Two-Fluid Modeling of Bubbly Flows around Surface Ships Using a Phenomenological Subgrid Air Entrainment Model

Jingsen Ma, Assad A. Oberai, and Mark C. Hyman
Donald A. Drew, Richard T. Lahey, Jr

Abstract

The quantitative prediction of bubbly flow around a maneuvering surface ship is critical in determining its hydrodynamic performance and its acoustic and optical signatures. It is a challenging multiscale problem that relies heavily on subgrid models of turbulence and air entrainment. In this manuscript we analyze this problem using a phenomenological air entrainment model that predicts the location and rate of air entrainment around a surface ship. This subgrid model was coupled with a two-fluid Reynolds averaged Navier Stokes (RaNS) bubbly flow model and used to evaluate the flow field around a naval surface ship in straight ahead and turning motions. For straight ahead motion the predicted void fraction distributions aft of the stern were compared with experiments at three different ship speeds and good agreement was found. The qualitative differences in the location of the air entrainment and the resulting bubbly flow between straight ahead and steady turning motions were discussed and compared with experimental observations at sea. To our knowledge this study presents the first quantitative numerical prediction of void fraction distributions around a full-scale surface ship, well matching the experimental measurements.

Key words: subgrid air entrainment model, bubbly flows, multiphase simulation, ship hulls.

1 Corresponding author. E-mail: oberaa@rpi.edu, Phone:+1 518-276-3386, Fax:+1 518-276-4886

Preprint submitted to Elsevier 8 September 2011
1 Introduction

A maneuvering surface ship at sea typically entrains large quantities of air due to breaking bow waves, unsteady transom flows and wave/structure interactions along the hull. This is evidenced by the “white water” wake that extends many ship lengths behind the stern. Bubbly flow around a naval surface ship is important because it determines its acoustic and optical signatures and may alter its drag characteristics. Thus the simulation and prediction of air entrainment and the resulting bubbly flow around and behind a surface ship is of great interest in naval and marine hydrodynamics.

Over the last few decades significant progress (e.g., [1-3]) has been made in predicting bubbly flows around surface ships by utilizing the two-fluid models described in [4] and [5]. Some other interesting efforts on directly solving two-phase equations for ship-hull flows has also been made recently[6]. However, to date quantitative comparisons with experimental data have not been feasible, partly due to the lack of a robust and accurate subgrid air entrainment model. Clearly the location and rate of air entrainment play a critical role in determining the accuracy of a multiphase simulations. Moraga et al. [3] have proposed a subgrid air entrainment model and applied it to simulating the flow around a surface ship with some success. This model assumed that bubble sources are present everywhere close to the free surface and they are entrained once the local downward liquid velocity exceeds the bubble rise velocity. However, this model does not account for the fact that if the free surface moves downward with the same velocity as the liquid, no bubbles are entrained. Furthermore, it only provides an expression for the location of air entrainment and not its rate, which must be chosen empirically. Several models for determining air entrainment rate have also been proposed for canonical flows such as plunging liquid jets (e.g., [7–9]) and hydraulic jumps (e.g., [8]), and they can be used in select regions around a surface ship, but none that can capture all the bubble sources simultaneously.

Recently, we have developed a subgrid air entrainment model [10,11] that assumes that the turbulence in the liquid produces a rough air/liquid interface with cavities of a certain size. These cavities may be entrained into the liquid once the downward liquid velocity exceeds the downward velocity of the interface. This leads to a simple expression for the location and rate of air entrainment that is proportional to the liquid’s local turbulent kinetic energy times the gradient of the liquid velocity in the downward (i.e., pointed away from the interface and into the liquid) direction. This model has been implemented in a two-fluid framework [4,5] for bubbly flows and applied to the canonical problems of a plunging liquid jet and a hydraulic jump. In both cases this air entrainment model gave accurate results [10,11]. In this manuscript we apply the model to the two-phase flow around the research vessel (RV) Athena.
We consider both straight ahead motion and a steady turn. For the former case we compare our predictions with at-sea measurements of void fraction. For the latter no such data is available but we are able to make qualitative comparisons between the two motions with aerial photographs from a sea test.

The format of the remainder of this manuscript is as follows. In Section 2 we describe the RaNS two-fluid model and the sub-grid air entrainment model that were used in this study. In Section 3, we present simulations of bubbly flows around the RV *Athena*, in both straight ahead and steady turning motions. We discuss the results and compare the predicted void fraction profiles with the available experimental data. We end with Conclusions in Section 4.

