Computers and Fluids 163 (2018) 7–19

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

Multi-phase SPH modelling of air effect on the dynamic flooding of a damaged cabin

X.Y. Cao[a], F.R. Ming[a], A.M. Zhang[a,*], L. Tao[a,b]

[a] College of Shipbuilding Engineering, Harbin Engineering University, Harbin 150001, China
[b] Department of Naval Architecture, Ocean & Marine Engineering, University of Strathclyde, Glasgow G4 0LZ, UK

ARTICLE INFO

Article history:
Received 14 April 2017
Revised 3 November 2017
Accepted 21 December 2017
Available online 22 December 2017

Keywords:
Flooding
Multi-phase SPH model
Air flow
Dynamic response

ABSTRACT

The air flow may take effects on the responses of the damaged ship in the dynamic flooding process. It not only relates to the amount of inflow but also the stability of the ship. In order to accurately predict the responses of a damaged ship, it is essential to take the air into account. In this study, a multi-phase SPH model combined with a dummy boundary method is proposed. One of the advantages of the new SPH model in solving this nonlinear problem is that, it does not rely on other algorithms to track the interface of different phases but can easily deal with breaking, splashing and mixing. The stability and accuracy of the numerical model are verified by comparing with experimental and published numerical results. The air captured in the flooding process is further studied with focus on the exchange of air and water near the opening. Finally, the effects of the sizes and number of the deck openings on the air flow are analyzed. It is found that the air flow can reduce the kinematic energy of inflow water, leading to decreases in the dynamic moment formed by the flooding water and sinking rate of damaged cabin.

1. Introduction

The flooding of a ship is a complex multiphase flow problem and also a fluid-structure interaction problem [1]. Once the hole occurs on the outer plate caused by accident or other damages, the water may flow into the ship cabin rapidly and impact on the inner structures, meanwhile the air inside the ship will be compressed or pushed out. Due to the large deformation, splashing and breaking of free surface, air bubbles may be trapped in water and coalesced at the interface. Besides, for the rapid flooding case, the air cushion above the water level can be formed in an airtight cabin. In some worse scenarios, these complex fluid flows are coupled with the nonlinear motion of the ship hull [2, 3], so it is very difficult to forecast the complicated hydrodynamic process. During the flooding process, the dynamics of the air flow play a dominant role and compromise the stability of the damaged ship, which may lead to severe consequence [4] and warrants thorough study.

Palazzi and De Kat [5] found by experiment that the air flow will introduce additional damping and the air compressibility will cause the energy dissipation, thus the fluid motion and resonance amplitude are restrained. Hearn et al. [6] developed the stiffness expression for the internal air to study the aerostatic influences of the air on the motion of a damaged ship. Different models for the internal air stiffness were proposed to satisfy the adiabatic equation. Smith [7] measured the wave loads of a damaged cabin model under the forced vibration and found that the air increases the motion damping of the model. Ruponen et al. [8–10] pointed that the air flow and air pockets have significant influences on the dynamic flooding process of a damaged cabin in certain conditions by comparative studies of experiment and numerical simulation. Strasser et al. [11] used a CFD code based on RANSE and VOF method to model the flooding considering with air compression, and noticed that flooding in ship without large air ventilations slowed down when the damaged opening was fully submerged. Air flow and air compression may further complicate the dynamic flooding process. In most studies on the topic reported to date, the flooding water in the cabin is assumed as a flat surface parallel to the sea level, then the dynamic flooding motion and the exchange of the air and water nearby the opening are ignored. However, for a damaged cabin flooding with a high rate or flooding with fluid breaking and coalescing, the assumption can result in a large discrepancy compared to the actual situation. In order to obtain the realistic flooding process, it is necessary that the numerical model for dynamic flooding is applied with the above-mentioned factors being considered properly. The meshless smoothed particle hydrodynamics (SPH) method in solving the dynamic flooding process, which relates to free surface sloshing [12,13], slamming [14,15] and coupling with structures [16–18], has shown its distinct advantages. With the development of SPH technique [19–21], the approaches for the

* Corresponding author.
E-mail address: zhangaman@hrbeu.edu.cn (A.M. Zhang).

https://doi.org/10.1016/j.compfluid.2017.12.012
0045-7930/

interface of different phases [22–24] and computational efficiency [25,26] have been improved.

