

# Numerical Technique for Sloshing Velocity in a Partially Filled Moving Tank



Atul Bhattad, B. V. Dharmendra

**Abstract:** Numerical simulations have been carried out on a partially filled rectangular tank using volume of fluid method. The tank has been given to and fro motion. Numerical simulation has been carried for a two dimensional case having laminar flow. The effect of sloshing on velocity at different times has been observed using ANSYS software. The study was conducted for two sec. Variations in the velocity has been observed with the time period.

**Keywords:** Sloshing, Laminar flow, ANSYS, VOF, Velocity, Numerical.

## 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 [5-7]. Study of sloshing phenomenon is essential for turning or braking applications in moving tanks [8]. Sakai et al. [9] theoretically investigated the sloshing behavior of oil tanks. They studied the surface interaction between a roof and the fluid. Nielsen [10] also studied different aspects of sloshing in

an excited container. Sloshing can be reduced by lengthening fundamental natural frequencies [11] and by using baffles [12].

Numerical studies have been performed to see the effect of sloshing [13, 14]. Most of the researchers either developed a code or used commercial CFD packages for numerical simulations. Kim et al. [15] studied sloshing in partially filled rectangular tanks using the iterative solution procedure. Kucukarslan et al. [16] 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 effect of sloshing behavior on velocity vector 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 with 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. 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. 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.

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. E-mail: atul45007@gmail.com

**B. V. Dharmendra**, Mechanical engineering department, Koneru Lakshmaiah Education Foundation, Vaddesvaram, Andhara Pradesh-522502, India. E-mail: dharmbv@gmail.com

© The Authors. Published by Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP). This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>)

# Numerical Technique for Sloshing Velocity in a Partially Filled Moving Tank

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 given in FLUENT database.

## 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.

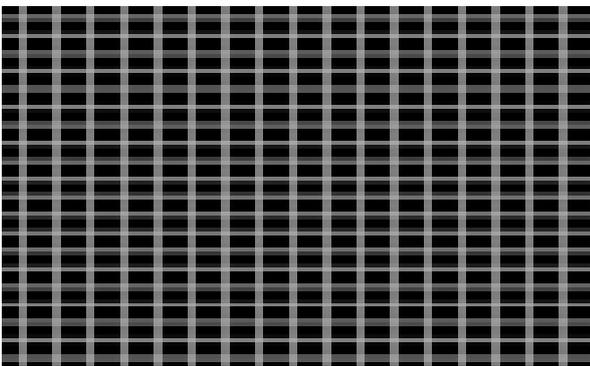


Fig. 1. Computational grid

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. PISO is a non-iterative method used for unsteady flow. 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.

## 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. Figs. 3-7 shows variation of velocity in x-direction with varying time (0-2 sec.). The red color shows the highest velocity zone and blue color denotes the lowest velocity zone.

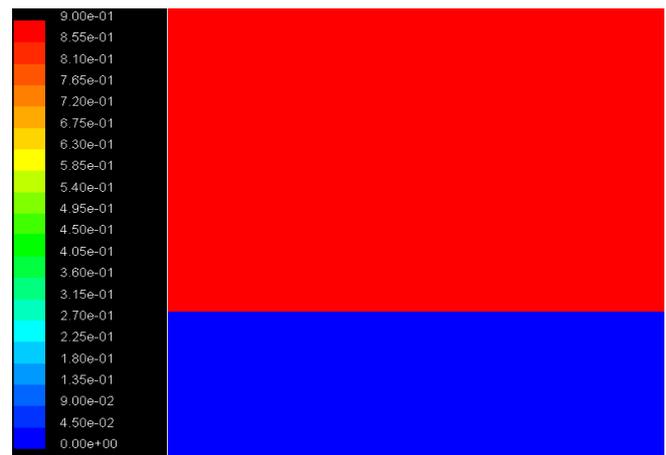


Fig. 3. Velocity contour at t = 0 sec.

