# A Robust Mass-Momentum Flux Scheme for Simulation of Breaking Ship Waves

Cheng Liu[1], Changhong Hu[2], Decheng Wan[1]

[1]*School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China. Email: chengliu@sjtu.edu.cn; dcwan@sjtu.edu.cn*
[2]*Research Institute for Applied Mechanics, Kyushu University, Fukuoka 816-0811, Japan. Email: hu@riam.kyushu-u.ac.jp*

## 1. Introduction

The violent free surface flows associated with wave breaking is a common and complicated physical process in ship and ocean engineering problems. Complicated phenomena including two-phase turbulence, wave energy dissipation, and mass transfer around the interface, are important for such violent flows. A number of experiments have been performed to reveal the physics of typical air-water two phase flows [1], in which high-speed cameras, LDV, PIV, and acoustic measurement, etc. are used for the measurements. However, it is still difficult to obtain important statistical information such as void fraction, energy dissipation, and bubble or droplet distribution, through the experiments. CFD approach has also been tried to simulate such turbulent multiphase flows, but there are still many difficulties. One of the challenges is the numerical stability at the interface in simulating large-density-ratio two-phase flow problems. If no smoothing treatment for density field is used, spurious velocity field may appear adjacent to the interface [2]. Many researchers believed that the numerical instability comes from the inaccurate discretization of pressure gradient and from the decoupling of pressure-velocity [3]. We have found through our previous researches that most of the error sources may come from the inconsistent advection of mass-momentum flux computation, and to develop a mass-momentum consistent scheme is the motivation of this study.

In this extended abstract, the proposed scheme is described briefly. The robustness of the proposed scheme in preserving momentum flux of large-density-ratio problems is demonstrated by the numerical simulations of breaking waves from a KCS model.

## 2. Mathematical Formulation

The governing equations for two-phase flow problems, including the continuity, momentum, and phase-fraction advection equation, can be given as

$$\nabla \mathbf{U} = 0, \tag{1}$$

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \mathbf{F}_g, \tag{2}$$

$$\frac{\partial C}{\partial t} + \nabla \cdot C = 0, \tag{3}$$

where $\mathbf{U}$ is the velocity vector, $p$ is the pressure. In this study, only the immiscible Newtonian fluid is considered, thus the shear stress tensor can be written as $\boldsymbol{\tau} = \mu[\nabla \mathbf{U} + (\nabla \mathbf{U})^T]$. The density $(\rho)$ and viscosity $(\mu)$ for each cell are obtained using the phase fraction $C$ as shown by the following equations.

$$\rho = \rho_l C + \rho_g (1 - C), \ \mu = \mu_l C + \mu_g (1 - C). \tag{4}$$

Here the phase fraction $C$ is used to determine whether the cell is occupied by the gas/liquid phase or the mixture of two phases. The gravity force $\mathbf{F}_g = \mathbf{g}$ is considered in Eq. (2).

For implementation, the hydrostatic pressure contribution is isolated, an alternative pressure $\acute{p}$ defined by following is solved instead of $p$,

$$\acute{p} = p - \rho(\mathbf{g} \cdot \mathbf{h}).$$

Therefore,

$$-\nabla p = -\nabla\acute{p} - \nabla\big(\rho(\mathbf{g}\cdot\mathbf{h})\big) = -\nabla\acute{p} - \rho\mathbf{g}\cdot\nabla\mathbf{h} - \mathbf{h}\cdot\nabla(\rho\mathbf{g}) = -\nabla\acute{p} - \rho\mathbf{g} - \mathbf{gh}\cdot\nabla\rho, \quad (5)$$

are derived, in which $\nabla\mathbf{h} = \mathbf{I}$ and $\nabla\mathbf{g} = 0$ have been used. Substitute Eq. (5) to the momentum equation (2), we obtain

$$\frac{\partial\rho\mathbf{U}}{\partial t} + \nabla\cdot(\rho\mathbf{U}\otimes\mathbf{U}) = -\nabla\acute{p} + \nabla\cdot\boldsymbol{\tau} - \mathbf{gh}\cdot\nabla\rho. \quad (6)$$

Eq. (6) is the target momentum equation of the two-phase flow problems that will be used for the next-stage numerical discretization.

## 3. Mass-Momentum Flux Computation Method

Due to the hyperbolic property of the non-linear term, the information propagation by advection is along the characteristic velocity direction. Therefore, a class of upwind scheme has been proposed for the discretization of advection terms, in which various flux limiters are used to avoid the over/undershoot of the solution profile. One of the most popularly used limited upwind methods, the WENO (Weighted Essentially Non-Oscillatory) scheme, utilizes 3 sets of localized low-order-interpolation stencils to build a high-order reconstruction scheme [4].

