Proceedings of the Twenty-sixth (2016) International Ocean and Polar Engineering Conference **www.isope.org**
Rhodes, Greece, June 26-July 1, 2016

ISBN 978-1-880653-88-3; ISSN 1098-6189

Comparative studies of 3-D LNG tank sloshing based on the VOF and IMPS methods

Jianhua Wang[1,2]*, Decheng Wan*[1,2]**, Gang Chen*[1,2]*, Wenhua Huang*[3]
1 State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University,
2 Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai, China
3 School of Science, Huzhou University, Huzhou, China
*Corresponding author

ABSTRACT

The present work is focused on the comparative study of two numerical methods, i.e. MPS method and VOF method, for the prediction of sloshing in LNG tank. Numerical simulations are carried out by an in-house meshless solver MLParticle-SJTU based on improved Moving Particle Semi-Implicit (IMPS) method, and VOF based CFD solver naoe-FOAM-SJTU developed on the open source platform OpenFOAM. Several sloshing conditions with different kinds of tanks are applied to validate the present two numerical methods. The time histories of impact pressure and flow patterns are presented and compared to the experimental data. For the rectangular tank, sway motion with different excitation periods are taken into account to investigate the sloshing performance and validate the two simulation methods. For the membrane-type LNG tank, pitch motion with different excitation periods are simulated. According to the numerical results, the two methods can both predict the impact pressure compared with the experiment data. The VOF method and the MPS method show different flow patterns when encountering with breaking waves.

KEY WORDS: VOF; MPS; sloshing; MLParticle-SJTU solver; LNG tank; naoe-FOAM-SJTU solver

INTRODUCTION

Generally, sloshing is the motion of fluid with free surface in partially filled tanks and is of significant importance in the field of ship and ocean engineering. Sloshing flow is a highly nonlinear problem, which may involve complicated phenomena, such as breaking wave, high-speed impact on tank wall and overturning of free surface. Violent liquid sloshing in an oil or liquefied natural gas (LNG) ship can cause local breakage and global instability to the ship hull, and even lead to leakage of oil, and capsizing of ship (Shao et al., 2012). The influence of sloshing depends on the amplitude and excitation period of the tank motion, liquid-fill depth, liquid properties and tank geometry. When the excitation frequency is close to the highest natural frequency of the liquid, sloshing will be significantly violent, which is called resonance phenomenon. As mentioned above, it is essential to avoid the ship first natural frequency being too close to the dominant frequency of the environment condition to achieve a good motion performance.

Since sloshing can be a significant factor for the safety and stabilization of ship, many researchers have conducted a lot of corresponding research work. Early studies on sloshing are usually theoretical method based strong hypothesis (Faltinsen, 1978), where flow is irrational and geometry of tank is simple. Thus, analytical solution is invalid for sloshing in membrane-type LNG tank, especially when the tank is oscillated in resonance frequency. Traditionally, the experimental researches for sloshing problems are widely used (Akyildiz and Ünal, 2005; Bulian et al., 2014; Kim et al., 2015; Lugni et al., 2006) and experimental results can validate the numerical solutions.

With the development of computational fluid dynamics (CFD), numerical simulation has become an effective approach to study sloshing problem. In the past few years, many studies have been conducted on sloshing based on CFD methods. Kim (2001) applied the SOLA-SURF method to simulate sloshing flows in 2-D and 3-D containers and adopted a buffer zone concept to calculate the impact pressure on the tank ceiling. Kishev et al. (2005) proposed an improved constraint interpolation profile (CIP) method to investigate violent sloshing flow in a horizontally oscillating rectangular tank and validated with the experiment. Wu and Chen (2009) applied a finite difference method (FDM) solver to investigate sloshing waves in 3D liquid tank subjected to a range of excitation frequencies with motions that exhibit multiple degrees of freedom. Chen et al. (2009) analyzed the accuracy of numerically predicting impact pressure on the walls and ceiling of tanks based on level-set method. Ekachai and Chakrit (2012) used open source code OpenFOAM to simulate the three dimensional liquid sloshing model.

The above researches are based on the traditional CFD method, and an alternative approach to study the sloshing flow is the meshless method, such as SPH (Gingold and Monaghan, 1977; Lucy, 1977) and MPS (Koshizuka and Oka, 1996; Koshizuka et al., 1998). For meshless methods, the flow field is represented by a set of interacting particles. As particles have no fixed topography, meshless method is capable of handling large-deformed free surface problems. Furthermore, the fragmentation and coalescence of fluid can be naturally simulated in meshless methods. Another advantage for particle method is that there is no interface diffusion near the free surface since the particles are traced based on Lagrangian representation (Zhang and Wan, 2012; Zhang et al., 2014).