2 Multiphase Modeling Framework

In this section we summarize the derivation of the subgrid air entrainment model and describe its implementation within a RaNS-type, two-fluid, computational multiphase fluid dynamics (CMFD) code. For simplicity, we restrict our simulations to mono-dispersed bubbles and to one-way coupling. We model the fluid as composed of a continuous liquid phase and a dispersed gas phase comprised of bubbles of a given diameter. This diameter was selected to be the typical bubble diameter measured in the experiments. Readers interested in the effects of two-way coupling and polydisperse modeling are referred to the related work of Moraga et al. [3] and Ma et al. [12].

2.1 Mass conservation of the dispersed phase

Due to the assumption of momodispersed bubbly flow, bubble coalescence and air dissolution play no role. In this case, the conservation equation for the bubble number density, $N''_g$, for bubbles of a characteristic diameter, $D_g$, moving with a velocity of $u_g$, often referred to as the population balance equation, is given by [13]:

$$\frac{\partial N''_g}{\partial t} + \nabla \cdot \left( u_g N''_g \right) = E_g$$  \hspace{1cm} (1)

where $E_g$ is rate at which bubble density increases due to air ingestion and is determined by the subgrid air entrainment model described later. Note that other sources of bubbles on the right hand side of Eq. (6), such as those due to bubble breakup and coalescence, can also be included if appropriate.

The subgrid air entrainment model used here may be briefly derived as follows
The interested reader is referred to [10,11] for a detailed derivation. Consider a gas/liquid interface, Γi, with roughness and air cavities of height ≈ a (see Fig. 1). In addition, Γ is an imaginary surface below the interface at a distance a/2. We require that Γ moves with the same downward velocity as the interface. Thus any air that passes through Γ is entrained. The average volume of air passing through Γ per unit area and time is then given by:

\[ q = C_1 \langle u_n(a/2) - u_n(0) \rangle \]
\[ \approx C \langle \bar{u}_n(a/2) - \bar{u}_n(0) \rangle \]
\[ \approx \frac{C}{2} \langle \partial \bar{u}_n / \partial n \rangle (0) a \]  

where \( u_n \) is the downward liquid velocity at the interface, an overbar indicates an ensemble average, and,

\[ \langle f \rangle = \begin{cases} f, & f > 0 \\ 0, & f \leq 0 \end{cases} \]  

A simple approximation for the interface’s roughness is [14]: \( a \approx u^2 / g = k / g \), where \( k \) is the local turbulent kinetic energy and \( g \) is the acceleration due to gravity. Thus, Eq. 2 becomes:

\[ q = \frac{C}{2} \langle \partial \bar{u}_n / \partial n \rangle \frac{k}{g} \]  

Based on this result, the equivalent air sources per unit volume of water in a liquid layer of thickness \( \phi_{\text{ent}} \) is given by:

\[ Q = \frac{q}{\phi_{\text{ent}}} = \frac{C}{2} \langle \bar{u}_n \rangle \frac{k}{\phi_{\text{ent}} g} \]  

Thus the air source term for the number density of bubbles per unit volume of water is:

\[ \mathcal{E}_g = \frac{Q}{\bar{v}_g} = C_{\text{ent}} \langle \partial \bar{u}_n / \partial n \rangle \frac{k}{\phi_{\text{ent}} \bar{v}_g g} \]  

where \( \bar{v}_g \) is the volume of an average bubble, \( C_{\text{ent}} = \frac{C}{2} \) is the model coefficient that depends on the physical properties of the liquid and the gas in the multiphase flow. Here we have assumed that the flow is mono-dispersed, that is all the bubbles are of the same diameter \( D_g \).
2.2 Momentum balance of the dispersed phase