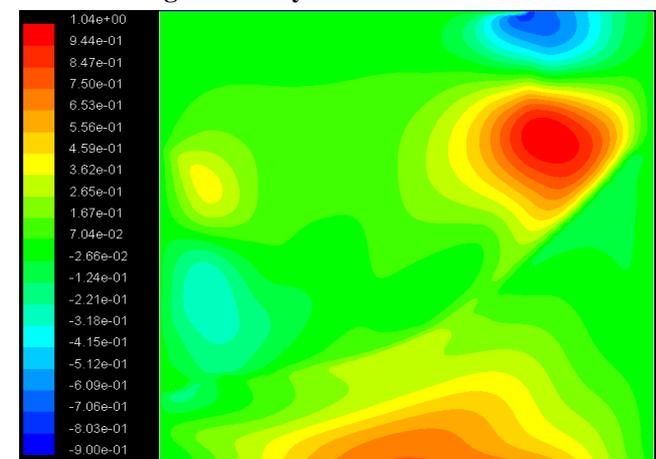


Fig. 4. Velocity contour at t = 0.5 sec.

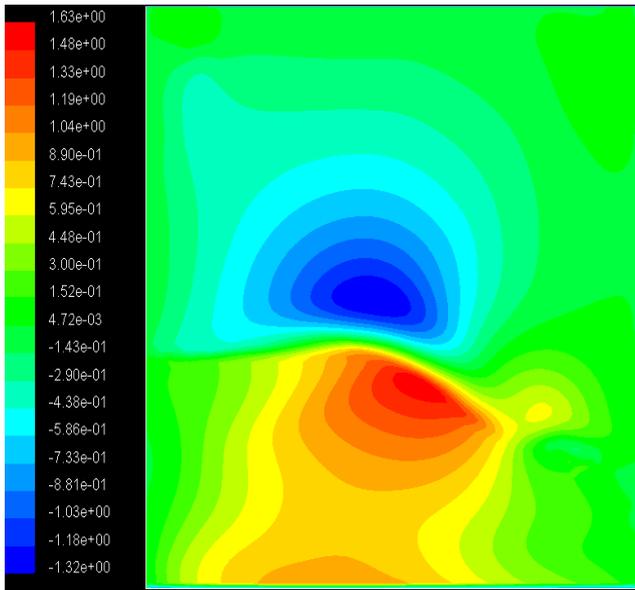


Fig. 5. Velocity contour at t = 1.0 sec.

vectors with time from 0 to 2 sec. for fill level of 200m.m.is shown in figs. 8-11.

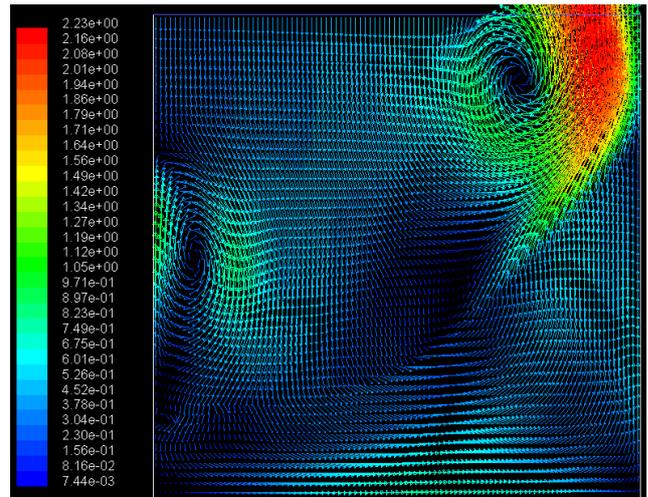


Fig. 8. Velocity vector at t = 0.5 sec.

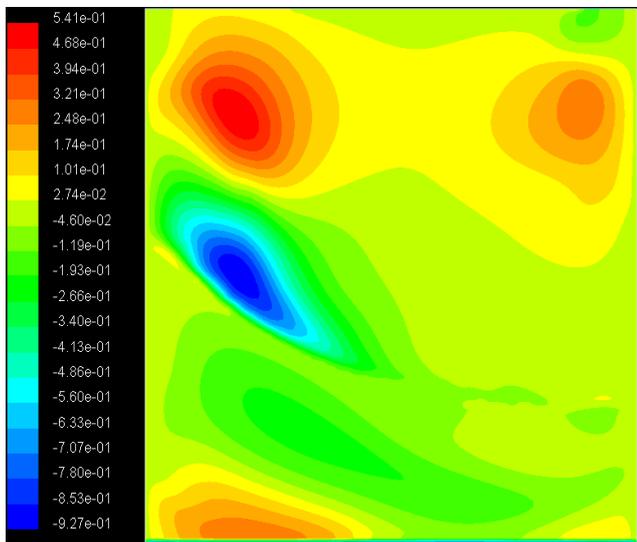


Fig. 6. Velocity contour at t = 1.5 sec.

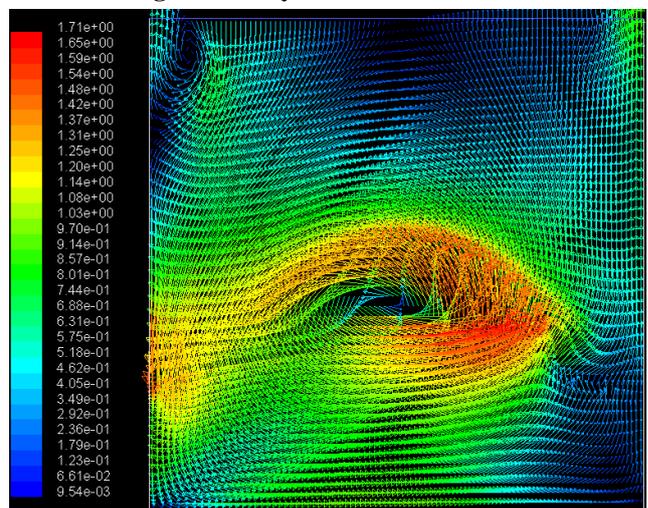


Fig. 9. Velocity vector at t = 1.0 sec.

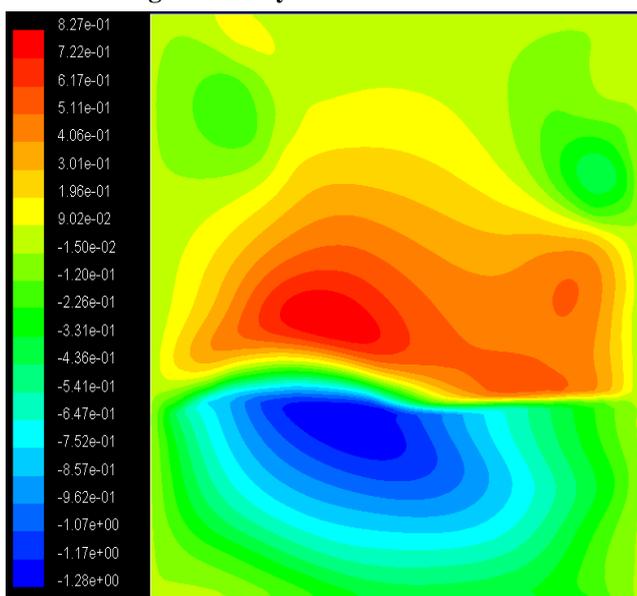


Fig. 7. Velocity contour at t = 2.0 sec.

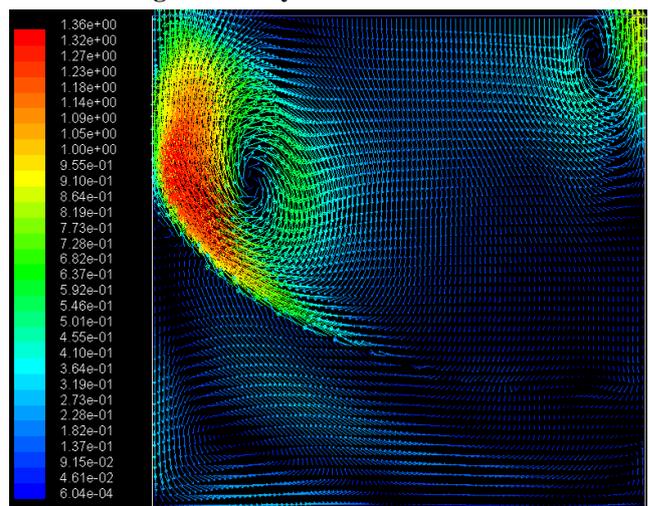


Fig. 10. Velocity vector at t = 1.5 sec.

