Water Sloshing in Rectangular Tanks – An Experimental Investigation & Numerical Simulation

Lyes Khezzar lkhezzar@pi.ac.ae
Mechanical Engineering
Petroleum Institute, P.O. Box 2533
Abu Dhabi, UAE

Abdennour Seibi aseibi@pi.ac.ae
Mechanical Engineering
Petroleum Institute, P.O. Box 2533
Abu Dhabi, UAE

Afshin Goharzadeh agoharzadeh@pi.ac.ae
Mechanical Engineering
Petroleum Institute, P.O. Box 2533
Abu Dhabi, UAE

ABSTRACT

This paper presents the steps involved in designing a test rig to study water sloshing phenomenon in a 560 x 160 x 185 mm PVC rectangular container subjected to sudden (impulsive) impact. The design encompasses the construction of the testing facility and the development of a proper data acquisition system capable of capturing the behavior of pre- and post impact water motion inside the tank. Fluid motion was recorded using a video camera for flow visualization purpose. Two water levels of 50 and 75% full as well as two driving weights of 2.5 and 4.5 kg were used. The experimental study was supplemented by a computational fluid dynamics study to mimic the fluid motion inside the tank. Examination of CFD capability to predict the behavior of the free surface of the fluid during the container initial motion and after impact is the focus of this paper. The flow fields, obtained using the numerical code, are in reasonable agreement with those from experiments. Both experimental and numerical results indicated the presence of a single traveling wave before impact, contrary to what was observed in previous studies.

Keywords: water sloshing, computational fluid dynamics, flow visualization.

1. INTRODUCTION

The problem of water sloshing in closed containers has been the subject of many studies over the past few decades. This phenomenon can be described as a free surface movement of the contained fluid due to sudden loads. Olsen [1] classified the free surface fluid motion in three different slosh modes consisting of i) lateral sloshing, ii) vertical sloshing, and iii) rotational sloshing (Swirling). Sloshing is a phenomenon that can be found in a wide variety of industrial applications such as Liquefied Natural Gas (LNG) carriers and their new design, rockets and airplanes fuel reservoirs and road tankers.

The design of this equipment requires detailed understanding of liquid motion during sloshing. Sloshing can be the result of external forces due to acceleration/deceleration of the containment body. Of particular concern is the pressure distribution on the wall of the container reservoir and its local temporal peaks that can reach as in road tankers twice the rigid load value. In road tankers, the free liquid surface may experience large excursions for even very small motions of the container leading to stability problems.

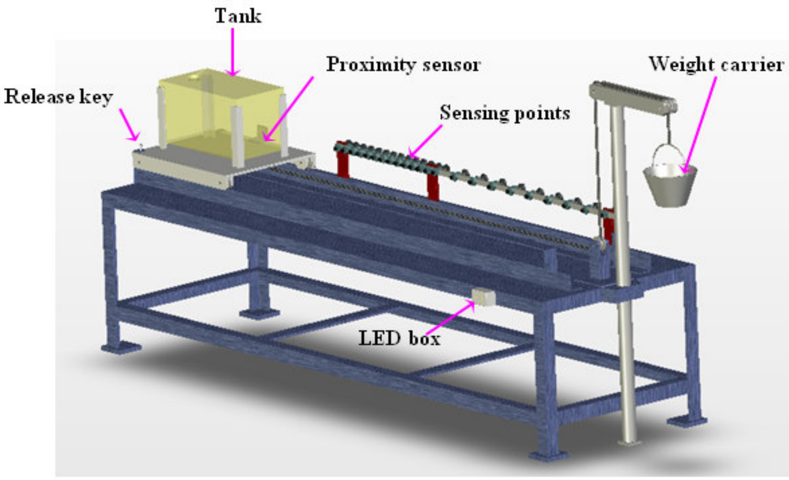
Several studies were conducted on sloshing of fluids and the extensive review by Ibrahim et al. [2] and Ibrahim [3] provide a thorough review of the subject liquid sloshing dynamics. Initial work started in the early 1960's with the study of the influence of liquid propellant sloshing on the flight performance of jet propelled vehicles. Chwang & Wang [4] applied nonlinear theory to calculate the pressure force in accelerating rectangular and circular container. It was found that during the initial stage of the impulsive motion, no traveling free-surface waves are present and the fluid simply piles up on one side of the container. Moreover, Popov et al. [5, 6] studied the effect of acceleration and curvature on the fluid motion in rectangular containers and observed that the dynamic coefficient is influenced by the aspect ratio of fluid height to length. The study revealed that maximum sloshing occurs in square containers with 30 – 60% fluid level and that maximum forces occur at a fluid level ranging between 75 – 93%. A similar study conducted by Ye and Birk [7] investigated the pressure variation at the walls of a horizontal cylindrical vessel during and after impact where fluid sloshing takes place at fluid levels less than 95% full. The study revealed that the pressure in the tank increases as the fluid level inside the tank increases. Faltinsen et al. [8] studied the transient loads on sloshing tanks and observed five distinct transient phases with different amplitudes. Chen and Chiang [9] conducted a simulation study on a simple two-dimensional rectangular tank with rigid walls subjected to horizontal and vertical accelerations using an inviscid and incompressible fluid to examine the nonlinear behavior of fluid motion. The study revealed that the fundamental frequency of the flow is strictly dependent on tank width and fluid depth. The effect of fluid viscosity was studied by Faltinsen and Rognebakke [10] and revealed that viscosity becomes prominent in small amplitude excitations and high fluid levels. Moreover, Bass et al. [11] found that viscosity has a minor effect on sloshing with large excitation amplitudes.