To investigate the flooding process considering the air flow, the multi-phase SPH model is developed in this paper. Validated by two benchmarks, i.e., the classical dam-break and the sinking process of an intact box, the multi-phase SPH model is then applied to the comparative study of a damaged cabin flooding with or without air flow to reveal the influencing mechanism. Finally, the dynamic exchange of air and water for the damaged cabin with different openings on the upper deck is studied and the effects of air flow are examined.

2. Numerical methods

2.1. Basic equation of the multi-phase SPH

To solve the dynamic flooding process of interest, a multi-phase model is developed based on SPH model in Colagrossi and Landrini [24], which is a simple procedure to predict the interactions between different phases. The multi-phase SPH model is based on the assumption of the weakly-compressible fluid and can be solved in an explicit way. The air is assumed to be iso-entropic thus it is not necessary for solving the energy equation and the pressure is the function of density. The Navier–Stokes equations for weakly compressible fluids in a Lagrangian form are as follow:

$$\begin{aligned} \frac{D\rho}{Dt} &= -\rho\nabla\cdot\boldsymbol{v} \\ \frac{D\boldsymbol{v}}{Dt} &= \boldsymbol{g}-\frac{\nabla P}{\rho} \\ \frac{D\boldsymbol{x}}{Dt} &= \boldsymbol{v} \\ P &= P(\rho) \end{aligned} \tag{1}$$

where ρ,$\boldsymbol{v}$,$\boldsymbol{x}$,$\boldsymbol{g}$ and P denote the density, the velocity, the coordinate, the gravity acceleration and the pressure, respectively. In order to solve Eq. (1), the derivatives in it can be firstly written as an integral in the domain through the kernel approximation. For a certain vector function $\boldsymbol{f}(\boldsymbol{x})$, the divergence can be transformed as [27]

$$<\nabla\cdot\boldsymbol{f}(\boldsymbol{x})>\approx\int_{\Omega}\boldsymbol{f}(\boldsymbol{x}')\cdot\nabla W(\boldsymbol{x}-\boldsymbol{x}')d\boldsymbol{x}' \tag{2}$$

where $\boldsymbol{x}'$ is an adjacent position of $\boldsymbol{x}$ and $W(\boldsymbol{x}-\boldsymbol{x}')$ is the kernel function.

When the problem domain is discretized into particles, the particle approximation can be applied to discretize the integral by the weighting summation of neighbouring particles in the support domain [27]. For the divergence of a vector function $\nabla\cdot\boldsymbol{f}(\boldsymbol{x})$, the approximation at the particle i can be carried out by the neighbouring particles j as

$$\nabla_i\cdot\boldsymbol{f}(\boldsymbol{x})=\int_{\Omega}\boldsymbol{f}(\boldsymbol{x}')\cdot\nabla W(\boldsymbol{x}-\boldsymbol{x}')d\boldsymbol{x}'=\sum_{j\in\Omega}\boldsymbol{f}(\boldsymbol{x}_j)\cdot\nabla_iW_{ij}V_j \tag{3}$$

where W is selected as the renormalized Gaussian kernel function [16,28]; V is the volume of the particle, which changes with the density in the simulation. Thus, it is easy to obtain that the divergence of a constant vector function is zero. There holds

$$\boldsymbol{f}(\boldsymbol{x}_i)\cdot(\nabla_i\cdot\mathbf{1})=\boldsymbol{f}(\boldsymbol{x}_i)\cdot\sum_{j\in\Omega}\nabla_iW_{ij}V_j=\sum_{j\in\Omega}\boldsymbol{f}(\boldsymbol{x}_i)\cdot\nabla_iW_{ij}V_j=0 \tag{4}$$

Thus, the equation in a symmetric form is obtained by subtracting Eq. (4) from Eq. (3)

$$\nabla_i\cdot\boldsymbol{f}(\boldsymbol{x})=\sum_{j\in\Omega}\left[\boldsymbol{f}(\boldsymbol{x}_j)-\boldsymbol{f}(\boldsymbol{x}_i)\right]\cdot\nabla_iW_{ij}V_j \tag{5}$$

For the gradient of a scalar function $f(\boldsymbol{x})$, it can be written as

$$\nabla_if(\boldsymbol{x})=\int_{\Omega}f(\boldsymbol{x}')\nabla W(\boldsymbol{x}-\boldsymbol{x}')d\boldsymbol{x}'=\sum_{j\in\Omega}f(\boldsymbol{x}_j)\nabla_iW_{ij}V_j \tag{6}$$

Conducting the similar transformation as Eq. (4), the gradient of the scalar function can be derived as Eq. (7).

$$\nabla_if(\boldsymbol{x})=\sum_{j\in\Omega}\left[f(\boldsymbol{x}_j)+f(\boldsymbol{x}_i)\right]\nabla_iW_{ij}V_j \tag{7}$$

It should be underlined that different forms for the gradient of the scalar function and the divergence of the vector function are used. The form of Eq. (7) allows an anti-symmetric property which is crucial for maintaining the momentum conservation when $f(\boldsymbol{x})$ represents the pressure, while the form of Eq. (5) can decrease the errors introduced by the kernel truncations near the free surface, see more analyses in Colagrossi et al. [29].

Through the transformation of Eq. (5) and Eq. (7), the continuity and momentum equation in Eq. (1) can be expressed as:

$$\frac{D\rho_i}{Dt}=\rho_i\sum_j(\boldsymbol{v}_i-\boldsymbol{v}_j)\cdot\nabla_iW_{ij}V_j \tag{8}$$

$$\frac{D\boldsymbol{v}_i}{Dt}=\boldsymbol{g}-\sum_j\frac{1}{\rho_i}(P_i+P_j)\nabla_iW_{ij}V_j+\boldsymbol{f}_v \tag{9}$$

where the subscripts i and j represent a pair of interacting particles, $\boldsymbol{g}=(0,-9.81)m/s^2$ and $\boldsymbol{f}_v$ is the viscous force. In solving the multi-phase flow problem, there exist density differences across the interface of different phases, which may lead to approximation errors of the particle density so that pressure oscillations usually occur near the interface. In Eqs. (8) and (9), for the pairing particles i and j from different phases, only the volume of the particle from another phase contributes to the approximation but not the density. Also, because the uniform volume is used for different phases at the initial time, so it benefits the smooth approximation of the fields at the interface, e.g., the pressure and the velocity fields.

To solve the Eqs. (8) and (9), the relationship between pressure and density is normally used in SPH method. For the fluid of different phases, the Tait equation [28] improved by Nugent and Posch [30] is adopted:

$$\begin{aligned} P_w &= B[(\frac{\rho_w}{\rho_{w0}})^{\gamma_w}-1]+P_0 \\ P_a &= B[(\frac{\rho_a}{\rho_{a0}})^{\gamma_a}-1]+P_0-\chi\rho_a^2 \end{aligned} \tag{10}$$

where the subscript w represents the heavier phase and a represents the lighter phase. For water: $\gamma_w=7$, the initial density is $\rho_{w0}=1000\text{kg/m}^3$; while for air: $\gamma_a=1.4$ [24], and the initial density is $\rho_{a0}=1.29\text{kg/m}^3$. The same coefficient $B={c_a}^2\rho_{a0}/\gamma_a={c_w}^2\rho_{w0}/\gamma_w$ is proposed by Colagrossi and Landrini [24] for ensuring the identical initial pressure for both phases at the interface. P_0 is the background pressure.

Eq. (10) gives the explicit relationship between the pressure and the density, which leads to an explicit algorithm developed in the multi-phase SPH model. The predictor-corrector scheme [31] and a constant time step satisfying the Courant-Friedrichs-Lewy (CFL) condition are used. According to the CFL condition, the time steps will be too small when the real sound speed of water is used. Due to this consideration, a suitable artificial sound speed is always used in weakly compressible SPH to approximate the free surface flow problems. The assumption of weakly compressible fluid satisfies that the variation of the density field, proportional to the square of the Mach number (see Monaghan [32]), is less than 1%, i.e., $\Delta\rho/\rho\sim(v_{\max}/c_0)^2=Ma^2$. It should be noted that it is not the real situation [24, 26], where the lighter phase has a larger sound