A MULTIPHASE CFD METHOD FOR PREDICTION OF FLOODWATER DYNAMICS

Zhiliang Gao[1], Dracos Vassalos[1], Qiuxin Gao[1]
[1] Department of Naval Architecture and Marine Engineering,
Universities of Glasgow and Strathclyde, Glasgow, G4 0LZ, UK,
Correspondence Email: zhiliang.gao@strath.ac.uk

ABSTRACT

When a vessel is damaged, seawater floods into the damaged compartments and subsequently influences the motion of the vessel. Interactively, the vessel behaviour would affect the motion of floodwater. In order to predict this coupled motion effectively, it is required to accurately calculate the floodwater dynamics. In this study, a numerical tool based on multiphase Computational Fluid Dynamics (CFD) method is developed to predict the dynamics of floodwater. The Volume of Fluid (VOF) method is used to capture the fluid interface. The governing equations are discretized by Finite Volume (FV) method. The SIMPLE algorithm is employed for pressure-velocity coupling. In order to verify the method, the dam break problem, tank sloshing problem and damaged compartment flooding problem are solved. The numerical results are satisfactory in comparison with experimental results, analytical results or published numerical results.

Keywords: *floodwater dynamics, CFD method, VOF method*

1. INTRODUCTION

For damaged vessel, proper lifesaving measures and evacuation procedures are vital to safety of human life. In order to establish relevant guidelines, clear understanding of the dynamic behaviour of a damaged vessel and the process of water flooding into it is essential. In this process, vessel motion affects water flooding and water sloshing in the compartment and conversely, liquid loads due to water sloshing in the compartment influence the vessel motion. To solve this interactive dynamic problem effectively, it is necessary to correctly predict the dynamics of floodwater.

The classical hydraulic model is widely used to calculate the floodwater dynamics in previous studies (Santos et al., 2002; Palazzi and De Kat, 2004; Ruponen, 2007). The inflow and outflow of water through the damaged opening is determined by the modified empirical Bernoulli's equation. The motion of floodwater inside the compartment is ignored and its free surface is assumed to be horizontal. An improved model for the internal water motion was proposed by Papanikolaou et al. (2000), in which the internal water is considered as a lump mass moving freely over a specific path surface, yet the surface of water is assumed to remain flat. A more sophisticated model for calculating the internal water dynamics is using shallow water equation (Santos and Guedes Soares, 2003; Valanto, 2006; Santos and Guedes Soares, 2008). Although all the approaches mentioned above are practical and efficient to predict the floodwater motion and its impact on the vessel, there are some limitations. Firstly, a simple hydraulic model drives the water ingress/egress through the opening, and hence the relevant physics of the flow are not fundamentally simulated. Secondly, the surface of floodwater in the compartment is assumed to be either horizontal or flat. In the case when the vessel undergoes large amplitudes of motion, these approaches lack the ability to model the violent flows with non-linear free surface, even though employing the shallow water theory. Thirdly, the above models can not fully consider the influences of the geometry of damaged opening and the internal layout of complex compartment on the motion of floodwater. Therefore, it is required to find more effective and accurate method to predict the floodwater dynamics.

During the past two decades, the method of Computational Fluid Dynamics (CFD) developed rapidly. It has been widely and successfully used to solve ship hydrodynamic problems. Although numerical simulation of water flooding into damaged vessel based on CFD method is very time consuming, some works have been presented in the past few years. González et al., (2003) and Skaar et al., (2006) used Smoothed Particle Hydrodynamics (SPH) method to model the water flooding process. It is shown that the SPH method is suitable to treat complex free surface motion. However, the required number of particles is large and the modelling of boundaries of the computational domain is still problematic. Compared to the novel SPH method for practical use, the Volume of Fluid (VOF) method, proposed by Hirt and Nichols (1981), has become the most popular method for calculating free surface flows. Many studies have shown that the VOF method is capable to capture sharp interface even with large scale overturning and deforming. Therefore, it is feasible and promising to apply CFD method based on VOF technique to prediction of floodwater dynamics, and some efforts have been made on this application (Van't Veer and De Kat, 2000; Gao, 2001; Woodburn et al., 2002; Gao et al., 2004; Cho et al., 2005).

In this study, a numerical tool based on multiphase CFD method is developed to predict the dynamics of floodwater. The VOF method is used to capture the fluid interface. The governing equations are discretized by Finite Volume (FV) method. The SIMPLE algorithm is employed for pressure-velocity coupling. In order to verify the method proposed, it is used to solve the dam break problem, tank sloshing problem and damaged compartment flooding problem. The results obtained are compared with the experimental results, analytical results or published numerical results, which shows that the present method is effective for predicting the floodwater dynamics.

