Computers & Fluids 103 (2014) 86–99

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Sloshing impact simulation with material point method and its experimental validations ☆

J.G. Li [a], Y. Hamamoto [b], Y. Liu [a], X. Zhang [a,*]

[a] School of Aerospace, Tsinghua University, Beijing 100084, China
[b] Structural Strength Department, Research Laboratory, IHI Corporation, Yokohama 235-8501, Japan

ARTICLE INFO

Article history:
Received 24 January 2014
Received in revised form 24 June 2014
Accepted 26 July 2014
Available online 6 August 2014

Keywords:
Material point method
Contact algorithm
Liquid sloshing
Impact pressure

ABSTRACT

Liquid sloshing is usually associated with violent changes and breakups of free surfaces and strong fluid structure interactions. These phenomena present unique challenges for traditional computational fluid dynamics methods. In this paper, the material point method (MPM) is extended to solve the dynamic behavior of sloshing liquids in a moving container and a numerical scheme is developed to calculate impact pressure based on a contact algorithm over background grids. Moreover, a weakly compressible equation of state which employs an artificial sound speed is incorporated into the MPM to compute the pressure field of the liquid phase and a special scheme is employed to apply harmonic excitation to the particle-discretized container. The performance of the improved MPM in prediction of liquid impact pressure is verified by modeling a water block dropping test onto an aluminum plate. To further validate the proposed scheme, liquid sloshing experiments in a partially filled tank are conducted. The slosh-induced impact pressures on the vertical walls of the tank obtained from the MPM simulation are in good agreement with the experimental results.

1. Introduction

The phenomenon of liquid sloshing is a problem of practical interests in several industrial fields, such as mechanical engineering, marine engineering, in-land, aerospace transportation and civil engineering [1]. All containers that carry liquids must address the issue of sloshing. For example, ocean going liquefied natural gas (LNG) vessels may suffer from external wave impacts and inertial loads during the voyage, which can result in local damage as well as global collapse to the main hull structure, and can then further lead to leakage of fuel, or even overturn of the vessel. Sloshing pressure is also an important parameter in the assessment of safety of aeronautic and astronautic vehicles to prevent incidents induced by violent sloshing of liquefied fuel inside the fuel tank. The liquid oscillations in large storage tanks caused by earthquakes can produce tremendous impact pressure on the walls, which may result in serious casualties and economic losses. Hence, accurate predictions of the sloshing impact loads on offshore structures, space vehicles, storage tanks, water reservoirs, road vehicle tanks and ships are of great concern to engineers, designers, physicists, and mathematicians.

In the past decades, extensive mathematical formulations, experimental studies and more recently numerical simulations have been performed to deal with sloshing problems [2]. The early research efforts of this issue focused on two dimensional linear problems in containers with simple geometry, which can be solved by analytical methods [3]. With the advent of the modern theory of nonlinear dynamics, nonlinear models [4–7] with viscous damping based on potential flow theory have been developed to further study both two dimensional and three dimensional complex surface dynamic phenomena of sloshing liquid in moving containers. However, these researches are usually valid for simple cases with linear or weakly nonlinear liquid sloshing dynamics, and analytical techniques for predicting large-amplitude sloshing are still not fully developed. Furthermore, liquid sloshing dynamics generally involve strong nonlinear phenomena such as wave breaking, particle splash, and jet flow. Thus, taking into account all these factors analytically is extremely difficult.

Sloshing pressure is an important parameter in an assessment of safety of hull design when ships containing liquids sail through rough seas. In order to find the characteristics of the impact pressure [8], researchers have to conduct large-scale experiments (even full scale tests) to reflect what happens in the tank and find

☆ Supported by the Tsinghua-IHI Cooperation Project (92), the National Natural Science Foundation of China (11272180), and Specialized Research Fund for the Doctoral Program of Higher Education (SRFDP) of Chinese Ministry of Education (20110002120045).

* Corresponding author.
E-mail address: xzhang@tsinghua.edu.cn (X. Zhang).

http://dx.doi.org/10.1016/j.compfluid.2014.07.025
0045-7930/© 2014 Elsevier Ltd. All rights reserved.

phenomena which only occur in large-size tanks due to the three dimensional effects since there is no proper scaling law [9] which can transfer the model results to prototype conditions perfectly, when it comes to impact loads. For these reasons, laboratory experiments usually have problems of long experiment cycle and high cost.

As mentioned above, experimental works are generally expensive and sometimes certain physical phenomena related to liquid sloshing cannot be scaled in a practical experimental setup. Recently more and more researches on liquid sloshing are focused on numerical simulations with the fast advancement of the computer hardware and numerical algorithms. A number of researchers have provided comprehensive reviews on the problem of liquid sloshing, and the related two dimensional and three dimensional numerical simulation methods [10–12]. Most of the numerical simulations are focused on Eulerian and Arbitrary Lagrangian–Eulerian grid-based methods, such as finite difference method (FDM) [13,14], finite volume method (FVM) [15,16], finite element method (FEM) [17,18] and boundary element method (BEM) [5,6]. As a complex fluid motion, sloshing processes usually involve the complex variation of liquid surface, violent turbulence and vortex, as well as the strong coupling between the fluid and container walls, which bring many difficulties to traditional grid-based numerical methods when dealing with liquid sloshing problems. These goals and simulation challenges have led to the development of hybird methods building upon the work of others. For example, traditional FDM and FVM often require special algorithms such as volume-of-fluid (VOF) method [19,20] or level-set method [21] to capture changing surfaces or moving interfaces for certain classes of problems in liquid sloshing dynamics.

