Proceedings of the Twenty-second (2012) International Offshore and Polar Engineering Conference
Rhodes, Greece, June 17–22, 2012

ISBN 978-1-880653-94–4 (Set); ISSN 1098-6189 (Set)

www.isope.org

Apply MPS Method to Simulate Liquid Sloshing in LNG Tank

Yuxin Zhang and Decheng Wan
State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering,
Shanghai Jiao Tong University, Shanghai, China

ABSTRACT

In the present study, liquid sloshing in a 2D membrane-type LNG tank is simulated based on Moving Particle Semi-Implicit (MPS) method, which is a meshless method. However, the traditional MPS method suffers from strong unphysical pressure oscillation. To overcome this, the present MPS employs some improvements, such as: nonsingular kernel function, mixed source term for pressure Poisson equation (PPE) and an accurate surface detection method. Smooth pressure field is obtained based on the present MPS method. The tank is forced to move at various modes: sway and roll motion. The effects of excitation period and amplitude on the flow are investigated. It is shown that the impact behavior is significantly affected by excitation period and amplitude. When excitation period is near resonance period, resonance phenomenon is observed. The flow is violent and a periodic impact behavior shows two large pressure peaks in each period. A case by combining the horizontal, vertical and roll motions is simulated. The predicted pressure on the wall of the LNG tank by MPS method shows a good agreement with experimental data and other numerical results. The impact behavior induced by liquid sloshing is accurately numerically predicted. In addition, violent free surfaces are observed. The present MPS method shows good flexibility in dealing with such complicated flows.

KEY WORDS: Sloshing; MPS; impact pressure; LNG tank; free surface flows

INTRODUCTION

Sloshing is a kind of fluid motion in partially filled tank. It is of great importance in design of ships that carry liquid cargo, such as LNG(Liquefied Natural Gas) ship, since the impact load induced by liquid sloshing may cause large damage on the tank and affect the safety of ship. So there is a strong need to predict the impact loads on the structure in design of LNG ship.

Sloshing flow is a highly nonlinear problem, which may involve complicated phenomena, such as breaking wave, high-speed impact on tank wall and overturning of free surface. Due to the complexity, analytical solution of such problem is quite difficult. Early theoretical studies on sloshing are usually based strong hypothesis(Faltinsen, 1974), such as flow is irrational and geometry of tank is simple. Therefore, analytical solution is invalid for sloshing in membrane-type LNG tank, especially when the tank is oscillated in resonance frequency.

CFD(Computational Fluid Dynamics), which numerically solves the flow field, has been an effective approach to predict impact load induced by sloshing. In the past two decades, there have been many studies on sloshing based on CFD, such as Veldman et al.(2007) employed a finite difference method(FDM) solver combined with volume of fluid(VOF) to investigate the influence of sloshing on the dynamics of spacecraft. Chen et al.(2009) analyzed the accuracy of numerically predicting impact pressure on the walls and ceiling of tank based on level-set method. Kim(2001) applied the SOLA-SURF method to simulate sloshing flows in 2-D and 3-D containers and adopted a buffer zone concept to calculate the impact pressure on the tank ceiling. Liu and Lin(2008) developed a 3-D two phase fluid flow model, which can simulate violent sloshing with broken free surface and strong turbulence under six DOF excitations. Hu et al.(2004) employed an improved constraint interpolation profile(CIP) method to investigate violent sloshing flow in a sway tank.

All of the above mentioned numerical studies on sloshing is based on grid system. An alternative approach in CFD is the meshless method, such as SPH(Smoothed Particle Hydrodynamics, Gingold and Monaghan, 1977; Lucy, 1977) and MPS(Moving Particle Semi-Implicit, Koshizuka and Oka, 1996; Koshizuka et al., 1998). In meshless method, the flow field is represented by a set of interacting particles, which have mass, momentum and energy, etc. The basic idea of the meshless method is to solve the flow field based on a set of Lagrangian particles, which are interacting with each other. As particles have no fixed topography, meshless method can easily trace large-deformed flow field. The fragmentation and coalescence of fluid are naturally simulated in meshless method(Zhang and Wan, 2011a; Zhang et al. 2011). Another advantage for meshless method is that there is no numerical diffusion near the free surface since the particles are traced based on lagrangian representation.

There has been some work on liquid sloshing based on particle method,