

FLUID MECHANICS

SIMULATION OF WATER-ENTRY AND WATER-EXIT PROBLEMS USING A MOVING MESH ALGORITHM

ROOZBEH PANAHI

*Marine Transportation and Tehcnology Department,
Transportation Research Institute, No. 65, Hekmat St., Noori Highway,
Tehran, Iran, Post Code: 14639-17151,
e-mail: panahi@rahiran.ir*

[Received 25 May 2010. Accepted 27 February 2012]]

ABSTRACT. Simulation of the water-entry and water-exit particularly, at the interface of two phases i.e. water and air due to the effect of flow-induced loads, gravity force and trapped air cushion presence is very complicated. This paper attempts to introduce a finite volume-based moving mesh algorithm in order to simulate such problems in a viscous incompressible two-phase medium.

The algorithm employs a fractional step method to deal with the coupling between pressure and velocity fields. Interface is also captured by solving a volume fraction transport equation. A boundary-fitted body-attached mesh of quadrilateral Control Volumes (CVs) is implemented to record hydrodynamic time histories of loads, motions and interfacial flow changes around the structure. Forced water-exit of a cylinder is simulated based on the introduced algorithm, together with free symmetric and asymmetric water-entry of wedges. Results show that the presented algorithm is favorably capable of assessing such complexities when comparing to experimental data.

KEY WORDS: Interfacial flow, finite volume, body-attached mesh, slamming.

Nomenclature

α	volume fraction	$\vec{a}$	body linear acceleration vector
$\vec{u}$	fluid velocity vector	I	body moment of inertia
P	pressure	$\vec{\alpha}$	body angular acceleration vector
$\vec{g}$	gravity acceleration	$\vec{\omega}$	body angular velocity vector
m	body mass	$\vec{c}$	$\vec{u}$ relative to CV velocity vector
V	volume of a CV	$\vec{n}$	unit outward vector normal to surface
$\vec{A}$	area vector	$\vec{r}$	position vector
$\vec{\tau}$	tangential stress	G	body mass center
k	CV face over the body	$\vec{\nabla}$	gradient vector
i or j	Cartesian direction	t	time

1. Introduction

Fluid-structure interaction problem at water surface is a complicated phenomenon in marine hydrodynamics. It can be divided into two main categories of water-entry and water-exit in which the former case is more complex than the later one. Green-water and wave run-up effects on platforms and ships are evident examples of such a category. Water-entry in the case of high normal relative speed causes large impact loads. This is called slamming and involves local loads changing rapidly in time and space. The duration of slamming load at one location is in the order of milliseconds. In addition, the position of peak load changes with respect to time. It must be mentioned, that such loads threaten the safety of marine structures both abruptly and progressively. Meanwhile, structures may suffer local damage from the impact loads or large-scale buckling. Slamming on ships is also recognized in four types of problems including water-entry, wet-deck slamming, green water and sloshing [1].

Various approaches such as experimental, analytical and numerical were used by researches to investigate the problem in which the former one, numerical approach, has been used more than others, recently. These approaches are deeply reviewed and discussed by Faltinsen [1, 2] and Bertram [3]. Here, original studies are:

- Linear slamming theory based on expanding thin plate approximation by Von Karman in 1929 [4] and Wagner in 1932 [5];

- Simple non-linear slamming theory based on self-similar flow by Dobrovolskaya in 1969 [6];
- Slamming theory including air trapping by Bagnold in 1939 [7] and Verhagen in 1967 [8];
- Effect of water compressibility by Korobkin in 1992 [9];
- 3D slamming theory by Watanabe in 1986 [10];
- Hydroelasticity in slamming by Kvaalsvold in 1994 [11] and Kvaalsvold and Faltinsen in 1995 [12].

In practice, it can be assumed that each structure is an elastic body with complex shape, the effect of gravity is considerable, the interaction between trapped air pockets and water is important and the water surface includes complicated jets. In such cases, analytical solutions are very difficult or even impossible. Therefore, Computational Fluid Dynamics (CFD) can be used as a reliable and appropriate tool. The first CFD applications to slamming were employed in 1980s as the required computer facilities weren't available since that time. Real progress was achieved with field methods like finite volume rather than boundary methods like boundary element. Coupling of rigid body motions with interfacial flow is among the recent developments using CFD including (a) velocity and pressure fields, (b) rigid body motions and (c) free surface.

Methods for solving main flow equations which govern velocity and pressure fields in the case of incompressible viscous fluid are typically categorized as: predictor-corrector, artificial compressibility and fractional step [13]. Fractional step category is based on using distinct steps to reach a divergence-free flow field without outer iterations, using the Hodge decomposition concept. This category is implemented in this study according to its characteristic in unsteady problems [13].

Another important issue is the appropriate representation of free surface. Existing categories to handle interface are surface and volume approaches [13]. Using the later approach, here a scalar transport equation is solved to determine volume fraction of phases in each CV. This enables one to simulate large deformations.

Solution of aforementioned problems yields to velocity and pressure distributions over a body surface. Integration of such stresses over the body gives the loads acting on it. Then, the equations of conservation of linear and angular momentum have to be solved to calculate Degrees of Freedom (DoF) of the body motions and consequently to find out its new location and

orientation. At this stage, a strategy is required to introduce such movements into the computational domain and to update it for the next time step. There are a wide variety of strategies [14] such as overlapping mesh [15]. Here, a boundary-fitted grid which follows the time history of the body motions is used [16].

In this paper, after a brief introduction to governing equations in the next section, a finite volume algorithm is presented for simulation of nonlinear interface-structure interaction in two-phase flow in section 3. Water-exit simulation of a horizontal circular cylinder is then carried out using the developed code and presented in section 4. Symmetric and asymmetric free falling of wedges are also investigated.

2. Governing equations

The computational mesh has to be adapted to the instant position of the free surface using boundary conditions when implementing surface methods in simulating the free surface. This issue is dramatically difficult when strong deformation of the free surface appears. Another type of method, which is among the so called volume method, is more suitable to analyze complex free surface and therefore has been taken in this study. This method treats the flow as a mixture of two phases (e.g. water and air). The following equation is solved to determine the volume fraction α implying the availability of two phases in each CV. A moving mesh is implemented in this study, so the equation is based on Arbitrary Lagrangian-Eulerian (ALE) approach [16]:

$$(1) \quad \frac{\partial \alpha}{\partial t} + \vec{\nabla} \cdot (\alpha \vec{c}) = 0.$$

So, the computational domain extends over both water and air phases; the volume fraction is initially set to be equal to 1 for CVs filled by water (phase 1) and 0 for CVs filled by air (phase 2). Therefore, one can treat both phases as a single effective phase whose properties like density ρ_{eff} and viscosity ν_{eff} vary in space according to the volume fraction of each phase ($\rho_{eff} = \alpha\rho_1 + (1-\alpha)\rho_2$; $\nu_{eff} = \alpha\nu_1 + (1-\alpha)\nu_2$) [17].

Afterwards, Navier-Stokes and continuity equations are solved for an incompressible viscous effective fluid based on ALE approach:

$$(2) \quad \frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j} (u_i c_j) = -\frac{1}{\rho_{eff}} \frac{\partial P}{\partial x_i} + \nu_{eff} \frac{\partial}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} \right) + g_i,$$

$$(3) \quad \frac{\partial u_j}{\partial x_j} = 0 \quad \text{where} \quad i = 1, 2 \quad \text{or} \quad 3 \quad \text{and} \quad j = 1 \quad \text{to} \quad 3.$$

Therefore, it is assumed that if one CV is partially filled with one phase and partially filled with the other phase both phases have the same pressure and velocity. In addition, the effect of surface tension based on the Weber number of evaluated test cases is neglected. The Weber number is important only if it is of the order of unity or less, which typically occurs when the surface curvature is comparable in size to the liquid depth, e.g., in droplets or capillary flows [18]. However, the effect of surface tension could be taken into account by transforming the resultant force into a body force at Navier-Stokes equations [19].

Movements of the body are estimated under the influence of interfacial flow-induced as well as other external loads by solving the linear and angular momentum equations:

$$(4) \quad \vec{F} = \vec{W} + \vec{F}_{flow} = m\vec{g} + \sum_{k=1}^n (-P_k \vec{n}_k + \vec{\tau}_k) A_k = m\vec{a},$$

$$(5) \quad \vec{M}_G = \vec{M}_{G_{flow}} = \vec{M}_{G_{ext}} + \sum_{j=1}^n (\vec{r}_k - \vec{r}_G) \times (-P_k \vec{n}_k + \vec{\tau}_k) A_k \\ = I_G \vec{\alpha} + \vec{\omega} \times I_G \vec{\omega}.$$

All equations and vectors are expressed for a non-deformable moving CV with an arbitrary velocity in an inertial reference system [18].

