Ocean Engineering 147 (2018) 68–76

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical study of damaged ship's compartment sinking with air compression effect

Zhiliang Gao [a,b,*], Yaling Wang [c], Yan Su [a], Lin Chen [d]

[a] Key Laboratory of High Performance Ship Technology (Wuhan University of Technology), Ministry of Education, Wuhan, 430063, China
[b] School of Transportation, Wuhan University of Technology, Wuhan, 430063, China
[c] CCCC Second Harbor Consultants Co., Ltd, Wuhan, 430071, China
[d] Donghai Science and Technology College, Zhejiang Ocean University, Zhoushan, 316000, China

ARTICLE INFO

Keywords:
Damaged ship
Flooding
Sinking
Air compression
CFD

ABSTRACT

The computational fluid dynamics (CFD) method was applied to study the dynamics of damaged ship during flooding and sinking process, in which the air compression effect was taken into account. The studied case was a ship's compartment with bottom damage sinking into calm water. To investigate the air flow effect, the compartment was set to be fully ventilated, partially ventilated and airtight for different cases. In the numerical simulation, the air was treated as ideal gas. The volume of fluid method was adopted to capture the air and water interface. The shear stress transport model was employed for turbulence modelling. The technique of dynamic layering mesh was used to deal with the ship sinking motion. All the numerical schemes were implemented with the CFD solver ANSYS-Fluent. The benchmarking results were presented and discussed, including the time and gird dependence studies, the turbulent model against the laminar model, the air compressible model against the incompressible model and the computation against the model test. The effect of air flow on water flooding and ship sinking was also analysed.

1. Introduction

When a ship is damaged, the sea water floods into the damaged compartment and subsequently flows to other compartments through the internal openings. The interactive dynamics between the floodwater and damaged ship is very complex. It becomes even more complex if the floodable compartment is not fully ventilated. On one hand, the entrapped air could impede floodwater spreading uniformly over the internal compartments. In such case, the ship may capsize in a short time due to highly asymmetric distribution of internal water. On the other hand, the entrapped air could slow the flooding progression and consequently prevent the ship sinking rapidly. Hence, the inclusion of air compressibility is required for the study of damaged ship's stability and survivability.

Since the 1980s, a variety of numerical methods have been developed to calculate the damaged ship dynamics by different researchers (Spouge, 1986; Vassalos and Turan, 1994; Spanos and Papanikolaou, 2001; Kong, 2009; Rodrigues and Guedes Soares, 2015; Manderbacka and Ruponen, 2016), but only a few methods accounted for the air compression effect. Based on the quasi-static assumption, Palazzi and de Kat (2004) developed a numerical method to simulate the flooding process of a damaged frigate with different ventilation conditions. In their study, the flow rate through an opening was governed by the modified Bernoulli's equation. The internal water was assumed to settle down instantaneously with a horizontal surface. The air behaviour obeyed Boyle's law. The comparison between computational and experimental results confirmed that the accuracy of computation model was improved when the air compression effect was included. However, large discrepancy between the two results was still observed. To improve the performance of quasi-static model, Ruponen (2007) proposed a pressure-correction method to calculate the volume of floodwater implicitly. Yet the determination of discharge coefficient needs to make use of other means such as model test (Stening et al., 2011) or computational fluid dynamics (CFD) simulation (Ruponen et al., 2012). The improved model was successfully applied to simulate the progressive flooding of a damaged barge with airtight compartment arrangement. Its performance was further validated in the flooding test of an attack craft equipped with small ventilation pipes (Ruponen et al., 2013). A simplified version of Ruponen's method was developed by Dankowski (2013) to enhance the computational efficiency. Further modification is proposed by Lee (2015)

* Corresponding author. Key Laboratory of High Performance Ship Technology (Wuhan University of Technology), Ministry of Education, Wuhan, 430063, China.
E-mail address: zlgao@hotmail.com (Z. Gao).

https://doi.org/10.1016/j.oceaneng.2017.10.035
Received 11 June 2017; Received in revised form 3 September 2017; Accepted 16 October 2017

to improve numerical stability and accuracy. In general, the quasi-static flooding model can yield satisfactory results in the intermediate and final flooding stages, but it lacks the ability to reproduce the transient phenomenon in the initial stage.

To model the flooding and damaged ship dynamics intrinsically, the CFD method has been applied to relevant study in the past decade. For the case where the air compression is negligible, the CFD method is capable of modelling the floodwater dynamics (Strasser, 2010; Gao, 2012; Zhang et al., 2013; Sadat-Hosseini et al., 2016; Bašić et al., 2017). If the air compression effect is crucial for the flooding process, coupling the velocity, pressure and density fields in the computation encounters difficulty (Cho et al., 2006; Hashimoto et al., 2011; Shen, 2012). To avoid unphysical solution, the air was normally treated as incompressible or even ignored in the CFD simulation, but the numerical error shown in the previous study (Gao et al., 2011) cast doubt on such practical treatment or simplification. For the CFD model itself, its reliability can be guaranteed only if the contribution to the numerical error is clarified. Thus, taking air compressibility into account is essential for flooding simulation to minimize the uncertainty of numerical results, especially in case of transient flooding with large amplitude motion of ship.

