

24th DAAAM International Symposium on Intelligent Manufacturing and Automation, 2013

CFD Analysis of a Kerosene Fuel Tank to Reduce Liquid Sloshing

Vaibhav Singal^{a*}, Jash Bajaj^b, Nimish Awalgaonkar^a, Sarthak Tibdewal^c

^a Energy Division, School Of Mechanical and Building Sciences, VIT University, Vellore - 632014, Tamil Nadu, India

^b Mechanical Department, BITS Pilani, Pilani-333031, India

^c Chemical Division, School of Mechanical and Building Sciences, VIT University, Vellore-632014, Tamil Nadu, India

Abstract

Sloshing is a phenomenon of fluid- structure interaction. It occurs due to agitation of fluid in fractionally filled container. A case study of a fuel tank partially filled with Kerosene was simulated in this paper using Volume of Fluid (VOF) multiphase model. Simulations compared the amplitude of sloshing in tank with and without the use of baffles. CFD analysis was carried out using a commercial finite volume package ANSYS FLUENT 12.0. The tank was set into motion by giving an acceleration of 9.81 m/s^2 in the +X direction for 1.5 seconds. The fuel was also under the action of gravity in $-Z$ direction. It was found from the CFD transient simulations of the Kerosene liquid interface that the sloshing in the fuel tank was significantly reduced with the introduction of baffles in the fuel tank. The damping effect of the baffles was clearly visible in the obtained results. Further research can be undertaken in improving the optimum configuration and dimensions of the baffles in order to further reduce the sloshing in the fuel tank.

© 2014 The Authors. Published by Elsevier Ltd. Open access under [CC BY-NC-ND license](https://creativecommons.org/licenses/by-nc-nd/4.0/).

Selection and peer-review under responsibility of DAAAM International Vienna

Keywords: baffles; CFD; fuel tank ; Kerosene; Sloshing

1. Introduction

Sloshing occurs in all vehicles that undergo accelerated motion [1]. The study of this phenomenon of fluid-structure interaction is of great practical significance for the automotive as well as aerospace industry [2]. Slosh refers to the movement of liquid inside another object. Violent Sloshing occurs in partially filled tanks of rockets,

*Corresponding author. Tel.: +91-7200309978.

E-mail address: vaibhavsingal@hotmail.com

satellites, submarines, trucks, etc. Some fuel containers are so huge and contain fuel in so large amount that the large liquid movement generates acute hydrodynamic stresses that causes instabilities and breakdown of the structure and may even affect the stability of the vehicle carrying the liquid resulting in unwanted hazards. Based on the shape of the container and type of disturbance, the liquid experiences different types of motions such as planar, non-planar, symmetric, asymmetric, rotational, quasi-periodic, chaotic etc [3]. So, it is very important with regard to safety and environment to study the behavior of fluids in partially filled containers. Improper designs of fuel tanks often accounts for large scale sloshing. Designs of fuel tank are often improved by addition of anti-slosh baffles that prevent pick-up pipe intake from the abrupt fuel starvation when fuel is not in proximity of the intake due to sloshing [4].

The phenomenon of Sloshing waves have been studied for past several decades. Many of these studies have focussed on the linear and theoretical solutions of Sloshing. However, these theoretical studies have been lacking in explaining the topics such as breaking in the case of violent liquids and wave overturning wherein the liquid viscosity and flow turbulence could not be ignored. In such cases, numerical simulations and analysis has helped in addressing the aforementioned problems. In recent times, the Computational Fluid Dynamics (CFD) analysis is playing a vital role in analysing the different design models and helps in saving time and cost by eradicating the need for construction of several prototypes in the design and testing phase. The CFD modelling approaches are also able to overcome flow limitations experienced in theoretical studies. Also, the complicated boundary problems often encountered in reality can be handled through CFD analysis [5]. A more general modelling technique is the solution of the Navier–Stokes equations using CFD [6]. Several numerical techniques exist for analysis of liquid storage tanks, of which the famous ones are explicit finite element, implicit Lagrangian – Eulerian, hybrid finite element and volume of fluid (VOF) [7]. The VOF method is a very robust and flexible method than other methods for simulating complicated free surface. It is able to capture complex surface geometries having overturning waves and splashing. It is a challenging task to study sloshing due to the presence of complex flow interactions with the container. In this regards, Pal et al. (2003) numerically investigated sloshing in cylindrical tanks with and without baffles [8]. Also, Kim (2001) [9] and Kim et al. (2004) [10] solved the Navier-Stokes equation with free boundary with a comparison of impact pressure in tanks with and without baffles. Their analyses were based on a finite difference method.