3. Numerical solution

3.1. Discretization

All governing equations are integrated over a hexahedral CV with an arbitrary velocity and a time interval before implementing the finite volume discretization. For example, the volume fraction transport equation becomes:

$$(6) \quad \int_t^{t+\Delta t} \left(\int_V \frac{\partial \alpha}{\partial t} dV \right) dt + \int_t^{t+\Delta t} \left(\int_V \vec{\nabla} \cdot (\alpha \vec{c}) dV \right) dt = 0,$$

here, the simplest approximation for the spatial discretization of the first term (unsteady term) is to replace it by the product of the value of the integrand at the CV center and the volume of the CV. The first-order Euler implicit interpolation is the obvious choice about its temporal discretization. After investigating different interpolation techniques [20], The Compressive Interface

Capturing Scheme for Arbitrary Meshes (CICSAM) [17] is used for spatial discretization of the convection term. In addition, Crank-Nicholson interpolation for its temporal discretization is used based on the authors' investigation [20].

Navier-Stokes equations would be ready for discretization by integrating over a CV and using the Gauss theorem. The diffusion term (second term in r.h.s. of Eq. (2)) is estimated by a summation over six CV faces using the midpoint rule, where each compartment is estimated by the over-relaxed interpolation [21]. The convection term of Eq. (2) (second term in l.h.s. of Eq. (2)) is approximated as well as its counterpart in Eq.(6) and the Gamma interpolation is used to approximate the required fluid velocity at the CV face. The pressure term (first term in r.h.s. of Eq. (2)) is also discretized using a new interpolation (Piecewise Linear Interpolation (PLI)) which is introduced by the authors. It works well in the presence of two phases with high density ratio [22].

3.2. Velocity and pressure coupling

A fractional step method is implemented [23] to compute pressure and velocity fields. Here, Crank-Nicholson scheme is used for the temporal discretization of the convection term in contrast to Adams-bashforth scheme used in the original method. In addition, convection term is linearized using Picard iteration method instead of the Newton's method in [23]. This is expressed as a flowchart in [16].

3.3. Solution algorithm

As mentioned in previous sections, one encounters three distinct sub-problems in CFD simulation of an interface-structure interaction problem. First sub-problem including Navier-Stokes and continuity equations is solved after a pre-processing (mesh generating and initializing of the parameters). Its outputs i.e. velocity and pressure fields, are used to calculate total loads acting on the body. Then, 6-DoF rigid body motion equations are solved in second sub-problem based on available loads. This yields to rigid body linear and angular velocities. At this step, constraints based on the difference between two sequential iterations calculated loads are checked. Fulfillment of such conditions let the procedure to continue and to move the body as well as the computational mesh. Otherwise, it will be back to the first sub-problem again. This inner loop provides a strongly coupled solution in the whole computational domain between rigid body motions and flow field. After all, the last sub-problem (volume fraction transport equation) is solved to represent the free surface in the position-updated mesh. Such an algorithm is continued

to capture a desired time history. The body-attached mesh motion strategy is clearly shown in Fig. 1(a) by representing a computational domain before and after the rigid body motions. Fig. 1(b) also displays a general view of described solution algorithm.

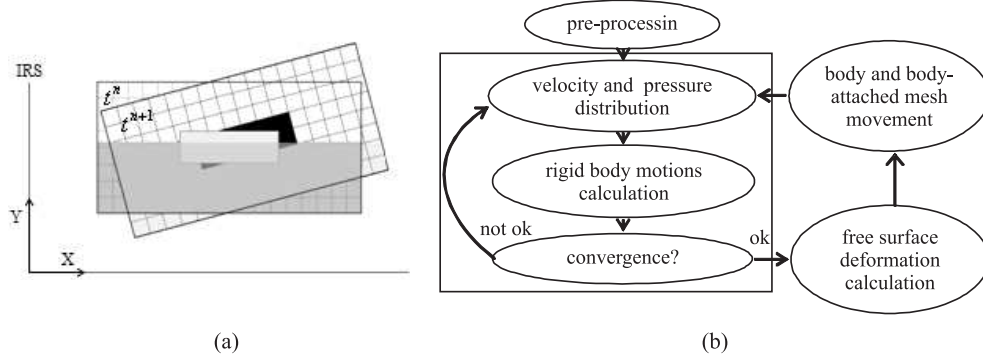


Fig. 1. Solution algorithm (a) motion simulation strategy at t^n and t^{n+1} ,
(b) relation between sub-problems