2. MATHEMATICAL MODELS

The present paper considers incompressible flow with two different fluids (water and air). One-fluid formulation for a two-phase flow is used here. According to this model, continuity equation and Navier-Stocks (NS) equation are stated as below:

$$\nabla \cdot \boldsymbol{u} = 0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(\rho_m \boldsymbol{u}\right) + \nabla \cdot \left(\rho_m \boldsymbol{u}\boldsymbol{u}\right) = \nabla \cdot \left(\mu_m \nabla \boldsymbol{u}\right) - \nabla p + \rho_m \boldsymbol{g} \tag{2}$$

where $\boldsymbol{u} = (u, v, w)$, u, v, w are the velocity components; $\rho_m = f\rho_w + (1-f)\rho_a$ is the mixture density; ρ_w, ρ_a are the density of water and air, respectively; $\mu_m = f\mu_w + (1-f)\mu_a$ is the mixture viscosity; μ_w, μ_a are the viscosity of water and air, respectively; f is the water 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; $\boldsymbol{g} = (0, 0, g)$, g is gravitational acceleration.

The conservative form of the scalar convection equation for the volume fraction f is as follow:

$$\frac{\partial f}{\partial t} + \nabla \cdot (\boldsymbol{u} f) = 0 \qquad (3)$$

It should be noted that since turbulent effects on the cases studied here are small, turbulence modelling is omitted.

3. NUMERICAL METHOD

In the present study, an in-house CFD code is used to calculate the dynamics of a multiphase flow. The governing equations (1) to (3) can be solved numerically. For this purpose, the equations are discretized by FV method based on non-staggered grid. Rhie-Chow (RC) interpolation (Rhie and Chow, 1983) is used to suppress the pressure oscillation. The diffusion term in NS equation is calculated by central differencing scheme, while the convection term is calculated by hybrid scheme which switches between central differencing scheme and first-order upwind scheme. The SIMPLE algorithm is employed for pressure-velocity coupling. The full implicit scheme is used to evaluate the time integral. The detail of discretization formats and algorithms used above may refer to Versteeg and Malalasekera (1995). In order to retain the physical profile of interface, the modified High Resolution Interface Capturing (HRIC) scheme (FLUENT User's Guide, 2006), which is originally proposed by Muzaferija et al. (1998), is used to evaluate convective term in the scalar equation for the transport of the volume fraction.

Boundary conditions for Eqs. (1) to (3) depend on the specific test case. In this paper, no-slip wall boundary condition which requires the fluid to stick to the wall and pressure outlet boundary condition which requires the pressure is fixed at a constant value are used. For transient calculations, all the variables need to be initialized.

4. NUMERICAL RESULTS

In order to verify the present method, it is firstly applied to the dam break problem. The numerical results are compared with experimental results and published numerical results. Then the method is used to solve the tank sloshing problem. The numerical results are compared with analytical results and experimental results. Finally, the method is used to solve damaged compartment flooding problem. The numerical results are given and compared with the experimental results.

4.1. Dam break problem

Firstly, the method proposed is applied to the dam break problem. Experiment has been performed for dam break flow in a tank of 3.22×1×1.8 m (Zhou et al., 1999), as shown in Fig. 1. After the flap was lifted, the water with an initial water height (h) equal to 0.6 m freely flowed into the void. During the experiment, water heights and pressure have been measured by using two water height probes (H1 and H2) and one pressure gauge (P1).

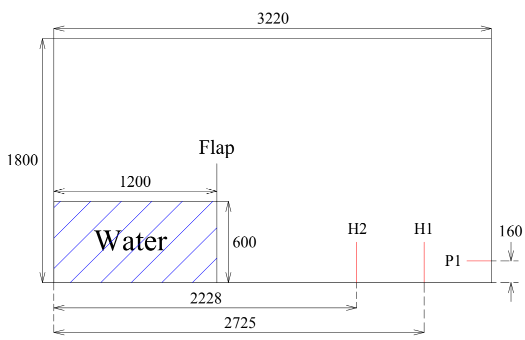


Figure 1. General description of the dam break problem (units: mm).