2. Computational study

The fuel tank was configured with (fig.1) and without baffles (fig.2) and comparison was made between the two using CFD analysis tools. The dimensions of the fuel tank are given in the Table.1. The dimensions of the baffles are shown in Fig.3. Both the configurations (with and without baffles) were set into same accelerated translation motion of 9.81 m/s^2 in the direction of +X axis. As the tank's motion actuates and bears acceleration for 1.5 seconds in the direction of +X axis, Kerosene experiences force in opposite direction of same magnitude according to the Newton's third law of motion.

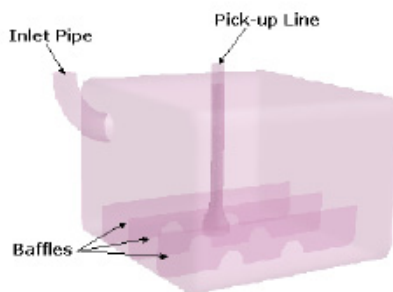


Fig. 1. Tank with baffles.

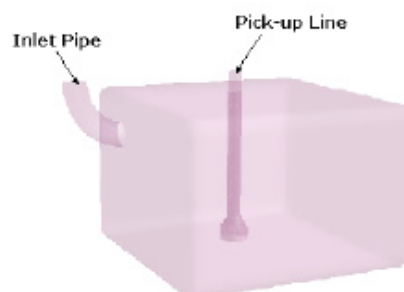


Fig. 2. Tank without baffles.

Table 1. Dimensions of the fuel tank (in meters).

Name of parameter	Value (in meters)
Length of tank (in +X axis)	50
Breadth of tank (in +Y axis)	50
Height of tank (in +Z axis)	30
Inlet pipe diameter	4.8
Pick-up pipe diameter	3
Pick-up pipe length	34

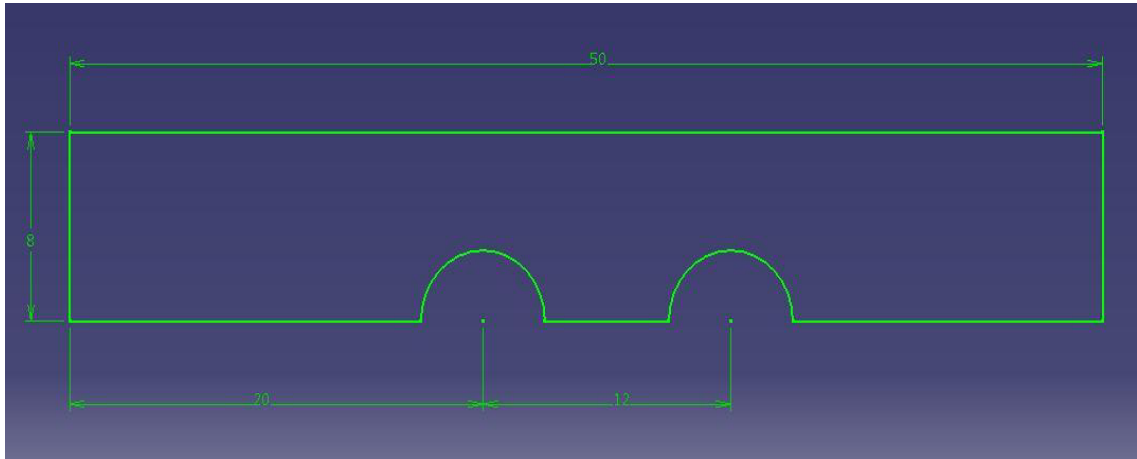


Fig. 3. Dimensions of the baffles used in the Kerosene fuel tank (in meters).