The ensemble-averaged balance of momentum equation for the dispersed phase is given by (see Moraga et al. [15]):

\[ \frac{\partial (\bar{v}_g N'' \rho_d \mathbf{u}_g)}{\partial t} + \nabla \cdot (\bar{v}_g N'' \rho_d \mathbf{u}_g \otimes \mathbf{u}_g) = \bar{v}_g N'' \rho_d \mathbf{g} + \mathbf{M}'_g \]

where \( \bar{v}_g \) is the volume of a bubble with diameter \( D_g \), \( \rho_d \) is the dispersed air bubble’s density, and \( p_c \) is the pressure of the continuous liquid phase. Note that \( \mathbf{M}'_g \), the fluctuating interfacial force density, must be constituted. This term determines momentum transfer between the continuous phase and the dispersed phase. It may be partitioned to account for different types of interactions,

\[ \mathbf{M}'_g \approx \mathbf{M}^D_g + \mathbf{M}^L_g + \mathbf{M}^{TD}_g + \mathbf{M}^{VM}_g + \mathbf{M}^W_g \]

where the right hand side contains contributions due to drag (D), lift (L), turbulent dispersion (TD), virtual mass (VM) and wall-induced forces (W). For the specific expression for each term and the constants involved, all are the same with the work of Hyman et al. [2] and Moraga et al. [3], and are not repeated here.

2.3 Conservation laws for the continuous phase

It is assumed that the continuous liquid phase is incompressible and is not affected by the dispersed phase (i.e., one-way coupling). As a result the continuity equation for the liquid phase simplifies to,

\[ \nabla \cdot \mathbf{u}_c = 0. \]

The ensemble-averaged balance of linear momentum for the continuous phase is [3]:

\[ \frac{\partial \rho_c \mathbf{u}_c}{\partial t} + \nabla \cdot \rho_c \mathbf{u}_c \otimes \mathbf{u}_c = \nabla \cdot (\mathbf{T}_c + \mathbf{T}^{Re}_c) + \rho_c \mathbf{g} \]
Where \( T_c = -p_c I + 2\mu_c D_c \) is the Cauchy stress for a Newtonian fluid, and \( T_c^{Re} \) is the Reynolds stress tensor, which was modeled as:

\[
T_c^{Re} = -\left(\frac{2}{3}\rho_c k\right) I + 2\mu_t D_c,
\]

In the expressions above \( I \) is the identity tensor, \( \mu_c \) is the liquid’s viscosity, \( D_c \) is the rate of strain of the liquid phase, and \( k \) and \( \mu_t \) are the local turbulent kinetic energy and the turbulent viscosity, respectively. In this work, a blended \((k - \epsilon)/(k - \omega)\) turbulence model developed by Menter [16] was used to determine \( k \) and \( \mu_t \), and hence to construct \( T_c^{Re} \). All model coefficients were given by Menter [16]. For further details on the implementation of this model, the reader is referred to the previous work of Moraga et al. [3].

### 2.4 Modeling the free surface

The free surface of the air/water mixture is represented using a single-phase level set function, \( \phi \), described by Sussman et al. [17], which represents the signed distance from the free surface, with the level set \( \phi = 0 \) representing the free surface. Its evolution is governed by:

\[
\frac{\partial \phi}{\partial t} + u_c \cdot \nabla \phi = 0
\]

For more details on the application of the single-phase level set method to bubbly flow simulations the reader is referred to the work of Carrica et al. [18] and Moraga et al. [3].

### 3 Simulation of bubbly flow around the RV *Athena*

Using the methodology described in the previous section we have performed simulations of bubbly flows around the hull of the RV *Athena* which is operated by the Carderock Division of the US Navy Naval Warfare Center. Simulations of RV *Athena* have been previously used by Hyman et al. [2] and Moraga et al. [3] as a test case for verifying CMFD models. We first simulate the straight ahead motion of *Athena* at three different ship speeds and compare the void fraction profiles aft of the transom stern with the experimental measurements reported by Terrill et al. [19]. Then we simulate the flow around *Athena* during a steady turn. Some interesting differences between the straight ahead and steady turning motions are observed and discussed.
3.1 Straight ahead ship motion

We modeled one-half of the ship by assuming symmetry about the hull and ignoring any asymmetrical instabilities in the flow. The grid consisted of $3.7 \times 10^6$ cells with one and two levels of nested Chimera overset grids that were used to resolve the hull appendages (i.e., the skeg and rudder). This grid emphasizes a heavily refined overset block in the near wake region but not in the bow region since our focus was on the transom flows for which experimental measurements were reported [19]. The reader is referred to the work of Hyman et al. [2] and Moraga et al. [3] for further details about the grid and boundary conditions employed in our numerical simulations.

