iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 8 **Issue:** XI **Month of publication:** November 2020

DOI: https://doi.org/10.22214/ijraset.2020.32116

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue XI Nov 2020- Available at www.ijraset.com

Review of Numerical Methods for Sloshing

Pradeepkumar K S[1], Selvan V[2], Satheeshkumar K R P[3]
[1]II Year Student, ME Structural Engineering, Department of Civil Engineering, Kumaraguru College of Technology
[2]Head of Department, Department of Civil Engineering, Kumaraguru College of Technology
[3]Assistant Professor II, Department of Civil Engineering, Kumaraguru College of Technology

Abstract: Sloshing refers to the oscillating movement of free surface of the liquid. This paper presents various methods proposed for simulation of sloshing phenomenon in partially filled liquid containers. Understanding slosh dynamics has several applications in different fields of engineering. The impact pressure on the container due to liquid sloshing governs numerous designs of machinery ranging from simple liquid carrying truck to propellant tanks in spacecrafts and tanks in ships and the design of liquid retaining structures prone to seismic excitation. Experimental method, analytical approach and numerical methods are used to predict sloshing responses. This paper focuses on various numerical methods proposed, validated, and are widely used in liquid sloshing. The methods differ from each other in ways of treating the free surface of the liquid, discretization methods used, assumptions made and consequently the governing equations of flow chosen.

Keywords: Sloshing, Turbulence model, Violent sloshing, Sloshing review, Numerical methods for sloshing.

I. INTRODUCTION

Sloshing phenomenon can be found almost everywhere, where the fluid has a free surface, and the system is susceptible to disturbance. Though sloshing effects at all systems are not much important, there are some cases where understanding sloshing is necessary. Some important engineering applications of sloshing are:

A. Trucks carrying liquid in automobile industry,
B. Fuel tanks in aircrafts,
C. Propellant tanks in space crafts,
D. LNG carriers,
E. Liquid retaining structures prone to earthquake.

The effects of sloshing are studied by any of the following three methods:
1) Experimental methods,
2) Analytical methods,
3) Numerical methods.

Out of the three, experimental approach to study the effects prove to be the accurate and provides exact data than the other two methods. But experimental method is not feasible due to the cost and inability to use widely. Moreover, it is difficult to scale down in models, all the physical phenomena involved and extend it to full scale predictions.

On the other hand, analytical and numerical methods are economical and can generate solutions with little modifications in their formulations, whenever there are different requirements and scenarios. The analytical approaches are applicable to linear sloshing and small amplitude sloshing. These approaches fail when sloshing become violent, breaking, and overturning of waves occur. These methods are also limited to certain simple geometries.

II. NUMERICAL METHODS

To overcome these effects, numerical simulations are widely used. With increasing computational capabilities, numerical method shave gained a lot of importance and extensive research are being carried out to obtain realistic simulations. The fluid flow can be viewed using either Eulerian approach or Lagrangian approach.

The former approach considers a location in space through which the fluid flows, while the latter tracks the individual fluid particles in the fluid flow. The discretization of the flow domain through meshes or grids falls under Eulerian approach and meshless methods falls under Lagrangian approach. The Arbitrary Eulerian-Lagrangian approach has also been used in some literatures for sloshing problems.

III.EULERIAN APPROACH

Numerical methods such as boundary element method, finite volume method, finite element method and finite difference method are used to obtain solutions in meshed methods. The numerical approach for laminar sloshing flow is based on potential flow theory. But the methods based on potential flow theory cannot handle wave breaking and violent sloshing. For violent or turbulent sloshing flow, mass conservation and momentum equations (Navier-Stokes equations) are used to characterize the flow. Non-linearity arising due to moving boundary condition is avoided by using a dynamic mesh in the sloshing studies.

A. Finite Element Method

G.X.Wu et al.[1] analysed sloshing waves in 3D tank using finite element method based on fully non-linear potential flow theory. Free surface elevation of the 2D cases are compared with experimental and analytical data. Numerical simulation results are in better agreement with experimental data than the analytical results.