The traditional approaches that have been used to assess sloshing loads include linear and nonlinear potential flow theory, direct experimentation on scaled models and more recently the use of Computational Fluid Dynamics (CFD) investigated by Godderidge et al. [12, 13]. The results showed that the sloshing natural frequency and the inertia of the system are affected by the fluid level. Potential flow theory has some limitations and cannot model fluid fragmentation or merging. CFD is thus increasingly being considered as a viable tool for the study of such flows and is currently being tested and validated as a design method as described by Celebi and Akyildiz [14], Kassisnos and Prusa [15] and Ibrahim [2]. A comparative study, conducted by Cariou and Casella [16], has established that non-impulsive phenomena are correctly simulated but impacts and pressure peaks are still far more difficult to assess and need improvement.

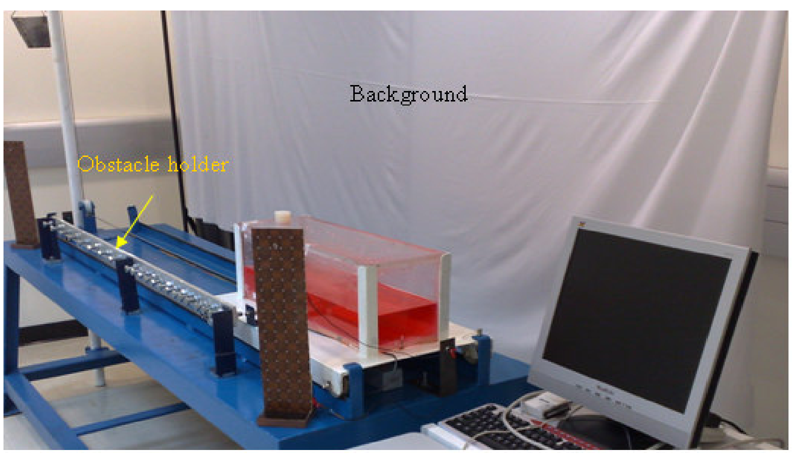
The present work focuses on the liquid flow dynamics inside a model accelerating rectangular container subjected to a sudden (impulsive) impact. The container is meant to represent a road tanker in motion and suddenly colliding with another object resulting in an impact and sloshing of the fluid inside it. The study involves experimental flow visualization and CFD modeling based on a two-dimensional geometry using a commercial CFD package. The next two sections contain the experimental set up, measuring, image processing, and computational techniques. These are followed by a result and discussion section and a conclusion.

2. EXPERIMENTAL STUDY

The experimental study aims at capturing the fluid motion before and after impact through visualization. Figure 1 shows a solid model and the actual experimental set up with proper data acquisition system. The test rig consists of an acrylic rectangular tank installed with proper instrumentation for data collection before and after impact. Measurements related to the tank displacement using proximity sensors (Omron E2A-M12) from which the tank speed and acceleration are determined as well as water sloshing behavior through visualization were recorded. Due to the high accuracy of the proximity sensor, the experimental error on time and space measurements are estimated to be less than 1%. Tank motion was recorded at i) the beginning of the accelerated motion, ii) while moving, and iii) after impact using a digital camera of 7.2 Mega pixels, which can capture up to 30 frames per second, and two Light Emitting Diodes (LEDs). The two LEDs were used as a trigger indicator of the release of the tank at the beginning of the experiment and tank impact when it hits the backstop at the end of the motion. Adjustable obstacles (bars) were aligned on the side of the track path where the first 14 bars have a center-to-center distance of 20 mm and the other 8 bars have a center-to-center distance of 50 mm (see Figure 1). Two magnets were installed on the impact wall in order to stop bouncing of the tank after it hits the stopper. A data acquisition system using an interface card (NI PCI-6221) was designed to record the feedback signals from all the sensors used for measurement.



a) Solid model of experimental set-up



b) Actual experimental set-up

FIGURE 1: Experimental setup and data acquisition system

2.1 Testing Procedure

The experimental set up, shown in Figure 1, consists of a rectangular Perspex container (175x175x550 mm) filled with water, the working fluid, and seated on a trolley that runs horizontally on two rails. Plastic wheels were used to minimize friction at the rail/wheels interface. The trolley was driven by a steel cable attached to a counterweight where a specified dead weight can be placed to achieve a desired trolley-water acceleration that reflects heavy trucks' motion carrying large tanks filled with fluid.