In this study, the CFD method, which is based on solving the Reynolds-averaged Navier-Stokes (RANS) equations with the volume of fluid (VOF) method and the dynamic mesh technique, was used to simulate a damaged ship's compartment flooding and sinking in calm water. The air was treated as compressible fluid so as to account for different ventilation conditions. The grid and time dependence tests were first conducted. Then the numerical results were compared with the experimental data. The air flow effect on the motion of floodwater and damaged ship was also investigated. It is shown that the adopted mathematical model and numerical schemes are effective to predict the interactive dynamics between the air, water and ship during transient flooding process.

2. Methodology

2.1. Governing equations of fluid motion

The present method considers the air and water as compressible and incompressible fluids, respectively. The fluid motion is governed by the continuity, RANS and energy conservation equations described in the Cartesian coordinate system are as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial \tau_{ij}}{\partial x_j} - \frac{\partial P}{\partial x_i} + \rho g_i \tag{2}$$

$$\frac{\partial}{\partial t}(\rho T) + \frac{\partial}{\partial x_j}(\rho u_j T) = \frac{\partial}{\partial x_j}\left[\frac{\lambda}{C_p}\frac{\partial T}{\partial x_j}\right] \tag{3}$$

with

$$\tau_{ij} = \mu_{\text{eff}}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\delta_{ij}k \tag{4}$$

$$\mu_{\text{eff}} = \mu + \mu_t \tag{5}$$

where t is the time; x_i ($i = 1, 2, 3$) is the coordinate component; u_i is the mean velocity component in the x_i-direction; $\rho = \alpha\rho_1 + (1-\alpha)\rho_2$ is the effective density; ρ_1 and ρ_2 are the densities of water and air, respectively; $\mu = \alpha\mu_1 + (1-\alpha)\mu_2$ is the effective viscosity; μ_1 and μ_2 are the viscosities of water and air, respectively; α is the fluid volume fraction, which is set to 1 in the water region, 0 in the air region and between 0 and 1 for the interface; P is the pressure; g_i is the component of gravitational acceleration in the x_i-direction; T is the temperature; λ is the thermal conductivity; C_p is the specific heat capacity; δ_{ij} is the Kronecker delta; k is the turbulence kinetic energy; μ_t is the turbulent viscosity.

The shear stress transport (SST) k-ω model is utilized for turbulence modelling and is stated as follows:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G_k - \rho\beta^* k\omega \tag{6}$$

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_j}(\rho u_j \omega) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + \frac{\alpha_1 F_1 + \alpha_2(1-F_1)}{\mu_t}\rho G_k - \rho[\beta_1 F_1 + \beta_2(1-F_1)]\omega^2 + 2(1-F_1)\frac{\rho}{\omega\sigma_{\omega,2}}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \tag{7}$$

$$\Gamma_k = \mu + \mu_t[F_1/\sigma_{k,1} + (1-F_1)/\sigma_{k,2}] \tag{8}$$

$$\Gamma_\omega = \mu + \mu_t[F_1/\sigma_{\omega,1} + (1-F_1)/\sigma_{\omega,2}] \tag{9}$$

$$\mu_t = \frac{\rho k}{\omega}\frac{1}{\max\left[\frac{1}{\alpha^*}, \frac{SF_2}{a_1\omega}\right]} \tag{10}$$

$$F_1 = \tanh\left\{\min\left[\max\left(\frac{\sqrt{k}}{0.09\omega y_w}, \frac{500\mu}{\rho y_w^2\omega}\right), \frac{4\rho k}{\sigma_{\omega,2}D_\omega^+ y_w^2}\right]\right\}^4 \tag{11}$$

$$F_2 = \tanh\left[\max\left(\frac{2\sqrt{k}}{0.09\omega y_w}, \frac{500\mu}{\rho y_w^2\omega}\right)\right]^2 \tag{12}$$

$$D_\omega^+ = \max\left(\frac{2\rho}{\sigma_{\omega,2}\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}, 10^{-10}\right) \tag{13}$$

$$\alpha^* = \frac{0.144\mu\omega + \rho k}{6\mu\omega + \rho k} \tag{14}$$

$$S = \left[\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right]^{1/2} \tag{15}$$

$$G_k = \min(\mu_t S^2, 10\rho\beta^* k\omega) \tag{16}$$

where ω is the specific dissipation rate; y_w is the distance to the nearest wall; the constants for turbulence model are given as: $\sigma_{k,1} = 1.176$, $\sigma_{k,2} = 1.0$, $\sigma_{\omega,1} = 2.0$, $\sigma_{\omega,2} = 1.168$, $a_1 = 0.31$, $\alpha_1 = 0.556$, $\alpha_2 = 0.44$, $\beta^* = 0.09$, $\beta_1 = 0.075$, $\beta_2 = 0.0828$.

According to the VOF based free-surface capturing method, the scalar transport equation for the volume fraction is written as follows:

$$\frac{\partial \alpha}{\partial t} + \frac{\partial}{\partial x_j}(\alpha u_j) = 0 \tag{17}$$

Finally, the state equation of ideal gas is employed for the closure of equations and reads as follows:

$$P = \rho RT \tag{18}$$

where R is the ideal gas constant;

2.2. Governing equations of rigid body motion

The body motion in the present study involves a ship's compartment vertically sinking in calm water. Only one degree of freedom (DOF) of the moving body, i.e., heave motion, is considered in the numerical simulation. The heave motion of rigid body is determined by solving the following equations: