Proceedings of the Twenty-fifth (2015) International Ocean and Polar Engineering Conference
Kona, Big Island, Hawaii, USA, June 21-26, 2015

ISBN 978-1-880653-89-0; ISSN 1098-6189
www.isope.org

Numerical Study on Liquid Sloshing in Three-dimensional Rectangular Tanks with Different Filling Rates and Fixed Baffle

*Chonghong Yin, Jianwei Wu and Decheng Wan**

State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University,
Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai, China
*Corresponding author

ABSTRACT

Tank sloshing is a phenomenon of liquid motion in partially filled container, arising from an external excitation, which attracts more and more attention with the development of large LNG carriers and other tankers. In this paper, we will build a three-dimensional rectangular tank based on the experimental model, and make an investigation numerically using RANS-VOF method by utilizing our in-house CFD solver naoe-FOAM-SJTU, which is based and developed on OpenFOAM. Firstly, numerical cases with different water filling rates: 30%, 50%, 70%, and 80% at the resonance frequency are simulated, results are compared with the experimental data from Daewoo to confirm the reliability of our calculations. And then fixed baffle to reduce the sloshing are simulated and compared.

KEY WORDS: Tank sloshing; VOF method; Baffles; RANS; naoe-FOAM-SJTU solver.

INTRODUCTION

Tank sloshing is a crucial issue faced in engineering applications. With increasing energy demand and LNG production, prediction of sloshing-induced impact loads on hull structures become one of the research interests in ship hydrodynamics. Recent years, with the rapid development of computer technology, Computational fluid dynamics (CFD) has become a powerful tool to the research of tank sloshing.

Many theoretical studies and numerical simulations corresponding with experimental validation are carried out on tank sloshing during last decades. A numerical non-linear method of sloshing in tanks was introduced by Faltinsen (1978). Wu GX et al.(1998) did the numerical simulation of sloshing waves in a 3D tank based on a finite element method. At the same time, different measures to reduce the sloshing-induced impact loads are also studied both numerically and experimentally. Three-dimensional liquid sloshing in tanks with baffles was studied by Liu (2009), C.G. Koh (2013) did the experiment and numerical simulations by CPM method about liquid sloshing with constrained floating baffle (CFB) .

The aim of our work in this paper is to construct a three-dimensional model of the sloshing tank, and study the liquid phenomenon of sloshing numerically under an external excitation. Both experimental observations and numerical simulations are carried out, and the results of experiments from Daewoo et al. are used for validation of our calculations. Tank models from Daewoo with different water filling rates 30%, 50%, 70%, 80% at the resonance frequency are simulated, despite the comparison of free surface with experiment, we also decorate several pressure points to get impact loads on the bulkhead, and compare with experimental results. Next, a vertical baffle at filling rate 70% is simulated and analyzed. In this paper, all the computations are performed by our in-house CFD solver naoe-FOAM-SJTU (Shen and Wan, 2012), which is based on the open source code OpenFOAM and secondary developed for solving hydrodynamics problems of ship, tank sloshing and ocean engineering.

MATHEMATICAL EQUATIONS

Governing Equations

In this paper, Reynolds-Average Navier-Stokes (RANS) equations coupled with VOF method are adopted for incompressible viscous fluid flows in sloshing tanks. The governing equations are:

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

$$\frac{\partial \rho \boldsymbol{U}}{\partial t} + \nabla \cdot (\rho(\boldsymbol{U} - \boldsymbol{U}_g)\boldsymbol{U}) = -\nabla p_d - \boldsymbol{g} \cdot \boldsymbol{x} \nabla \rho + \nabla \cdot (\mu \nabla \boldsymbol{U}) \tag{2}$$

where $\boldsymbol{U}$ and $\boldsymbol{U}_g$ are velocity fields for fluid and grid nodes respectively; $p_d = p - \rho \boldsymbol{g} \cdot \boldsymbol{x}$ is dynamic pressure, and $\boldsymbol{g}$ is the gravity acceleration vector; ρ and μ are the mixture density and dynamic viscosity for two-phase fluid.