ANSYS ICEM-CFD™ 12.0 software was used to mesh the computational domain of the fuel tank under consideration. The volume of fuel tank as the control volume was the domain taken in consideration for the subsequent analysis. The mesh consists of 90772 numbers of cells in flow domain. As seen from the fig.4 the inlet pipe supplies fuel inside the fuel tank whereas, pick up pipe supplies fuel from fuel tank to the engine of the vehicle. Also, as seen in the fig.3, triangular unstructured meshing was done for the given control volume. ANSYS FLUENT™ 12.0 commercial package was used to perform the CFD simulations.

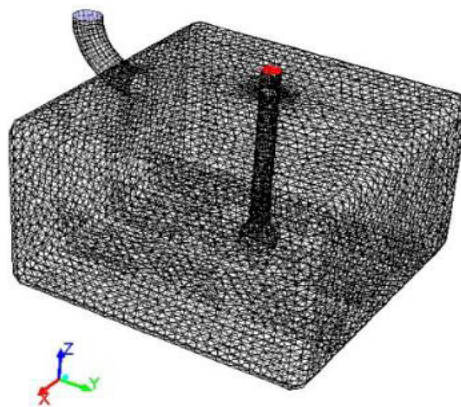


Fig. 4. Computational Domain of the fuel tank with baffles.

3. Computational modelling

A. Computational Model

Volume of Fluid (VOF) multiphase model in ANSYS FLUENT™ 12.0 was used to predict the motion of the Kerosene fuel inside the tank when the tank is under accelerated motion. The VOF model was designed to capture the position of interface between two or more immiscible fluids (air and Kerosene). The Kerosene was assumed to occupy about 25% of the total volume of fuel tank. The properties of the Kerosene fuel are given in Table.2. Volume fraction of each of the fluids in each computational cell is tracked throughout the domain by sharing a single set of momentum equations between the fluids. The model relies on the fact that the fluids are not interpenetrating. In each control volume, the volume fractions of all phases add to unity.

Table 2. Properties of Kerosene.

Property (units)	value
Density (kg/m ³)	780
Specific heat (j/kg-k)	2090
Thermal conductivity (w/m-k)	0.149
Viscosity (kg/m-s)	0.0024

For a pth fluid's volume fraction in the cell denoted as vf_p , following three conditions exist:

- $vf_p=0$: The cell is empty (of the pth fluid).
- $vf_p=1$: The cell is full (of the pth fluid).
- $0 < vf_p < 1$: The cell contains the interface between the pth fluid and one or more fluids.

B. Governing Equations

Therefore, the relevant properties and variables are assigned to every control volume as per the local value of vf_p within the domain.

Volume Fraction Equation:

$$\frac{1}{\rho_q} \left[\frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q) \right] = S_{\alpha_q} + \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp})$$

Where, \dot{m}_{qp} is mass transfer from phase q to phase p and \dot{m}_{pq} is mass transfer from phase p to phase q.

The volume fraction equation was solved using explicit time discretization. In the explicit approach, standard finite interpolation schemes were applied to the volume fraction values computed at the previous time step.

$$\frac{\alpha_q^{n+1} \rho_q^{n+1} - \alpha_q^n \rho_q^n}{\Delta t} V + \sum_f (\rho_q U_f^n \alpha_q^n, f) = \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) + S_{\alpha_q}] V$$

Where,

n+1 = index for new (current) time step

n = index for previous time step

face value of the qth volume fraction, computed from the first- or second- order upwind, QUICK, modified HRIC, compressive, or CICSAM scheme.