4. Numerical results

4.1. Water-exit of a horizontal circular cylinder

Here, a cylinder is pulled out of the water with a constant speed of 0.87 m/s from an initial depth of 0.3 m below the calm water surface. A 60×60 cm computational domain is used including 18900 CVs in this simulation while its diameter is 11 cm. No-slip boundary condition is used on cylinder wall which means that the fluid at the wall takes its velocity. Here, extrapolation is used for pressure. Hydrostatic pressure condition is then applied on the left and the right boundaries. In other words, the hydrostatic pressure is calculated at these boundaries with respect to the undisturbed free surface level. The velocity at the boundary is also obtained with the Neumann boundary condition by a zero-gradient. The upper boundary is high enough above the water level so the fluid is at rest. The pressure at this boundary is unknown and its boundary value is extrapolated from the interior of the flow domain. However, the gradient of the pressure at this boundary is small, and it is sufficient to apply a zero-gradient boundary condition. For the lower boundary, an outlet condition is implemented. Here, extrapolation of all the flow quantities is an appropriate choice. On the other hand, it is practical to calculate the pressure distribution from hydrostatic at this boundary and apply a zero-gradient for the velocity. Zero-gradient is used for volume fraction at all aforementioned boundaries [17].

The computational domain is also initialized as it is at rest at the beginning of simulation. So, hydrostatic and static situations are implemented for water and air velocity and pressure fields, respectively. The results are compared to the experiments [24] (Fig. 2). The numerical simulation has successfully predicted the dominating phenomena during the cylinder water-exit. The water above the cylinder is lifted and thin layer is formed subsequently around the above part of the cylinder. In this test case, the moving mesh has a constant speed of 0.87 m/s upward with respect to the forced motion of the cylinder. So, solving rigid body motions equations (Eq. (4) and Eq. (5)) are not necessary in this simulation.

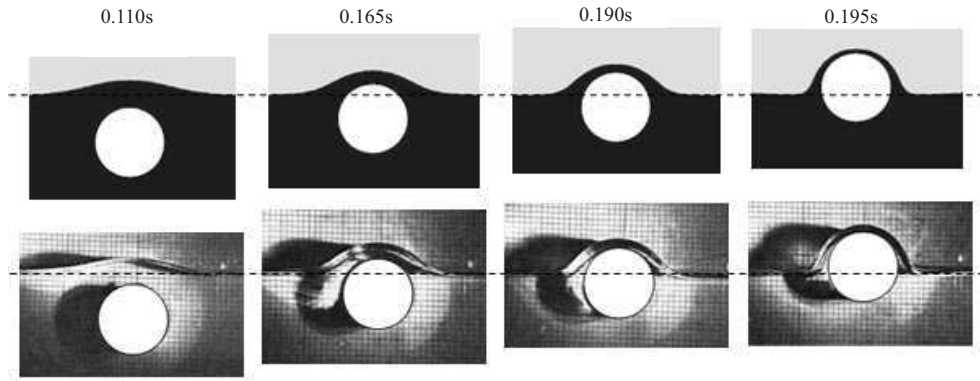


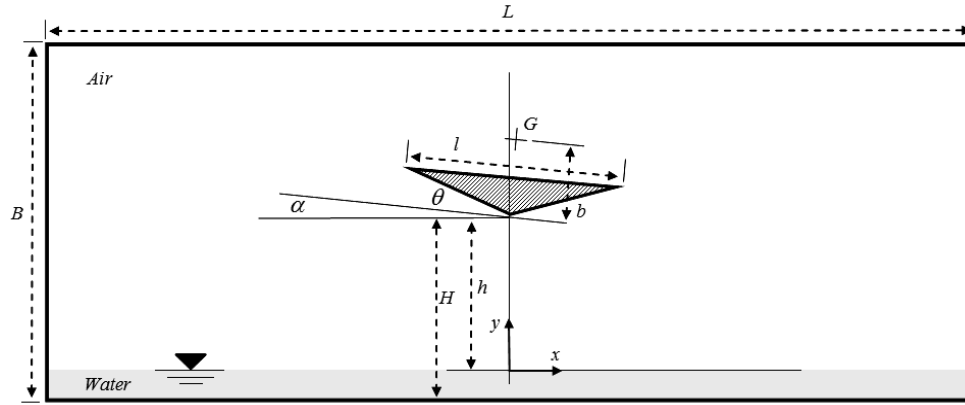
Fig. 2. Free surface deformation in water-exit of a horizontal circular cylinder; (up): simulation, (down): experiment [24]

4.2. Symmetric and asymmetric wedge water-entry

Wedge slamming is an important case study among water-entry simulations. It is a typical case for motion of ships in waves. In this section, symmetric and 1-DoF impact, as well as asymmetric and 2-DoF impact, of free-falling wedges are simulated with characteristics shown in Fig. 3. In other words, after releasing the wedge from an initial position without linear/angular velocity, it falls and reaches a speed just before the impact under the influence of gravity. Impact speed is 5 m/s and 3.46 m/s in symmetric and asymmetric cases, respectively. Initial and boundary conditions are applied analogously as in the water-exit case. Figure 4(a) shows the time history of wedge water-entry after symmetric impact, which is simulated using three meshes and compared to experimental data [25]. It is obvious that using mesh of higher quality reduces the error. But, there is still a gap between simulation and experiment,

even in the case of 18000 CVs mesh, which could be predicted earlier. In other words, using meshes which are finer than 18000 CVs can't significantly improve the accuracy of numerical results as investigated in this study.

The implemented algorithm neglects the effect of components such as compressibility and hydroelectricity and the variance in results is a direct consequence of such assumptions. However, the accuracy is still acceptable. Time history of slamming load is also presented in Fig. 4(b) using such meshes. Effect of the mesh quality is obvious, especially in calculation of the maximum point in both axes. Calculated force in the case of fine mesh is decreased approximately 25.8% in comparison to that of coarse mesh. Also, the time of maximum force occurrence is moved backward. Such results do not vary using finer meshes.



Type of Impact	[m]						[–]degree		[kg]	[kg.m ²]	[–] CV	[s]
	B	L	H	H	B	L	θ	α	weight	I _{zz}	Mesh Quality	Time Step
Symmetric	0.2	1.2	1.3	2.0	3.0	3.6	25	0	94	–	18000	0.002
Asymmetric	0.216	0.61	0.61	0.7	1.4	2.8	20	5	124	8.85	40000	0.0005

Fig. 3. Symmetric and asymmetric impact of free-falling wedges; problems characteristics

Finally, the effect of considering/neglecting the speed reduction after the wedge water-entry is investigated, see Fig. 5. The simulation is repeated using constant speed of 5 m/s when the wedge passes through water (0-DoF). Maximum impact load is 6.6 times greater in the case of 0-DoF in comparison to that of 1-DoF which consequently yields to a meaningful overestimation in structure design. Free surface deformation is also compared in such cases,