In the present study, both grid-based and meshless methods are used for the simulation of sloshing in LNG tanks. The numerical methods including improved MPS method, VOF based CFD method will be introduced first, where governing equations, free surface capturing and numerical schemes will be presented in detail. Then the next section will be the numerical simulations, where two tanks, namely rectangle type and membrane type LNG tank, are employed to numerically study the sloshing problem. Wave evolution and impact pressure will be presented and compared with experiment results. Finally, a summary of the study is drawn.

IMPROVED MPS METHOD

Governing Equations

For the MPS method, governing equations include the mass conservation equation and momentum conservation equation, which can be written as following:

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

$$\frac{D\boldsymbol{U}}{Dt} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\boldsymbol{U} + \boldsymbol{g} \tag{2}$$

where $\boldsymbol{U}$ is the velocity field, ρ is the fluid density， p is the pressure， ν is the kinematic viscosity, $\boldsymbol{g}$ is the gravity acceleration.

Particle Interaction Models

In the particle method, governing equations are transformed to the equations of particle interactions based on the kernel function. In the present work, we applied the improved kernel function (Zhang and Wan, 2012):

$$W(r) = \begin{cases} \dfrac{r_e}{0.85r + 0.15r_e} - 1 & 0 \le r < r_e \\ 0 & r_e \le r \end{cases} \tag{3}$$

where $r = |r_i - r_j|$ denotes the distance between two particles, r_e is the supported radius of the influence area of each particle. particle number density (PND) is defined as (Koshizuka et al., 1998):

$$< n >_i = \sum_{j \ne i} W(|\boldsymbol{r}_j - \boldsymbol{r}_i|) \tag{4}$$

The differential operators in the equation (1), (2) can be discretized by gradient model, divergence velocity model and Laplacian model respectively.

$$< \nabla P >_i = \frac{D}{n^0}\sum_{j \ne i}\frac{P_j + P_i}{|\boldsymbol{r}_j - \boldsymbol{r}_i|^2}(\boldsymbol{r}_j - \boldsymbol{r}_i)\cdot W(|\boldsymbol{r}_j - \boldsymbol{r}_i|) \tag{5}$$

$$< \nabla^2\phi >_i = \frac{2D}{n^0\lambda}\sum_{j \ne i}(\phi_j - \phi_i)\cdot W(|\boldsymbol{r}_j - \boldsymbol{r}_i|) \tag{6}$$

$$\lambda = \frac{\sum_{j \ne i} W(|\boldsymbol{r}_j - \boldsymbol{r}_i|)\cdot|\boldsymbol{r}_j - \boldsymbol{r}_i|^2}{\sum_{j \ne i} W(|\boldsymbol{r}_j - \boldsymbol{r}_i|)} \tag{7}$$

where D is the number of space dimensions, n^0 is the initial particle number density for incompressible flow, r represents the coordinate λ is a parameter introduced to keep the increase of variance equal to that of the analytical solution.

Incompressible Model

In traditional MPS method, the incompressible condition is achieved by keeping the particle number density constant. In the improved MPS method, the source term for PPE (Poisson equation of pressure) is consist of the divergence-free condition and the constant particle number density (Lee et al., 2011; Tanaka and Masunaga, 2010), and we adopted the following mixed source term:

$$< \nabla^2 P^{n+1} >_i = (1-\gamma)\frac{\rho}{\Delta t}\nabla\cdot V_i^* - \gamma\frac{\rho}{\Delta t^2}\frac{< n^* >_i - n^0}{n^0} \tag{8}$$

where： γ is a blending parameter with a value between 0 and 1, and we set $\gamma = 0.01$ for all MPS computations in the present work,.

