ISOPE 2009 Sloshing Comparative Study: Simulation of Lateral Sloshing with Multiphase CFD

Bernhard Godderidge\textsuperscript{a}, Stephen Turnock\textsuperscript{a}, Nick Cowlan\textsuperscript{b}, Mingyi Tan\textsuperscript{a}

\textsuperscript{a}School of Engineering Sciences, Ship Science
University of Southampton.
Southampton, United Kingdom

\textsuperscript{b}BMT SeaTech Ltd.
Southampton, United Kingdom

ABSTRACT

As part of the ISOPE 2009 sloshing comparative study, the eight specified test problems are analyzed using the CFD code CFX-11. The computational models and meshes are based on the methods developed in previous studies. 20 oscillations are simulated for each case and the mean, maximum and the mean of the 1/10\textsuperscript{th} highest pressures are computed for each specified pressure sensor. The flow features and their influence on the impact pressure magnitude are then considered and both hydrodynamic impacts and impacts with air entrapment are observed. It is found that the peak pressures are up to 170 kN/m\textsuperscript{2}, with time durations of the order of one millisecond. The resolution of such small time scales and the occurrence of wave breaking and air entrapment, which influence the pressure significantly, require a robust multiphase model capable of simulating phase mixing.

KEY WORDS: Sloshing; CFD; multiphase model; fluid impact; sloshing comparative study.

INTRODUCTION

Sloshing occurs when a tank is partially filled with a fluid and subjected to an external excitation force (Olsen, 1976). Ships with large ballast tanks and liquid bulk cargo carriers, such as very large crude carriers (VLCCs), are at risk of exposure to sloshing loads during their operational life (Rizzuto and Tedeschi, 1997). The inclusion of structural members within the tanks dampens the sloshing liquid sufficiently in all but the most severe cases. However, this approach is not used for Liquefied Natural Gas (LNG) carriers and the accurate calculation of the sloshing loads is an essential element of the LNG tank design process (Bass et al., 1980; Knaggs, 2006). Recent increases in vessel size have renewed interest in methodologies for the simulation of the sloshing loads experienced by the containment system (Han et al., 2005; Card and Lee, 2005). Recent incidents of sloshing damage onboard LNG carriers (Hine, 2008) have added further urgency to the improvement of sloshing analysis techniques in LNG carrier and floating LNG design.

The work of Abramson (1966) summarizes the methods available in modern sloshing analysis, and Ibrahim (2005) gives an up-to-date survey of analytical and computational sloshing modeling techniques. A more general modeling technique is the solution of the Navier-Stokes equations using Computational Fluid Dynamics (CFD). Some recent examples of CFD sloshing simulation include Hadzic et al. (2002), Standing et al. (2003) and El Moctar (2006).

This study simulates eight sloshing problems in different tanks and at various filling levels as set in the ISOPE 2009 sloshing comparative study using the commercial CFD code ANSYS CFX-11\textsuperscript{1}. Pressure data at predetermined points are compared to experimental data at ISOPE 2009.

SLOSHING PROBLEM

The tank cross sections used in this study are shown in Fig 1(a) and 1(b). The longitudinal section is 1.7488 m long and 1.0703 m high. The transverse cross section is 1.516 m long and has the same height as the longitudinal cross section. Both low (20\%) and high (70\%) filling levels are considered. The tank motion is defined in the form $a \sin(\omega t)$, where amplitude $a$ and frequency $\omega$ for each case are given in Table 1.

\begin{figure}[h]
\centering
(a):longitudinal cross-section \hspace{1cm} (b): transverse cross-section
\caption{Sloshing tank sections}
\end{figure}

\begin{table}[h]
\centering
\begin{tabular}{|c|c|c|}
\hline
Case & Filling Level \% & Frequency $\omega$ (rad/s) \\
\hline
1 & 20 & 0.15 \\
2 & 70 & 0.15 \\
3 & 20 & 0.30 \\
4 & 70 & 0.30 \\
5 & 20 & 0.45 \\
6 & 70 & 0.45 \\
7 & 20 & 0.60 \\
8 & 70 & 0.60 \\
\hline
\end{tabular}
\caption{Filling levels and frequencies used in the simulations}
\end{table}

The locations of the pressure taps are given in Table 2 where the origin of the coordinate system is at the bottom left of a longitudinal tank with length and height of those in Figs 1(a) and 1(b). There are usually 5 mesh elements resolving the location of each pressure sensor in the refined regions.

\begin{table}[h]
\centering
\begin{tabular}{|c|c|c|}
\hline
Case & Filling Level \% & Pressure Tap Location (mm) \\
\hline
1 & 20 & (200, 200) \\
2 & 70 & (200, 200) \\
3 & 20 & (400, 400) \\
4 & 70 & (400, 400) \\
5 & 20 & (600, 600) \\
6 & 70 & (600, 600) \\
7 & 20 & (800, 800) \\
8 & 70 & (800, 800) \\
\hline
\end{tabular}
\caption{Pressure tap locations}
\end{table}

\footnote{1 The simulations were carried out using the Iridis 2 computational facility at the University of Southampton}