
Kangjian He1, Cong Ye1,2, Decheng Wan1*

1 Computational Marine Hydrodynamics Lab (CMHL), School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China
2 China Ship Scientific Research Center, Shanghai, China
*Corresponding author

ABSTRACT

Water-air-bubble mixed flow is a complex multiphase flow generated by the intense interaction between marine structures and surrounding fluids. It involves remarkable and complicated influence on the hydrodynamic performance of marine structures. This paper studied the influence of uniformly mixed water-air-bubble incoming flow on hydrodynamic performance of a two-dimensional NACA0012 hydrofoil by using the Computational Fluid Dynamics (CFD) method. Two-fluid method (TFM) is adopted to solve the mixed water-air-bubble multiphase flow field with the open-source CFD toolkit OpenFOAM. Validation of the numerical model is conducted in two aspects. On the one hand, numerical results with single-phase incoming water flow are compared with experimental and numerical results in literatures. On the other hand, mesh convergence test is conducted to further verify the convergence and robustness of this model. On the basis, the influence of water-air-bubble flow for the condition of 10% air volume fraction on velocity and pressure fields around the hydrofoil is presented. Coefficients of lift and drag of the hydrofoil are discussed comparing with single-phase water flow condition. This research can provide a valuable reference for studying the complex interaction between multiphase water-air-bubble mixed flow and marine structures.

KEY WORDS: Water-air-bubble mixed flow; two-fluid method; hydrofoil; hydrodynamic performance.

INTRODUCTION

Water-air-bubble mixed flow is a complicate multiphase flow and usually generated by the intense interactions between navigating ships and free surface flow. Water-air-bubble mixed flow usually involves lots of bubbles with multiple sizes and little-scale interface. Lots of bubbles in the water-air-bubble mixed flow will sweep down along the ship and cluster near the propulsion system, which will affect the hydrodynamic performance of hull-propeller-rudder. The cavitation of propeller will generate cavitation bubbles and affects the hydrodynamic performance of the rudder. Otherwise, the submarine usually set holes at the stern to release flue gas. The flue gas released from holes will generate water-air-bubble mixed flow, which changes the back flow field and affects the hydrodynamic performance of rudder and propeller. The effect on the hydrodynamic performance of marine structures, such as propeller performance and ship maneuverability, is complicated and unclear. It is significant to adopt multiphase method to study complicated water-air-bubble mixed flow and analyze the effect.

Numerical simulation has been widely adopted to model the multiphase flow. There are three main multiphase method: interface capture method, Euler-Lagrange method, and Euler-Euler method. Interface capture method such as Volume of Fluid (VOF) method (Hirt and Nichols, 1981) and Level-Set method (Carrica et al., 2007) is the most commonly used method in naval architecture and ocean engineering, which involves large-scale water-air interface (Zhang et al., 2020). However, this method usually requires finer mesh resolution relative to the scale of water-air interface to accurately capture the physical process (Wardle and Weller, 2013). Euler-Lagrange method (Ma et al., 2015) uses spherical Lagrange particles to model micro bubbles and solves the liquid phase based on Euler method. Euler-Lagrange method provides a relatively precise method to model multiphase flow and can capture the detail motion of bubbles (Zhang et al., 2020). However, this method will require high computational cost with the increasing number of Lagrange particles. Euler-Euler method, or two-fluid method (TFM) treats the dispersed phase as continuous and solve the dispersed phase with Navier-Stokes equations. TFM is adaptive to be used to model high air volume fraction multiphase flow due to relatively low computational cost. TFM has been broadly validated and applied in the simulation of bubble columns (Adam Mühlbauer, et al., 2019), bubble flow around the ship (Li et al., 2019; Ma et al., 2011), and other complex two-phase flows. Water-air-bubble mixed flow usually involves numerous micro bubbles and micro-scale water-air interface. It is difficult to be modeled with interface capture method and Euler-Lagrange method. Therefore, TFM is adopted to model the water-air-bubble mixed flow in this paper.

This paper is organized as follows: The numerical computational method and calculation settings are introduced first. Then, the validation and verification are conducted to show the correctness of the computational method. Finally, the influence of uniformly mixed water-air-bubble incoming flow on hydrodynamic performance is discussed and conclusions are summarized.