International Journal of Offshore and Polar Engineering (ISSN 1053-5381)
Copyright © by The International Society of Offshore and Polar Engineers
Vol. 16, No. 2, June 2006, pp. 104–111

Sensitivity Study on Computational Parameters for Prediction of Slosh-induced Impact Pressure

Yonghwan Kim*
Department of Naval Architecture & Ocean Engineering, Seoul National University, Seoul, Korea

Yung-Bum Lee, Yong-Soo Kim and Jungmoo Lee
Research Institute, Daewoo Shipbuilding & Marine Engineering Inc., Koje, Korea

In this study, based on careful observation of experimental data, physics-based numerical models are developed for sloshing flows in ship cargo. The particular scheme of interest is a finite difference method based on the SOLA-SURF method. The technical issues of conventional methods are outlined, and the corresponding remedies are introduced. The present numerical method is validated by comparing computational results with the experimental data measured in model tests. In particular, sensitivity to critical computation parameters, e.g. mesh size and time segment, is observed. The comparison shows a fair agreement of overall fluid motions and hydrodynamic pressures.

INTRODUCTION

Recent activity in building large LNG carriers and designing coastal LNG platforms increases the demand for an accurate prediction of sloshing flow and corresponding hydrodynamic loads. Compared to the studies on sloshing flows of the late ’70s and early ’80s, the recent studies have a distinct difference: the analysis tool. That is, most studies now rely on numerical methods. Due to the dramatic development of computational resources in the last 2 decades, numerical skills are widely used in many engineering fields. Further, many CFD codes are available in the commercial market.

Despite a mature computational environment, the direct application of numerical techniques to the ship-sloshing problem is not easy. A primary reason is the occurrence of impact on the tank ceiling and side walls. When hydrodynamic impact is involved, a very careful analysis is required. In particular, in our engineering problem, an accurate prediction of slosh-induced loads on ship structures is the ultimate goal of sloshing analysis. We need then to predict the actual magnitude of impact pressure as well as the kinematics of the sloshing flow. For this, the general-purpose computational programs are not considered to be an adequate tool.

Some representative studies based on numerical methods have been conducted by Faltinsen (1978), Bridges (1982) and Mikelis (1984). Some examples of 3-dimensional analyses can be found in the recent works of Arai et al. (1994), Wu et al. (1998) and Kim (2001, 2002). However, only a limited number among these has actually demonstrated the capability to predict impact.

This study aims to develop physics-based numerical models for an accurate prediction of slosh-induced impact loads, and to observe sensitivity to computational parameters. The numerical method considered in this study is a finite difference method based on the SOLA-SURF scheme (Hirt et al., 1975). For 2-dimensional tanks, Mikelis (1984) presented an excellent comparison between experimental data and numerical solution based on the SOLA-SURF method. For 3-D tanks, Kim (2001, 2004) has recently solved the sloshing problem in rectangular and prismatic tanks. This study is based on Kim’s studies.

This study focuses on simulating violent sloshing flows, including the occurrence of impact. When the fluid motion is very violent, some local physical phenomena are extremely difficult to analyze. For example, splash and wave breaking are typical phenomena in violent flows, yet simulating those phenomena requires great computational effort. Hence, the primary concern in this study targets impacts generated by the global fluid motion.

The computational results are compared with experimental data. The model tests introduced in this paper were carried out by Daewoo and Hitachi (Kim, 1993). Two model tanks under forced sway motions were considered in the experiment, and the time histories of hydrodynamic pressure at 9 points were recorded. The study compares the quantities of peak pressure for various computational parameters. Since the computational results are influenced by mesh size, time segment and size of buffer zone, the sensitivity to those parameters is systematically observed.

FORMULATION

The mathematical formulation of sloshing flow inside a partially filled tank under force excitation is well known. This study considers 2 equations, the continuity and Navier-Stokes equations, in fluid domain. For the application of a finite difference method, let’s consider the discretization of tank domain into the Cartesian staggered grids, as shown in Fig. 1.

The continuity and Navier-Stokes equations can be combined and written to the following discrete forms:

$$\begin{aligned}\vec{u}^{*}_{ijk} = \vec{u}^{(n)}_{ijk} + \Delta t\left[-\frac{1}{\rho}(\widehat{\nabla} p)^{(n)}_{ijk} + \nu(\widehat{\nabla}^2 \vec{u})^{(n)}_{ijk} + (\vec{F})^{(n+1)}_{ijk}\right] \\ \left. - \Delta t\{\vec{u}\cdot\widehat{\nabla}\}^{(n)}_{ijk}\vec{u}^{(n)}_{ijk}\right\}\end{aligned} \tag{1}$$

for the computation of velocity components, and:

$$\Delta p_{i,j,k} = -\frac{\gamma(\widehat{\nabla}\cdot\vec{u}^{*}_{ijk})}{2\Delta t(1/\Delta x^2 + 1/\Delta y^2 + 1/\Delta z^2)} \tag{2}$$

---

*ISOPE Member.

Received September 5, 2005; revised manuscript received by the editors April 12, 2006. The original version (prior to the final revised manuscript) was presented at the 15th International Offshore and Polar Engineering Conference (ISOPE-2005), Seoul, June 19–24, 2005.

KEY WORDS: Sloshing, hydrodynamic impact, computational fluid dynamics, buffer zone, sloshing experiment.