The dam break problem stated above can be considered as two-dimensional flow problem. In the present computation, the employed mesh system has 240 uniform horizontal meshes and 135 uniform vertical meshes. The computational time step is 0.001 s. No-slip wall condition is imposed on the whole boundary. The present numerical results are compared with experimental results and numerical results of SPH method (Colagrossi and Landrini, 2003). Figs. 2 and 3 show the comparison of the water heights at H1 and H2, respectively. It can be seen that the present results are satisfied well with the published results in the first initial stage of the test. The results disagree after the water wave breaks ($t\sqrt{(g/h)}>6$). Since it becomes difficult to measure the water level accurately in the experiment after the water wave breaks, it is hard to explain the differences. The comparison of the pressure history at P1 is shown in Fig. 4. As can be seen, the numerical results compare well with the experimental results and SPH results. The first peak around $t\sqrt{(g/h)}=6$, resulting from the broken wave hitting the water surface, is better predicted by the present method. The comparison of interface profile calculated by the present method and SPH method are shown in Fig. 5. At the initial stage, the flow is smooth and with a simple interface. The interface profiles obtained by the two methods are identical. After the water wave over turns and breaks, the flow becomes chaotic. The differences of interface profile obtained by the two methods are obvious.

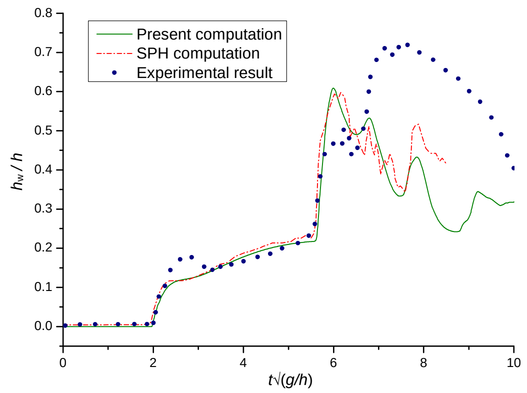


Figure 2. Comparison of the time history of vertical water height at H1.

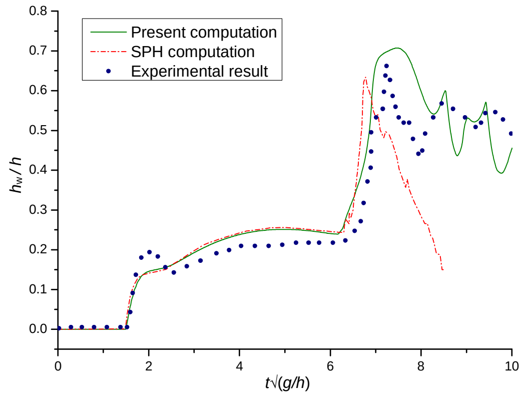


Figure 3. Comparison of the time history of vertical water height at H2.