Free Surface Capturing

In the traditional MPS, the particle with small PND can be considered as the free surface particle. Different from this idea, Zhang proposed an accuracy free surface detection method based on the asymmetric distribution of neighbor paraticles. In particular, a vector function is defined as follow (Zhang and Wan, 2011):

$$< \boldsymbol{F} >_i = \frac{D}{n^0}\sum_{j \ne i}\frac{1}{|\boldsymbol{r}_i - \boldsymbol{r}_j|}(\boldsymbol{r}_i - \boldsymbol{r}_j)W(\boldsymbol{r}_{ij}) \tag{9}$$

If particle satisfying:

$$<|\boldsymbol{F}|>_i \;>\; \alpha \tag{10}$$

is considered as surface particle, where α is a parameter, and has a value of $0.9|\boldsymbol{F}|^0$ in this paper, $|\boldsymbol{F}|^0$ is the initial value of $|\boldsymbol{F}|$ for surface particle.

It should be pointed out that Eq. (9) and Eq. (10) are not valid for splashed particle which has no or few neighbor particles, so it is only used for particles with number density between $0.8n^0$ and $0.97n^0$. Particles with number density lower than $0.8n^0$ is definitely surface particles, while those with number density higher than $0.97n^0$ should get pressure through Poisson equation.

VOF BASED CFD METHOD

Governing Equations

Generally, Navier-Stokes equations are used to describe the motion of fluid continuum. In terms of the unsteady incompressible two-phase fluid, the governing equations adopted here is the Unsteady Reynolds-Average Navier-Stokes (URANS) equations coupled with the volume of fluid (VOF) method. The equations can be written as a mass conservation equation and a momentum conservation equation, which are listed below:

$$\nabla \cdot \mathbf{U} = 0 \tag{11}$$

$$\begin{aligned}\frac{\partial \rho \boldsymbol{U}}{\partial t} + \nabla \cdot \left[(\rho \boldsymbol{U} - \boldsymbol{U}_g)\boldsymbol{U} \right] = -\nabla p_d - \boldsymbol{g} \cdot \boldsymbol{x} \nabla \rho + \nabla \cdot (\mu_{eff} \nabla \boldsymbol{U}) \\ + (\nabla \boldsymbol{U}) \cdot \nabla \mu_{eff} + f_\sigma\end{aligned} \tag{12}$$

where $\mathbf{U}$ represents the fluid velocity field shared by the two phase fluids throughout the flow domain and $\mathbf{U}_g$ the velocity of mesh points; $p_d = p - \rho \mathbf{g} \cdot \mathbf{x}$ is the dynamic pressure, obtained by subtracting the hydrostatic component from the total pressure; ρ is the mixture density of the two-phase fluid; $\mathbf{g}$ is the gravity acceleration; $\mu_{\text{eff}} = \rho(\nu + \nu_t)$ is the effective dynamic viscosity, in which ν and ν_t are the kinematic viscosity and kinematic eddy viscosity respectively, the latter one is obtained by the SST $k-\omega$ turbulence model (Menter, 1994); f_σ is a source term due to surface tension.

Free Surface Capturing

The volume of fluid (VOF) method with artificial compression technique is applied for locating and tracking the free surface (Hirt and Nichols, 1981). In the VOF method, each of the two-phase is considered to have a separately defined volume fraction (α), where 0 and 1 represent that the cell is filled with air and water respectively and $0 < \alpha < 1$ stands for the interface between two-phase fluid. The surface tension term in Eq. (12) is defined as $f_\sigma = \sigma \kappa \nabla \alpha$, where σ is the surface tension coefficient ($0.07 kg / s^2$ in water). κ is the curvature of surface interface, determined from the volume of fraction by $\kappa = -\nabla \cdot \left(\nabla \alpha / |\nabla \alpha| \right)$. The density and dynamic viscosity for the mixed fluid can be presented as:

$$\begin{aligned}\rho &= \alpha \rho_1 + (1-\alpha)\rho_2 \\ \mu &= \alpha \mu_1 + (1-\alpha)\mu_2\end{aligned} \tag{13}$$

The volume fraction function can be determined by solving the advection equation:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot \left[\left(\boldsymbol{U} - \boldsymbol{U}_g \right) \alpha \right] + \nabla \cdot \left[\boldsymbol{U}_r (1-\alpha) \alpha \right] = 0 \tag{14}$$

where the last term on the left-hand side is an artificial compression term to limit the smearing of the interface and $\boldsymbol{U}_r$ is a relative velocity used to compress the interface. This term has no meaning in the continuum formulation but is suitable to compress the interface in the discrete formulation, especially when the interface is not sharp enough.

Numerical Algorithm

The VOF based CFD solver applied in this paper is naoe-FOAM-SJTU(Shen and Wan, 2012), which is based on the open source CFD platform OpenFOAM. It is developed to deal with complex motion problems such as large-amplitude ship motion in waves, moving rudder and rotating propeller in ship self-propulsion and free maneuvering. In the present study, we use the 6DOF module to handle with the prescribed motion of LNG tanks. During the calculation, the URANS equations and VOF transport equation are discretized by the finite volume method (FVM), and for the discretized URANS equations, the merged SIMPLE- PISO (PIMPLE) algorithm is adopted to solve the coupled equation of velocity and pressure. The Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm allows to couple the Navier-Stokes equations with an iterative procedure and the Pressure Implicit Splitting Operator (PISO) algorithm enables the PIMPLE algorithm to deal with the pressure-velocity correction. More detailed description for the SIMPLE and PISO algorithm can be found in Ferziger and Peric (2012) and Issa (1986). Additionally, several built-in numerical schemes in OpenFOAM are used in solving the PDEs. The convection terms are discretized by a second-order TVD (Total Variation Diminishing Scheme) limited linear scheme, and the diffusion terms are approximated by a second-order central difference scheme. Van Leer scheme (van Leer, 1979) is applied for VOF equation discretization and a second-order backward Euler scheme is applied for temporal discretization.

NUMERICAL SIMULATIONS

In the present work, two kinds of LNG tanks, i.e. rectangle-type and membrane-type, are used to numerically study the sloshing problems. The rectangle-type tank is obtained from (Kang and Lee, 2005), and the model is built following the experimental setup. The membrane-type tank with its experimental data is from (Cai et al., 2011), and the main parameters are given in the following sections.

In this section, liquid sloshing in two kinds of tanks subjected to linear and angular motion are simulated to validate meshless method solver MLParticle-SJTU and the VOF based solver naoe-FOAM-SJTU in terms of flow patterns and impact pressure on probe locations.

Rectangular Tank

Numerical Model

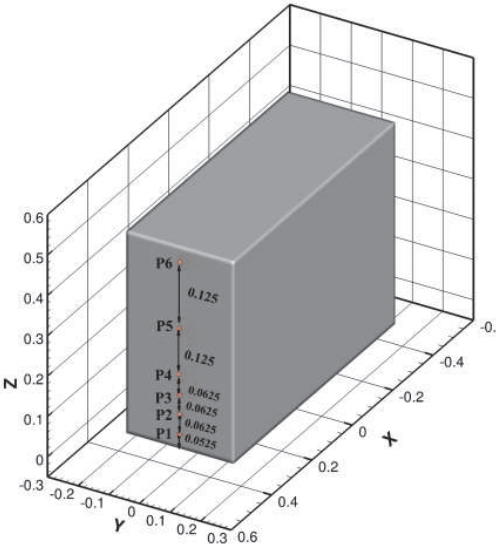


Fig. 1 Schematic of rectangle tank and the sensor location

Fig. 1 shows the geometry of the rectangle tank, which is the same as the experimental model given by Kang and Lee (2005). The length of the tank is L=0.8 m, the width of tank is W=0.35 m and its height is

H=0.5 m. The depth of water is d=0.35 m, corresponding filling ratio is 70%. The pressure probe locations are place on the left side wall, and the detailed information of sensor locations are shown in Fig. 1.

The tank is subject to sinusoidal horizontal motion:

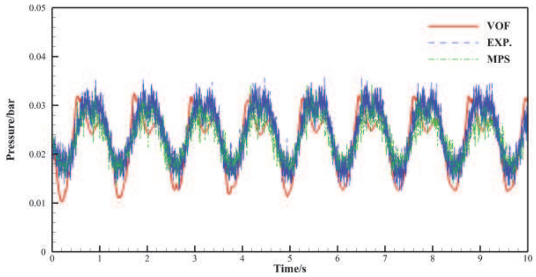
$$x = -A\cos(\omega t) \tag{15}$$

where: A=0.02 m， ω is excitation frequency, here $\omega = 5.8125$ rad/s is the first order resonant frequency of fluid motion. In this case, three excitation frequencies, namely w/w_n=0.9, w/w_n=1.0, w/w_n=1.1, are employed to validate the present two approaches in simulating sloshing problems.

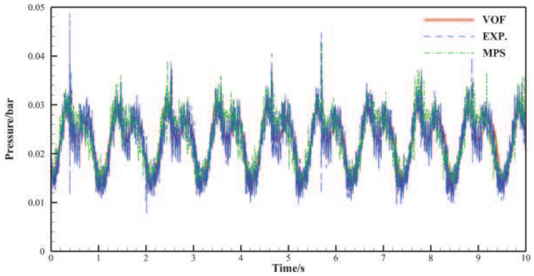
Numerical Results

In MPS simulation, 2D calculation is applied and the number of particles is 19287, among which 17313 are fluid particles, and the corresponding particle initial spacing is 0.004m. For the VOF calculation, the grid number is 0.14M, which is generated by the OpenFOAM utility blockMesh and all the cells are hexahedron type.

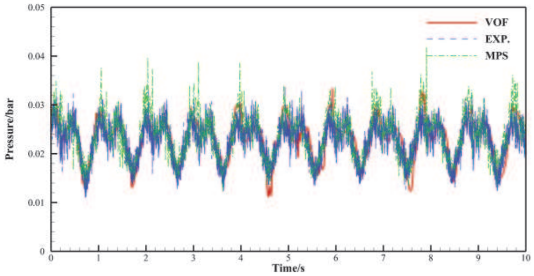
The numerical simulation for this case is conducted on the IBM nx360M4, which consist of 20 CPUs/node with 64GB memory per node, processor clock speed of 2.8GHz, and bandwidth of 56Gbps. The calculation are decomposed into 10 processors and the simulation time for VOF is 5420s and IMPS is about 7000s.



a) w/w_n=0.9



b) w/w_n=1.0

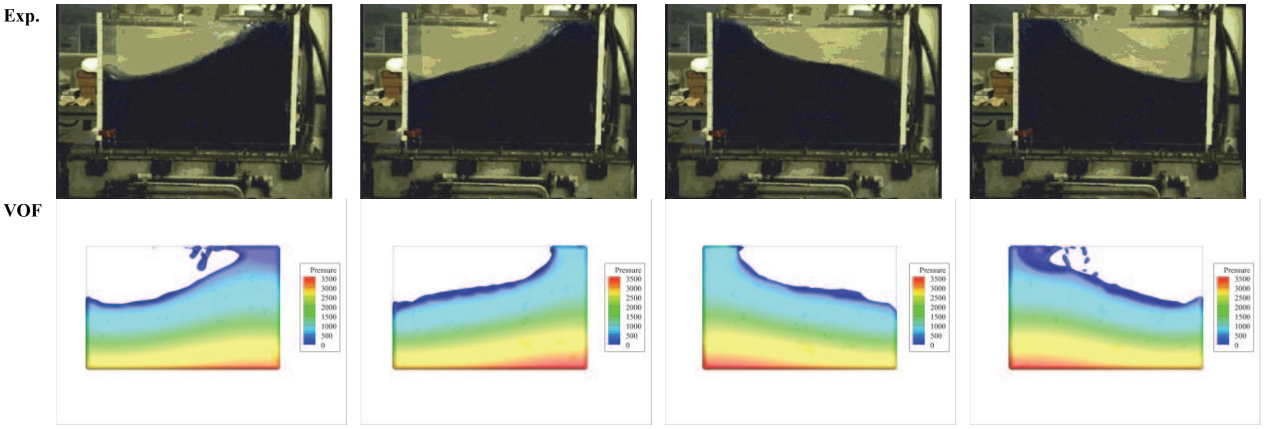


c) w/w_n=1.1

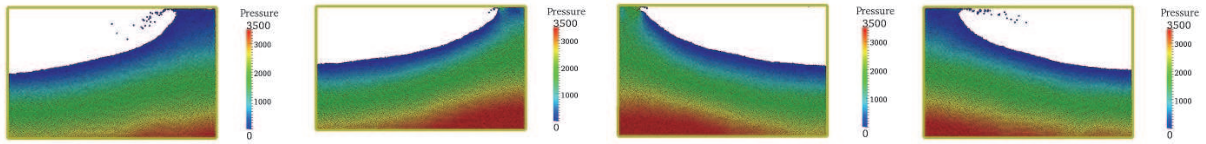
Fig. 2 Comparison of the impact pressure

Fig. 2 shows the time histories of impact pressure of P2 obtained by MPS method, VOF method and experiment, respectively. From Fig. 2 we can see that both results of MPS method and VOF method agree well with the experiment data. The 2D MPS results show high frequency oscillation and the VOF results are smoother. This is mainly due to the fact that only 2D simulation is carried out by IMPS and it is expected that the oscillation amplitude and frequency can both be reduced by 3D calculation according to our previous works on 3D liquid sloshing flows(Zhang et al., 2014). The results show that the present methods, i.e. MPS method and VOF method, are applicable for the sloshing simulations.

