International Conference on Computational Methods in Marine Engineering
MARINE 2007
P. Bergan, J. García, E. Oñate and T. Kvamsdal (Eds)

NUMERICAL SIMULATION AND MODEL EXPERIMENTS OF SLOSHING IN LNG TANKS

R. WEMMENHOVE*, R. LUPPES*, A.E.P. VELDMAN* AND T. BUNNIK†

*Department of Mathematics, University of Groningen
P.O. Box 800, 9700 AV Groningen, The Netherlands
e-mail: r.wemmenhove@math.rug.nl

†Maritime Research Institute Netherlands (MARIN)
P.O. Box 28, 6700 AA Wageningen, The Netherlands

Key words: Hydrodynamic wave loading, Two-phase flow, Numerical simulation, Volume-of-Fluid, Compressibility.

Summary. A two-phase model has been developed to simulate hydrodynamic wave loading in different offshore environments. The numerical model is based on an improved Volume-Of-Fluid (iVOF) method, that is designed to simulate the interaction between waves and three-dimensional structures. In this paper, special attention is paid to the velocity field and pressure distribution around the interface of air and water. By using a newly-developed discretisation, spurious velocities at the free surface due to an incorrect density treatment are prevented. Compressibility of the gas phase is included by using an adiabatic equation of state. Some results of the validation of the numerical model through sloshing model experiments are shown.

1 INTRODUCTION

During all types of weather conditions, partially filled LNG tanks are facing varying pressure loads on their walls. Especially during more violent weather conditions, the interaction between ship movement and the motion of LNG inside tanks is worthwhile to investigate. The fluid distribution in the tanks strongly depends on both the tank filling ratio and the weather conditions the tanker is subjected to. Inside the tanks, the fluid is generally a complex mixture of different fluids, with a strong mixing under violent flow conditions.

The simulation of the fluid behaviour in LNG tanks shows an analogy with the simulation of hydrodynamic wave loading on other offshore structures [2], so similar models can be used. Most models focus on specific aspects of free-surface flows, such as the wave impact of aerated flows on walls or the velocity field under breaking waves.

When flow conditions are getting more violent, using a two-phase flow model is strongly recommendable. However, the small spatial and temporal scales of entrapped and entrained air in the flow are a serious problem in simulations. Existing two-phase models mainly focus on single bubbles or on quite regular waves. Keeping track of the dynamics of the air phase in sloshing experiments is difficult. The required resolution in combination with long time series in the experiments induces the need to use powerful computers, but even then long calculation times remain an issue. The present paper shows the results of two-phase numerical simulations of sloshing in a tank. The simulations are validated on a series of 1:10 scale sloshing model experiments. In comparison with earlier sloshing model experiments, the current validation experiments are on a larger model scale. Moreover, in the numerical model the experimentally measured global motion time-traces of the sloshing tank are used as input for the simulations.

2 COMPUTATIONAL MODEL

The advection of the two phases in the numerical model is based on the Volume-Of-Fluid (VOF) algorithm as developed by Hirt and Nichols [1]. A local height function [3] is used to improve the mass conservation properties of the fluid displacement algorithm. For a one-phase approach, the incompressible Navier-Stokes equations are solved with a free-surface boundary condition at the liquid-gas interface. This is in contrast with the two-phase approach, where this interface is no longer considered as a free surface, although the interface is still reconstructed using the VOF algorithm. Numerically important is the density jump across the liquid-gas interface, as the density can easily increase or decrease by a factor 1000, which imposes a challenge to the numerical stability and accuracy of the model.

The discretisation of the Navier-Stokes equations uses a staggered Cartesian grid with a spatial discretisation by means of a first- or second-order upwind scheme. The Forward Euler method is used for the discretisation in time.

Compressibility is introduced in the numerical model by the compressible continuity equation $\frac{\partial \rho}{\partial t} + \nabla \rho u = 0$. The pressure is solved by a pressure-Poisson equation, which now includes additional time derivatives and spatial gradients of the density:

$$dt\nabla(\frac{1}{\rho^n}\nabla p^{n+1}) = \frac{1}{\rho^n}\frac{\rho^{n+1}-\rho^n}{dt} + \frac{u^n}{\rho^n}\nabla\rho^n + \nabla\cdot\tilde{u}^n \qquad (1)$$

where ρ denotes the density, p the pressure, u the velocity, n the old time level and $n+1$ the new time level. The tilde velocity $\tilde{u}$ contains the convective and diffusive terms of the momentum equation. The 'compressible' terms in the Poisson equation are numerically dangerous, as the density value characteristically jumps a few orders of magnitude at the free surface. Therefore, these terms are reduced by splitting up the density into a liquid and a gas part and by applying $\frac{DF_s}{Dt} = 0$, where F_s denotes the liquid cell fraction. As a result, only the (less varying) gas density ρ_g remains in the compressible terms of the Poisson equation. This gas density is computed by the adiabatic equation of state.

Given the staggered grid properties, special attention should be paid to the discretisation of the density at different cell positions. The momentum equation is defined around cell edges, whereas the density is calculated in cell centers. Therefore, the density has to be averaged at cell edges in some way. The most straightforward method is to average between the densities of directly neighbouring cells. However, this corresponds to an averaging 'stencil' twice as wide as a momentum cell. An alternative method is to consider only the area between the two cell centers, i.e. the momentum control volume. In each case, the correct determination of the normal to the free surface is a prerequisite for the density averaging. Considering the pressure term $\nabla p/\rho$ in the momentum equation, the simple averaging method with a wide averaging stencil may lead to spurious velocities [4]. By using the smaller averaging domain, a gravity-consistent density averaging method can be obtained, with which these spurious velocities are prevented.