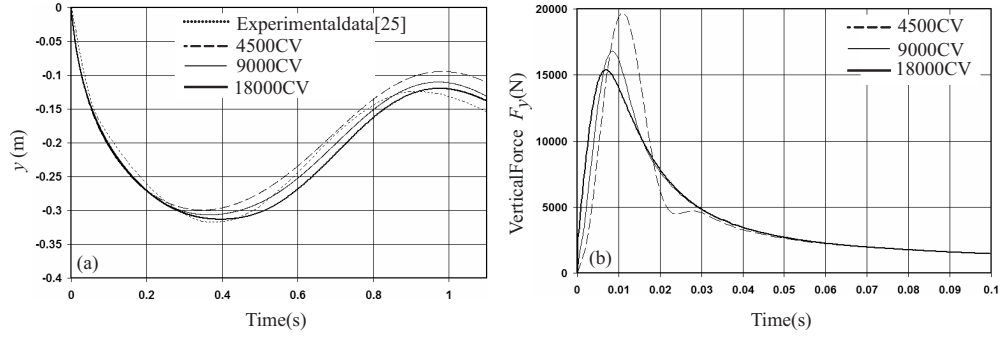


Fig. 4. Wedge-water entry, (a): vertical displacement, (b): impact load

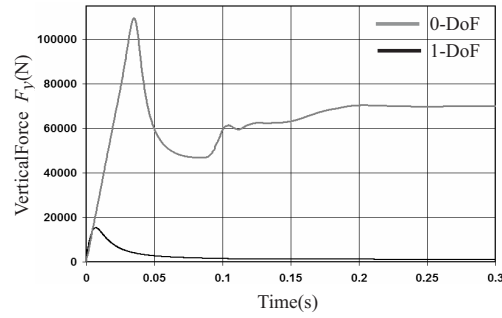


Fig. 5. Impact load in two cases of 0-DoF and 1-DoF symmetric slamming

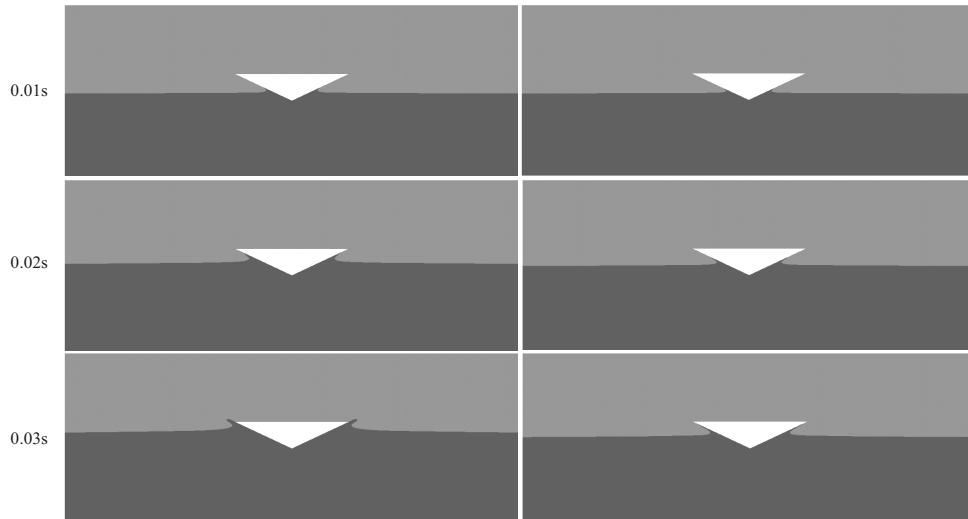


Fig. 6. Free surface deformation in symmetric slamming; (left): 0-DoF, (right): 1-DoF