Fig. 3 shows some snapshots in experiment and simulations at first order resonant frequency, respectively. It can be seen that the flows are violent and breaking waves are observed when liquid impacts on the side wall of the tank. Both MPS and VOF method can predict well of the flow patterns with the experiment.



MPS



a) T+1/4T b) T+3/8T c) T+5/8T d) T+3/4T

Fig. 3 Comparison of the flow patterns between experiment and numerical simulation

Membrane-type LNG tank

Numerical Model

The 3D membrane-type tank is obtained from Cai et al (2011), and the geometry of the tank is shown in Fig. 4, the main dimensions of the tank is L=0.834 m long, H=0.477 m high and W=0.664 m wide. The rotation center is 0.229m from the bottom and the filling ratio in this study is 70%. The detailed information of pressure probe locations are shown in Fig. 4.

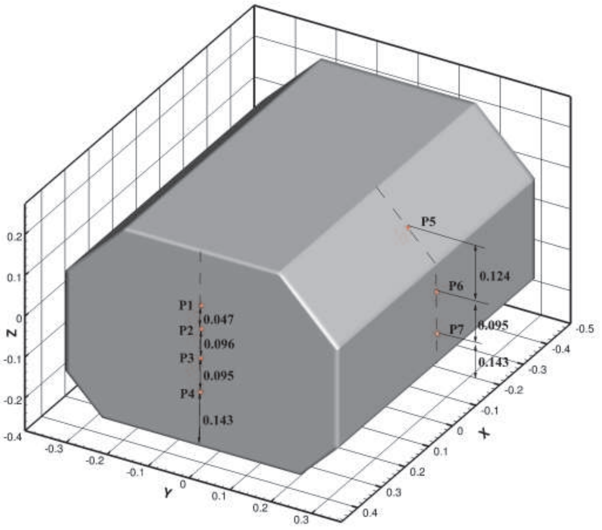


Fig. 4 Schematic of membrane-type tank and the sensor locations

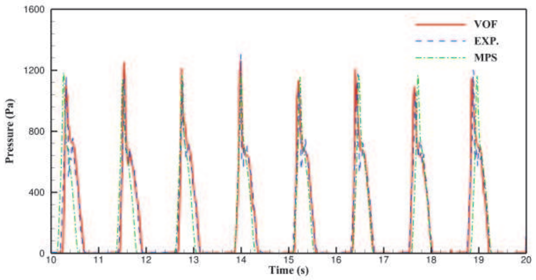
The tank motion is pure pitching which follows the sinusoidal function given by:

$$\theta = \theta_0 \sin(\omega t) \qquad (16)$$

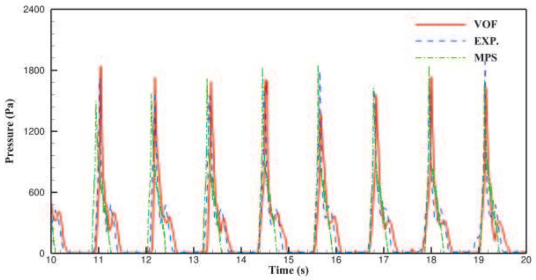
where θ_0 is the angular amplitude and ω is the circular frequency of pitching motion, here θ_0=12° and three frequencies are applied in the simulation, i.e. f = 0.8Hz, f = 0.85Hz and f = 0.9Hz.

In this case, 3D calculation by MPS is applied due to the complex geometry and the number of particles is 0.523M, among which 0.396M are fluid particles, and the corresponding particle initial spacing is 0.005m. For the VOF simulation, the grid number in the computational domain is 0.27M. All the computations are running in 10 processors and the simulation time for VOF is 26370s and IMPS is about 189200s.