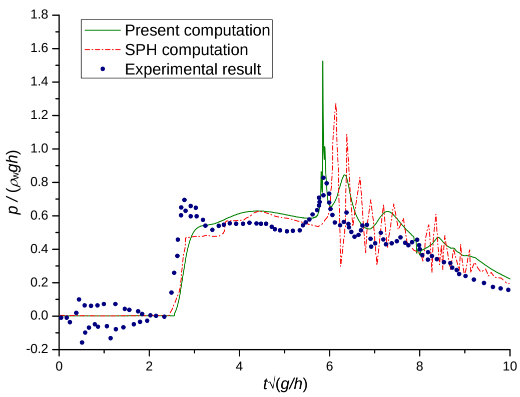
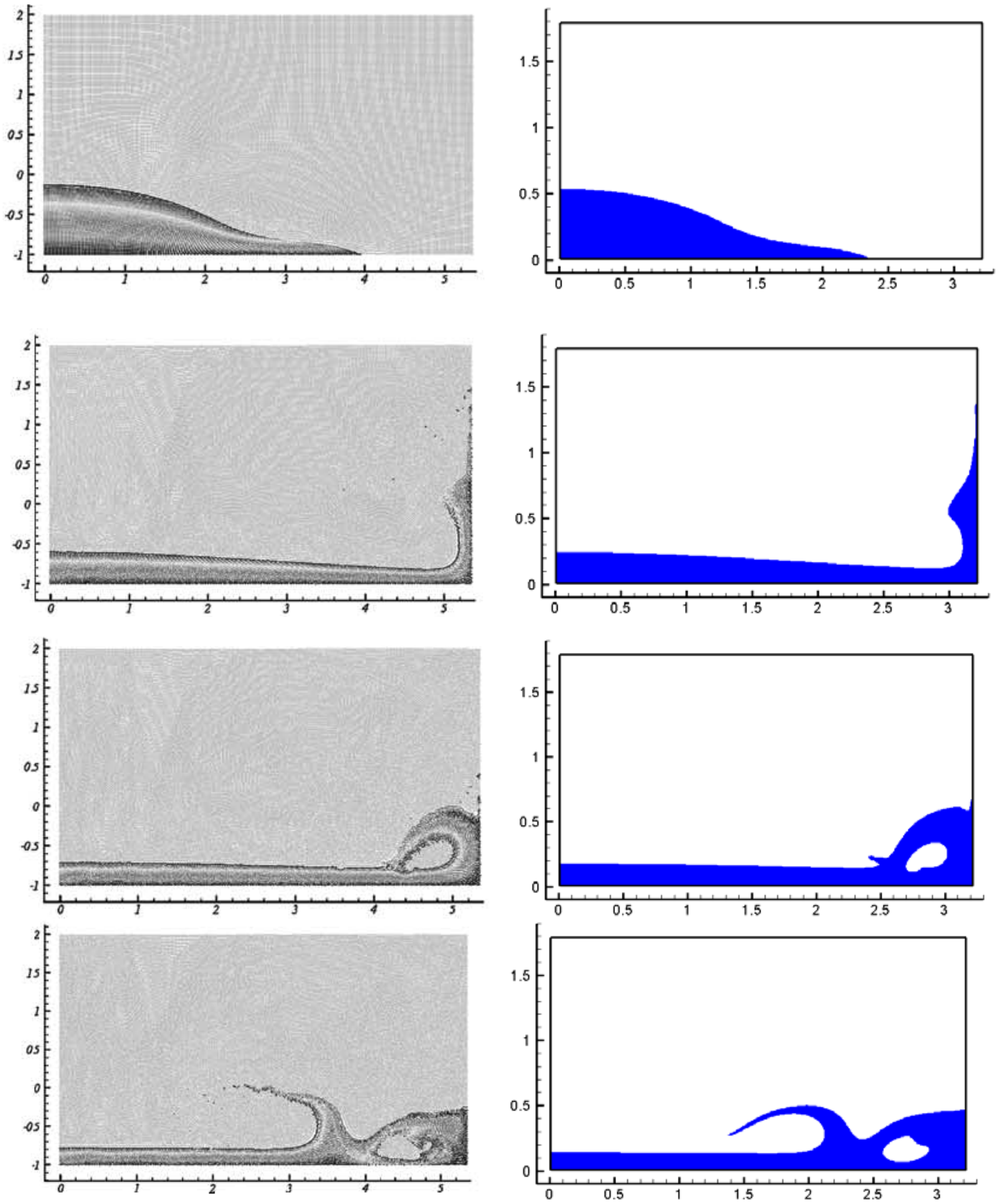


Figure 4. Comparison of the time history of pressure at P1.



(a) SPH computation (b) Present computation

Figure 5. Comparison of interface profile at $t\sqrt{(g/h)}$=1.66, 4.81, 6.17 and 7.37.

4.2. Tank sloshing problem

Secondly, the present method is used to solve the tank sloshing problem. The experiments of liquid sloshing in a horizontally excited hexahedron tank have been conducted (Liu and Lin, 2008). The tank is 0.57 m long, 0.31 wide and 0.3 m high. The still water depth (h) is 0.15 m. So the lowest natural frequency (ω_0) of liquid in the tank is 6.0578 s^{-1}. The tank is secured on a shaker whose movement follows the sinusoidal function: $x=-a\sin(\omega t)$. The shaking amplitude (a) is 5 mm and the frequencies (ω) are $0.583\omega_0$ and $1.0\omega_0$, corresponding to non-resonance and resonance case, respectively. The tank is equipped with three wave gauges to measure the elevation of water surface. The setup of experiment is shown in Fig. 6.

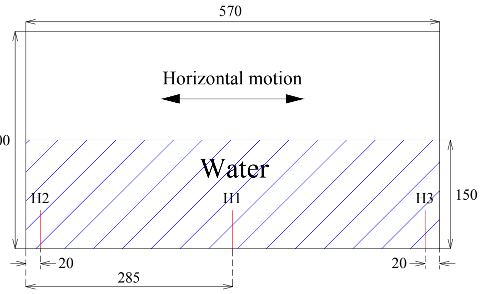


Figure 6. General description of the tank sloshing problem (units: mm)

