Computers & Fluids 63 (2012) 9–26

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

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

Numerical study of sloshing liquid in tanks with baffles by time-independent finite difference and fictitious cell method

Chih-Hua Wu[a], Odd Magnus Faltinsen[b], Bang-Fuh Chen[a,*]

[a] Department of Marine Environment and Engineering, National Sun Yat-sen University, Kaohsiung, Taiwan
[b] Centre for Ships and Ocean Structures & Department of Marine Technology, NTNU, Trondheim, Norway

ARTICLE INFO

Article history:
Received 6 May 2011
Received in revised form 13 January 2012
Accepted 21 February 2012
Available online 30 April 2012

Keywords:
Tank sloshing
Baffle
Time-independent finite difference
Fictitious cell

ABSTRACT

The numerical analysis of liquid sloshing in tanks is a big challenge when the fully nonlinear and viscous effects are all included in the analysis. The analysis will become more complicate as the tank is attached with internal structures, such as baffles. The width of the baffle is very thin compared with the breadth length and the numerical technique used to capture the detailed flow phenomenon (vortex generation and shedding) around the baffle is very rare in the literatures. In this paper, a time-independent finite difference scheme with fictitious cell technique is used to study viscous fluid sloshing in 2D tanks with baffles. The Navier–Stokes equations in a moving coordinate system are derived and they are mapped onto a time-independent and stretched domain. The developed numerical model is rigorously validated by extensive comparisons with reported results. An experiment setup was also made to validate the present numerical sloshing results in a tank with baffles. The method is applied to a number of problems including impulsive flow past a flat plate, sloshing fluid in a 2D tank with a surface-piercing baffle, sloshing fluid in 2D tanks with bottom-mounted baffles. The effects of baffles on the resonant frequency are discussed. The present developed numerical model can successfully analyze the sloshing phenomenon in 2D tanks with internal structures and can be easily extended to 3D model.

1. Introduction

Sloshing waves in moving tanks have been studied numerically, analytically and experimentally in the past several decades, and these studies have explored significant phenomena such as linear and nonlinear effects of the sloshing wave and the effect of fluid viscosity. Numerous analytical, numerical and experimental analyses of sloshing fluid in a tank have been published. Ibrahin et al. [23] provide a detailed survey of the CFD research and a general insight into sloshing problems. The earliest analyses were simply linear, weakly nonlinear and inviscid analyses (such as Nakayama and Washizu [30], Barton and Parker [3] among many others). In the years following 1990, fully nonlinear free surface boundary conditions, complete primitive Navier–Stokes equations and fluid viscosity were included. If the interior of the tank is smooth, and the fluid viscosity plays a minor role, an inviscid/irrotational potential flow solution is suitable for analysis of the sloshing liquids in a tank. A potential formulation of the problem was employed by Nakayama and Washizu [30], Waterhouse [35] and Ockendon et al. [31] amongst others. The series of studies by Faltinsen and his co-workers constitutes a major contribution to this field of study [12–17]. Besides the potential flow approaches, many numerical studies of the problem with primitive variables have been made, particularly when the fully nonlinear effects of the waves on the free surface are included. Papers that described the modeling of two-dimensional or three-dimensional sloshing include Chen and Chiang [6], Frandsen [19], Chen [7], Chen and Nokes [8], Akyildiz [1,2] and more recent papers by Liu and Lin [28] and Wu and Chen [36]. The comprehensive discussions of the advantages and disadvantages of various computational fluid dynamics (CFD) methods were reported by Faltinsen and Timokha [18].

The tuned liquid dampers (TLDs) are used to suppress horizontal vibrations of structures. The TLD consists of a tank partially filled with water. The lowest frequency of sloshing is tuned to a structural natural frequency. A high damping of the sloshing is desirable. Warnitchai and Pinkeaw [34] studied the mathematical model compared with experimental investigations for a rectangular tank with flow-damping devices. The vertical flat plate and the wire mesh screen can cause significant damping effects on sloshing waves. Isaacson and Premasiri [24] developed the mathematic solutions and experiment investigations to describe the hydrodynamic damping due to baffles in a fluid-filled rectangular tank undergoing horizontal motions. The average rate of energy dissipation due to flow separation around baffles and the total energy of sloshing waves are used to estimate the hydrodynamic damping.

---

* Corresponding author.
E-mail address: chenbf@mail.nsysu.edu.tw (B.-F. Chen).

0045-7930/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2012.02.018

For numerical studies associated with TLD, the finite element method is a popular numerical method in solving sloshing liquid in a baffled tank. Cho and Lee [9,10], studied the sloshing liquid is a baffled tank by a nonlinear finite element method. Biswal et al. [5] applied FEM on computing the nonlinear sloshing response of liquid in a two-dimensional rectangular tank and a circular cylindrical container with rigid baffles. The effect of baffle parameters including length, numbers and position on sloshing response were discussed. Isaacson and Premasiri [24] further studied the hydrodynamic damping due to baffles in the tank. A boundary element method (BEM) model for liquid sloshing in a baffled tank was adopted by Firouz-Abadi et al. [20]. However, the potential flow assumption used in BEM cannot predict the effect of energy dissipation due to viscous effect and flow separation. Kim [26] and Kim et al. [27] employed the SLOA scheme to study the liquid sloshing with baffles and compared the impact pressure with that in an un-baffled tank. Liu and Lin [29] investigated liquid sloshing in a baffled tank with large-eddy-simulation (LES). In their study, the vertical baffle is a more effective tool in reducing the sloshing amplitude. Panigrahy et al. [32] did a series of experiment in a developed liquid sloshing with and without different types of baffles under various fill depths. They concluded that the introduction of baffles in the tank effectively decreases the sloshing displacement because the sharp-edged baffles could dissipate the kinetic energy by generating turbulence in the flow.