Tank motion was initiated by placing pre-defined dead weights (2.5 and 4.5 kg) in the weight carrier and releasing the pin attaching the base plate to the frame. The tank was filled with colored water to a certain level (25, 50, and 75% full). Tank motion was measured using a proximity sensor and a sensing range of 4-mm as it travels from the starting point till impact. The data acquisition system (DAQ) was integrated into the experiment to collect data related to tank movement from the proximity sensor and the LED's. Labview software was used to read and record the measurements with time. When the tank starts moving, the closed electric circuit of LED1 opens and generates a logic pulse captured by the DAQ, at the same time, LED1 lightens up. While traveling, data from the proximity sensor, which is attached to the base of the tank was recorded over time. At impact, LED2 lightens up indicating the time at which tank impact took place and the corresponding logic pulse is transmitted to the DAQ (see Figure 2). The test was repeated several times for each combination of water volume and dead weight.

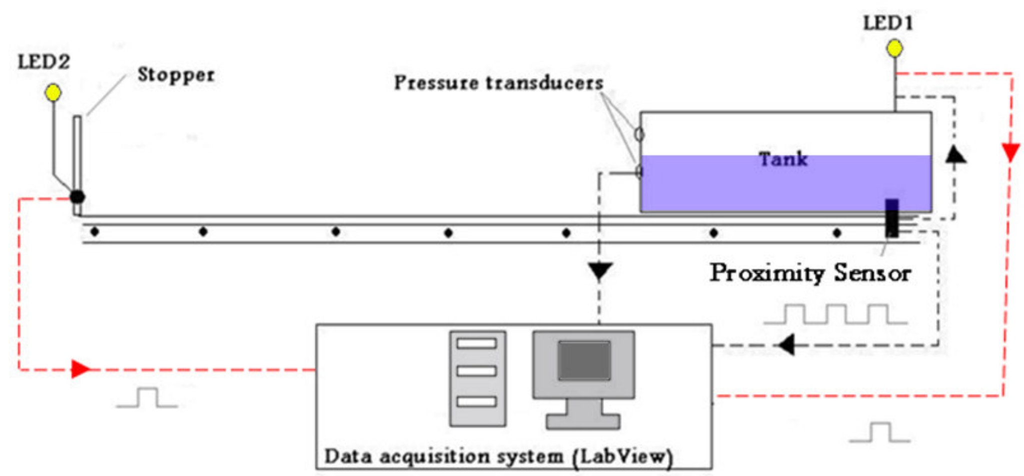


FIGURE 2: Schematic of the experimental setup

2.2 Flow Visualization and Image Processing

This section treats a particular case of fluid-tank system for further flow visualization and image processing of other cases. The working fluid is water and the container is filled up to the height of 87.5 mm with colored water as shown in Figure 3. In order to study the dynamics of the air-water interface, a 2D visualization of flow inside the moving tank was accomplished. The fluid and air-water interface motion was examined using a video camera (Canon A520). The experimental setup was illuminated with normal light and the video camera was installed perpendicular to the direction of motion of the container in order to record the entire interfacial region and the colored water distribution during the sloshing period. The entire interfacial region of the tank was scanned during impact. Full-frame images of 92 x 35 pixels were acquired and transferred to a computer for processing. The calculated errors for water level are based on the uncertainty of measured heights from reconstructed images, which is on order of 8 %.

In order to quantitatively characterize the observed states before and after impact, an image processing method using MATLAB software was developed. The colored image (Fig. 4a) was filtered in order to remove colors and additional noises created from the reflected light. The image processing enhances the sharpness and the contrast of the image (Fig. 4b). The filtered grayscale image was used to identify the position of the air-water interface (Fig. 4c). Finally the reconstructed image is colored in blue and white to illustrate the position of the water front (Fig. 4d). A sequence of reconstructed images is presented in the result section and compared with numerical computations.