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 velocity

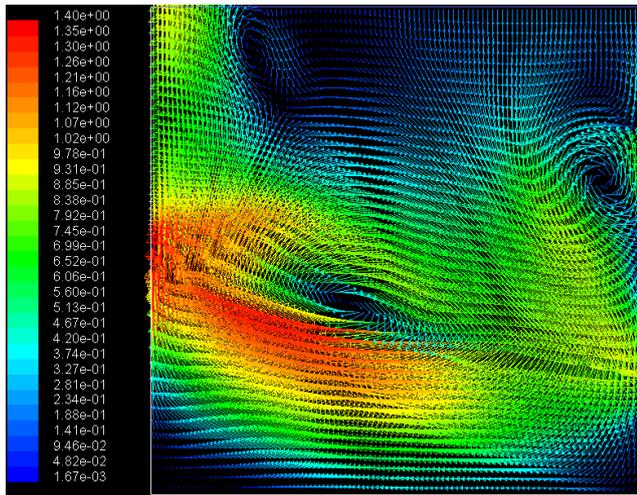


Fig. 11. Velocity vector at  $t = 2.0$  sec.

## IV. CONCLUSIONS

The flow inside a tank undergoing a periodic sway motion has been simulated using FLUENT's VOF model. The study was conducted for two sec. time period. Variations in the velocity has been observed with the time period.

## REFERENCES

1. M.S. Celebi, and H. Akyildiz, "Nonlinear modeling of liquid sloshing in a moving rectangular tank". *Ocean Engineering*, 29, 2002, 1527–1553.
2. H. Akyildiz, and N.E. Unal, "Experimental investigation of pressure distribution on a rectangular tank due to the liquid sloshing". *Ocean Engineering*, 32, 2005, 1503–1516.
3. H.N. Abramson, "The dynamic behavior of liquids in moving containers". NASA SP-106, National Aeronautics and Space Administration, Washington, DC, 1966.
4. V. 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.
5. N.E. Mikelis, and J.M.J. Journee, "Experimental and numerical simulations of sloshing behavior in liquid cargo tanks and its effect on ship motions". National Conference on Numerical Methods for Transient and Coupled Problems, Venice, Italy, 1984.
6. Y. Kim, "Numerical simulation of sloshing flow with impact load". *Applied Ocean Research*, 23, 2001, 53–62.
7. H. Akyildiz, and N.E. Unal, "Sloshing in a three dimensional rectangular tank: numerical simulation and experimental validation". *Ocean Engineering*, 33, 2006, 2135–2149.
8. 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.
9. F. Sakai, M., Nishimura, and H., Ogawa, "Sloshing behavior of floating roof oil storage tanks". *Journal of Composites and Structures*, 19, 1984, 183–192.
10. B.N. Nielsen, "Numerical prediction of green water loads on ships". Ph.D. Thesis, Department of Mechanical Engineering, Technical University of Denmark, 2003.
11. M.G. Koller, and P.K. Malhotra, "Seismic evaluation of unanchored cylindrical tanks". In 13th World Conference of Earthquake Engineering. Vancouver, Canada, 2004, 2534.
12. V.A. Buzhinskii, "Vortex damping of sloshing in tanks with baffles". *Journal of Applied Mathematics and Mechanics*, 62, 1998, 217–224.
13. N.C. Pal, "Coupled slosh dynamics of liquid filled laminated composite containers". Ph.D. Thesis, Department of Aerospace Engineering, IIT Kharagpur, India, 1999.
14. 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.
15. 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.

16. 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.



**B.V. Dharmendra** is working as a professor in the mechanical engineering department of Koneru Lakshmaiah Education Foundation, Vaddesavaram, Andhra Pradesh, India. He has teaching experience of 32 years. He worked as an incharge principal in Government college of Engineering Raichur, Karnataka and worked as a HOD in IP department in University BDT college of engineering in Karnataka. He is M.E. in machine tools from PSG tech Coimbatore and Ph.D. in mechanical from IIT Kanpur.