V = volume of cell

U_f = volume flux through the face, based on normal velocity.

Momentum Equation:

$$\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = - \nabla p + \nabla \cdot [\mu (\nabla \vec{v} + \nabla \vec{v}^T)] + \rho \vec{g} + \vec{F}$$

Energy Equation:

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot [(k_{eff} \nabla T)] + S_h$$

C. Boundary conditions and Solution Method

The boundaries in the computational domain were named baf1, baf2, baf3 for the baffles and inlet for the inlet pipe and pick-out for fuel exit from the tank through pick-up pipe. The boundary type for inlet and pick-out are wall and pressure outlet respectively. The only difference in the boundary types for the two configurations of the tank is that the boundary type for baffle surfaces baf1, baf2 and baf3 is wall for the tank with baffles and interior for the tank without baffles. Pressure-based solver is used in transient condition. Fractional step algorithm is used as Pressure-Velocity Coupling Method as the flow is time dependant. Non-Iterative Time Advancement is enabled to make computation less CPU intensive. Green-Gauss Node Based spatial discretization scheme was used.

4. Results and discussion

From the CFD simulation it was observed that the pick-up pipe was not completely submerged in the liquid fuel present in the fuel tank after 0.45 seconds (Fig. 4 and Fig. 5) and 1.25 seconds (Fig. 6 and Fig. 7) of acceleration. This confirmed that the pick-up pipe in the tank without baffles was not submerged in fuel while the baffles in the tank created recirculation zones at the lower z plane where the intake of the pick-up pipe is located. Hence, baffles in the tank prevented Kerosene from escaping the lower z plane, causing the pick-up pipe intake to be submerged in Kerosene all the times. On the other hand, the liquid interface at time 0.45 seconds and 1.25 seconds for the tank without baffles showed the existence of large scale sloshing, as a result of which the pick-up pipe is not submerged in the Kerosene, preventing the continuous intake of Kerosene.

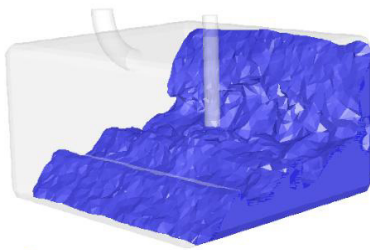


Fig.5 Liquid Interface at t = 0.45 seconds
(for tank with Baffles).

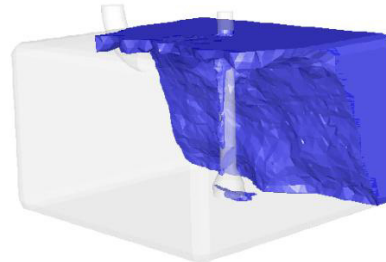


Fig. 6 Liquid Interface at t = 0.45 seconds
(for Tank without Baffles).

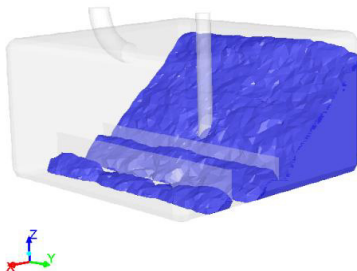


Fig.7. Liquid Interface at t = 1.25 seconds
(for tank with Baffles).

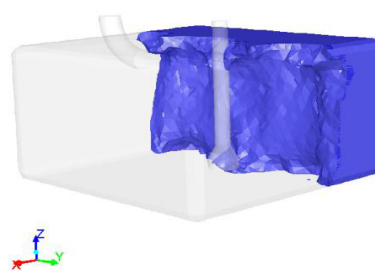


Fig.8 Liquid Interface at t = 1.25 seconds
(for tank without Baffles).