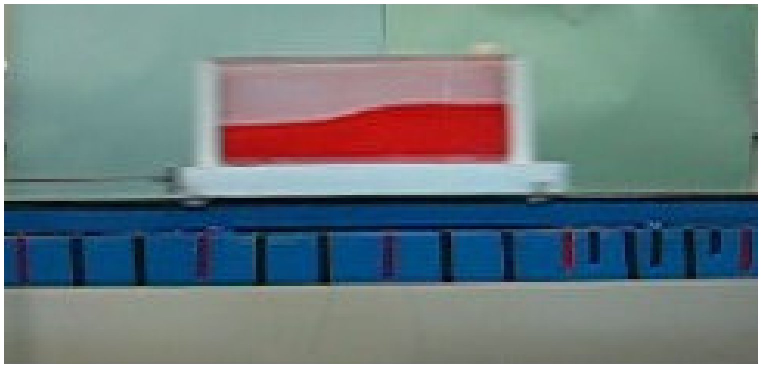
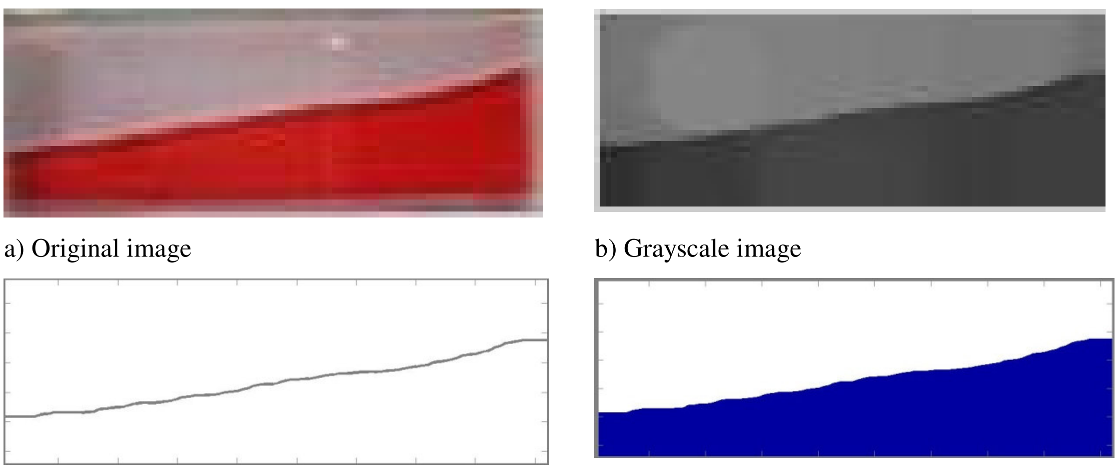


FIGURE 3: Photograph showing the fluid (in red) before impact.



a) Original image

b) Grayscale image

c) Front position

d) Reconstructed image

FIGURE 4: Image Processing

3. COMPUTATIONAL FLUID DYNAMICS

The dynamics of fluid at hand was modeled using FLUENT to mimic the experimental results obtained through visualization. The free surface fluid motion inside the tank can modeled in two parts i) rectilinear motion with zero initial speed and ii) post-impact fluid sloshing. During the entire motion the fluid has a free surface and is unsteady and incompressible. Since the aspect ratio of tank length to width is low, the fluid motion was modeled using a two-dimensional rectangular Cartesian mesh of 90x150. During the first part of the tank motion, the fluid behaves as a rigid body, but after impact the sloshing motion is violent. Nevertheless the flow regime is assumed to be laminar throughout. This is a reasonable assumption since in this type of flow the phenomena at play are largely inviscid, some localized turbulence effects may be generated during sloshing but at the sheared interface, which should not affect the fluid-wall interactions during sloshing and global fluid behavior. In addition it has the advantage of reducing the execution time and represents a good compromise between accuracy and CPU time.

The flow is considered incompressible, laminar and unsteady. The modeling of this type of motion which includes a moving frame denoted by x_{0i} attached to the tank relative to an inertial frame, was done using the body force approach described by Godderidge et al [13] and Kassinos and Prusa [15]. The effects of the linear acceleration of the tank on the fluid particles inside it were introduced into the governing equations as body forces:

$$g_{0i} = g_i - \ddot{r}_i \tag{1}$$

The relationship between the moving coordinate system and inertial frame is given by:

$$x_{oi} = x_i - r_i \tag{2}$$

The conservation equations of momentum and mass within the moving frame of reference are thus written as:

$$\frac{\partial u_{0n}}{\partial t} + u_{0i}\frac{\partial u_{0n}}{\partial x_{0i}} = -\frac{1}{\rho}\frac{\partial p}{\partial x_{0n}} + \frac{\mu}{\rho}\delta_{ij}\frac{\partial^2 u_{0n}}{\partial x_{0i}\partial x_{oj}} + g_{0n} \tag{3}$$

$$\frac{\partial u_{0i}}{\partial x_{0i}} = 0$$

Where p is the pressure, μ is the fluid viscosity, u_{0i} is the velocity vector, g_{0i} is the total net body force, g_i is the body force within the moving frame and $\ddot{r}_i$ is the inertia force resulting from the motion of the tank. The acceleration $\ddot{r}_i$ was calculated from the measured motion of the tank. The vector r_i denotes the position of the moving frame attached to the tank with respect to the inertial frame of reference. The solution will therefore consist of solving the equations relative to the inertial frame with modified body forces that result from the acceleration of the tank.

For this particular case two body forces were taken into account, the gravity in the vertical direction and a horizontal one due to the acceleration of the tank. The horizontal body-force was set to zero for the periods at and after impact. In addition, the absolute velocity of the fluid was modified at impact and set to:

$$u_i = u_{0i} + \dot{r}_i \tag{4}$$

Where u_{oi} denotes the fluid velocity field relative to the moving frame calculated before impact and $\dot{r}_i$ is the velocity of the tank at impact. This change is an instantaneous jump and leads to the strong nonlinear features of the slow behavior after impact. The velocity $\dot{r}_i$ was calculated from the measured motion of the tank. The integration time is for 4 seconds and the time step was chosen constant and equal to 0.01 second.

