International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878 (Online), Volume-8 Issue-6, March 2020

Simulation of Partially Filled Liquid in A Moving Tank

Atul Bhattad, Payal Bhattad, Ankur Bhattad, Kshitij Maheshwari, Ayush Maheshwari

***Abstract*: Numerical simulations have been carried out on a rectangular tank filled partially with liquid using volume of fluid technique. The tank has been given to and fro motion in one direction. Numerical simulation has been carried for a two dimensional case having laminar and unsteady flow. The changes in free surface displacement and dynamic pressure at different times has been observed using ANSYS software. The study was conducted for two sec. It was observed that free surface displacement of fluid increases with velocity. Also, with an increase in volume of liquid the sloshing effect decreases.**

***Keywords*: Sloshing, Laminar flow, Surface displacement, VOF method, Pressure, Simulation.**

I. INTRODUCTION

Sloshing of liquid constitutes various problems of great real-world significance concerning the safety of transport systems, such as oil tank trucks on roadways, liquid cargo in ocean, liquid tank carriages on rails and in liquid rocket engines [1,2]. The partially filled tanks are disposed to violent splashing under certain circumstances. The massive fluid motion leads to intensive impact and pressure on tank walls causing structural damage. It also affects the stability of the carriage vehicles. When a partially filled tank is accelerated, the free surface present in the tank develops a sloshing phenomenon. During such kind of movement, it provides energy to endure the sloshing phenomenon causing severe motion in the free surface.

The first attempt towards the dynamic pressures analysis over the internal walls of the fuel carriers has been carried out by Abramson [3]. Application of linear theories were also seen to examine the liquid motion in cylindrical and spherical tanks with compartments [4]. Experimental tests have been carried out for studying the free surface oscillation and validated with mathematical models and simulations. Study of sloshing phenomenon is essential for turning or braking applications in moving tanks [5]. Sakai et al. [6] theoretically investigated the sloshing behavior of oil tanks. They studied the surface interaction between a roof and the fluid. Nielsen [7] also studied different aspects of sloshing in an excited container. Sloshing can be reduced by lengthening fundamental natural frequencies [8] and by using baffles [9].

Numerical studies have been performed to see the effect of sloshing [10, 11]. Most of the researchers either developed a code or used commercial CFD packages for numerical simulations. Kim et al. [12] studied sloshing in partially filled rectangular tanks using the iterative solution procedure. Kucukarslan et al. [13] applied finite element method to investigate the transient characteristics of dam-reservoir interaction for a sudden excitation and observed a decrease of pressure with time.

From, the survey it has been observed that sloshing study is of much importance for avoiding the liquid carrier accidents. The numerical study on tank fill depth and dynamic pressure acting on container walls is less and that to using VOF (volume of fluid) technique with ANSYS software is not available. Hence, author decided to investigate the sloshing behavior in a partially filled rectangular tank using VOF technique.

II. DESCRIPTION OF PROBLEM AND METHOD

The present study has been focused on computer modeling to simulate the response of liquid storage tanks using fluent software. The primary objective is to observe the variation of free surface displacement and pressure in a 2-D rectangular tank filled partially with water as a liquid at different time levels. The flow considered is unsteady and laminar. The liquid used for working is water. The tank is field partially with water and partially with air. Tank is open from the top; hence atmospheric pressure is acting on it.

The VOF model is appropriate for multiphase flows and can accurately capture the interface between the different fluids. Sway motion (To and Fro) is given to the tank having dimensions 600 mm x 600 mm x 600 mm. Here the analysis is done for the 2-D case. Simulations have been performed using ANSYS software with a 2-D quadrilateral mesh consisting of 10000 cells and 10201 nodes. Simulations have started with the fluid initially at rest. Time-step of 2 ms has been used. This sway motion (to and fro) has been implemented by a user-defined function (UDF) in unsteady case.