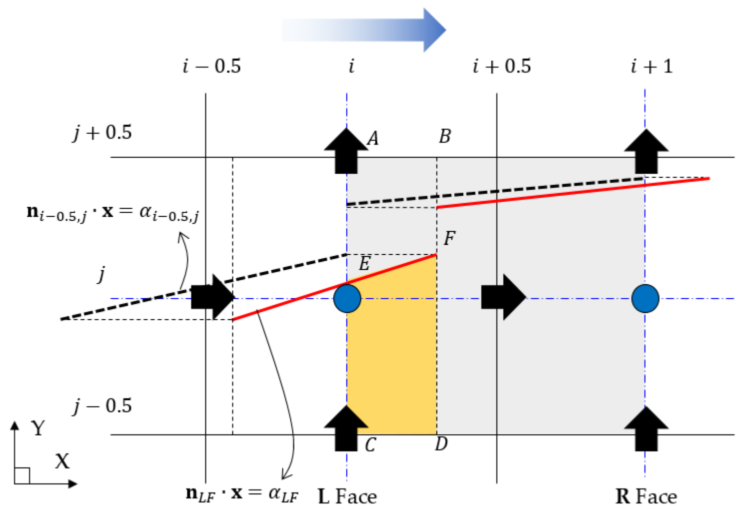



**Fig. 1** Sketch for demonstration of the mass flux computation by a geometrical method.

In the WENO scheme for the advection term of Eq. (6), the face flux is computed through upwind interpolation (often with flux limiter to reduce unphysical over- or under shoot). In this study, the discretization of the advection term is redesigned to ensure that the mass-flux in momentum equation (6) is consist with the scheme for VOF transport equation (3). In order to maintain the consistence, a novel geometric approach based on staggered mesh is proposed, in which the evaluation of the momentum flux and the reconstruction of VOF function are following the same procedure to update the solution.

One of the most crucial parts of the advection scheme for Eq. (6) is how to compute the momentum flux on the cell faces while preserving consistence with mass flux, which is a quite challenging task for a staggered mesh. In this study, a geometric method similar to PLIC-VOF is proposed. The mass flux

$M_{i,j}^{n}$ crossing L face in $\Delta t$, which is written by

$$M_{i,j}^{n} = \int_{t^n}^{t^n+\Delta t}\int_{x^n}^{x^n+\Delta x}\left[Q(\mathbf{x},t)\rho_l + (1-Q(\mathbf{x},t))\rho_g\right]dV\,dt, \tag{7}$$

can be obtained by $M_{i,j}^{n} = \rho_l S_{CDFE} + \rho_g S_{EFBA}$, as $S_{CDFE} = C_{LF}$ and $S_{EFBA} = r - C_{LF}$ (Fig. 1), finally we get

$$M_{i,j}^{n} = \rho_l C_{LF} + \rho_g (r - C_{LF}). \tag{8}$$

Similar procedure is used for the computation of the R face mass flux $M_{i+1,j}^{n}$. By rotating the x-y coordinate frame, mass flux of B face and T face, $M_{i+0.5,j-0.5}^{n}$ and $M_{i+0.5,j+0.5}^{n}$ can be derived without additional difficulties.

The proposed mass-momentum scheme has been validated by a series of benchmark problems. Numerical tests have shown the robustness for the simulation of violent two-phase flow. The fake velocity at the interfaces accused by inaccurate two-phase flux computation can be avoided.