This type of flow involves two fluids air and water and a shared interface or free surface. The Volume of Fluid Method (VOF) of Hirt and Nichols [18] is used to track the variation of the interface by solving the continuity or advection equation for the volume fraction of the secondary phase given by:

$$\frac{\partial \gamma}{\partial t}+\frac{\partial \gamma u_{0i}}{\partial x_i}=0 \tag{5}$$

The above equation will not be solved for the primary phase; the primary phase fraction will be computed based on the following constraint:

$$\sum_{q=1}^{2}\gamma_q=1 \tag{6}$$

The finite-volume method was used in FLUENT to solve the momentum and continuity equations. The solver used is implicit and segregated with a time integration being first order implicit. The PISO (Pressure-Implicit-Splitting Operator) algorithm was used for pressure-velocity coupling and the Volume-of-Fluid (VOF) method where the equations are solved on a fixed mesh with the surface being located via the use of a void fraction. The free surface was taken to be the contour of the void fraction and was used as the multiphase model to track the shape of the air-water interface. The first order upwind discretization scheme was used for the momentum and volume fraction equations. At time t = 0, all the fluid was assumed to be at rest and the pressure inside the tank was assumed to be equal to the atmospheric pressure.

4. RESULTS AND DISCUSSION

Four experiments with different water levels and dead weights required to start tank motion were conducted. The aim of the experiment was to i) measure the distance travelled by the tank over time using a proximity sensor and ii) visualize the fluid motion before and after impact. Figure 5 shows a typical raw data for a 50% water level driven by a 2.5 kg dead weight. The output signals were obtained from the proximity sensor from which the tank displacement and elapsed time can be obtained. The responses of the proximity sensor and the two LEDs are shown in Figure 5 where the travel time is the time taken by the tank from the beginning of the motion till impact. In the flow visualization, it is the time between the lighting of the first LED (LED1) and the lighting of the second LED (LED2). The obtained displacements were used to develop expressions for the tank velocity and acceleration for different testing conditions. A summary of four tests is presented in Table 1 which includes regression expressions with accuracy of 99.99% for the velocity and acceleration needed for the computational fluid dynamics study using FLUENT.

The velocity and acceleration included in Table 1 were used in CFD as initial conditions to simulate the flow behavior in similar conditions to the experimental study. The motion before impact lasts for about 1.98 seconds. Comparison between the fluid motion obtained experimentally and numerically for a typical case (50% full and 4.5 kg dead weight) is shown in Figure 6. During that time the bulk of the fluid moves towards the right hand side wall to reach a maximum height at t=0.67 seconds. It then moves towards the opposite wall before coming back at t=1.14 seconds, as seen in Figure 6, with a traveling wave contrary to what Chwang and Wang [4] observed. It was observed that the free surface inclination changes direction twice. Subsequently, the fluid builds up on the right hand side wall to reach a maximum level just before impact as observed by Chwang and Wang [4].

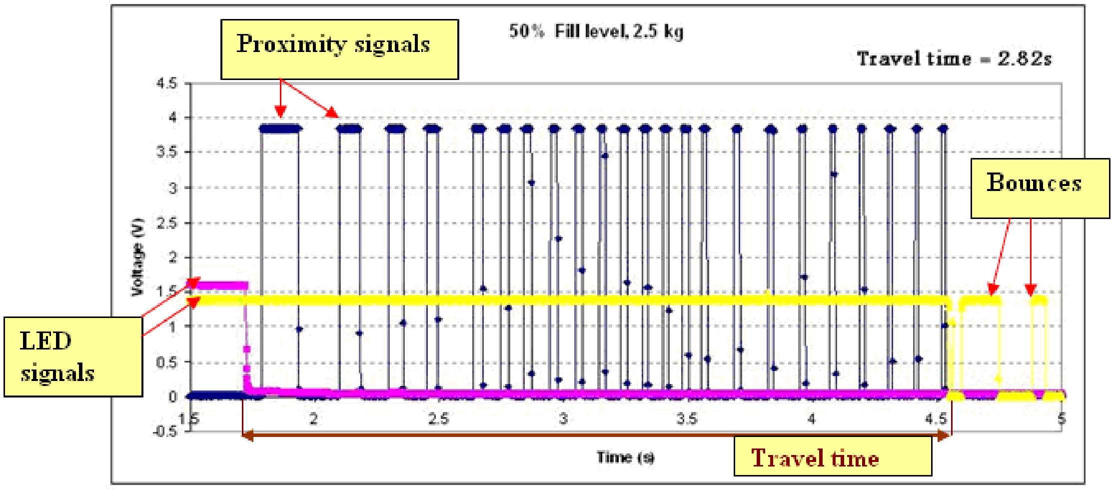


FIGURE 5: Pulse curves for 50% full water level and 2.5 kg dead weight.