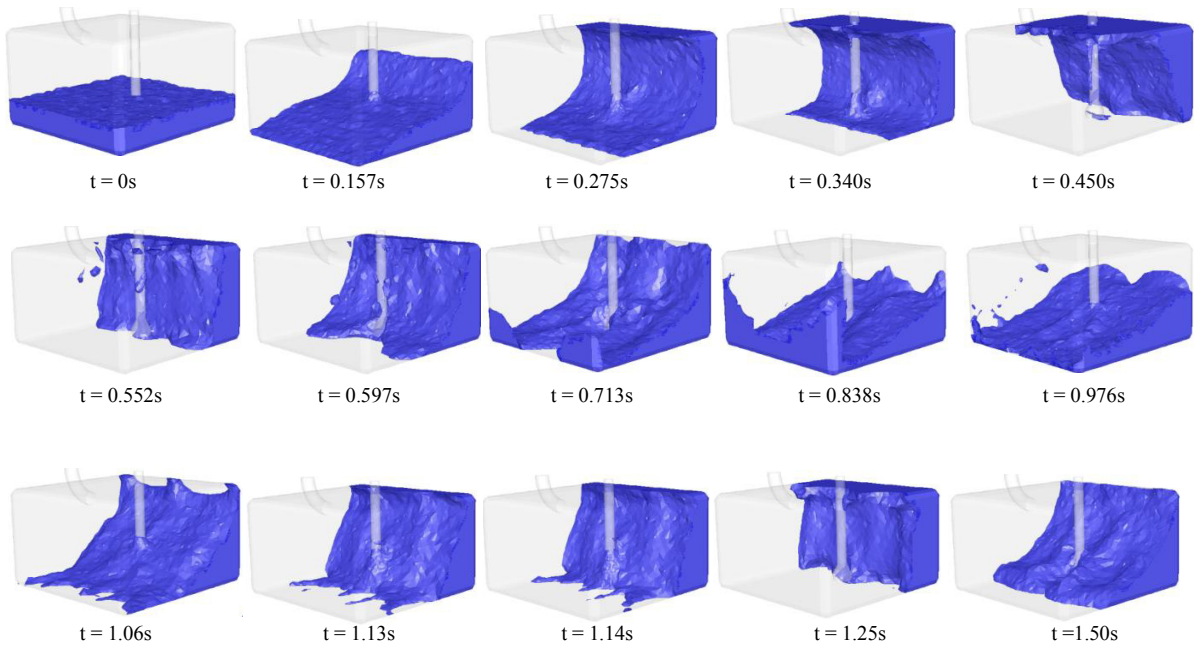


Fig. 9. Liquid interface at various time steps for Tank without baffles.