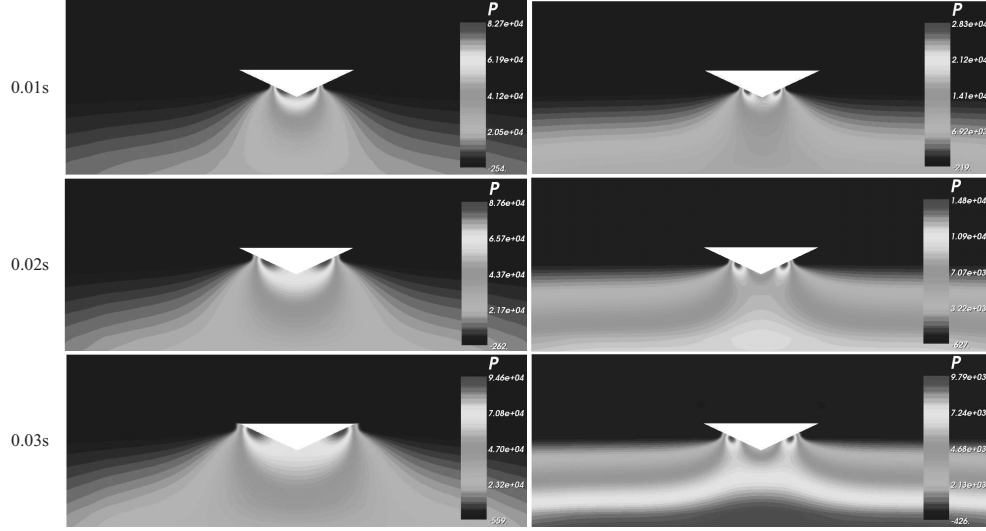


Fig. 7. Pressure contours in symmetric slamming; (left): 0-DoF, (right): 1-DoF

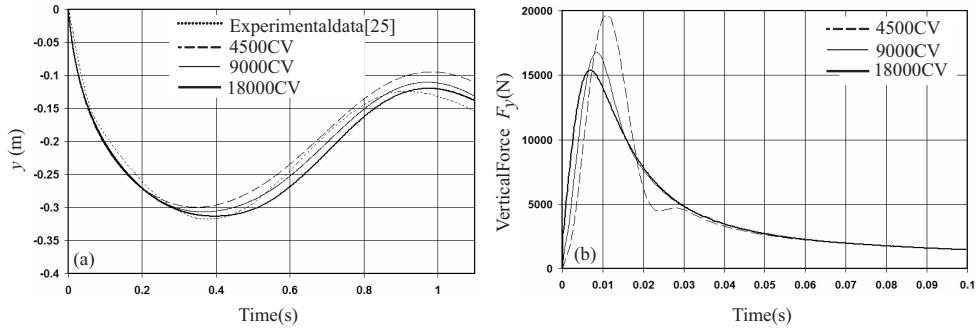


Fig. 8. 2-DoF asymmetric wedge slamming (left): linear acceleration (right): angular acceleration

as well as pressure contours in Figs. 6 and 7, respectively. In the case of 1-DoF slamming, pressure contours return to an approximately hydrostatic distribution rapidly after the impact, while the disturbance is strongly retained in the case of 0-DoF impact. This is the direct factor of aforementioned load overestimation.

Accelerations in 2-DoF asymmetric slamming are compared with experimental data [26] and another numerical simulation using SPH method [27] in Fig. 8. The error is greater in the case of angular acceleration calculation

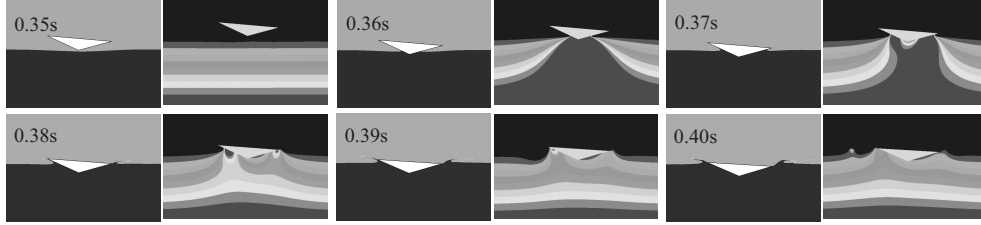


Fig. 9. 2-DoF asymmetric wedge slamming; free surface and pressure contours

rather than the linear one. This can be interpreted as the more important effect of implemented algorithm simplifications in angular motion. Free surface deformation and pressure contours are also shown in Fig. 9.

5. Conclusion

The present method shows good accuracy and capability for simulation of highly nonlinear interface-structure interaction problems. It simply provides the possibility of including all rigid body motions in a simulation procedure. This enables the algorithm to be applied in practical water-entry and water-exit problems. Different case studies are presented and discussed. The ability to simulate large interface deformations is appreciable in addition to accurate calculation of impact loads and motions time histories. Development to 3D problems is also straightforward by discretizing over a hexahedral CV instead of a quadrilateral one.

REFERENCES

- [1] FALTINSEN, O. M., M. LANDRINI, M. GRECO. Slamming in Marine Applications. *J. Eng. Math.*, **48** (2004), 187–217.
- [2] FALTINSEN, O. M. Hydroelastic Slamming. *J. Marine Sci. Tech.*, **5** (2000), 49–65.
- [3] BERTRAM, V. Practical Ship Hydrodynamics, Butterworth-Heinemann, Oxford, 2000.
- [4] VON KARMAN, T. H. The Impact on Seaplane Floats During Landing. NACA TN 321, 1929.
- [5] WAGNER, H. Über Stoss- und Gleitvorgänge an der Oberfläche von Flüssigkeiten. *Zeitschrift für angewandte Mathematik und Mechanik*, **12** (1932), No. 4, 193–215.