VOF method

VOF method(Rusche, 2002) is applied to capture the free surface in this work. This method has the advantage of being able to control numerical diffusion and can also provide high accuracy, which is suitable for the simulation of liquid sloshing. The transport equation for α is defined as follows:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot \left[\left(\boldsymbol{U} - \boldsymbol{U}_g \right) \alpha \right] + \nabla \cdot \left[\boldsymbol{U}_r \left(1 - \alpha \right) \alpha \right] = 0 \tag{3}$$

where the first two terms represent the volume fraction term in continuous equation, and the last term stands for the compressible term when free surface is considered. α denotes the volume fraction which is the volume percentage of liquid in one cell. To all of the cells, the value of α varies between 0 and 1:

$$\begin{cases} \alpha = 0 & \text{air} \\ \alpha = 1 & \text{water} \\ 0 < \alpha < 1 & \text{interface} \end{cases} \tag{4}$$

with the introduction of α, ρ and μ can be defined as:

$$\rho = \alpha \rho_l + (1 - \alpha) \rho_g \tag{5}$$

$$\mu = \alpha\mu_l + (1-\alpha)\mu_g \quad (6)$$

where the subscripts of *g* and *l* represent gas and liquid separately. The free surface term in equation (2) can be written as

$$f_\sigma = \sigma\kappa\nabla\alpha \quad (7)$$

where σ is the free surface tension whose value is 0.07 kg/s^2, and the curvature of free surface is denoted by $\kappa = -\nabla\cdot(\nabla\alpha/|\nabla\alpha|)$.

Discretization

In this work, both RANS equations and VOF transport equation are discretized by the FVM method. The computation domain is divided into separate cells. The flow field data are stored in the center of the cell; interpolation calculation is then operated to get the data in the faces of the cell, and area integral is performed to obtain the data in the whole domain. The discretization schemes applied in equation (2) are second-order upwind scheme for the convection term and second-order central scheme for the diffusion term; Van Leer scheme is used in equation (3) and the time derivative term in both equations is discretized by Euler scheme.

Velocity-pressure decoupling

The PISO algorithm(Issa, 1986) is utilized for velocity-pressure decoupling (VPD). The algorithm is originally used for unsteady and compressible flow, but later imported to solve steady and incompressible fluid problems. The whole process is divided into three steps: predict – correct – re-correct.

SLOSHING IN RECTANGULAR TANK

Model and Computational Domain

The sloshing tank model chosen in this paper is an experiment model from Daewoo Shipbuilding by Y.B. Lee and D.H. Kang (2005). Size parameters of the rectangular tank are shown as Fig.1. We also place several pressure measurement points on the bulkhead. The tank is 800mm length and 500mm high, also has a width of 350mm. Fig.2 shows the computational domain of the tank by the example of filling-rate 70%. During the experiment, the tank moved along the longitudinal direction. The tank position is defined as:

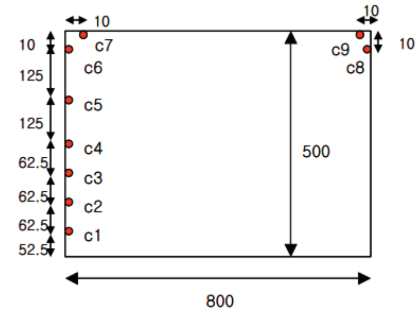
$$X = \zeta \sin\omega t \quad (8)$$

Where, ζ is the excitation amplitude, which is 20mm in this work, ω is the excitation frequency. The excitation frequency ω is determined based on the natural frequency ω_n of the fluid motion in the tank and ω_n is obtained by the linear theory as follow:

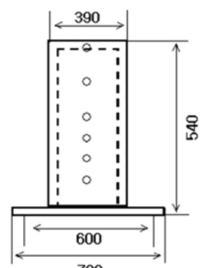
$$f = \frac{1}{2\pi}\sqrt{\frac{n\pi}{L} g \tanh\left(\frac{n\pi h}{L}\right)} \quad (9)$$

Corresponding to four different filling ratios 30%, 50%, 70% and 80% of the sloshing tank, we get the frequency ω calculated from the linear potential theory as shown in (9), which are respectively 4.513, 5.386, 5.819, 5,941.