B. Finite Difference Methods

Seung Hee Lee et al. [2] simulated three dimensional sloshing phenomena occurring in liquid cargo tanks. The governing equations of flow are Navier-Stokes equations along with continuity equation. The governing equations are discretized using finite difference method with rectangular fixed staggered mesh system. Marker density method is adopted for constructing the free surface with air pressure variations (IU-SLOSH). Sub-grid scale model is used to consider turbulence effects in the simulation. A good agreement of pressure histories compared with experimental data, and good agreement with modal approach under simple conditions reveals the practical applicability of the numerical model. Hakan Akyildiz et al. [3] developed a numerical model to analyse the sloshing effects in a 3D rectangular tank. The governing equations of flow are Navier-Stokes equations and continuity equation. The governing equations are discretized using finite difference method. The model solves the Navier-Stokes equation using SOLA scheme. Effects of turbulence, wave breaking and surface tension are ignored in the model.

1) *Free Surface Motion*: Another problem in violent sloshing is the non-linear motion of free surface. Some of the methods developed to predict the distorted free surface of the flow such as marker and cell (MAC), volume of fluid (VOF), level set method (LS) and coupled level set-volume of fluid method (CLSVOF). The volume of fluid method is widely used to capture the interface because of its mass conservation characteristics. But it lacks precision for direct computations of curvature and normal vector. In level set method, the curvature and normal vector can be accurately calculated with ease. But mass conservation is often violated in LS method. To overcome the shortcomings of these methods, a coupled level set- volume of fluid method is used in sloshing problems. Elahi et al. [4] developed a numerical model to simulate sloshing in 2D tank considering liquid free surface deformation, liquid viscosity, and surface tension. The governing equations of flow are Navier-Stokes equations and continuity equation with advection equation for free surface using volume of fluid method. The governing equations are discretized using finite difference methods. Virtual body force method is used to couple the fluid and solid dynamics. The free fall motion of fluid in the container and simulation of fluid flow under linear acceleration are examined to validate the accuracy of the numerical model. Hakan Akyildiz [5] studied the effects of vertical baffle in liquid sloshing in 2D rectangular tank. The governing equations of flow are Navier-Stokes equations and continuity equation. The free surface behaviour due to non-linear sloshing is modelled by volume of fluid method. The governing equations are solved using SOLA scheme under cartesian staggered grid. Surface tension, wave breaking, and effects of turbulence are ignored in the model. The pressure data obtained from numerical simulation agree well with the experimental data.
2) *Turbulence Models*: Various turbulent models to include the turbulent effects in the sloshing and to improve the accuracy of the solution are RANS (Reynolds averaged Navier-Stokes equations) one equation closure model, RANS two equation closure models (k-□, k-□, SST), LES(Large Eddy Simulation) approach and DNS (Direct numerical simulation) in the increasing order of complexity. DNS is extremely costly and not feasible since it resolves all range of turbulent length scales. Ling Hou et al. [6] simulated sloshing flow in 2D rectangular tank using ANSYS Fluent under single and coupled external excitations. Standard k-□ RANS closure model is used for turbulence modeling. Free surface is captured using volume of fluid method. The interface is constructed using piecewise linear scheme. Transport equation for volume fraction is solved by explicit time marching scheme. External excitation is applied through dynamic mesh. Liu and Lin [7] developed NEWTANK model to simulate 3D non-linear sloshing with broken free surfaces. This model solves the spatially averaged Navier-Stokes equations on non-inertial reference frame. LES approach with Smagorinsky sub-grid scale model is used to model the turbulence. The governing equations are solved using finite difference method on a staggered grid mesh. VOF method is used to capture the free surface. Liu and Lin [8] extended the previously developed NEWTANK model to simulate sloshing in a tank with baffles. This model