In order to obtain the effect of moving tank acting on the fluid and avoid the treatment of complicated boundary condition on moving wall, a source term ($-a\omega^2\rho_m\sin(\omega t)$) is added to the right hand side of the *x*-component NS equation. The above sloshing problem can be simplified as two-dimensional problem. No-slip wall condition is imposed on the whole boundary. For the non-resonance case ($\omega=0.583\omega_0$), the employed mesh system has 114 uniform horizontal meshes and 50 non-uniform vertical meshes with 20 uniform meshes ($\Delta z=0.001$ m) being arranged near the free surface. The computational time step is 0.001 s. The numerical solutions of free surface elevation in comparison with the linear analytical solutions (Faltinsen, 1978) and the experimental results are shown in Figs. 7 to 9. As can be seen, the numerical results are in good agreement with the published results. For the resonance case ($\omega=1.0\omega_0$), the employed mesh system has 114 uniform horizontal meshes and 60 uniform vertical meshes. The computational time step is 0.001 s. Figs. 10 to 12 show the comparison of free surface elevation. As can be seen, the linear analytical solution fails to predict the non-linear wave, while the resonant phenomenon is well reproduced by the present method and fairly good agreements between the numerical results and experimental results are obtained. It is shown that the present method is effective to predict the non-linear sloshing motion.

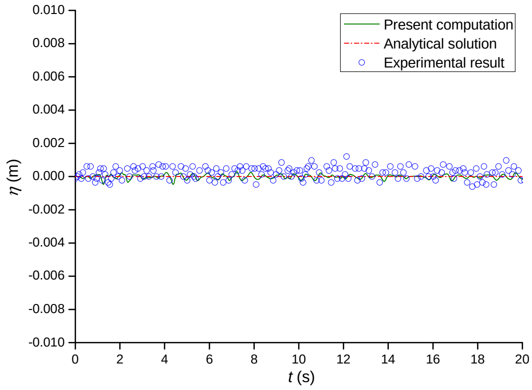


Figure 7. Comparison of the time history of free surface elevation at H1 ($\omega=0.583\omega_0$).

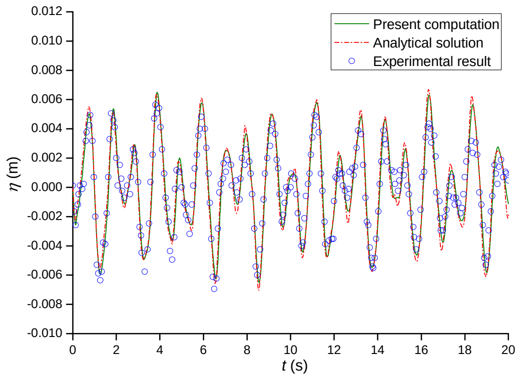


Figure 8. Comparison of the time history of free surface elevation at H2 ($\omega=0.583\omega_0$).

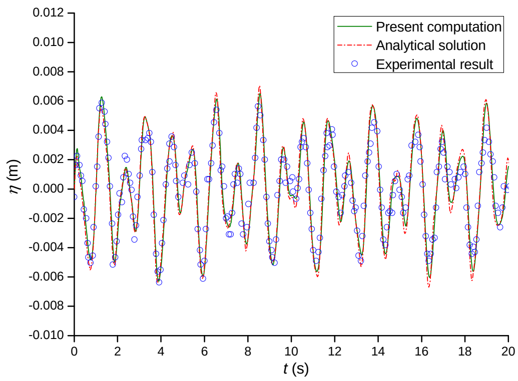


Figure 9. Comparison of the time history of free surface elevation at H3 ($\omega=0.583\omega_0$).

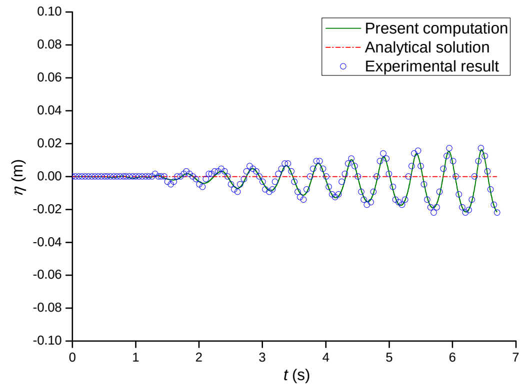


Figure 10. Comparison of the time history of free surface elevation at H1 (ω=1.0ω_0)

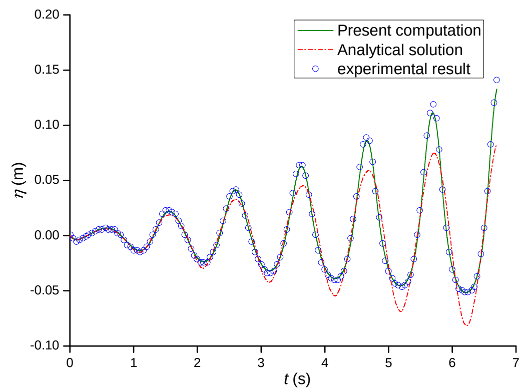


Figure 11. Comparison of the time history of free surface elevation at H2 (ω=1.0ω_0)