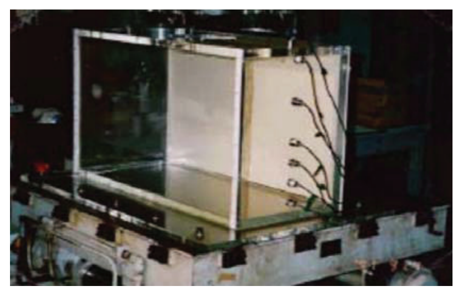
The utility *blockMesh*, provided by OpenFOAM, is employed to generate mesh and cell number is about 1.72 million.



(a)



(b)



c

Fig. 1 Size parameters of the experiment model

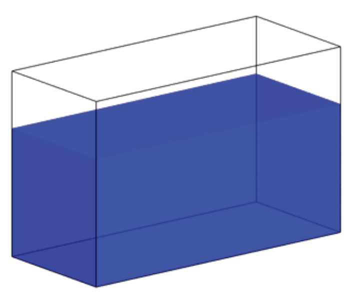


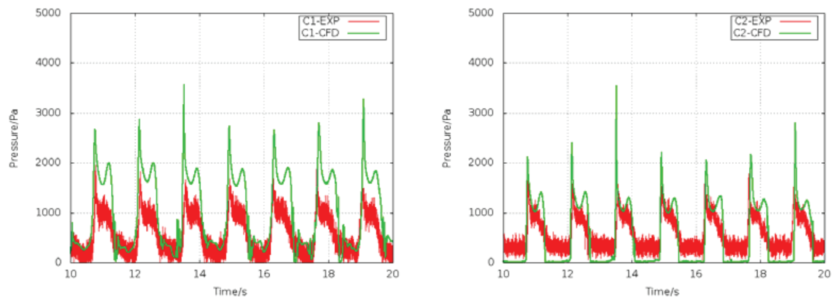
Fig. 2 Computational model

Results and Comparison

We did the numerical simulations for four tanks with different filling rate from 30% to 80%, and then time history of pressure at different measuring points are obtained and then analyzed.

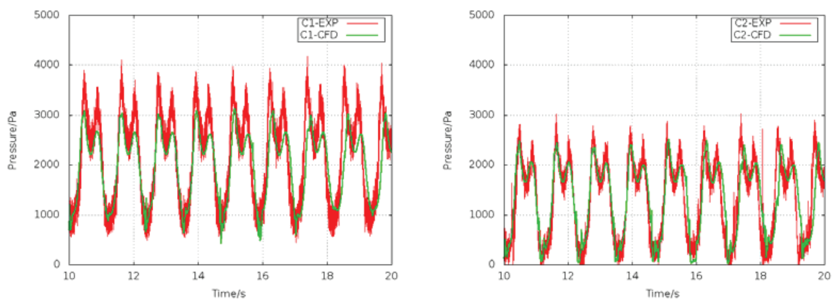
Figs.3~5 shows the time history of pressure curves at measuring points C1 and C2 in sloshing tanks from filling rate 30% to 70%. In Fig. 6, we show pressure curves at points C1~C4 under the 80% filling rate.

By comparing the numerical results and experiment results, we find that our simulation results under water filling rate 30% are not very satisfactory. Our CFD results are about 30% -50 % larger than experiment data. Also, as can be seen, the pressures obtained by C1, C2 under the water filling rate 50% and 70% show a good agreement with the experimental result in Fig. 4~5. Considering the lager error occurred in 30% condition, the main reason can be that in the case of fixed pressure measurement points, the 30% filling rate tank has a lower still water free surface, so the total pressure actual is mainly produced by slamming, the main component of slamming pressure is the dynamic pressure, which has a larger relative error.