We have simulated the straight ahead motion of Athena at 6, 9 and 10.5 knots, which corresponds to Reynolds numbers of $1.46 \times 10^8$, $2.18 \times 10^8$ and $2.55 \times 10^8$, respectively, and Froude numbers of 0.14, 0.21 and 0.25, respectively. These numbers are based on the ship’s speed and length, $L = 47.2m$. Fig. 2(a) presents a snapshot of free surface elevation around the ship hull at the speed of 10.5 knots. From it, we can very clearly see the peaks and troughs of bow waves induced by the interaction between the flow and the bow. The most striking phenomenon is the formation of ship wake behind the stern. Close to the contact line of ship stern, it is seen that the free surface first slightly falls down, then suddenly rises up within a region of a width close to that of the stern. The width of the wake increases as it transports downstream, due to the effects of diverging waves starting from the two edges of transom stern. These are further demonstrated in Fig. 2(b), which plots the profile of time averaged free surface elevation along transverse distance to the ship center at 2m and 4m aft of the transom. In the plot, it is also shown the predicted results by a finer mesh consisting of $6.1 \times 10^6$ cells, which has twice resolution in the overset block for the wake region. We see negligible difference, which tells that the mesh used is fine enough to reach a RANS solution of grid-convergence. Fig. 3 displays the turbulent kinetic energy and streamwise velocity in the ship wake. It is found that some turbulence was developed along the side walls of the ship hull due to the boundary layer effects and transported downstream. However, the most significant part of turbulence was produced by the wake flow behind the transom, which had a magnitude of at least two order larger than that from the side walls. The ship wake turbulence transported to very far behind the ship, also spreaded transversely according to the diverging waves. The velocity contour shows there is strong recirculation near where the sudden rise-up of free surface occurs. This is very similar to what is usually observed in hydraulic jumps, as further discussed later. All of these predictions are consistent with at-sea observations [19] and related CFD single phase modeling of ship hull flows by other researchers. In order to focus on the major objective study, i.e., prediction of air entrainment and bubbly wake, no further discussion of the single phase flow analysis will be made. Readers interested in the single
phase modeling of ship flows are referred to the work of others, e.g., Carrica et al. [18,20] and Wilson et al. [21]. In addition, for the sake of computational efficiency, all the following two-phase simulations were conducted by using the coarse mesh.

The air entrainment model was switched on once a statistically stationary state for the liquid flow was obtained. A uniform bubble diameter of 0.08mm was assumed, which was selected to be the typical bubble size measured in experiments [19]. For the air entrainment model, the entrainment depth was set to $\phi_{\text{ent}} = 0.002 L$, which fell in the range of 2 $\sim$ 5 times of mesh size as recommended in the work of [11], and $C_{\text{ent}} = 11.8$ was used, which was calibrated by the void fraction value at only one point under one condition then uniformly applied to everywhere for all the cases. Specifically, in this study, the data point at 2m aft of the transom, 0.9m deep of the ship center plane and under a ship speed of 9 knots was used for this calibration purpose. Using a time step of 0.005 in non-dimensionalized units, which translates to a dimensional time interval of about 0.06s, it took about 300 steps for bubbles to traverse the computational domain. We started to collect data after this interval and monitored the evolution of time-averaged values for the bubble velocities and void fractions until they attained a statistically-stationary state.

Fig. 4 is a snapshot of the bubble source term, $E_g$, close to the free surface. We observe strong entrainment just aft of the transom stern and also see some scattered sources in the near-wake region. In addition to these regions, air entrainment was also observed along the hull-air-water contact line and at the masker. All these regions of air entrainment are consistent with visual observations at sea [19]. The only exception is the bow region, where we did not predict the expected entrainment because we were unable to resolve the breaking bow wave due to insufficient grid resolution [2,3]. From our previous studies [10,11], we know that in order to have our model work for the case of air entrainment due to a plunging sheet (or jet) of liquid, it needs the impact plunge to be well captured, which requires very high computational cost. However, in the current study neglecting the air entrainment of breaking bow wave had little effect on the two-phase flows at the stern which is far away downstream. Furthermore, for R/V Athena it has been found that [22] below a speed of 14 knots the breaking bow wave is very small, thus only entraining few air bubbles.

The time-averaged plots of void fraction near the stern at the center plane along the ship length are shown in Fig. 5. The zoomed-in view near the transom stern region indicates the presence of a large vortical structure that is responsible for much of the air entrainment and is reminiscent of the air entrainment mechanisms in a hydraulic jump [10,11].