Manuscript received on February 10, 2020.
Revised Manuscript received on February 20, 2020.
Manuscript published on March 30, 2020.
* Correspondence Author
Atul Bhattad*, Mechanical engineering department, Koneru Lakshmaiah Education Foundation, Vaddesvaram, Andhara Pradesh-522502, India. Email: atul45007@gmail.com
Payal Bhattad, faculty of physics in Rabindra Bhartiya Public School, Vijayawada, Andhra Pradesh,
Ankur Bhattad , AQR Capital, Bengluru, India
Kshitij Maheshwari, Manager Department of, Development Benelli, Hydrabad, India.
Ayush Maheshwari, senior design engineer in Larsen and Toubro Construction, Chennai, India.

The segregated solver is used which is also known as a pressure-based solver. This is used because it is good for solving multiphase problems. Operating conditions have been provided for the operating density of air which is a primary fluid. Air is lighter than water. To set the operating density as the density of the lighter fluid is good because it excludes the hydrostatic pressure gradient within the lighter phase and hence augments the accuracy during rounding off the momentum. The reference pressure location is provided as (0, 0) which is a region where one of the fluid is always 100%. This is an essential condition for the smooth convergence of the solution. Patching of the domain is needed to show that how much portion contains water and how much portion contains air. Here portion having water is given volume fraction as 1 and which contains only air is assigned volume fraction as 0.

A. Solution Methodology

For modelling and analysis, Ansys software is used. Water and air are used as fluid medium whose properties are as follows:

For Water: Density = 998.2 kg/m^3, Viscosity = 1.003 e-03 kg/m-s, C_p = 4.182 kJ/kg-K, Thermal conductivity = 0.6 W/m-K

For Air: Density = 1.225 kg/m^3, Viscosity = 1.789 e-05 kg/m-s, C_p = 1.006 kJ/kg-K, Thermal conductivity = 0.0242 W/m-K

Atmospheric pressure (1.0132 bars) is acting at the top as pressure inlet.

B. Boundary Conditions

- Moving Wall: To describe the boundary conditions a grid is shown in Fig. 1. The left, bottom and right edges are given as a wall that is moving. It is moving in the x-direction with a velocity of 0.9 m/s.
- Pressure Inlet: The upper edge is given as pressure inlet. It is opened to the atmosphere and hence atmospheric pressure is acting on it.

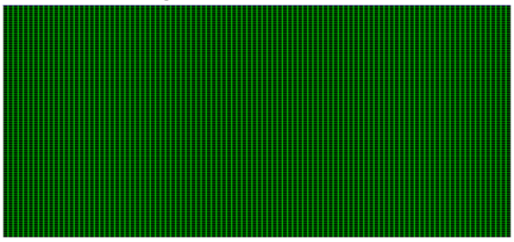

Fig. 1.Computational grid

The inside domain has been filled with fluid. Here the tank is partially filled with water and air. So, it is a case of multiphase. Air is acting as primary fluid and water as a secondary fluid. Here is a case where water is filled at the level of 200 mm. This is shown in Fig. 2.

Fig. 2.Fill level of 200m.m.

C. Discretization Scheme

The implicit scheme is used for unsteady formulation because it takes the recent values for further calculations which are obtained from previous steps. The upwind scheme is used for Navier-Stokes equations. It is used for the discretization of the momentum equation. The upwind scheme is designed to simulate numerically the propagation of information in a flow field along a path. If unwinding is carried out correctly, the solution of sharp discontinuity problems is possible. PISO is a non-iterative method used for transient cases. It relies on the accuracy gained by discretization. In transient cases, all time-dependent terms are retained in the momentum and continuity equations. It is used to calculate velocity and pressure fields at different time levels. It gives good results in comparison to other methods for transient flow.

The volume of fluid (VOF) model is defined as the ratio of final volume of secondary fluid to initial volume of secondary fluid. The scheme used under the VOF model is a geo-reconstruct scheme which is the most accurate scheme for interface tracking. The fundamental equations used for the simulation of sloshing of liquid are continuity and momentum equations. Here working fluid used is water which is incompressible.

For iterations, Time step size provided is 0.002 s. Number of Time steps provided is 1000. Adaptive Time (Variable) stepping method is used because our flow is unsteady.