Besides the grid-based methods, particle methods (or meshfree methods) in which fluid particles are followed in a Lagrangian manner provide innovative alternatives to solve complex fluid dynamics problems such as liquid sloshing dynamics. To name a few, Idelsohn et al. [22] developed the PFEM (particle finite element method) to solve incompressible flows with free-surfaces and breaking waves. Koshizuka et al. [23] and Lee et al. [24] used the MPS (moving particle semi-implicit method) to study violent free surface motions and impact loads. Nestor et al. [25] established the FVPM (finite volume particle method) for meshfree simulation of viscous flow problems in engineering.

More recently, smoothed particle hydrodynamics (SPH) is being increasingly used to simulate fluid motions for its simple calculation and easy implementation [26]. The SPH method was originally developed in the late 1970s to solve astrophysical problems in three-dimensional open space [27]. In 1990s, Libersky et al. [28] extended it to high strain hydrodynamics problems with material strength. Since then, SPH has been extensively studied and extended to different problems in science and engineering, including high explosive detonation and explosion [29,30], high velocity impact and penetration [31], multi-phase [32], free surface flows [33,34] and sloshing type problems [35,36]. After being continuously improved, SPH was extended first to weakly compressible flows [37,38], then to strongly compressible [39] and truly incompressible hydrodynamics [40] as well. In SPH, a series of particles possessing individual material properties are used to represent the state of a system and these particles are capable of moving in the space according to internal particle interactions and external forces. Since the collective movement of those particles is similar to the movement of a liquid or gas flow, SPH can give a very good description of violent fluid motion phenomenon like sloshing. As a purely Lagrangian meshfree method, SPH can naturally track material interfaces, free surfaces and moving boundaries, and the history of flow field variables (such as pressure, velocity, density) can be easily obtained by approximating the governing equations which are discretized on the particles. However, special care are required to the solid boundary treatment and the low accuracy of the pressure measurement on solid walls when dealing with fluid motions [2]. Moreover, a neighboring particles search is needed at each time step, which makes the SPH computation much expensive.

Among all kinds of particle-oriented methods, the material point method (MPM) [41,42] is an extension to solid mechanics problems of a hydrodynamics code named FLIP [43]. In MPM, the material domain is discretized with a set of Lagrangian material points (particles), each with local mass and other state variables in order to model history-dependent behaviors. A spatial background grid that provides an Eulerian description of the material domain is predefined to calculate the gradient and integrate the momentum equation. At each time step, the particles are rigidly attached to background grid and move with the grid. Kinematic variables are firstly mapped from particles to grid nodes to establish the momentum equations on background grid, and the solutions of the momentum equation are then mapped from grid nodes back to particles to update their velocities and positions. At the end of each time step, the deformed grid is discarded and a new regular computational grid is set up for the next time step so that there is no mesh distortion or element entanglement associated with the FEM, while numerical dissipation normally associated with Eulerian methods is reduced.

MPM combines the advantages of both Lagrangian and Eulerian methods [44]. These features make it fairly attractive in modeling liquid sloshing dynamics, which is usually associated with changes of free surfaces and violent fluid–structure interactions, wave interactions with other structures especially in ocean and coastal hydrodynamics and offshore engineering. Typical applications of MPM in this area include fluid–structure interaction [45–47], ice dynamics [48], multiphase flows [49]. According to the Courant–Friedrich–Levy (CFL) condition, the critical time step size in MPM depends on cell size of the spatial background grid, rather than the particle space in SPH, so that the time step used in MPM is much larger than that of SPH. Furthermore, there is no neighboring particles search which is very time consuming. Therefore, MPM is much more efficient than SPH [50–52].

This paper aims at predicting sloshing impact pressure by extending MPM to the simulation of dynamic behaviors of fluid under external excitations. In this study, the liquid medium is assumed to be weakly compressible by employing an artificial equation of state (EOS) to relate the pressure and density, which is the most commonly used procedure in particle-based method. A novel algorithm based on contact algorithms proposed by Bardenhagen et al. [53,54] and Huang et al. [55] is then developed for accurate calculation of impact pressure over the background grid while a special scheme is employed to apply harmonic excitation to the particle-discretized container.

The remaining parts of the paper are organized as follows. Section 2 gives a brief description of the mathematical formulation of MPM and its time integration scheme. Several numerical strategies for the simulation of liquid sloshing dynamics are presented in Section 3. In Section 4, the improved MPM is first verified by a water block dropping test and then validated by fluid sloshing experiments in a partially filled tank. Numerical results are in good agreement with experimental observations. Finally, conclusions are drawn in Section 5.

2. MPM formulations

The MPM discretizes a material domain by a set of particles, as shown in Fig. 1. Each particle carries the position, velocity, mass, density, stress, strain and all other internal state variables required for the constitutive model. A background computational grid is used to calculate the gradient terms.