solves the spatially averaged Navier-Stokes equations on non-inertial reference frame. LES approach with Smagorinsky sub-grid scale model is used to model the turbulence. The governing equations are solved using finite difference method on a staggered grid mesh. The internal baffle is modelled by virtual boundary force (VBF) method. VOF method is used to capture the free surface. Dongxi Liu et al. [9] studied sloshing using laminar model and turbulence models and assessed the shortcomings of each model. Though RANS turbulence models provide significant improvement in accuracy, they still cannot model some important unsteady effects in violent sloshing. LES approach has been proved to improve the accuracy of results than RANS models. But the computational cost of LES approach is high, and it is very hard to find a grid independent LES solution. To reduce the computational cost of LES approach and to improve the accuracy of RANS model, VLES (very large eddy simulation) approach combining these two methods is developed. A coupled level set-volume of fluid method is used to model the free surface. Dongxi Liu et al. [10] simulated the 3D sloshing flow in rectangular tank with and without baffles using the newly developed hybrid RANS/LES approach. To improve the drawbacks of Speziale's hybrid method, a new hybrid resolution control function is proposed. The model is validated by comparing with the experimental data and some other numerical models without baffles. Then the hybrid RANS/LES approach is extended to sloshing in rectangular tank with horizontal and vertical baffles. Dongxi Liu et al. [11] studied liquid sloshing using laminar model and turbulence models with CLSVOF method to track the free surface. The effect of choice of turbulence models and interface capturing method in numerical results are studied. Rhee [12] developed a numerical model to simulate liquid sloshing at low filling levels. The standard k-□□RANS closure model is used for considering turbulence. VOF of fluid method is employed to construct the free surface. The governing equations are discretized by second order accurate schemes on unstructured grids. Tae-Won Kang et al. [13] studied sloshing in rigid cylindrical tank using shear stress transport model (SST). The SST model has the merits of both RANS k-□ and k-□ models. The k-□ model is effective in predicting the flow near the wall and k-□ is effective in predicting the flow in the state separated from the wall. The free surface is modelled using volume of fluid method.

C. Finite Volume Methods

Sloshing based on RANS turbulence models result in unphysical motion on the free surface due to excessive turbulence at interface. So,Li et al. [14] developed two eddy viscosity eliminators, one based on sigmoid functions and another based on polynomials and combined them with RANS two closure models (k-□ model, k-□ model, k-□□SST model),to reduce this effect. J. H. Jung et al. [15] investigated the effect of vertical baffle on liquid sloshing in 3D rectangular tank. Two equation RANS k-□□closure model is used to consider the turbulence effects. Motion of free surface is captured by volume of fluid method. The numerical model discretized the governing equations using finite volume method. The time histories of pressure are compared with previously available experimental data. The results agree reasonably well with the experimental data.

Li et al. [16] simulated sloshing using the buoyancy modified k-□SST model and modified isoAdvector method. To reduce the excessive turbulence levels at the interface buoyancy modified turbulence model is adopted. This avoids the unphysical motion at the transition region at the interface due to excessive turbulence. A new geometric VOF (isoAdvector method) method compatible with dynamic meshing is used to capture the free surface motions. The governing equations are discretized using finite volume methods. Moving velocity correction for face interface intersection line (FIIL) is adopted to update the volume fraction. Milovan Perić and Tobias zorn [17] simulation sloshing in moving tanks using RANS k-□ and k-□ turbulence models. An effective interface capturing scheme is adopted to consider wave breaking and overturning of waves. Finite volume method is used to solve the governing equations. The results show that this method can predict the sloshing loads in moving tanks with sufficient accuracy. Chia-Ren Chu et al. [18] analyzed the hydrodynamic loads in a water with multiple baffles. LES approach with Smagorinsky sub-grid scale model (SGS) is used for turbulence moelling. The motion of free surface is captured by volume of fluid method. The computational domain and the governing equations are discretized by finite volume methods.

IV.LAGRANGIAN APPROACH

The sloshing phenomenon can also be simulated using Lagrangianapproach. Being Lagrangian, these methods discretize the domain into discrete particles and so a meshless method is obtained. Moving particle semi-implicit method (MPS), consistent particle method (CPM), smoothed particle hydrodynamics (SPH) lattice Boltzmann method (LBM) are some of the meshless methods used for simulating fluid flow.