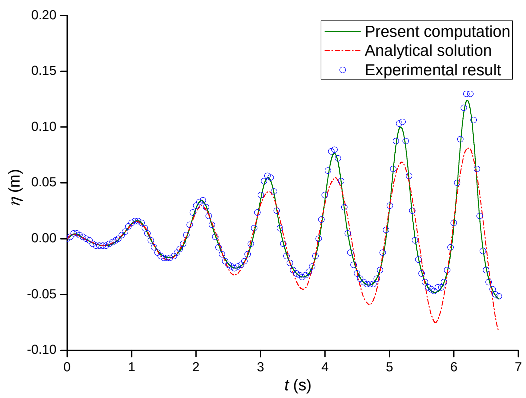


Figure 12. Comparison of the time history of free surface elevation at H3 (ω=1.0ω_0).

4.3. Damaged compartment flooding problem

Finally, the present method is used to simulate water flooding into damaged compartment. Cho et al. (2005) have conducted a series of model tests for the damaged compartment of Ro-Ro passenger ferry. Two compartment models shown in Fig. 13 are adopted in their study. The floating positions of the models are fixed in the experiments. The length (L), breadth (B), height (H) and draft (D) of the model is 0.55 m, 0.515 m, 0.186 m and 0.132 m, respectively. The damaged opening (0.17 m in length) is on the starboard side.

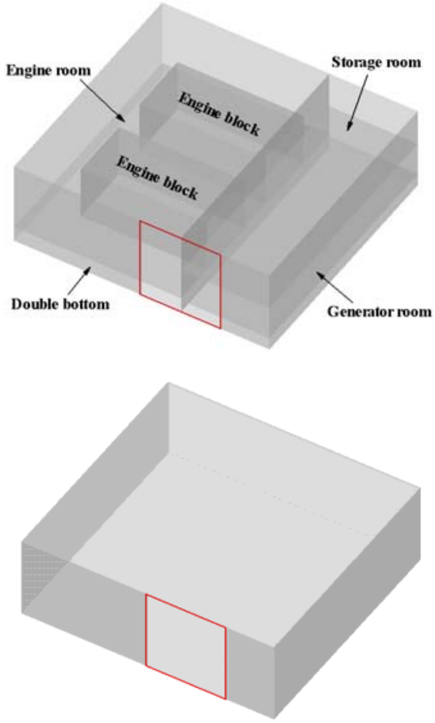


Figure 13. Models of damaged compartment (Upper: Real model, Under: Simple model).

In this study, the simple model is used for the numerical simulation. The outline of the computational domain is shown in Fig. 14. The computational region is extended to 3L on the left and right, 3B on the front and back, 1H on the bottom. No-slip wall boundary condition is imposed on the left, right, front, back and bottom of the computational domain and on the compartment boundary. Pressure outlet boundary condition is imposed on the top of the computational domain. The total number of mesh arranged in the numerical simulation is

$100 \times 101 \times 26 = 262600$. The computational time step is 0.001 s. The comparison of z-direction force history which indirectly reflects the flow magnitude through the opening is shown in Fig. 15. As can be seen, the trend of the calculated *z*-force changing with the time is in good agreement with that of experimental results. It is indicated that the ingress of floodwater calculated by the present method is consistent with that in the experiment. It may be observed that the values of *z*-force predicted by the present method are about 4% less than those obtained by the experiment. Since the averaged water level outside the compartment would drop 2% in the numerical simulation, but it remains constant in the experiment. So the existence of error between the numerical results and the experimental results is acceptable. Fig. 16 shows the numerical prediction of the motion of floodwater in the compartment. As can be seen, the water floods into the damaged compartment promptly and reaches to the opposite wall of the inlet in 0.43 s. After that, overturning wave is formed and the flow becomes violent. The flow is back to the inlet in 1.29 s.

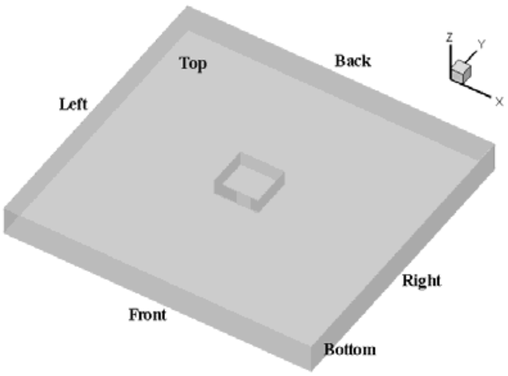


Figure 14. Computational domain of compartment flooding problem.