III. COMPUTATIONAL RESULTS

The effect of sloshing is discussed for 200 mm fill level of liquid in a tank and contours are drawn at different time for dynamic pressure and free surface displacement. From time t=0 to t=1 sec. tank goes in forward direction and from t=1 to t=2 sec. it travels backward direction which completes one cycle of the tank. Variation of dynamic pressure with respect to time for unsteady case is discussed. The fill level of water in this case is 200 mm. As the tank moves in forward direction, due to state of rest, water moves in backward direction. At t=0 water is at rest and at t=1 water strikes to the back wall. Maximum pressure is shown by red color, minimum by blue color and rest of the colors show intermediate pressure. Figs. 3-7 shows variation of dynamic pressure with respect to time t=0-2 sec.

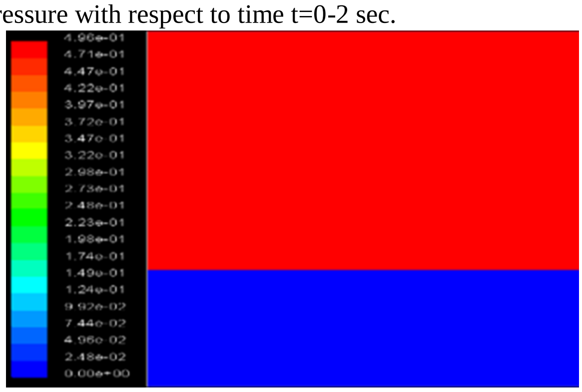

Fig. 3.Dynamic Pressure contour at t=0 sec.

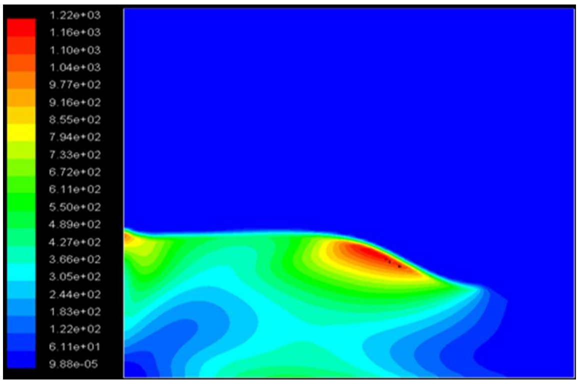

Fig. 4.Dynamic Pressure contour at t=0.5 sec.

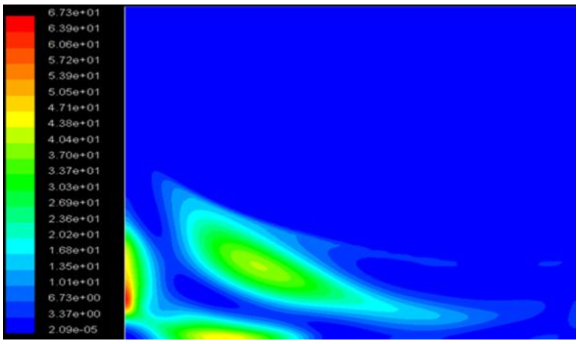

Fig. 5.Dynamic Pressure contour at t=1.0 sec.

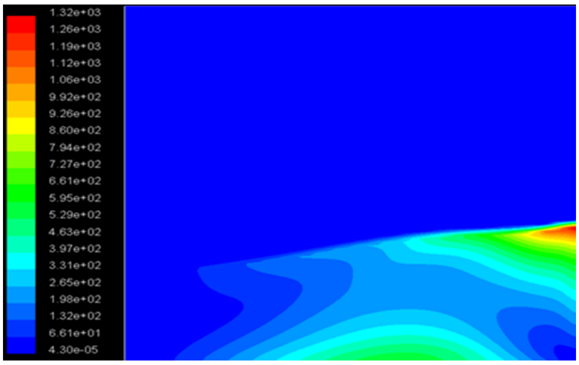

Fig. 6.Dynamic Pressure contour at t=1.5 sec.