We have computed the time-averaged void fraction data along the center plane
of the ship, 2m aft of the transom. We compare our simulations with the at-sea measurements for RV *Athena* [19]. This comparison was performed at three different ships speeds and the results are presented in Fig. 6 The predicted results are in good agreement with measured data. It should be stressed that we achieve this agreement using the same value of the air entrainment parameter $C_{\text{ent}}$ at all speeds. The main deviation occurs well below the interface, at a depth greater than 1m, where our simulation appears to under-predict the void fraction, likely because we did not perform polydispersed prediction and thus did not predict the smaller bubbles which tend not to rise to the interface. Also the measured void fraction in this region is very low (around $10^{-4}$), and thus it is prone to experimental error (i.e., an intrusive probe rake was used to measure the at-sea void fraction). Furthermore, some of these bubbles may have been generated due to cavitation at the propeller, which was not modeled in our simulation. In any event our simulation appears to accurately predict the void fraction distribution in the transom stern region.

\textbf{3.2 Steady turn}

In order to further test the developed CMFD model, it is interesting to see how it works for flows under more complex ship motions. For this purpose, we have also computed the flow around RV *Athena* undergoing a steady starboard turn, with a turning radius of $R_t = 4L$. Same as in the previous work of Hyman et al. [2], during the computation, the whole grid including the ship hull is moving according to this prescribed motion with respect to an inertial frame of reference. As already demonstrated by Carrica et al. [20], the main advantage of using an inertial frame to describe the turn is that the Coriolis and centrifugal accelerations present in a rotating frame do not have to be accounted for. In contrast, the time derivatives of any transported quantity have to be modified to account for the fact that in the conservation equations time derivatives are taken keeping constant the coordinates in physical space and not the transformed computational space coordinates. A full description of the transformed conservation equations is beyond the scope of this work and can be found elsewhere [20]. For this simulation the flow was asymmetric and a grid around the entire ship was generated having approximately 6.5 million points in 100 computational blocks. As in the previous case with the ship moving straight-ahead, this grid included all the control surfaces but not the propeller-related appendages. There were 3 levels of chimera nesting to include appendages and refinement astern of the transom and in the region around the propellers. The rudders were deflected 20° and the ship was given a sideslip angle of 8°. These conditions approximately correspond to a quasi-steady $4L$ turn. In the steady part of a turn, the rudders are nearly unloaded and the hull itself provides most of the lateral force needed to maintain the turn.
The ship speed during the turn was $U = 10.5$ knots, resulting in a Reynolds number of $2.55 \times 10^8$ and a Froude number of 0.25. The values used for the bubble diameter and the air entrainment coefficient were the same as for straight ahead ship motion. The background grid extended sufficiently far away from the ship for the assumption of undisturbed waters in the far-field boundaries to hold. As a result the boundary conditions in the far field were the same as for straight ahead motion. The simulation was initiated with the ship and the background grid advancing in the prescribed turn and used a time step of $\Delta t = 0.01L/U$. This condition was continued for 100 time steps in order to dissipate the startup transient. At time $t = L/U$, the bubble entrainment model was switched on and it continued on for another 100 time steps.

Fig. 7 shows a snapshot of the predicted bubble source strength close to the free surface around RV Athena during a steady turn. We observe a wave originating on the starboard (inside turn) corner of the transom. By comparing Figs. 4 and 7 we conclude that in both the straight ahead and steady turn cases air entrainment is seen at the contact line along the hull, at the masker and particularly at the transom. This entrainment leads to a region of “white water” aft of the transom. Nevertheless, some significant differences are observed between the bubbly wake of the straight ahead and steady turn cases and they are highlighted in Figs. 8 and 9. First, as expected, the bubbly wake for the steady turn case is curved toward the path traced by the ship. Second, the void fraction distribution in the near wake is markedly different. For the straight ahead case we observe two symmetric streaks of high void fraction region (void fraction over 20%) perpendicular to the transom, originating at the the two ends of the transom. This was also observed in the sea tests and is believed to be caused by the diverging waves due to the transom’s geometry (see Fig. 9). In the turning case these streaks are also observed, however they are closer together and they last longer, implying higher rates of air entrainment. In addition a third streak was observed at the inside corner of the turn though it is shorter and weaker than the former two (as seen in the left plot of Fig. 8). It appears that this extra entrainment is caused by the corner wave induced by the turning motion of the ship. Once again this pattern was also observed in the sea tests (see Fig. 9). In general it appears that the void fraction distribution for the steady turn case is more heterogeneous than for straight ahead ship motion. In Fig. 9 we note that for the turning ship case there is some entrainment ahead of the transom, toward the left side of the picture, that is missing from the simulation. This is likely due to the interaction of the wavy sea with the contact line which is not captured because we are simulating steady seas.
4 Conclusions