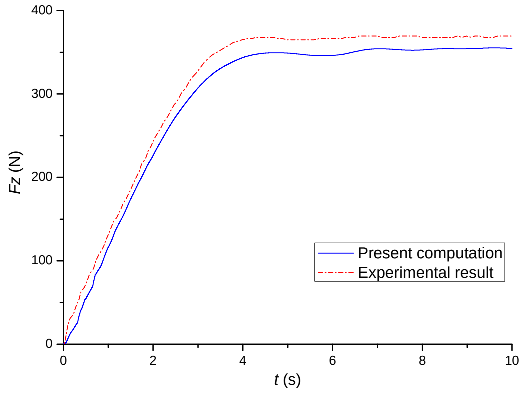
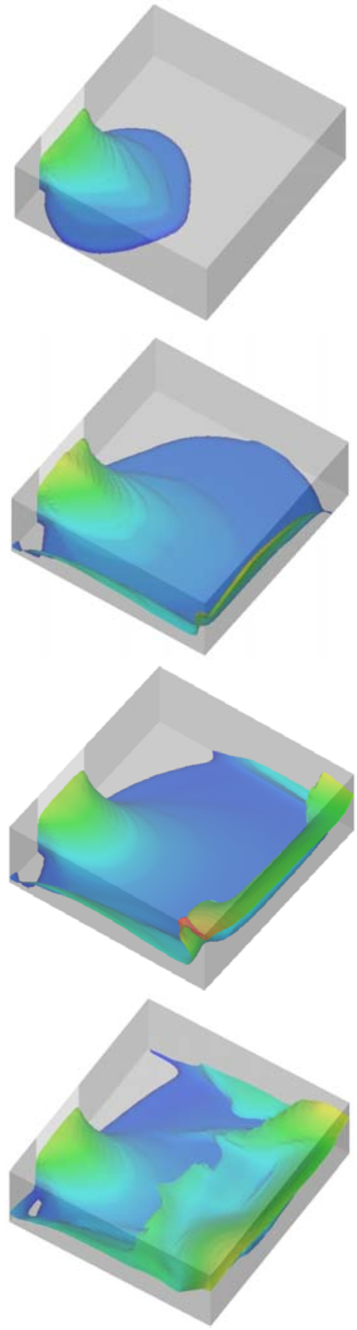


Figure 15. Comparison of the time history of *z*-direction force.



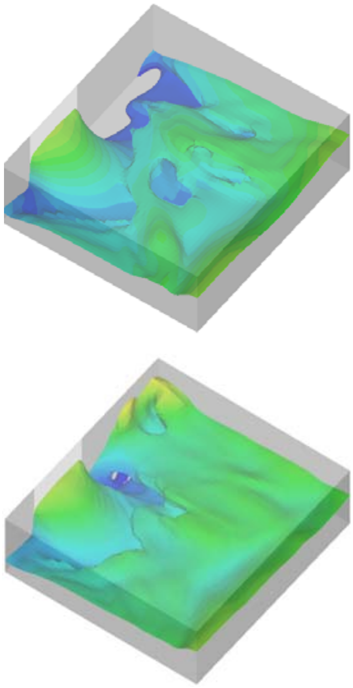


Figure 16. Snap shots of water flooding at t=0.215, 0.43, 0.645, 0.86, 1.075 and 1.29 s.

5. CONCLUSIONS

A numerical tool based on CFD method is developed to predict the dynamics of floodwater. The VOF method is used to capture the fluid interface. The governing equations are discretized by FV method. The SIMPLE algorithm is employed for pressure-velocity coupling. In order to verify the method proposed, it is firstly applied to the dam break problem. The numerical results are found to be satisfactory in comparison with experimental results and published numerical results. Then the present method is used to solve both the non-resonance and resonance liquid sloshing problems. In the non-resonance test, the results obtained are in good agreement with analytical solutions and experimental results. In the resonance test, the numerical results differ greatly with the linear analytical solutions which fail to predict the non-linear sloshing motion, while fairly good agreements between the numerical results and experimental results are obtained. Finally, the present method is used to simulate water flooding into damaged compartment. The comparison between numerical results and experimental is given and discussed. The study presented in this paper shows that the proposed method is capable to predict the floodwater dynamics.

Further work will be carried out in the future to simulate water flooding into real compartment. The effect of geometry of damaged opening on floodwater dynamics will also be investigated.