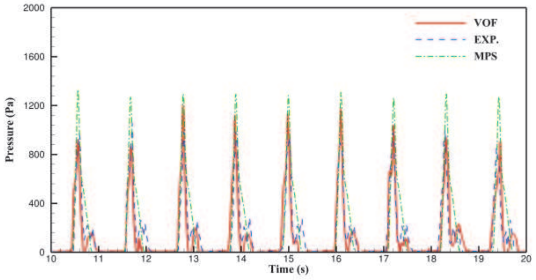
Numerical Results



a) f=0.80Hz



b) f=0.85Hz

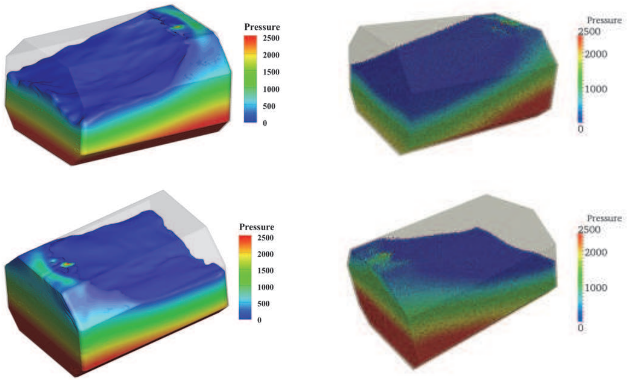


c) f=0.90Hz

Fig. 5 Comparisons of the impact pressure in different frequencies

Fig. 5 compares the time histories of impact pressure at P2 among experiment data, MPS and VOF results. Similarly, the liquid impact behavior is well predicted in all the simulations. Two pressure peaks are observed in each periodic impact behavior except for the MPS method. The first peak is larger but with a shorter duration, which is due to the impact on the side wall of the tank. After the liquid falls down and hits the underlying liquid, the second peak appears. The VOF results for the second peak is larger than that of the experiment due to the violent splashing and the MPS results for the second peak can be barely captured due to the less particles splashing. The amplitude of impact pressure under different frequencies shows highly different in Fig. 5, which illustrates that the excitation period has significant effect on the sloshing performance. The impact pressure at frequency of 0.85Hz is largest, and this implies that the the resonant frequency of filling ratio of 70% is around 0.85Hz. According to the analytical equation of rectangle tank, the first order of resonant frequency is 0.89Hz, which shows that the geometry of the tank plays an important role in the natural frequency.

Several special moments of the sloshing flow by two methods are shown in Fig. 6 from the figure we can see that both MPS and VOF method can produce the large free surface deformation when the fluid impact the upper tank wall.



a) VOF results b) MPS results

Fig. 6 Comparisons of the flow patterns between VOF and MPS

CONCLUSIONS

In this paper, the particle solver MLParticle-SJTU basded on IMPS and VOF method solver naoe-FOAM-SJTU are applied to simulate sloshing flows. The numerical results of the rectangular tank imply that the present two methods are both reliable for violent liquid sloshing problems. Furthermore, the flow patterns obtained by these two methods agree well with that of the experiment. In addition, we take the membrane-type tank into consideration and predict the impact pressure in pitching motion with three different frequencies. The numerical results show that the effects of excitation period on the sloshing are significant. Both experimental and numerical results show that the first order resonant frequency is affected by the geometry of the tank and the resonant frequency of the membrane-type tank is around 0.85Hz at the filling ratio of 70% according to both experimental and numerical results.

Future work will be focused on the sloshing flows in more complicated and severe conditions, including coupling motions and baffle in tanks. More validation on the two numerical methods will be carried out through some other cases with large free surface deformation.

ACKNOWLEDGEMENTS

This work is supported by the National Natural Science Foundation of China (51379125, 51490675, 11432009, 51579145, 11272120), Chang Jiang Scholars Program (T2014099), Program for Professor of Special Appointment (Eastern Scholar) at Shanghai Institutions of Higher Learning (2013022), Innovative Special Project of Numerical Tank of Ministry of Industry and Information Technology of China (2016-23) and Foundation of State Ley Laboratory of Ocean Engineering (GKZD010065), to which the authors are most grateful.

REFERENCES

Akyildiz, H, and Ünal, E (2005). "Experimental investigation of pressure distribution on a rectangular tank due to the liquid sloshing," *Ocean Eng*, 32(11), 1503–1516.

Bulian, G, Botia-Vera, E, and Souto-Iglesias, A (2014). "Experimental sloshing pressure impacts in ensemble domain: Transient and stationary statistical characteristics," *Phys Fluids*, 26(3), 032102.

Cai, Z, Wang, D, and Li, Z (2011). "Influence of excitation frequency on slosh-induced impact pressures of liquefied natural gas tanks," *J Shanghai Jiaotong Univ Sci*, 16(1), 124–128.

Chaichanasiri, E, and Suvanjumrat, C (2012). "Simulation of Three Dimensional Liquid-Sloshing Models using C++ Open Source Code CFD Software," *Kasetsart J Nat Sci*, 46(6), 978–995.