Three dimensional (3-D) simulations of the bubbly flow around the naval research vessel *Athena*, was performed using a RaNS-based two-fluid modeling approach, and a phenomenological subgrid air entrainment model. Consistent with experimental observations, this model predicted entrainment at the masker, the contact line, and particularly at the transom of the ship. For straight ahead motion the time-averaged values of the predicted void fraction profiles near the transom were compared with experimental data for three different ship speeds and good agreement was found. A simulation was also performed for a steady turn of the ship and qualitative comparisons were made between the predicted straight-ahead and steady turn cases with the RV *Athena* visualization data taken at sea. It was found that the ship maneuvers in a turn leads to greater rates of air entrainment and to a more heterogeneous bubbly wake.

Future work includes extending the current numerical framework by incorporating DES (Detached Eddy Simulation) model [23,24] for turbulence and two-phase level set methods [6] for both continuous gas and liquid phases. In this way, large scale turbulence and entrapped air cavities will be directly resolved in the regions where these structures are much larger than the grid size, while sub-grid RaNS and air entrainment model will be activated elsewhere. We expect this will provide us more profound and insightful understanding of the flows, while maintaining the overall computational cost still affordable.

5 Acknowledgments

This work was sponsored by the Office of Naval Research (ONR), grant N00014-03-1-0826, under the administration of Dr. Patrick Purtell, and was supported in part by grants of computer time from the DOD High Performance Computing Modernization Program at the Maui High Performance Computing Center (MHPCC), US Army Engineering and Research Development Center (ERDC) and Arctic Region Supercomputing Center (ARSC). We owe thanks to Dr. Francisco Moraga of the General Electric Corporate Research Center for very helpful discussions about CMFD models and technical details concerning ship modeling. We wish to acknowledge the assistance of Dr. Pablo Carrica from the Iowa Institute for Hydraulic Research for his assistance with CMFD code development and very useful technical discussions. We also give thanks to Dr. Eric Terrill at Scripps Institution of Oceanography for providing the void fraction experimental data for RV *Athena*. 
References


Fig. 1. Schematic of a rough air/water interface.
Fig. 2. Free surface elevation of RV *Athena* at 10.5 knots: (a) Overview of predicted free surface contoured by elevation; (b) Time averaged profiles of elevation at 2m and 4m aft of ship transom.
Fig. 3. Distribution of Turbulence Kinetic Energy (a) and (b) streamwise velocity in the wake of RV Athena at 10.5 knots. (A log scale for TKE is used to better display its development near the side wall of the ship where TKE is very small comparing with that in the ship wake)
Fig. 4. Predicted rate of air entrainment close to the free surface for RV *Athena* at 9 knots: (a) Overall view in which the entrainment at the transom is clearly seen; (b) Magnified view of the masker; (c) Magnified view of the hull-air-water contact line.
Fig. 5. (a) Time-averaged void fraction aft of the transom stern along the ship’s center plane, at a ship speed of 9 knots; (b) Enlarged view of the region close to the transom, that is the boxed part of (a).
Fig. 6. Predicted and measured void fraction distributions 2m aft of the transom along the ship’s center plane, as a function of depth for straight ahead ship speeds of 6 knots (top), 9 knots (center) and 10.5 knots (bottom).
Fig. 7. Predicted rate of air entrainment close to the free surface for RV *Athena* in a steady turn: (a) Overall view in which non-symmetric air entrainment at the transom is clearly seen; (b) Magnified view of the masker; (c) Magnified view of the hull-air-water contact line.
Fig. 8. Void fraction close to the free surface for RV *Athen* in straight ahead motion (right) and during a steady turn (left).
Fig. 9. Top: predictions of void fraction close to the free surface for RV Athena in straight ahead (right) and steady turn (left) motion. Bottom: Corresponding sea test pictures of the bubbly wake taken along a similar orientation for same ship motions.