In the past ten years, several numerical analyses were reported. Since the width of the baffle is very thin compared with the breadth length and the reported numerical techniques are still unable to capture the detailed flow phenomenon (vortex generation and shedding) around the baffle and it still is a big challenge in the literatures. In the present study, the tank with internal structures (baffles) is considered and the treatment of the flow field around baffles is carried out by the combination of a fictitious cell approach and a time-independent finite difference method [8]. The second order upwind scheme is also used to deal with the convective terms. The phenomena of vortex generation and flow separation due to baffles are presented. The influence of baffles inside the tank on the natural frequencies of the tank is discussed in detail. Moreover, the damping effect of sloshing waves caused by internal structures mounted at the tank bottom is demonstrated and discussed in this study.

Section 2 introduces the equations of motion which are written in a moving frame of reference attached to the accelerating tank. The fully nonlinear free surface boundary conditions are listed in this section. In the following sections, the coordinate transformation functions that map the time-dependent domain into a fixed unit square, and allow for mesh stretching at the boundaries, are presented along with a sensible dimensionless governing equation. The proposed finite-difference method is also developed in Section 3 where the iterative procedure is introduced. Besides, the fictitious cell approach is implemented to deal with the interfaces of fluid and structures (baffle, tank bottom and tank walls). The comprehensive benchmark tests of the present numerical scheme are demonstrated in Section 4 and the setup of the experiment is included as well. Section 4 also presents detailed results of a baffled tank under surge motion and provides a discussion of the phenomena found in this study. The investigations of tune liquid damper (TLD) for 2D tanks are also dissected in the section. Section 5 summarizes the key conclusions.

2. Mathematical formulation

In this work, the sloshing phenomenon in a rigid tank with partially filled liquid is analyzed by a time-independent finite difference method. As illustrated in Fig. 1, the breadth of tank is L, and d_0 is the still water depth. The gas flow including the possibility of gas pockets is neglected. A fully nonlinear model of viscous 2-D waves in a numerical wave tank was developed. As the coordinate system is set to move with tank motions, the Navier–Stokes equations, can be derived and written as [6]

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}-\ddot{x}_c+v\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial z^2}\right) \tag{2.1}$$

$$\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+w\frac{\partial w}{\partial z}=-g-\frac{1}{\rho}\frac{\partial p}{\partial z}-\ddot{z}_c+v\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial z^2}\right) \tag{2.2}$$

The continuity equation for the incompressible flow is

$$\frac{\partial u}{\partial x}+\frac{\partial w}{\partial z}=0 \tag{2.3}$$

and the kinematic boundary condition on free surface is

$$\frac{\partial h}{\partial t}+u\frac{\partial h}{\partial x}=w \tag{2.4}$$

In the above equations, u and w are, respectively, the velocity components of fluid in x- and z-directions, $\ddot{x}_c$ and $\ddot{z}_c$ are the corresponding ground acceleration components, p is the pressure, h is the surface height,ρ is the fluid density, g is the gravitational acceleration and the kinematic viscosity of the fluid is v.

To fulfill the dynamic condition at the free surface, the conservation of linear momentum at the free surface should be held. In brevity, the discontinuity in the normal stress proportional to the mean curvature of the free surface by surface tension, and the continuity of tangential stress condition at the free surface must be held when the viscosity of fluid is considered. The above conditions can be written as follows [6,8]:

$$\frac{\partial u}{\partial z}=-\frac{\partial w}{\partial x}+\frac{4\frac{\partial w}{\partial x}\frac{\partial h}{\partial x}}{\left[\left(\frac{\partial h}{\partial x}\right)^2-1\right]} \tag{2.5}$$

$$p_0=\rho g(h-d_0)+2\mu=\frac{\left[1+\left(\frac{\partial h}{\partial x}\right)^2\right]}{\left[1-\left(\frac{\partial h}{\partial x}\right)^2\right]}\frac{\partial w}{\partial z} \tag{2.6}$$

Taking partial differentiation of (2.1) and (2.2) with respect to x and z, and summing up the results, one can obtain following Poission equation which is used to solve for pressure.

$$\nabla^2 p=-\rho\frac{\partial}{\partial x}\left(u\frac{\partial u}{\partial x}+w\frac{\partial u}{\partial z}\right)-\rho\frac{\partial}{\partial z}\left(u\frac{\partial w}{\partial z}+\frac{\partial w}{\partial z}\right) \tag{2.7}$$

The non-slip condition is applied at the boundary between fluid and solid, that is $u = 0$ and $w = 0$ at tank walls. However, the slip between liquid and solid is a convenient assumption to make in order to release the non-integrable stress singularity. The boundary treatment of contact line by Tang et al. [33] is adopted and has been successfully applied in Chen [6], Huang et al. [22] and Chen and Nokes [8] and will also be used in the present study. In Tang's assumption, the non-slip boundary condition is released at the region of the first two meshes beneath the free surface. Since the mesh size near the free surface is stretched, the slip condition is only applied in a very small region, less than 1% of the liquid depth.

To evaluate the free surface, the most famous schemes are SURF, MAC (Maker and Cell) and VOF (Volume of fluid) methods. The SURF scheme assumes a single valued surface profile and is potentially able to deal with a uniform representation of large free surface waves and even for overturning inception. The MAC method is a Largrangian concept and can treat overturning waves and reentry inception with a simple logic. While the VOF method tracks the volume occupied by the fluid rather than the free surface and the method is the most popular method used in the literatures. All the above methods could properly calculate the instant free