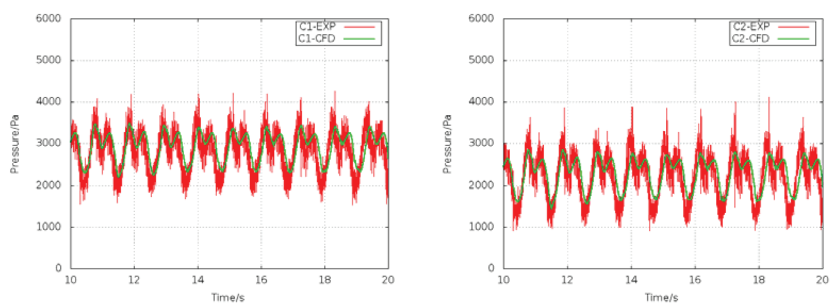
(a) (b)

Fig.3 Time history of pressure at point C1(a), and point C2(b), when filling rate is 30%



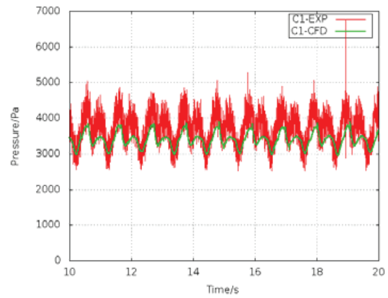
(c) (d)

Fig.4 Time history of pressure at point C1(c), and point C2(d), when filling rate is 50%

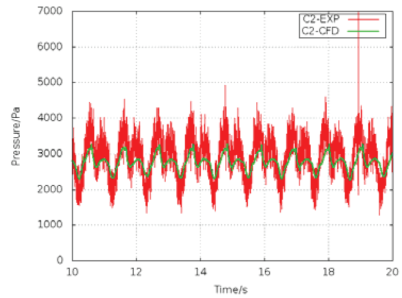


(e) (f)

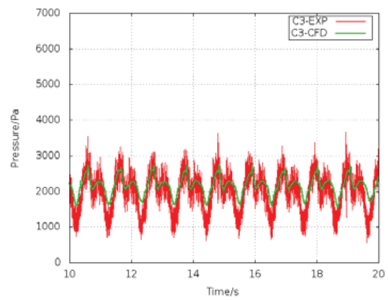
Fig.5 Time history of pressure at point C1(e), and point C2(f), when filling rate is 70%



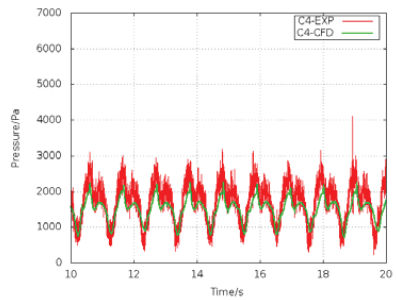
(g)



(h)



(i)



(j)

Fig.6 Time history of pressure at point C1 (g), point C2 (h), point C3 (i) and point C4 (j) when filling rate is 80%

During a period of the accelerate-slamming-scrolling process, the sloshing liquid produces two peaks in the time history of pressure curves. When the liquid just arrived at the far right of the tank bulkhead, the liquid with right direction movement changed to vertical instantaneously by the obstacle of the bulkhead, and form a jet. In this process, the liquid move upward along the bulkhead, also produce a maximum peak slamming on the measuring point C1 and C2. Slamming in the process is accompanied by broken, splash and other large free surface deformation.

The small peak of the pressure curve is generated by the secondary roll. Changes of the free surface within one sloshing period are shown in Fig. 7 taken filling-rate 70% as an example. We can also see that the changes of free surface by CFD simulations are very close to experimental results, but still not precise enough for simulations of liquid rolling and crushing.

With the increase in filling ratio, the main pressure components measured by the points gradually became the static pressure below the free surface, and the two pressure peaks shown in curves are mainly caused by ups and downs of the free surface, height difference between two peaks also gradually decreases.

In Fig. 4 and 6, we can see that the pressure signal at C1 has lager difference with experiment data than C2. Generally, impact on positions near and above the free surface is harder to capture than positions under water due to the complex phenomenon such as liquid-gas interaction and so on. As to the results in this paper, it may be caused by the RANS and VOF method using in this simulation, which don’t have sufficient accuracy to capture the pressure peak. And the greater the pressure peak, the larger the error.