Nan-Jing Wu et al. [19] developed a meshfree simulation model for liquid sloshing subjected to harmonic excitations based on potential flow theory. A time marching scheme is used for simulating free surface potential flow. A local polynomial collocation method is adopted to solve the governing Laplace equation. PAN Xu-jie et al. [20] used moving particle semi-implicit method

(MPS), a meshless method for simulating sloshing phenomenon. PPE is solved by incomplete Cholesky conjugate gradient method (ICCG). The mesh in traditional methods is substituted by particles in MPS method. The governing equations of flow are Navier-Stokes equations along with continuity equation. The relationships between the particles are established by kernel functions. The numerical density of the particles is used to track the free surface. The comparison with free surface elevation and time history of pressure obtained by VOF method showed the possibility of using MPS for simulating sloshing phenomenon. C. G. Koh et al. [21] used an improved consistent particle method (CPM) to simulate liquid sloshing. CPM is improved to reduce the spurious pressure fluctuations. Sloshing is simulated in a tank with a constrained floating baffle with a partitioned coupling procedure. Incompressibility is enforced by combining zero-density variation and velocity-divergence free condition. The surface profiles obtained by this method at various time instants agree well with the surface profiles from the experiment.

A. Smoothed Particle Hydrodynamics

Delorme et al. [22] studied the application of SPH method for sloshing simulation in LNG tankers. Unlike the other methods, no special treatment for free surface condition is needed as the position of the particles is the indication of free surface at any instant of time. The SPH method is applied to Navier-Stokes equation to simulate the sloshing flow. Special treatment for boundary condition is necessary in SPH. It is usually done by using repulsive particles or ghost particles. A method close to Monaghan and Gray's repulsive particle method is adopted in this paper. They concluded that there is scope for improvements in SPH formulations to use it for sloshing simulation. Z.Chen et al. [23] investigated the pressure on solid walls in 2D sloshing using SPH method. The improved SPH method used moving least square method for density re-initialization and coupled dynamic solid boundary treatment (SBT) method for improving pressure field near the boundary. Quintic kernel function is used to define the influence of neighboring particles in this method. Both the repulsive particles and ghost particles are placed in boundary and the combination resulted in satisfactory results. A new method for obtaining pressure at solid boundary by considering the correction term based on repulsive forces along with the physical pressure calculated from density is used.

J.R.Shao et al. [24] developed an improved SPH method to simulate liquid sloshing. Modified schemes for density correction and Kerner gradient correction are used to improve accuracy by smoothing pressure field. A coupled dynamic SBT algorithm is proposed to increase accuracy near boundary. Usually, Navier-Stokes equations are used for governing the flow in SPH. But to take the turbulence effects into account, RANS turbulence model is included in this method.

Hu Taian et al. [25] simulated sloshing flows with an elastic baffle using coupled numerical scheme between smoothed particle hydrodynamics and smoothed point interpolation method (SPH-SPIM coupled method. SPH is related to flow dynamics while SPIM is related to structure dynamics. The continuity equation and momentum equations are the flow governing equations. The coupled dynamic solid boundary condition is used, which has advantages of both repulsive and ghost particle method.

The validation criteria and the results obtained are presented in table 1.

Table 1: Validation Criteria And Results

Article reference number	Reference used for validation	Parameters compared	Results of validation	Conclusions
[6]	Experimental data	Pressure history at mid height under sway	Predicted numerical results are higher than experimental result by a maximum of 6.7%	Coupled sway, roll and heave under resonant conditions produce violent sloshing. Further extend 2d to 3d.
[7]	Analytical, experimental, and numerical data	Surface elevation	Predicted results agree well with the experimental data	For violent sloshing Analytical method resulted in a lower peak value, while numerical results agree well. Effect of including turbulence is negligible. Promising numerical tool to simulate highly non-linear liquid sloshing. Further study to quantify turbulence effect and surface breaking.
[8]	Other numerical data	Time history of surface elevation at right boundary of tank	Good agreement with the numerical data	For large amplitude sloshing without baffle, numerical result varies from analytical result but is close to experimental result.
[9]	Experimental data	Surface profile and pressure history	Surface profile and pressure data of VLES agrees well with experimental results better than laminar, RANS and LES models.	The appropriate choice of turbulent model influences the accuracy of sloshing simulation. LES and VLES models perform better than RANS and laminar flow models.
[10]	Experimental data and other turbulence models	Free surface elevation and Pressure histories at various heights of the rectangular tank	Very good agreements are obtained.	Performance of the present hybrid model in terms of accuracy and computational efficiency is higher than the other models.