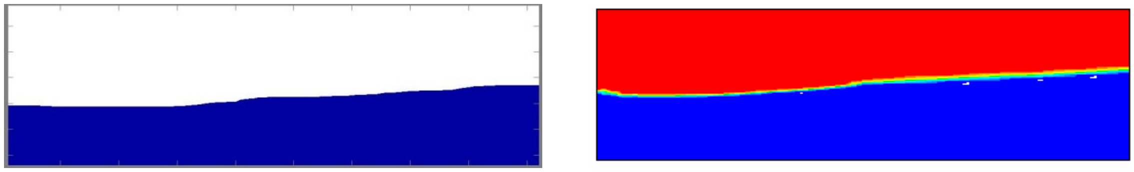
Characteristics/case	Case 1	Case 2	Case 3	Case 4
Water level (%)	50	50	75	75
Mass used (kg)	2.5	4.5	2.5	4.5
Displacement (m)	$x(t) = 0.218t^{1.85}$	$x(t) = 0.407t^{1.89}$	$x(t) = 0.180t^{1.84}$	$x(t) = 0.352t^{1.90}$
Velocity (m/s)	$v(t) = 0.0.403t^{0.85}$	$v(t) = 0.768t^{0.89}$	$v(t) = 0.33t^{0.84}$	$v(t) = 0.670t^{0.90}$
Acceleration (m/s^2)	$a(t) = 0.34t^{-0.15}$	$a(t) = 0.68t^{-0.11}$	$a(t) = 0.277t^{-0.16}$	$a(t) = 0.605t^{-0.10}$
Travel time t_r (sec)	2.82	1.98	3.14	2.12
Terminal velocity $v(t_r)$ (m/s)	0.969	1.41	0.864	1.32

TABLE 1: Cases considered in the experiment with their corresponding results

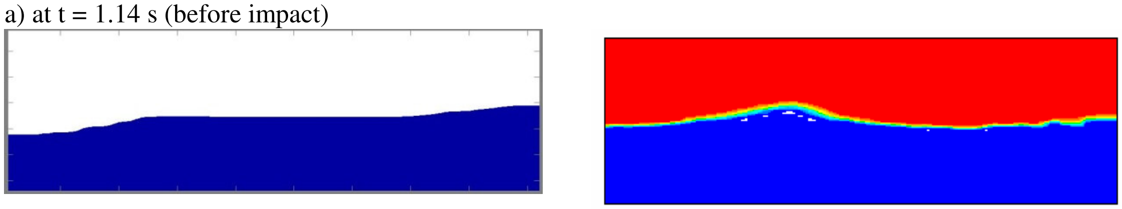
The experimental profiles compare well with the CFD results; both show the build up against the right, left then right – wall again of the fluid. At impact the fluid exhibits a violent sudden motion forward moving along the opposite vertical wall, around the top left corner and along the ceiling at t=2.17 seconds. At t=2.44 seconds the fluid accumulates in the left half of the container with large surface excursions of fluid and a displaced center of mass, before moving back to the right hand side with subsequent oscillations left-right. The CFD results show some discrepancies with the experimental photographs but the agreement is qualitatively encouraging knowing that the CFD model is two-dimensional and based on laminar flow. The CFD calculations show air entrapped within the bulk of the fluid. The experimental results do not show this entrapment and may due to slow speed of the camera used.

The fluid height on the right tank wall was traced experimentally and numerically during the pre- and post impact. Comparison between the simulation and experimental results showed a good agreement. The measured points are nearly similar to the simulated ones especially before impact. However, both results showed some discrepancies after impact due to minor bouncing of the tank (see Figure 7).

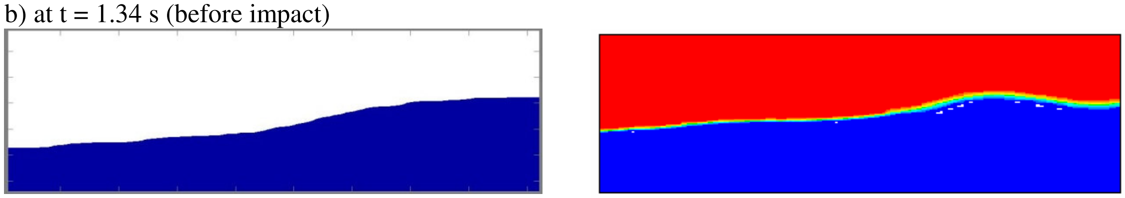
Experimental Results **Numerical Results**



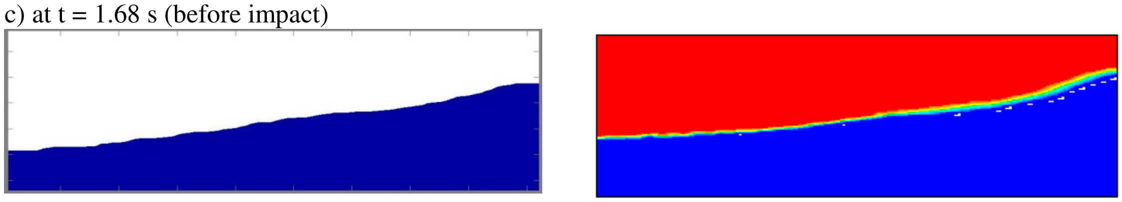
a) at t = 1.14 s (before impact)