6. REFERENCE

Cho, S. K., Hong, S. Y., Kim, Y. H., Lee, K. J., 2005, Investigation of dynamic characteristics of the flooding water of the damaged compartment of an ITTC RORO passenger ship, Proceedings of the 8th International Ship Stability Workshop, Istanbul, Turkey.

Colagrossi, A., Landrini M., 2003, Numerical simulation of interfacial flows by smoothed particle hydrodynamics, Journal of Computational Physics, 191 (2), 448-475.

Faltinsen, O. M., 1978, A numerical nonlinear method of sloshing in tanks with two-dimensional flow, Journal of Ship Research, 22 (3), 193-202.

FLUENT 6.3 User's Guide, 2006, Fluent Inc.

Gao, Q., 2001, CFD simulation of water ingress into damaged ships, Journal of Ship Mechanics, 5 (3), 8-17.

Gao, Q., Kara, F., Shigunov, V., Vassalos. D., 2004, Numerical simulation of damage ship flooding, Proceeding of the 7th Numerical Towing Tank Symposium, Hamburg, Germany.

González, V., Talens, M., Riola, J. M., Valle, J., Quesda, T., Espín, M., 2003, Numerical prediction of the dynamic behaviour of a Ro-Ro ship after a hull side damage, Proceedings of the 8th International Conference on Stability of Ships and Ocean Vehicles, Madrid, Spain, 215-227.

Hirt, C. W., Nichols, B. D., 1981, Volume of fluid method for the dynamics of free boundaries, Journal of Computational Physics, 39 (1), 201-225.

Liu, D., Lin, P., 2008, A numerical study of three-dimensional liquid sloshing in tanks, Journal of Computational Physics, 227 (8), 3921-3939.

Muzaferija S., Peric M., Sames P., Schelin T., 1998, A two-fluid Navier-Stokes solver to simulate water entry, Proceeding of the 22nd Symposium on Naval Hydrodynamics, Washington, United States, 638-651.

Palazzi, L., De Kat, J. O., 2004, Model experiments and simulations of a damaged ship with air flow taken into account, Marine Technology, 41 (1), 38-44.

Papanikolaou, A., Zaraphonitis, G., Spanos, D., Boulougouris, E., Eliopoulou, E., 2000, Investigation into the capsizing of damaged Ro-Ro passenger ships in waves, Proceedings of the 7th International Conference on Stability of Ships and Ocean Vehicles, Tasmania, Australia, 351-362.

Rhie, C. W., Chow, W. L., 1983, Numerical study of the turbulent flow past an airfoil with trailing edge separation, American Institute of Aeronautics and Astronautics Journal, 21 (11), 1525-1532.

Ruponen, T., 2007, Progressive flooding of a damaged passenger ship, PhD thesis, Helsinki University of Technology.

Santos, T. A., Guedes Soares, C., 2003, Investigation into the effects of shallow water on deck in ship motions, Proceedings of the 8th International Conference on Stability of Ships and Ocean Vehicles, Madrid, Spain, 81-96.

Santos, T. A., Guedes Soares, C., 2008, Study of damaged ship motions taking into account floodwater dynamics, Journal of Marine Science and Technology, 13 (3), 291-307.

Santos, T. A., Winkle, I. E., Guedes Soares, C., 2002, Time domain modelling of the transient asymmetric flooding of Ro-Ro ships, Ocean Engineering, 29 (6), 667-688.

Skaar, D., Vassalos, D., Jasionowski, A., 2006, The use of a meshless CFD method in modelling progressive flooding and damaged stability of ships, Proceedings of the 9th International Conference on Stability of Ships and Ocean Vehicles, Rio de Janeiro, Brazil, 625-632.

Valanto, P., 2006, Time dependent survival probability of a damaged passenger ship II - Evacuation in seaway and capsizing, HSVA Report, No. 1661, Hamburg.

Van't Veer, R., De Kat, J. O., 2000, Experimental and numerical investigation on progressive flooding in complex compartment geometries, Proceedings of the 7th International Conference on Stability of Ships and Ocean Vehicles, Tasmania, Australia, 305-321.

Versteeg, H. K., Malalasekera, W., 1995, An introduction to computational fluid dynamics - The finite volume method, Longman Scientific and Technical.

Woodburn, P., Gallagher, P., Letizia, L., 2002, Fundamentals of damage ship survivability, Transactions of the Institution of Naval Architects, RINA, 144, 143-163.

Zhou, Z. Q., De Kat, J. O., Buchner, B., 1999, A nonlinear 3-D approach to simulate green water dynamics on deck, Proceeding of the 7th Internal Conference of Numerical Ship Hydrodynamics, Nantes, France, 5.1, 1-15.