[11]	Experimental data	Velocity field is compared to sloshing wave shapes due to lack of experimental velocity field.	Agrees well with the experimental data	VLES model captures wave breaking better than RANS.
[12]	Experimental data	Free surface shapes, pressure histories and averaged impact pressure	Free surface shapes and pressure histories agree well with experimental results. Computed Averaged impact pressure varies from measured value about 8%.	The results show reasonable behaviour of liquid sloshing.
[13]	Experimental data	Time history of pressure	Predicted peak hydrodynamic pressures are within 6% simulation error.	Combination of Vertical and horizontal components of ground motion significantly increase the hydrodynamic pressure energy near the surface of water compared to the one with only horizontal components.
[14]	Experimental data	Surface elevation	RMSE of k-□ SST model with eddy viscosity eliminator based on sigmoid functions is the smallest.	RMSEs of two equation RANS closure models reduced when eddy viscosity eliminators are used. Eddy viscosity eliminator based on sigmoid functions perform better than the one based on polynomials.
[16]	Experimental data and other numerical methods	Time histories of wave elevation and forces	RMSEs of FVC isoAdvector method and buoyancy modified k-□ SST model are the lowest compared to other models.	FVC isoAdvector method agrees well with experimental data than MULES and FC isoAdvector method. Buoyancy modified k-□ SST model agrees well with experimental data than RANS k-□ SST model.
[18]	Experimental data	Maximum wave height and average wave height	Error of less than 2% for maximum wave height with and without baffles is obtained, while error less than 4% for average wave height with and without baffles is achieved.	The presence of baffles has some mitigating effect on sloshing phenomenon under close resonant condition.
[19]	Experimental data	Time history of free surface elevations in 2D rectangular tank, 3D square tank and 3D cylindrical tank.	The time histories agree well with the experimental results.	From the accurate estimation of velocity components, the pressure distribution can be found by solving PPE separately.
[21]	Theoretical and experimental data	Hydrostatic pressure, time history of pressure and free surface profiles under free sloshing and under CFB.	The time histories of pressure and free surface profiles agree well with experimental results.	The numerical model can predict the sloshing under CFB and, CFB is effective in reducing sloshing.
[23]	Experimental data	Flow pattern and pressure value on solid boundary.	Improved SPH with density correction and coupled dynamic SBT performed better than standard SPH and agree well with experimental data.	The MLS density re-initialization and coupled SBT algorithm keeps the pressure field smooth and stable. The new method for obtaining pressure in the boundary performs better.
[24]	Experimental data	Wave height	Improved SPH agree well with experimental data.	Improved SPH with density correction, kernel gradient correction, turbulence model and dynamic SBT algorithm performs better than traditional SPH and is capable of simulating violent sloshing.
[25]	Experimental data	Surface profile and time history of pressure in tank with water and surface profile of oil in tank with baffle.	Simulation results agree well with experimental result in all cases.	The coupled SPH-SPIM method is reliable in simulating sloshing flow and there is a scope for extending the 2D model to 3D sloshing.

V. CONCLUSION

Several numerical approaches, both meshed and meshless methods for simulation of sloshing phenomenon adopted widely are discussed along with their validations. The choice of method for simulation of sloshing depends upon the physical phenomena involved. Some key points that are to be considered while numerically simulating the sloshing phenomenon are,

A. The appropriate choice of free surface modelling using interface capturing or interface tracking method is necessary to achieve desired accuracy. VOF is simple and widely used.

