An Efficient Numerical Model to Predict the Onset of Sloshing in 3D Tanks

Yusong Cao and Fuwei Zhang
MARINTEK (USA) Inc.
Houston, Texas, USA

Aifeng Yao, Stergios Liapis and Steven Wu
Shell Global Solutions (US) Inc.
Houston, Texas, USA

ABSTRACT

This paper describes a simple, fast and effective numerical tool that can be employed to predict the onset of liquid sloshing in tanks and assess the severity of the hydrodynamic pressures applied on the tank walls. A potential flow model with linear boundary conditions was chosen to give an initial screening tool that provides reasonable accuracy and computational speed. The liquid in the tank is assumed incompressible and inviscid, and the flow is assumed irrotational. The flow can be then described by a velocity potential \( \Phi(t,x,y,z) \) which is governed by the Laplace equation. The flow problem can be formulated as an initial-boundary value problem. At each time step, the boundary value problem for the velocity potential is solved using a desingularized boundary integral method. A time-stepping approach is used, in which the kinematic and dynamic boundary conditions on the liquid free surface are integrated in time to update the surface elevation and the velocity potential for the next time instant. Three-dimensional problems are studied. The results of the potential flow model are validated with experimental results. For small forcing, the results using the linear model, agree very well with the experimental tests.

KEY WORDS: Liquid motion in tank; sloshing loads; potential flow; desingularized BIM; time-stepping approach.

INTRODUCTION

Sloshing of liquid cargo in a partially-filled tank can cause large pressures on the tank walls. In the case of wave periods close to the liquid natural periods, sloshing can become violent and may lead to structural failures. The underlying physics of the problem are very complex and not fully understood. A recent paper by Yung et al (2010) describes all the relevant parameters of the problem as well as the learning from an extensive model testing campaign. Advanced CFD methods (such as volume of fluid methods, level-set methods and SPH methods) have been proposed and developed in recent years for simulating liquid sloshing and providing better prediction of the phenomena (Yang and Lohner 2005, Lohner et al 2007, Pakozdi and Graczyk 2009, and Cao et al 2010). However, the CFD approaches are still very costly and too time consuming for practical applications (e.g. study of the liquid sloshing effects at the early stage of a vessel design; and use on board of a vessel for predicting the severity of sloshing effect, etc.). The present work is a continuation of the effort reported last year by Cao et al. (2010). In that study, two-dimensional sloshing problems were used as test cases to compare the suitability of a linear potential flow model, a semi-nonlinear potential flow model and computational fluid mechanics (CFD) methods for providing a fast predictive tool. It was found that the linear potential flow model and the semi-nonlinear potential flow model were superior over CFD simulations in terms of providing very good accuracy and requiring a reasonable computational time. Also relevant to the present work is the recent paper by Kim et al (2010) of employing experiments and a simple frequency domain algorithm to predict the sloshing intensity in liquid cargo tanks.

The objective of the present work is to develop a numerical algorithm that can be employed to identify the sea states that induce sloshing in liquid tanks and estimate the severity of the hydrodynamic pressures applied on the tank walls. After considering several sloshing algorithms, a potential flow model with linear free surface boundary conditions was chosen for the computer code development. This model is seen as a simple starting point that can be used to predict the onset of sloshing. The main reasons that made this code the best fit for this problem are: