ABSTRACT: Liquidised Natural Gas (LNG) is transported by LNG ships and it will slosh in partially filled tanks. This will cause damage to tank structures, e.g. cracks and fatigue, and affect ship’s stability. This paper presents numerical studies of LNG sloshing in partially filled membrane tank with a baffle on the bottom using Finite Volume Method to minimise impact pressure.

The software Gambit and Fluent are used to generate the mesh and simulate the LNG sloshing motion. The LNG sloshing behaviour is analysed considering a two-dimensional membrane tank which moved in one direction only for each case studies. Two cases of the tank motions are considered, i.e., sway and roll. The LNG tank is excited by a regular sinusoidal wave and the excitation frequency is set as the natural frequency. Liquid filling levels considered are 25%, 50% and 75% of the tank depth, respectively. Moreover, the numerical results of the impact pressure are compared with the published experimental results and show good agreement. Furthermore, a vertical baffle is inserted on the bottom of the tank to control the maximum impact pressure and results are compared with non-baffle cases:

The key findings are: the effect of liquid depth on the maximum impact pressure for rotational motion is more significant comparing with the horizontal motion and offer solutions for reducing impact pressure effectively. The maximum impact pressure for the tank with baffle is reduced nearly 50% comparing with the cases without baffle on the bottom of the tank.

KEY WORDS: LNG sloshing; Membrane tank; Baffles; Minimising sloshing impact.

1 INTRODUCTION

With the increasing arouse of environment protection, green energy attracts the global attention. Natural gas plays a significant role to replace the old energy system. The natural gas shrinks to 1/600 its volume after liquefaction, so LNG (Liquefied Natural Gas) is more convenient for transport or storage. Comparing with the pipeline and train, the most economical way to transport LNG is through the ship. Usually, at the beginning of the LNG carrier transport voyage, all the tanks are fully loaded up to 98%. Due to the emergence of the spot market, partial filling cases will occur. It is known that liquid will slosh when the tank is not full-filled [1]. Sloshing motion in a partially filled tank will be violent in certain condition, i.e. the frequency of the motion of the LNG tank is equal or closed to the natural frequency of the interaction between the LNG and the tank structure [2]. So, there is a demand that the LNG carrier should be safety operated at all liquid filling level.

Sloshing is an important issue in the marine industry and liquid sloshing in a LNG tank has been studied for years. Sloshing phenomenon in a ship tank can produce considerable impact forces which will lead to instability or rollover of the ship. Moreover, a violent sloshing motion inside the LNG tank will generate excessively high liquid impact pressure on the tank wall and cause damage of the ship structure [3], such as break internal pipelines. The damaged ship structure will induce oil spilling or explosion. These accidents could lead to serious human casualties and environmental pollution. Thus, the accurate prediction of the liquid sloshing phenomenon in a LNG tank is an essential element of the LNG tank design process. In this paper, methodologies used to analyse the sloshing phenomenon are reviewed and a brief summary of recent studies on utilization of the baffles to control the liquid sloshing is presented.

The study of fluid sloshing has a long history and continues to attract considerable attention because of its importance in application. Typically, there are mainly three approaches used to analyse the sloshing impact which are experimental approach [4] and [5], theoretical analytic approach [6] and [7] and computational approach [8-10].

Abramson et al., [4] made a great contribution to the experimental approach to analyse the sloshing impact load in LNG tank. Comparing with the other approaches, the experimental study can include more complicated physical phenomena associated with sloshing, such as compressibility effect of the liquid during impact, air entrapment and elastic effect of the ship wall etc. However, the drawbacks of the experimental approach are obviously. It is time consuming and requires lots of effort to run a test, even the shape of the model is simple. Moreover, it is difficult to correctly scale of the impact load from the model to full scale [4]. Currently, the experimental approach is mainly used by classification societies (Lloyd’s Register and the American Bureau of Shipping).

For the theoretical analytic approach, Graham and Rodriguez [6] developed a linear model for the aerospace industry. The linear potential theory and shallow water theory were used. Faltinsen [7] proposed a nonlinear analytic method
which was a third-order theoretical sloshing model. The limitation of theoretical analytic method is that the result is accurate as the sloshing motion is mild. When the sloshing motion becomes violent and highly non-linear phenomenon occurs, the result calculated by the theoretical analytic method is no more accurate.

The sloshing phenomenon in most cases involves violent fluid movement and nonlinear problem will occur. Since the analytic method of this highly-nonlinear problem does not exist, the computational methods which is known as computational fluid dynamic therefore plays an important role to support the experimental evidence [11]. Faltinsen & Timokha [12] use finite-difference scheme to simulate the liquid sloshing in LNG tank. Another popular method is finite-element method [8, 9, 10, 13]. The computational method is to modelling the numerical model by obtaining the solution of Navier-Stokes equation. Nowadays, the commercial CFD codes are well developed, such as Star-CCM and Fluent. Good agreement of the computational results and the experimental results shows that the computational approach is both easy to apply and accurate enough for simulate sloshing in LNG tank [14].

Another key issue in computational method is accurate prediction of the free surface of the liquid. The current available techniques are marker and cell method [15], volume of fluid [14], smooth particle hydrodynamics [16] and level set method. Among these techniques, the volume of fluid has been improved [17] and is wildly used today. The VOF method can simulate the violent deformation of the free surface such as wave breaking, overturning, etc. [11], which achieves better CFD performance than other techniques.

After reviewed the methodologies used to study the sloshing phenomenon, a brief summary of recent studies on utilization the baffles to control the liquid sloshing are presented.

Isaacson and Premasiri [18] analysed the theoretical prediction of hydrodynamic damping due to baffles in a rectangular tank with horizontal oscillations. They estimated the total energy damping due to flow separation around the baffles. Moreover, by carrying out the experimental measurements, the theoretical model was validated. By investigating the effectiveness of various baffle configurations, it was found that the baffles located close to the free surface give a higher damping than the other locations. However, this theoretical model could not capture the energy dissipation and breaking waves with violent liquid sloshing.

Cho and Lee [19] investigated the two-dimensional liquid sloshing in a baffled tank under the horizontal forced excitations, by using velocity–potential-based nonlinear finite element method. They found that the liquid motion and dynamic pressure variation above the baffle were more significant than those below the baffle. In addition, the liquid sloshing were strongly dependent on the configuration of the baffle designed. Further study was given by Cho et al. [20], they used the same numerical method proposed by Cho and Lee [19] to research the resonance characteristics of liquid sloshing in a 2D baffled tank subjected to forced lateral excitation based on the linearized potential flow theory. The limitation of this study was that they cannot solve the viscous and the rotational motion of the liquid sloshing because of the limitation of potential flow theory.

Liu and Lin [21] studied the liquid sloshing in a three-dimensional rectangular tank with baffles. The VOF method was used to track the free surface motion. They found that the vertical baffle was more effective than the horizontal baffle in reducing the impact pressure on the tank wall.

Recently, Jung et al. [22] also simulated a three-dimensional incompressible viscous two-phase flow, to investigate the effect of the vertical baffle height on the liquid sloshing. It was found that with a fixed liquid height, a higher baffle will suppressed the liquid sloshing more significantly.

Previously study of the effect of baffle on liquid sloshing are mainly depend on the rectangular tank. In this study, a vertical baffle is inserted at the centre of a two-dimensional octagon membrane tank bottom to numerically examine the baffle effect on the sloshing motion. The liquid sloshing motion in membrane tank is firstly simulated for without baffle cases. Both horizontal and rotational motions are considered. The membrane tank is excited by a regular sinusoidal wave and the excitation frequency is set as the natural frequency. Liquid filling levels are set as 25%, 50% and 75% of the tank depth, respectively. Special consideration is given at low liquid filling level. Then, by comparing with the result of without baffles cases, the liquid sloshing in membrane tank with baffle is simulate at the cases where the impact pressure is large and the ability of the baffle to control the maximum impact pressure in membrane tank is examined.