B. The inclusion of turbulence models in the problem is necessary to capture the effects of turbulence at interface. RANS k-□ model, k-□ model, k-□ SST model, LES approach, DNS methods are in the order of increasing complexities and thereby increasing accuracy.

C. The schemes for density correction, kernel gradient correction, treatment of boundary condition play a vital role in the accuracy of the results in SPH method.

Future scope for improved simulation of sloshing phenomenon also lies in these areas while using numerical techniques.

REFERENCES

[1] G. X. Wu, Q. W. Ma, and R. Eatock Taylor, “Numerical simulation of sloshing waves in a 3D tank based on a finite element method,” *Appl. Ocean Res.*, vol. 20, no. 6, pp. 337–355, 1998, doi: 10.1016/S0141-1187(98)00030-3.

[2] S. H. Lee, Y. G. Lee, and K. L. Jeong, “Numerical simulation of three-dimensional sloshing phenomena using a finite difference method with marker-density scheme,” *Ocean Eng.*, vol. 38, no. 1, pp. 206–225, 2011, doi: 10.1016/j.oceaneng.2010.10.008.

[3] H. Akyildiz and N. Erdem Ünal, "Sloshing in a three-dimensional rectangular tank: Numerical simulation and experimental validation," *Ocean Eng.*, vol. 33, no. 16, pp. 2135–2149, 2006, doi: 10.1016/j.oceaneng.2005.11.001.

[4] R. Elahi, M. Passandideh-Fard, and A. Javanshir, "Simulation of liquid sloshing in 2D containers using the volume of fluid method," *Ocean Eng.*, vol. 96, pp. 226–244, 2015, doi: 10.1016/j.oceaneng.2014.12.022.

[5] H. Akyildiz, "A numerical study of the effects of the vertical baffle on liquid sloshing in two-dimensional rectangular tank," *J. Sound Vib.*, vol. 331, no. 1, pp. 41–52, 2012, doi: 10.1016/j.jsv.2011.08.002.

[6] L. Hou, F. Li, and C. Wu, "A numerical study of liquid sloshing in a two-dimensional tank under external excitations," *J. Mar. Sci. Appl.*, vol. 11, no. 3, pp. 305–310, 2012, doi: 10.1007/s11804-012-1137-y.

[7] D. Liu and P. Lin, "A numerical study of three-dimensional liquid sloshing in tanks," *J. Comput. Phys.*, vol. 227, no. 8, pp. 3921–3939, 2008, doi: 10.1016/j.jcp.2007.12.006.

[8] D. Liu and P. Lin, "Three-dimensional liquid sloshing in a tank with baffles," *Ocean Eng.*, vol. 36, no. 2, pp. 202–212, 2009, doi: 10.1016/j.oceaneng.2008.10.004.

[9] D. Liu, W. Tang, J. Wang, H. Xue, and K. Wang, "Comparison of laminar model, RANS, LES and VLES for simulation of liquid sloshing," *Appl. Ocean Res.*, vol. 59, pp. 638–649, 2016, doi: 10.1016/j.apor.2016.07.012.

[10] D. Liu, W. Tang, J. Wang, H. Xue, and K. Wang, "Hybrid RANS/LES simulation of sloshing flow in a rectangular tank with and without baffles," *Ships Offshore Struct.*, vol. 12, no. 8, pp. 1005–1015, 2017, doi: 10.1080/17445302.2017.1301341.

[11] D. Liu, W. Tang, J. Wang, H. Xue, and K. Wang, "Modelling of liquid sloshing using CLSVOF method and very large eddy simulation," *Ocean Eng.*, vol. 129, no. November 2016, pp. 160–176, 2017, doi: 10.1016/j.oceaneng.2016.11.027.

[12] S. H. Rhee, "Unstructured grid based reynolds-averaged navier-stokes method for liquid tank sloshing," *J. Fluids Eng. Trans. ASME*, vol. 127, no. 3, pp. 572–582, 2005, doi: 10.1115/1.1906267.

[13] T. W. Kang, H. I. Yang, and J. S. Jeon, "Earthquake-induced sloshing effects on the hydrodynamic pressure response of rigid cylindrical liquid storage tanks using CFD simulation," *Eng. Struct.*, vol. 197, no. May, p. 109376, 2019, doi: 10.1016/j.engstruct.2019.109376.

[14] J. long Li, Y. xiang You, and K. Chen, "Applications of An Eddy-Viscosity Eliminator Based on Sigmoid Functions in Reynolds-Averaged Navier-Stokes Simulations of Sloshing Flow," *China Ocean Eng.*, vol. 34, no. 4, pp. 463–474, 2020, doi: 10.1007/s13344-020-0042-4.

[15] J. H. Jung, H. S. Yoon, C. Y. Lee, and S. C. Shin, "Effect of the vertical baffle height on the liquid sloshing in a three-dimensional rectangular tank," *Ocean Eng.*, vol. 44, pp. 79–89, 2012, doi: 10.1016/j.oceaneng.2012.01.034.

[16] J. Li, Y. You, K. Chen, and X. Zhang, "Numerical computations of resonant sloshing using the modified isoAdvector method and the buoyancy-modified turbulence closure model," *Appl. Ocean Res.*, vol. 93, no. May, p. 101829, 2019, doi: 10.1016/j.apor.2019.05.014.

[17] M. Perić and T. Zorn, "Simulation of sloshing loads on moving tanks," *Proc. Int. Conf. Offshore Mech. Arct. Eng. - OMAE*, vol. 3, pp. 1017–1026, 2005, doi: 10.1115/OMAE2005-67581.

[18] C. R. Chu, Y. R. Wu, T. R. Wu, and C. Y. Wang, "Slosh-induced hydrodynamic force in a water tank with multiple baffles," *Ocean Eng.*, vol. 167, no. July, pp. 282–292, 2018, doi: 10.1016/j.oceaneng.2018.08.049.

[19] N. J. Wu, S. C. Hsiao, and H. L. Wu, "Mesh-free simulation of liquid sloshing subjected to harmonic excitations," *Eng. Anal. Bound. Elem.*, vol. 64, pp. 90–100, 2016, doi: 10.1016/j.enganabound.2015.12.001.

[20] X. J. Pan, H. X. Zhang, and Y. T. Lu, "Numerical simulation of viscous liquid sloshing by moving-particle semi-implicit method," *J. Mar. Sci. Appl.*, vol. 7, no. 3, pp. 184–189, 2008, doi: 10.1007/s11804-008-7047-3.

[21] C. G. Koh, M. Luo, M. Gao, and W. Bai, "Modelling of liquid sloshing with constrained floating baffle," *Comput. Struct.*, vol. 122, pp. 270–279, 2013, doi: 10.1016/j.compstruc.2013.03.018.

[22] L. Delorme, a S. Iglesias, and S. A. Perez, "Sloshing Loads Simulation in Lng Tankers With Sph," *Int. Conf. Comput. Methods Mar. Eng.*, no. September, pp. 1–10, 2005.

[23] Z. Chen, Z. Zong, H. T. Li, and J. Li, "An investigation into the pressure on solid walls in 2D sloshing using SPH method," *Ocean Eng.*, vol. 59, pp. 129–141, 2013, doi: 10.1016/j.oceaneng.2012.12.013.

[24] J. R. Shao, H. Q. Li, G. R. Liu, and M. B. Liu, "An improved SPH method for modeling liquid sloshing dynamics," *Comput. Struct.*, vol. 100–101, pp. 18–26, 2012, doi: 10.1016/j.compstruc.2012.02.005.

[25] T. Hu, S. Wang, G. Zhang, Z. Sun, and B. Zhou, "Numerical simulations of sloshing flows with an elastic baffle using a SPH-SPIM coupled method," *Appl. Ocean Res.*, vol. 93, no. March, 2019, doi: 10.1016/j.apor.2019.101950.