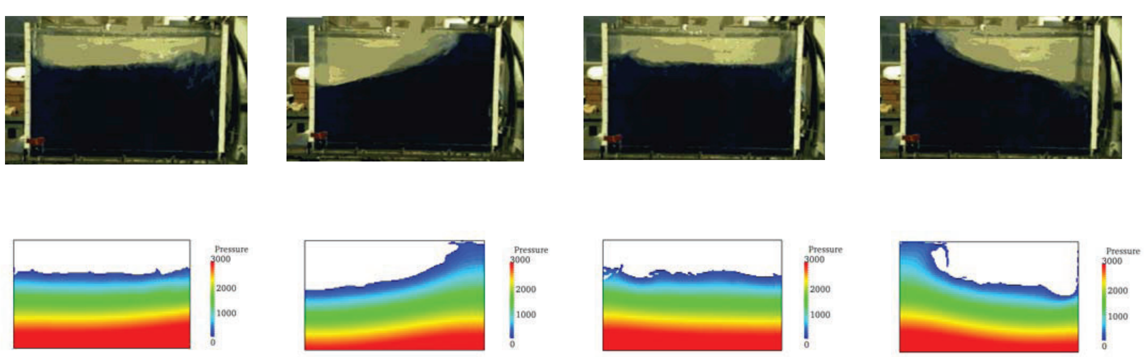


Fig.7 Wave profiles by CFD compared to experiment

SLOSHING IN RECTANGULAR TANK WITH BAFFLE

To explore the measures to reduce liquid sloshing, the effect of baffle in sloshing mitigation is investigated. We add a 100mm height vertical baffle at the middle of the sloshing tank. Take the 70% filling rate condition for example, the numerical computational model was shown as Fig. 8. Time history of pressure curves at point C1and C2 compared with experimental data and no baffle case results above was shown in Fig. 9.

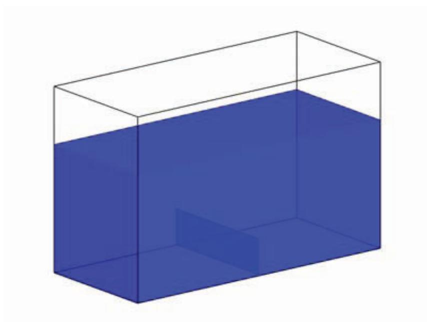
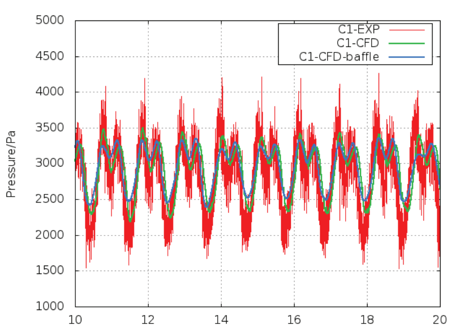


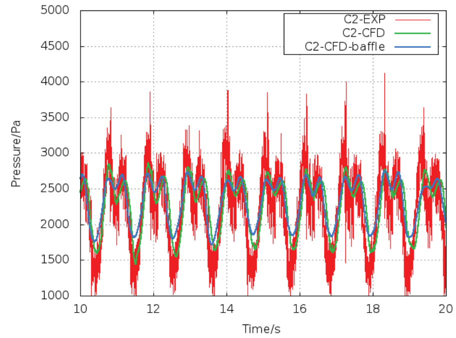
Fig. 8 Computational model with baffle

Although the height of the central baffle is only 100mm, less than one fifth of the sloshing tank height, it still played a significant effect in decreasing two pressure peaks, as can be seen in Fig. 9. Comparing to the impact on decreasing the two peaks of the time history of pressure curve, in this case, the vertical baffle played a more important role in increasing the pressure value at trough. This is because the baffle hindered the reflux liquid. At the same time, also because of the obstruction, after the first attack, the liquid will not drain quickly, so we can see that two pressure peaks come closer in the case with vertical baffle.