## 4. Numerical Results of the KCS Ship Flow

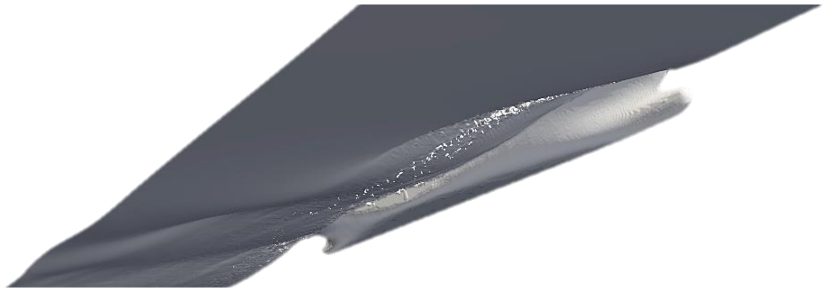
A CFD code based on the proposed scheme has been used to simulate breaking ship wave phenomena. The spilling and plunging breaking waves in the ship flows will generate a large amount of bubbles and droplets, which may reduce the hydrodynamic performance of propulsion. Besides, the bubbly wake behind the ship has obvious optical characteristics, the residence time and flow dynamics of entrained cavity deserve further investigation. In this study, we try to use the newly developed two-phase flow solver for the investigation of those problems. A KCS model with a scale ratio of 52.67 is considered, with the length between perpendiculars is 4.36m. The model experiment was already conducted by CSSRC, in which the free surface elevation of the bow wave is measured. Scars and spilling type of the ship wave breaking phenomenon is captured by high-speed camera. In this numerical study, additional important features of the KCS breaking wave, including the air entrainment and flow structures, which cannot be obtained directly by the model experiments are provided. Three cases with different Froude numbers (Fr=0.35, Fr=0.38 and Fr=0.4) are performed. Fig. 2 shows the underwater view of the entrained air cavity. The air cavity size distribution is given in Fig. 3. More details will be presented at the workshop.

## 5. Summary

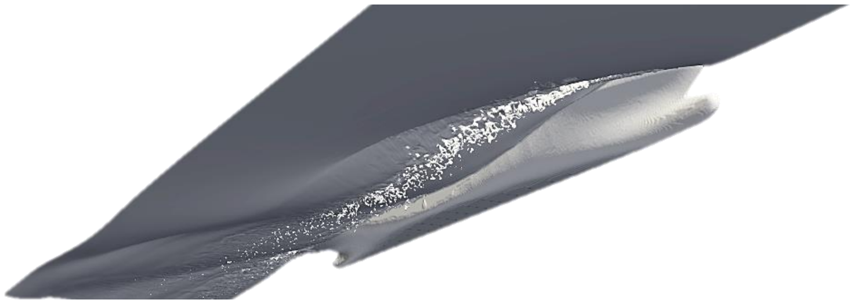
In this manuscript, we present a newly developed mass-momentum scheme for the simulation of large-density-ratio two-phase flows. A KCS breaking wave problems is considered as a preliminary validation. It shows that the present numerical approach is robust and efficient for simulation of violent ship flows. This is an ongoing development, more newly obtained numerical results will be presented at the workshop.

## Acknowledgements

This work is supported by the National Natural Science Foundation of China (11902199, 51979160, 51879159), Shanghai Pujiang Talent Program (19PJ1406100), The National Key Research and Development Program of China (2019YFB1704204, 2019YFC0312400), Chang Jiang Scholars Program (T2014099), Shanghai Excellent Academic Leaders Program (17XD1402300), and Innovative Special Project of Numerical Tank of Ministry of Industry and Information Technology of China (2016-23/09), to which the authors are most grateful.



(a) Fr=0.35



(b) Fr=0.4

**Fig. 2** Instantaneous wave profile of different Fr numbers.

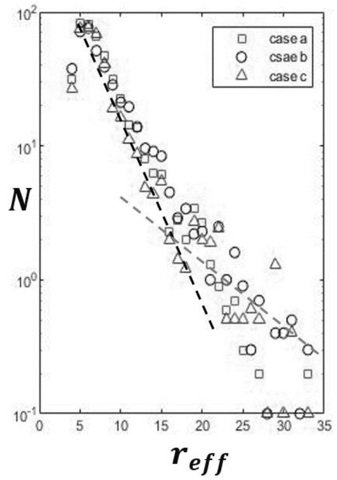



**Fig. 3** Air cavity size distribution for different Fr number cases (case a: Fr=0.35; case b, Fr=0.38; case c, Fr=0.4).

## Reference

[1]. Scardovelli R, Zaleski S. Direct numerical simulation of free-surface and interfacial flow. *Annual* Review of Fluid Mechanics, 1999, 31(1): 567-603.

[2]. Vaudor G, Ménard T, Aniszewski W, et al. A consistent mass and momentum flux computation method for two phase flows. Application to atomization process. Computers & Fluids, 2017, 152: 204-216.

[3]. Nangia N, Griffith B E, Patankar N A, et al. A robust incompressible Navier-Stokes solver for high density ratio multiphase flows. Journal of Computational Physics, 2019, 390: 548-594.

[4]. Liu C, Hu C. Adaptive THINC-GFM for compressible multi-medium flows. Journal of Computational Physics, 2017, 342: 43-65.