- [6] DOBROVOLSKAYA, Z. N. On Some Problems of Similarity Flow of Fluid with a Free Surface. *J. Fluid Mech.*, **36** (1969), 805–829.
- [7] BAGNOLD, R. Interim Report on Wave Pressure Research. *J. Inst. Civil Eng.*, **12** (1939), 202–226.
- [8] VERHAGEN, J. H. G. The Impact of a Flat Plate on a Water Surface. *J. Ship Res.*, **11** (1967), No. 4, 211–223.
- [9] Korobkin, A.A., Blunt-Body Impact on a Compressible Liquid Surface. *J. Fluid Mech.*, **244** (1992), 431–453.
- [10] WATANABE, I. Analytical Expression of Hydrodynamic Impact Pressure by Matched Asymptotic Expansion Technique. *T. West-Japan Soc. Naval Arch.*, 71, 1986.
- [11] KVAALSVD, J. Hydroelastic Modeling of Wetdeck Slamming. *Ph.D. Thesis*, Trondheim: Dept. Marine Hydrodynamics NTNU, 1994.
- [12] KVAALSVD, J., FALTINSEN, O. M. Hydroelastic Modeling of Wet Deck Slamming on Multihull Vessels. *J. Ship Res.*, **39** (1995), No. 3, 225–239.
- [13] FERZIGER, J., M. PERIC. Computational Methods for Fluid Dynamics, 3rd Rev. Ed., Springer Verlag, 2002.
- [14] PANAHI, R., M. SHAFIEEFAR. A Finite Volume Algorithm Based on Overlapping Meshes for Simulation of Hydrodynamic Problems. *J. Marine Sci. Appl.*, **8** (2009), 281–290.
- [15] PANAHI, R., M. SHAFIEEFAR. Towards a Numerical Hydrodynamics Laboratory by Developing an Overlapping Mesh Solver Based on a Moving Mesh Solver; Verification and Application. *Appl. Ocean Res.*, **32** (2010), No. 3, 308–320.
- [16] PANAHI, R., E. JAHANBAKHSH, M. S. SEIF. Development of a VoF Fractional Step Solver for Floating Body Motion Simulation. *Appl. Ocean Res.*, **28** (2006), No. 3, 171–181.
- [17] UBBINK, O., R. I. ISSA. A Method for Capturing Sharp Fluid Interfaces on Arbitrary Meshes. *J. Comp. Phys.*, **153** (1999), No. 1, 26–50.
- [18] WHITE, F.M. Fluid Mechanics, Mc Graw-Hill, 4th Ed., 2001.
- [19] BRACKBILL, J., D. KOTHE, C. ZEMAACH. A Continuum Method for Modeling Surface Tension. *J. Comp. Phys.*, **100** (1992), 335–354.
- [20] PANAHI, R., E. JAHANBAKHSH, M. S. SEIF. Comparison of Interface Capturing Methods in Two Phase Flow. *Iranian J. Sci. Tech., Transaction B: Technology*, **29** (2005), No. B6, 539–548.
- [21] JASAK, H. Error Analysis and Estimation for Finite Volume Method with Application to Fluid Flows. *PhD Thesis*, University of London, 1996.
- [22] JAHANBAKHSH, E., R. PANAHI, M. S. SEIF. Numerical Simulation of Three-Dimensional Interfacial Flows. *Int. J. Numer. Meth. Heat Fluid Flow*, **17** (2007), No. 4, 384–404.
- [23] KIM, D., H. CHOI. A Second-Order Time-Accurate Finite Volume Method for Unsteady Incompressible Flow on Hybrid Unstructured Grids. *J. Comp. Phys.*, **162** (2000), No. 2, 411–428.

- [24] GREENHOW, M., W. LIN. Nonlinear Free Surface Effects: Experiments and Theory. Report No. 83-19, Massachusetts Institute of Technology, 1983.
- [25] YETTOU, E. M., A. DESROCHERS, Y. CAMPOUX. Experimental Study on the Water Impact of a Symmetrical Wedge. *J. Fluid Dynam. Res.*, **38** (2006), No. **1**, 47–66.
- [26] XU, L., A. W. TROESCH, R. PETTERSON. Asymmetric Hydrodynamic Impact and Dynamic Response of Vessels. *Proceedings of 17th Int. Conf. Offsh. Mech. Arctic Eng.*, 98–320, 1998.
- [27] OGER, G., M. DORING, B. ALLESSANDRINI, P. FERRANT. Two-Dimensional SPH Simulations of Wedge Water Entries. *J. Comp. Phys.*, **213** (2006), No. **2**, 803–822.