Chen, YG, Djidjeli, K, and Price, WG (2009). "Numerical simulation of liquid sloshing phenomena in partially filled containers," *Comput Fluids*, 38(4), 830–842.

Faltinsen, OM (1978). "A numerical nonlinear method of sloshing in tanks with two-dimensional flow," *J Ship Res*, 22(3).

Ferziger, JH, and Peric, M (2012). *Computational methods for fluid dynamics* Springer Science & Business Media.

Gingold, RA, and Monaghan, JJ (1977). "Smoothed particle hydrodynamics: theory and application to non-spherical stars," *Mon Not R Astron Soc*, 181(3), 375–389.

Hirt, CW, and Nichols, BD (1981). "Volume of fluid (VOF) method for the dynamics of free boundaries," *J Comput Phys*, 39(1), 201–225.

Issa, RI (1986). "Solution of the implicitly discretised fluid flow equations by operator-splitting," *J Comput Phys*, 62(1), 40–65.

Kang, DH, and Lee, YB (2005). "Summary report of sloshing model test for rectangular model," *Daewoo Shipbuild Mar Eng Co Ltd South Korea*, (001).

Kim, Y (2001). "Numerical simulation of sloshing flows with impact load," *Appl Ocean Res*, 23(1), 53–62.

Kim, S-Y, Kim, K-H, and Kim, Y (2015). "Comparative study on pressure sensors for sloshing experiment," *Ocean Eng*, 94, 199–212.

Kishev, ZR, Hu, C, and Kashiwagi, M (2005). "Numerical simulation of violent sloshing by CIP method with experimental validation," *Proc 25th Int Offshore Polar Eng Conf*, Big Island, Hawaii, USA.

Koshizuka, S, and Oka, Y (1996). "Moving-particle semi-implicit method for fragmentation of incompressible fluid," *Nucl Sci Eng*, 123(3), 421–434.

Koshizuka, S, Nobe, A, and Oka, Y (1998). "Numerical analysis of breaking waves using the moving particle semi-implicit method," *Int J Numer Methods Fluids*, 26(7), 751–769.

Lee, B-H, Park, J-C, Kim, M-H, and Hwang, S-C (2011). "Step-by-step improvement of MPS method in simulating violent free-surface motions and impact-loads," *Comput Methods Appl Mech Eng*, 200(9), 1113–1125.

van Leer, B (1979). "Towards the ultimate conservative difference scheme. V. A second-order sequel to Godunov's method," *J Comput Phys*, 32(1), 101–136.

Lucy, LB (1977). "A numerical approach to the testing of the fission hypothesis," *Astron J*, 82, 1013–1024.

Lugni, C, Brocchini, M, and Faltinsen, OM (2006). "Wave impact loads: The role of the flip-through," *Phys Fluids*, 18(12), 122101.

Menter, FR (1994). "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA J*, 32(8), 1598–1605.

Shao, JR, Li, HQ, Liu, GR, and Liu, MB (2012). "An improved SPH method for modeling liquid sloshing dynamics," *Comput Struct*, 100, 18–26.

Shen, Z and Wan, DC (2012). "Manual of CFD solver for ship and ocean engineering flows: naoe-FOAM-SJTU." Technical Report No. 2012SR118110, Shanghai Jiao Tong University.

Tanaka, M, and Masunaga, T (2010). "Stabilization and smoothing of pressure in MPS method by quasi-compressibility," *J Comput Phys*, 229(11), 4279–4290.

Wu, C-H, and Chen, B-F (2009). "Sloshing waves and resonance modes of fluid in a 3D tank by a time-independent finite difference method," *Ocean Eng*, 36(6), 500–510.

Zhang, Y, and Wan, DC (2011). "Application of Improved MPS Method in Sloshing Problem," *Proc 23rd Chin Symp Hydrodyn*, Xi'an.

Zhang, Y, and Wan, DC (2012). "Apply MPS Method to Simulate Liquid Sloshing in LNG Tank," Proceedings of the Twenty-second International Offshore and Polar Engineering Conference, Rhodes, Greece, June 17–22, 2012, ISOPE, Vol 3, pp.381-391.

Zhang, Y, Wan, DC, and Takanori, H (2014). "Comparative study of MPS method and level-set method for sloshing flows," *J Hydrodyn Ser B*, 26(4), 577–585.