2 MATHEMATICAL MODEL AND NUMERICAL APPROACH

The commercial CFD package, Fluent v13.0, is used to numerically simulate the sloshing phenomenon in a two-dimensional LNG tank. The two-dimensional sloshing problem is governed by the continuity equation (1) and the Reynolds-Averaged Navier-Stokes equation (2),

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 
\]

\[
\rho \frac{D \mathbf{u}}{Dt} = F_i - \frac{\partial P}{\partial x_i} + \mu \Delta \mathbf{u} - \rho \left( \frac{\partial u_i'}{\partial x_i} \right) 
\]

\[
u_i = \bar{u}_i + u_i' 
\]

where \( x_i \) is Cartesian coordinates, \( u_i \) is corresponding velocity component, \( \bar{u}_i \) is the time-averaged value, \( u_i' \) is the fluctuating velocity component, \( \rho \) is the density, \( \mu \) is the viscosity, \( F_i \) is the external body force and \( \frac{\partial u_i'}{\partial x_i} \) is the Reynolds-stress term.

The Volume of Fluid (VOF) method is applied in this paper to capture the free surface profile of the two-phase model. VOF is a well-developed method and capable of resolving the interface between the mixture phases, so there is no need for additional modelling of the inter-phase interaction of the free surface layer. In this paper, the two-phase considered are air and liquid natural gas. In each cell, the volume of fraction \( \gamma \) is defined in equation (4)

\[
\gamma = \frac{\rho_{\text{grid}} - \rho_a}{\rho_l - \rho_a} 
\]
where, $\rho_{\text{grid}}$ is the density inside the grid, $\rho_a$ is the density of the air and $\rho_l$ is the density of the liquid natural gas. The volume fraction is equal to 1 for the grid is full of LNG, equal to 0 for the grid is full of air and between 0 and 1 for the interface grid contain both LNG and air.

The VOF is used to track the variation of the interface by solving the continuity equation for the volume fraction of the prim phase given in equation (5):

$$\frac{\partial \gamma}{\partial t} + \frac{\partial \gamma u_i}{\partial x_i} = 0$$

The CFD solution selections are based on two aspects, time effective and results accurate. For these two reasons, Standard k-ε model is chosen for the viscous model, PISO is selected for Pressure-Velocity Coupling, Green-Gauss Cell Based is chosen for gradient discretization, PRESTO! For pressure discretization, Second Order Upwind scheme for momentum discretization and Geometric Reconstruction Scheme for volume fraction. As the velocity of the flow in each grid varies throughout the simulation process, a variable time step during the simulation procedure is more efficient than the specification of a fixed time step. The variation of the time step is defined by the Courant, Friedrihs and Lewy number (CFL) given in equation (6),

$$CFL = U \frac{\Delta t}{\Delta x}$$

where, U is the magnitude of the velocity of the liquid flow, $\Delta t$ is time step and $\Delta x$ is the length of the grid. In this study, in order to obtain a higher accuracy quality of the numerical results, the CFL number is set to 0.5. More details of the numerical methods can be found in ANSYS FLUENT 12.0 User’s Guide [23].

Figure 1 shows the schematic diagram of a two-dimensional membrane tank with a vertical baffle insert on the bottom of the tank and the location of the pressure sensors to record the pressure variation during the simulation process. In order to verify the present numerical scheme, the numerical results calculated by present method is validated with available experimental results and numerical results [5] and [13]. The present numerical scheme is duplicated and used to calculate the same sloshing model as described in [5] and [13] (Case A). The liquid sloshing impact pressure is validated with [13] (Case A) given in Figure 2 and the phase shape of the liquid during the sloshing simulation is validated with [5] given in Figure 3.

In order to verify the present numerical scheme, the numerical results calculated by present method is validated with available experimental results and numerical results [5] and [13]. The present numerical scheme is duplicated and used to calculate the same sloshing model as described in [5] and [13] (Case A). The liquid sloshing impact pressure is validated with [13] (Case A) given in Figure 2 and the phase shape of the liquid during the sloshing simulation is validated with [5] given in Figure 3.
where, \( p \) is the impact pressure, \( T \) is the period of the excited.

By comparing with Figure 2 and Figure 3, it is observed that both the impact pressure and the phase shape of the liquid during sloshing motion calculated by the present method have a good agreement with the published numerical results and experimental results [5] and [13]. For the impact pressure comparison, the present numerical scheme can predict the high peak value of the liquid impact pressure accurately while for the low peak value calculated by the present method is slightly higher compared with the numerical results from Kim et al. [13], but the discrepancy is acceptable. To conclude, the present numerical scheme and parameter settings in FLUENT are verified to be correct and can predict both the sloshing impact pressure and liquid phase shape accurately. Then, this numerical method will be used to calculate the cases considered in this study.