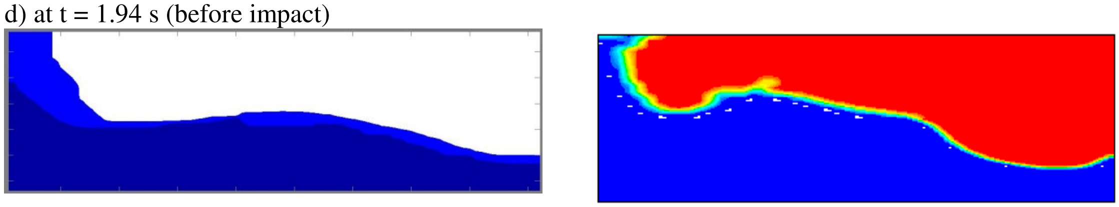
b) at t = 1.34 s (before impact)



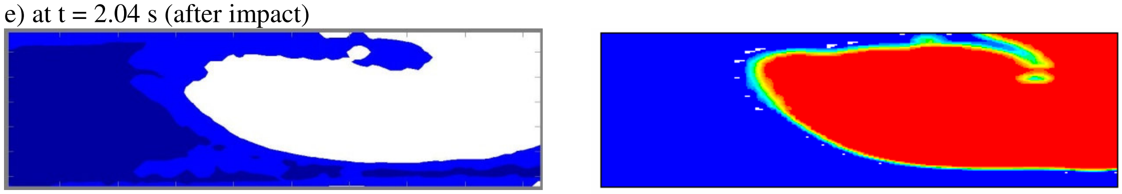
c) at t = 1.68 s (before impact)



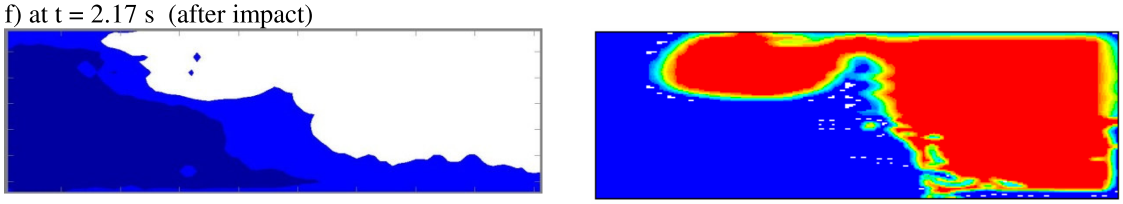
d) at t = 1.94 s (before impact)



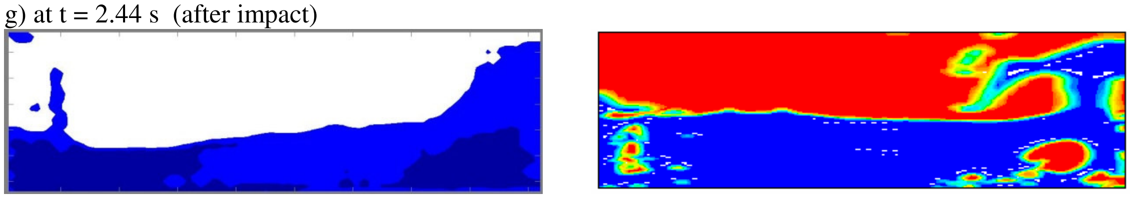
e) at t = 2.04 s (after impact)



f) at t = 2.17 s (after impact)



g) at t = 2.44 s (after impact)



h) at t = 2.97 s (after impact)

FIGURE 6: Comparison between experimental and numerical results for 50% fill level and 4.5 kg.