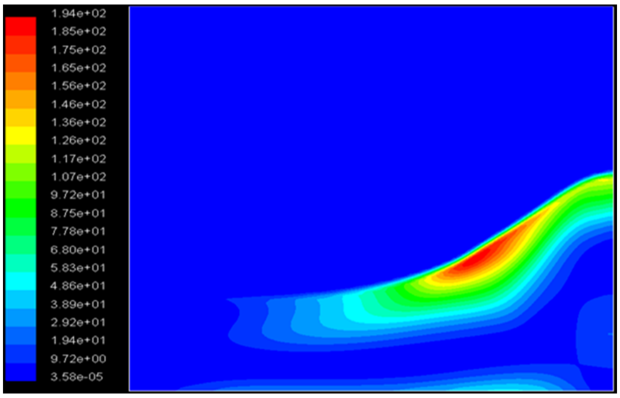

Fig. 7.Dynamic Pressure contour at t=2.0 sec.

When excitation is given to the static water, free surface of water starts moving. This movement of free surface increases with increase in velocity and time. The variation of free surface with time from 0 to 2 sec. for fill level of 200m.m.is shown in figs. 8-12.

Fig. 8.Free surface displacement contour at t=0 sec.

Fig. 9.Free surface displacement contour at t=0.5 sec.

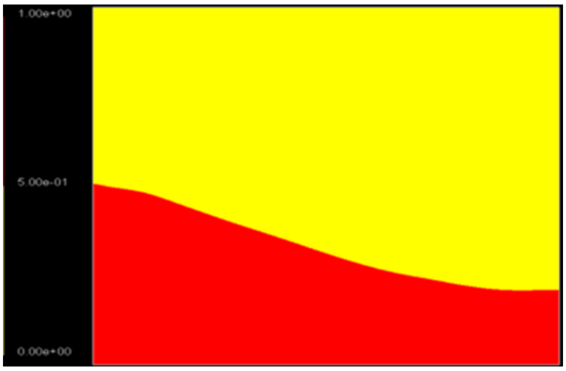

Fig. 10. Free surface displacement contour at t=1.0 sec.

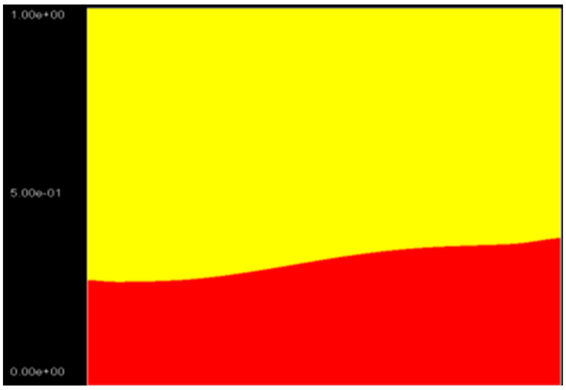

Fig. 11. Free surface displacement contour at t=1.5 sec.

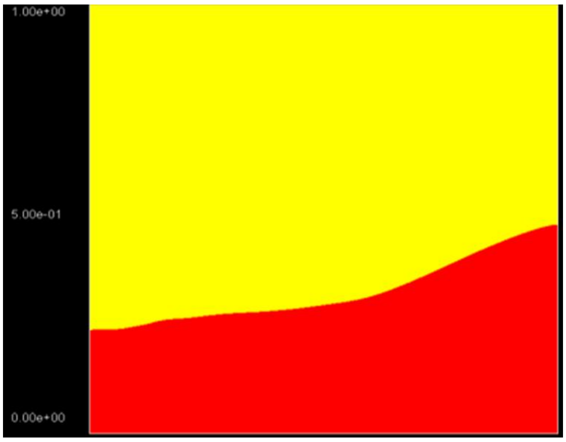

Fig. 12. Free surface displacement contour at t=2.0 sec.

IV. CONCLUSIONS

The flow inside a tank undergoing a periodic sway motion has been simulated using FLUENT's VOF model. The following conclusions have been made:

(a) Viscous and self-damping effects increase with increase in the mass of the liquid, which decreases the sloshing effects for a given excitation frequency.

(b) The pressure exerted on the walls varies in a similar nature as that of the applied excitation.

(c) Free surface displacement of fluid increases with an increase in the velocity.