The simulated cases considered in this study are tabulated in Table 1.

### RESULTS AND DISCUSSION

Figure 5. Impact pressure on sensor 3-5 with three liquid filling levels (25%, 50% and 75% of tank depth), sway motion with 0.02 motion amplitude.

Figure 6. Impact pressure on sensor 3-5 with three liquid filling levels (25%, 50% and 75% of tank depth), roll motion with 8 deg motion amplitude.
3.1 Without baffle sloshing cases

The liquid sloshing motion in membrane tank is firstly simulated for without baffle cases. Both horizontal and rotational motions are considered. The membrane tank is excited by a regular sinusoidal wave and the excitation frequency is set as the natural frequency. Liquid filling levels are set as 25%, 50% and 75% of the tank depth, respectively. The liquid impact pressure is recorded by the pressure sensors inserted on the tank wall. For different liquid depths, the impact pressure recorded by the sensors which can demonstrate a better variation of the impact pressure during the sloshing motion is presented, i.e. sensor 3 for 25% liquid filling depth, sensor 4 for 50% liquid filling depth and sensor 5 for 75% liquid filling depth.

Figure 5 and Figure 6 illustrate the impact pressure during the liquid sloshing motion with different liquid filling depth for sway and roll motion, respectively. By comparing the two figures, it is found that the liquid depth effect on the maximum impact pressure for rotational motion is more sensitive compared with the horizontal motion. For horizontal motion shown in Figure 5, with same motion amplitude and excitation frequency is set as natural frequency, an increasing liquid depth has very little effect on the impact pressure during sloshing motion. However, for rotational motion, a lower liquid filling level will induce a larger impact pressure as presented in Figure 6.

Figure 7 illustrate the liquid phase deformation during the sloshing motion. Case 1,4,5 demonstrate the liquid phase deformation for horizontal motion without baffle cases. As can be seen from these figures, an increasing liquid depth will increase the nonlinearity of the liquid motion. At a deeper liquid depth, the wave break and liquid splash phenomenon is observed.

Figure 7. The liquid sloshing in membrane tank for cases with and without baffle, the motion conditions for each case are given in Table 1.
3.2 Slashing control using baffle

The cases without baffle show that a lower liquid filling level will induce a larger impact pressure. Then, a vertical baffle is inserted in the middle of the tank bottom to control the maximum impact pressure on tank wall at a low liquid filling level. Figure 8 compares the impact pressure for roll motion with 13deg motion amplitude, it is obviously seen that the impact pressure for cases with baffle decrease significantly comparing with the non-baffle case. The Figure 9 shows the impact pressure reduction for other cases. It is conclude that the maximum impact pressure for the tank with baffle is reduced nearly 50% comparing with the cases without baffle on the bottom of the tank.

By inserting a baffle on the tank bottom, the natural frequency of the membrane tank is shifted. The value of the new natural frequency nearly doubled compared with the original one. Thus, the resonance phenomenon is avoided which occurred at original excited frequency.

![Figure 8. Impact pressure comparisons between cases with and without baffle, the liquid filling level of the two cases are 25% of tank depth with roll motion with 13deg motion amplitude.](image)

![Figure 9. Comparing the results of the maximum impact pressure for cases with and without baffle, liquid filling level for all cases are 25% of tank depth, motion direction and motion amplitude are given below each bar.](image)

4 CONCLUSION

From this study, three main conclusions have been drawn:

1) With graphical illustration, it is found that a increasing liquid filling level in the membrane tank will increase the nonlinearity of the liquid motion during the sloshing.

2) The effect of liquid depth on the maximum impact pressure for rotational motion is more significant comparing with the horizontal motion. For rotational motion a lower liquid filling level will induce a larger impact pressure.

3) Installing a vertical baffle at the middle of the membrane tank bottom can greatly reduce the liquid sloshing motion and sloshing impact pressure. The maximum impact pressure for the tank with baffle is reduced nearly 50% comparing with the cases without baffle on the bottom of the tank.

REFERENCES


[17] E. Loots, B. Buchner, W. Pastoor, T. Tveite., The numerical simulation of LNG sloshing with an improved volume of fluid method,
Proceedings of the 23rd international conference on offshore mechanics and artic engineering.