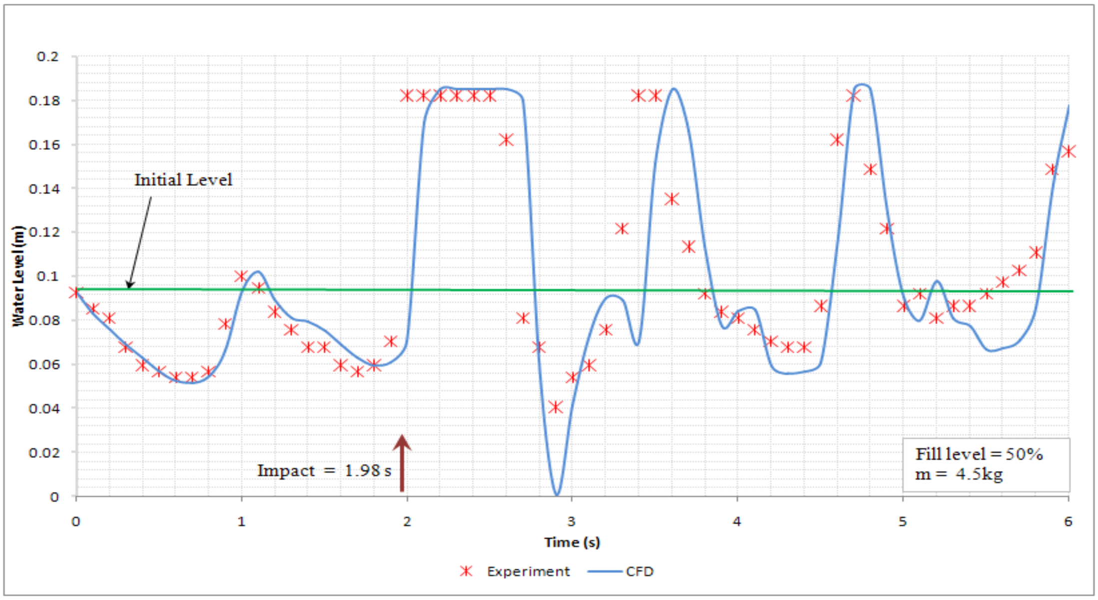


FIGURE 7: Right-side water level measurement for 50% fill level and 4.5 kg

5. CONCLUSIONS

The water sloshing phenomenon in a rectangular tank under sudden impact was investigated experimentally and numerically. Design of the testing rig and selection of proper sensors as well as data acquisition system was performed. Flow visualization of simulation and experimental results showed a good agreement. The water level for both simulated and experimental results compared well during motion and showed a minor discrepancy after impact which may be due to tank bouncing. Contrary to previous studies, both experimental and numerical results indicated the presence of a single traveling wave before the impact. Future study related to pressure measurements at the tank wall will be conducted for structural analysis purposes.

6. ACKNOWLEDGEMENT

Authors would like to acknowledge the support of our technicians Mr. Shrinivas Bojanampati and the late Mr. Allan Partridge for their help with the experimental set up.

7. REFERENCES

1. H. Olsen. “What is sloshing?” Seminar on Liquid Sloshing. Det Norske Veritas,1976.

2. R.A. Ibrahim, V.N. Pilipchuck, and T. Ikeda. “Recent Advances In Liquid Sloshing Dynamics” *Applied Mechanics Reviews*, Vol. 54, No. 2, pp. 133-199, 2001.

3. R.A. Ibrahim. “Liquid Sloshing Dynamics” *Cambridge University Press*, New York, 2005.

4. A.T. Chwang, and K.H. Wang. “Nonlinear Impulsive Force on an Accelerating Container” *J. of Fluids Eng.,* Vol. 106, pp. 233-240, 1984.

5. G. Popov, S. Sankar, T.S. Sankar, and G.H. Vatitas. "Liquid Sloshing In Rectangular Road Containers”, *Computers Fluids*, Vol. 21, No. 4, pp. 551-569, 1992.

6. G. Popov, S. Sankar, T.S. Sankar, and G.H. Vatitas. "Dynamics of liquid sloshing in horizontal cylindrical road containers," *Journal of Mechanical Engineering Science*, Vol. 207, 1993.

7. Z. Ye, and A.M. Birk. "Fluid Pressures in Partially Liquid-Filled Horizontal Cylindrical Vessels Undergoing Impact Acceleration," *Journal of Pressure Vessel Technology*, Vol. 116 No. 4, pp. 449-459, November 1994.

8. O. Faltinsen, O. Rognebakke, N. Alexander, and A.N. Timokha. "Resonant three dimensional nonlinear sloshing in a square-base basin, part 2. effect of higher modes", *Journal of Fluid Mechanics*, Vol. 523, pp. 199–218, 2005.

9. B.F Chen, and H.W. Chiang. "Complete 2D and Fully Nonlinear Analysis of ideal fluid in tanks", *Journal of Engineering Mechanics*, pp. 70-78, 1999.

10. O.M. Faltinsen, and O.F. Rognebakke. "Sloshing", In NAV2000: *International Conference on Ship and Ship Research*, Venice, 2000.

11. R.L. Bass, J.E.B. Bowles, R.W. Trundell, J. Navickas, J.C. Peck, N. Yoshimura, S. Endo, and B.F.M. Pots. "Modeling criteria for scaled LNG sloshing experiments", *Transactions of the American Society of Mechanical Engineers*, Vol. 107, pp. 272–280, 1985.

12. B. Godderidge, M. Tan, S. Turnock, and C. Earl. "A Verification and Validation Study of the Application of Computational Fluid Dynamics to the Modelling of lateral Sloshing" *Ship Science Report No 140, Fluid Structure Interaction Research Group*, University of Southampton, August 2006.

13. M.S. Celebi, and H. Akyildiz. "Nonlinear Modeling of Liquid Sloshing in a Moving Rectangular Tank", *Ocean Engineering*, Vol. 29, pp. 1527-1553, 2001.

14. A.C. Kassinos, and J. Prusa. "A Numerical Model for 3-D Viscous Sloshing in Moving Containers" *ASME- Publications FED*, Vol. 103, pp. 15-86, 1990.

15. A. Cariou, and G. Casella. "Liquid Sloshing in Ship Tanks: a Comparative Study of Numerical Simulation", *Marine Structures*, Vol. 12, pp. 183-198, 1999.

16. C.W. Hirt, and B.D. Nichols. "VOF method for the dynamics of free boundaries" *Journal of Computational Physics*, Vol. 39, pp. 201-225, 1981.