Next step, we choose the filling rate 50% of the sloshing tank to investigate the impact in sloshing mitigation of different heights vertical baffles. The simulations of this part are using 2D tank models. As shown in Fig. 10, by comparing four pressure-time curves corresponding to cases with different heights of baffles, we can find that the low height baffle's effect is not obvious, with the increasing of the baffle height from the 50 mm to 200 mm, the sloshing amplitude also gradually reduced. When the height of baffle reach to 200 mm, the pressure-time curve is no longer the previous waveform with two small peaks, but similar to form stable sine waveform.

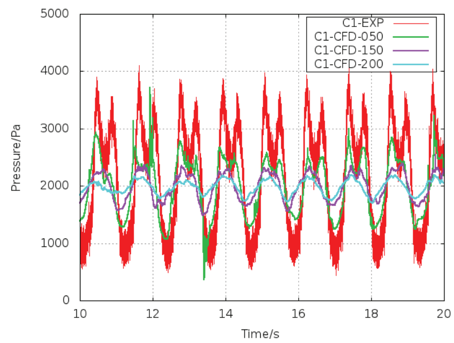


(a)

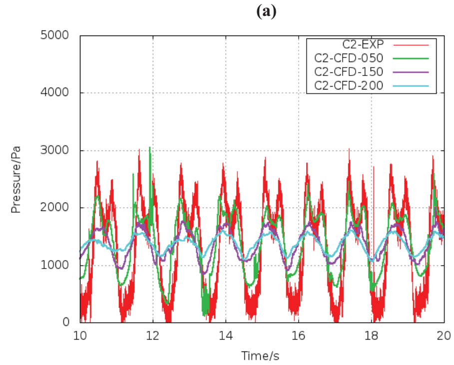


(b)

Fig.9 Time history of pressure at point C1(a), point C2(b) when filling rate is 70%



(a)



(b)

Fig.10 Time history of pressure at point C1 (a), point C2 (b) with different height of baffles when filling rate is 50%

In Fig. 11, we made a comparison of the velocity field in a period using sloshing tanks with and without baffle when filling rate is 50%, the height of the baffle is 100mm. The left column of Fig. 11 is the velocity field of the sloshing tank with baffle, and the right is no baffle situation. It can be found that for tank without baffle, the sloshing liquid has an obvious acceleration near the free surface; for sloshing liquid with vertical baffle, there will be a greater speed at the top of the baffle. But for the whole velocity field, the vertical baffle significantly reduced the acceleration of sloshing liquid caused by the forced vibration. Especially we can see that there are two low-speed areas in the bottom right corner and left corner of the rectangular tank, which did a good job to reduce the dynamic pressure of the liquid movement. By analysis of fluid's velocity field in the sloshing tank, we believe the baffle in tanks had a very big impact on the velocity field. The vertical baffle blocked the movement of the fluid from one side to the other to prevent velocity increases of the sloshing fluid, thereby reducing the dynamic pressure, and the amplitude of sloshing effects.

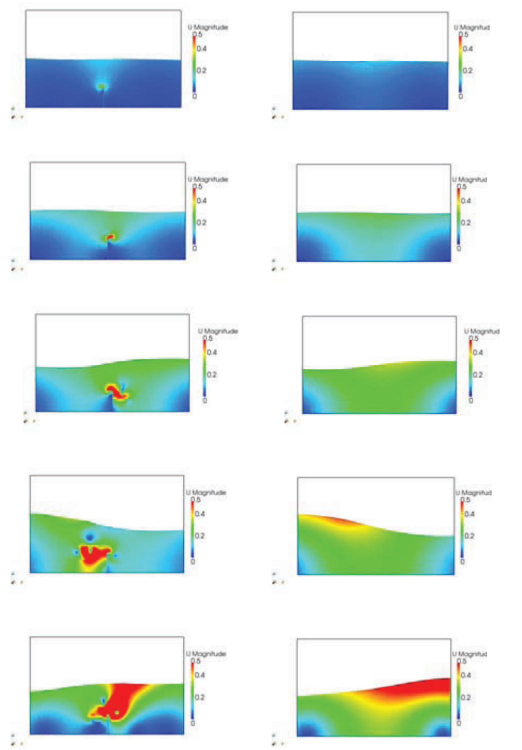


Fig.11 Velocity field comparison of the sloshing tank with and without baffle when filling rate is 50% at 5 typical times during a sloshing period.