REFERENCES

1. M.S. Celebi, and H. Akyildiz, "Nonlinear modeling of liquid sloshing in amoving rectangular tank". Ocean Engineering, 29, 2002, 1527–1553.
2. H. Akyildiz, and N.E. Unal, "Experimental investigation of pressure
3. distribution on a rectangular tank due to the liquid sloshing". Ocean Engineering, 32, 2005, 1503–1516.
4. H.N. Abramson, "The dynamic behavior of liquids in moving containers".
5. NASA SP-106, National Aeronautics and Space Administration. Washington, DC, 1966.
6. Armenio, And M.L. Rocca, "On The Analysis Of Sloshing Of Water In Rectangular Containers: Numerical Study And Experimental Validation". Ocean Engineering, 23 (8), 1996, 705–739.
7. G. Popov, S. Sankar, T.S. Sankar, and G.H. Vatistar, "Liquid sloshing in rectangular road container". Journal of Computer and Fluids, 21, 1992, 551–569.
8. F. Sakai, M., Nishimura, and H., Ogawa, "Sloshing behavior of floating roof oil storage tanks". Journal of Composites and Structures, 19, 1984, 183–192.
9. B.N. Nielsen, "Numerical prediction of green water loads on ships". Ph.D. Thesis, Department of Mechanical Engineering, Technical University of Denmark, 2003.
10. M.G. Koller, and P.K. Malhotra, "Seismic evaluation of unanchored cylindrical tanks". In 13th World Conference of Earthquake Engineering. Vanconver, Canada, 2004, 2534.
11. V.A. Buzhinskii, "Vortex damping of sloshing in tanks with baffles". Journal of Applied Mathematics and Mechanics, 62, 1998, 217–224.
12. N.C. Pal, "Coupled slosh dynamics of liquid filled laminated composite containers". Ph.D. Thesis, Department of Aerospace Engineering, IIT Kharagpur, India, 1999.
13. M. Eswaran, U.K. Saha, and D. Maity, "Effect of baffles on a partially filled cubic tank numerical simulation and experimental validation". Journal of Computers and Structures, 2008.
14. J.W. Kim, Y.S. Shin, and K.J. Bai, "A finite element computation for the sloshing motion in LNG tank". ISOPE 2002, Fukuoka, Japan.
15. S. Kucukarslan, C.B. Coskun, B. Taskin, "Transient analysis of dam–reservoir interaction including the reservoir bottom effects". Journal of Fluids and Structures, 20, 2005, 1073–1084.

AUTHORS PROFILE

Atul Bhattad is working as an Associate Professor in the Mechanical Engineering Department of Koneru Lakshmaiah Education Foundation, Vaddesavaram, Andhra Pradesh, India. He did his Ph.D. from Indian Institute of Technology Banras Hindu University Varanasi, Uttar Pradesh, India in 2019. He received his B.E. in Mechanical Engineering in 2007 from Pt. Ravishankar Shukla University, India and M.Tech. in 2009 in Fluid and Thermal from Indian Institute of Technology Guwahati, Assam, India. He is currently working on plate heat exchangers using hybrid nanofluids. His area of interest is Thermal. Fluid, and Heat exchangers.

Payal Bhattad is working as a faculty of physics in Rabindra Bhartiya Public School, Vijayawada, Andhra Pradesh, India. She did her Bachelor's in science from Nagpur University, India in 2013 and Master's in Physics from Gondwana University, India in 2015. Her area of interest is thermal physics and quantum physics.

Ankur Bhattad is working in AQR Capital, Bengluru, India as a vice-president. He did his Bachelor's in Technology from National Institute of Technology Silchar, Assam, India in 2009 from electrical engineering department. He worked as software engineer in many organizations like Infosys, Samsung and J P Morgan Chase. He has good hold in coding and optimization.

Kshitij Maheshwari is working as a Manager in Development department of Benelli, Hydrabad, India. He did his Bachelor's in Technology from National Institute of Technology Silchar, Assam, India in 2009 from Mechanical Engineering department. He worked in many Automobile firms like Subros limited, Hero and United Motors.

Ayush Maheshwari is working as a senior design engineer in Larsen and Toubro Construction, Chennai, India. He did his Bachelor's in Engineering from Bhilai Institute of Technology, India in Civil Engineernig Department and then did Master's in Technology from Indian Institute of Technology Guwahati, Assam, India in 2018.