3 MODEL RESULTS: SLOSHING IN LNG TANKS

As the fluid behaviour in partially-filled tanks needs investigation, the numerical model is validated on test problems with a sloshing motion. To study this sloshing behaviour, both numerical simulations and model scale experiments have been carried out. For two test cases with low and high tank filling ratios the numerical and experimental results will be compared.
The model tests on scale 1:10 have been carried out to generate validation material for the numerical model. To match simulation and experiment, the numerical simulations are carried out on the same scale as the model experiments. The experimental setup has been designed as a 2D cross-section of a sloshing LNG tank. Prior to the tests at DNV (Det Norske Veritas) in Oslo, the entire test setup (except the oscillator to move the tank) has been built and verified at MARIN.
The sloshing tank model was based on an LNG tank inside a No. 96 LNG carrier. The inner side of the tank (the part open to fluid) has a width of $3897mm$ and a height of $2697mm$ on model scale. The tank is filled with water, while the front side and back side are made of perspex to enable visualisation of the fluid motion inside the tank. Figure 1 shows screenshots of the

Figure 1: *Screenshots of the 10 percent (left) and 95 percent (right) filling ratio sloshing model experiment.*

sloshing tank for low and high filling ratios. As visible in this figure, the side walls of the tank are equipped with a number of measurement panels for pressure transducers, while the water height in the tank is measured by means of 12 water height probes. Furthermore, the tank is equipped with two cameras to monitor the fluid configuration. The time traces of the global motion of the tank are used as input for the simulations.
The fluid configurations in figure 1 show a strong sloshing motion of the fluid, which is induced by the tank oscillation. In the left picture (low filling ratio), the water height strongly increases during the impact of the sloshing liquid, with a runup at the tank wall up to about one meter. In the right picture (high filling ratio), only a small volume of air is present within the tank. The air in the top of the tank can be considered as entrapped, while air entrainment (the bubble cloud just below the free surface) is clearly visible as well.
The water height development about $1m$ from the left tank wall is shown in figure 2 for the low (a) and high (b) tank filling ratios. The agreement between simulations and experiment is good for the low filling ratio test case. However, the two-phase simulation shows a smaller amplitude than both the one-phase simulation and the experiment. This is partially caused by the numerical dissipation inherent to the first-order upwind scheme. For the high filling ratio

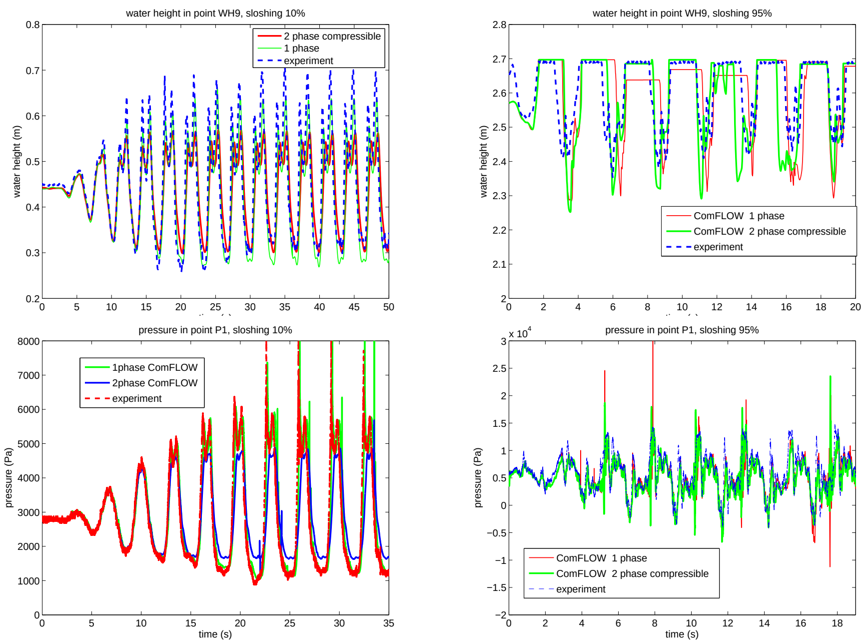

Figure 2: *Water height development for 10 % (a) and 95 % (b) filling ratio sloshing model experiments. Pressure level development at the tank wall for 10 % (c) and 95 % (d) filling ratio cases.*

test case, the water height is equal to the tank height during a part of the oscillation period. The two-phase simulation shows better agreement with the experimental data. This is probably due to the empty cells near the tank ceiling, which are not calculated in one-phase simulations. The pressure level development at the lowest and highest position at the right side wall, for the low and high filling ratio test cases is shown in figure 2(c) and 2(d), respectively.
The pressure signals of simulation and experiment show a good agreement, with a strongly repetitive pattern for low filling ratios and a spiky pressure signal (in both simulation and experiment) for the high filling ratio test case.

REFERENCES

[1] C.W. Hirt and B.D. Nichols. Volume Of Fluid (VOF) Method for the Dynamics of Free Boundaries. *J. Comp. Phys.*, **39**, 201–225, (1981).

[2] K.M.T. Kleefsman, G. Fekken, A.E.P. Veldman, B. Buchner and B. Iwanowski. A Volume-Of-Fluid Based Simulation Method for Wave Impact Problems. *J. Comp. Phys.*, **206**, 363–393, (2005).

[3] A.E.P. Veldman, J. Gerrits, R. Luppes, J.A. Helder and J.P.B. Vreeburg. The numerical simulation of liquid sloshing on board spacecraft. *J. Comp. Phys.*, (2007), in press.

[4] R. Wemmenhove, R. Luppes, A.E.P. Veldman and T. Bunnik. Numerical simulation of sloshing in LNG tanks with a compressible two-phase model. *26th International Conference on Offshore Mechanics and Arctic Engineering, paper no. 29294*, (2007).