CONCLUSIONS

In this paper, numerical simulations of the sloshing tank model from Kang D.H. and Lee Y.B with and without vertical baffles are presented. The computations are performed by our in-house CFD solver naoeFOAM-SJTU, which is developed based on the open source CFD package, OpenFOAM. Conditions with different filling rates and different heights of baffles are considered in this paper, and then we made a comparison on both time history of pressure and free surface.

First of all, simulations of a three-dimensional rectangular tank model with different water filling rates are compared with existing measurements data. Results indicate fairly agreement with the experimental measurements both on time history of pressure and free surface. Secondly, sloshing tank model is equipped with vertical baffle to investigate effects of baffles on liquid sloshing. The tank sloshing is computed by solving RANS equations. VOF method is used to capture the two phases interface. The numerical results show that the baffle in tanks had a very big impact on the velocity field, thereby influence the dynamic pressure and the amplitude of sloshing.

However, in current stage, only one simulation is conducted with the vertical baffle considered to reduce the liquid sloshing, further research can be conducted to simulate different tanks sloshing with different kinds of baffles. Considering the actual application in LNG tank, therefore, more internal structures and tank sloshing coupled with ship motions need to be computed in the future work to fully investigate the problem of ship motion coupled with LNG sloshing and reducing the slamming force on LNG tank.

ACKNOWLEDGEMENTS

This work is supported by National Natural Science Foundation of China (Grant Nos. 51379125, 51490675, 11432009, 51411130131), The National Key Basic Research Development Plan (973 Plan) Project of China (Grant No. 2013CB036103), High Technology of Marine Research Project of The Ministry of Industry and Information Technology of China, Chang Jiang Scholars Program (Grant No. T2014099) and the Program for Professor of Special Appointment (Eastern Scholar) at Shanghai Institutions of Higher Learning(Grant No. 2013022), to which the authors are most grateful.

REFERENCES

Faltinsen OM. (1978). "A numerical non-linear method of sloshing in tanks with two-dimensional flow." [J] *J Ship Res* 18:224–241.

Wu GX, Ma QW. Eatock-Taylor R (1998). "Numerical simulation of sloshing waves in a 3D tank based on a finite element method. " [J] *Appl Ocean Res* 20:337–355.

Dongming Liu, Pengzhi Lin (2009). "Three-dimensional liquid sloshing in a tank with baffles." [J] *Ocean Engineering* 36 202–212.

C.G. Koh, M. Luo, M.Gao, W. Bai (2013). "Modelling of liquid sloshing with constrained floating baffle." [J] *Computers and Structures* 122 270–279.

Kang D.H., Lee Y.B. (2005). "Summary Report of sloshing Model Test for Rectangular Model." *Daewoo Shipbuilding 7 Marine Engineering Co.*

Zhirong Shen and Decheng Wan (2012). "The manual of CFD solver for ship and ocean engineering flows: naoe-FOAM-SJTU" [R]. Shanghai Jiao Tong University.

Hakan Akyildiza, Erdem Unal (2005). "Experimental investigation of pressure distribution on a rectangular tank due to the liquid sloshing." [J] *Ocean Engineering* 32 1503–1516.

Shen Meng, Wang Gang, Tang Wenyong (2009). "Liguid Sloshing Analysis on Prismatic Tanks Based on Improved VOF Method" [J], *Shipbuilding of China*, 50(1), P1-9(Ch).

Yuxin Zhang, Decheng Wan (2012). "Numerical Simulation of Liquid Sloshing in Low-filling Tank by MPS" [J], *Chinese Journal of Hydroynamics*, 7(1), P100-107(Ch).