bubble diameter did not affect the peak position of the void fraction on the propeller disk area, while the void fraction of air bubbles flowing into the propeller increased as the air bubble diameter decreased. Comparison of the calculated and experimental results confirmed that the loss of propulsive efficiency due to air bubbles was negligible because the air bubbles flowed along the ship bottom toward the area above the propeller.

However, the resistance reduction mechanism of the air lubrication method have not yet been thoroughly examined, including the causes and effects of changes in fluid density and the turbulence modulation effects of air bubbles inside the boundary layer. CFD will play an important role in determining these causes by providing a detailed understanding of the physical phenomena.

References

1 Kodama, Y. et al., Experimental Study on Microbubbles and their Applicability to Ships for Skin Friction Reduction, International Journal of Heat and Fluid Flow, Vol.21, Issue 5 (2000) p.582

2 Kodama, Y. et al., A Full-Scale Air Lubrication Experiment Using a Large Cement Carrier for Energy Saving (Result and Analysis) , Conference Proceedings, the Japan Society of Naval Architects and Ocean Engineers, Vol.6 (2008) p.163

3 Murakami, A. et al., Numerical Simulation of Flow Around a Full Scale Ship Equipped with Bubble Generators, Conference Proceedings, the Japan Society of Naval Architects and Ocean Engineers, Vol.6 (2008) p.153

4 Takano, S. et al., Experimental Investigation of the Bubble Diameter of Injected Air on the Ship Bottom and Its Influence on Propeller, Conference Proceedings, the Japan Society of Naval Architects and Ocean Engineers, Vol.10 (2010) p.455