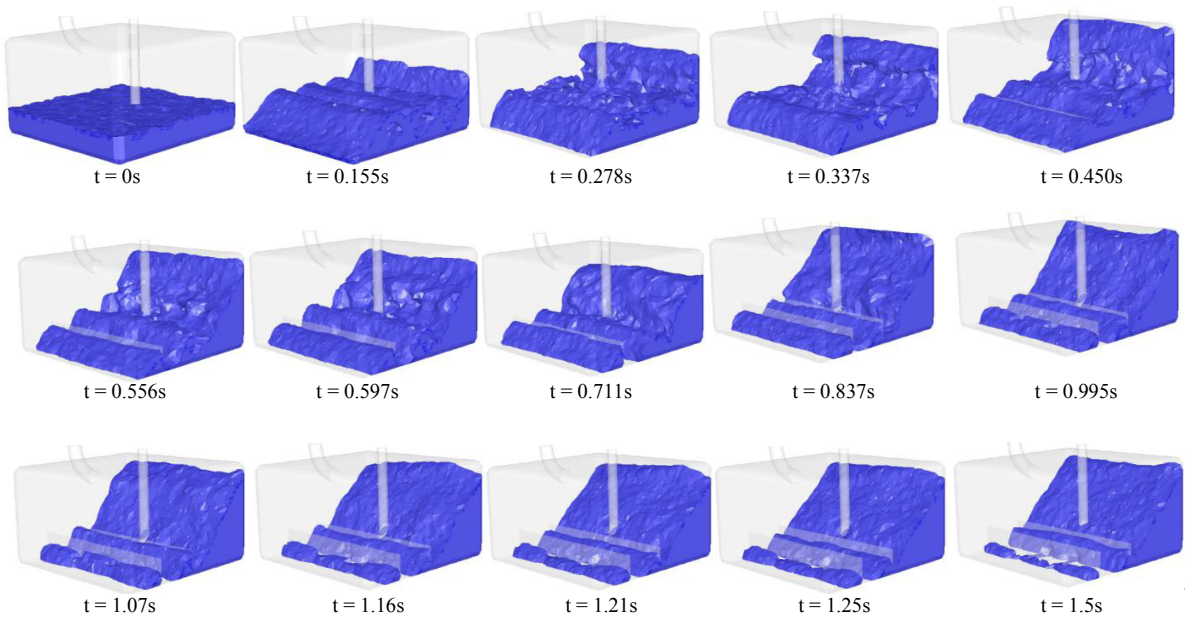


Fig. 10. Liquid interface at various time steps for Tank with baffles

5. Conclusion

The Kerosene liquid interface for the tank configurations with and without baffles was examined. It was evident from the CFD transient simulations of liquid interface carried out at various time steps for both configurations of tank that scale of sloshing reduces significantly by the use of baffles. The damping effect of baffles is clearly visible in the second case of analysis. The introduction of baffles in the fuel tank would therefore help in maintain the continuous supply of fuel from the engine to the fuel tank. In the further research, different other configurations of baffles can be analysed to optimize the design of the tank to further reduce the sloshing phenomenon. The dimensions of the baffles can also be optimized for further reduction in the sloshing. The CFD simulations so undertaken can therefore help in the designing of the future fuel tanks for the automotive and aerospace industries.

References

- [1] Olsen H. What is sloshing? In: Seminar on liquid sloshing. Det Norske Veritas;1976.
- [2] M. Eswaran, U.K. Saha and D. Maity, Effect of baffles on a partially filled cubic tank: Numerical simulation and experimental validation *Computers and Structures* 87, 198–205, 2009.
- [3] P.K. Panigrahy, U.K. Saha, D. Maity. Experimental studies on sloshing behavior due to horizontal movement of liquids in baffled tanks. *Ocean Engineering* 36, 213-222, 2009.
- [4] Seyyed M. Hasheminejad and M.M. Mohammadi, Effect of anti-slosh baffles on free liquid oscillations in partially filled horizontal circular tanks, *Ocean Engineering* 38, 49–62, 2011.
- [5] Ling Hou, Fangcheng Li, Chunliang Wu, A Numerical Study of Liquid Sloshing in Two-dimensional Tank under External Excitations, *Journal of Marine Science Applications* 11, 305-310, 2012.
- [6] Bernhard Godderidge, Stephen Turnock, Mingyi Tan and Chris Earl, An investigation of multiphase CFD modelling of a lateral sloshing tank, *Computers & Fluids* 38, 183–193, 2009.
- [7] S. Nicolici and R.M. Bilegan, Fluid structure interaction modeling of liquid sloshing phenomena in flexible tanks, *Nuclear Engineering and Design* 258, 51– 56, 2013.
- [8] Pal, N.C., Bhattacharyya, S.K., Sinha and P.K, Non-linear coupled slosh dynamics of liquid-filled laminated composite containers: a two dimensional finite element approach, *J. Sound Vib.* 261 (1), 729–749, 2003.
- [9] Kim and Y., Numerical simulation of sloshing flows with impact load, *Appl. Ocean Res.* 23, 53–62, 2001.
- [10] Kim, Y., Shin, Y.-S., Lee and K.H., Numerical study on slosh-induced impact pressures on three-dimensional prismatic tanks, *Appl. Ocean Res.* 26 (5), 213–226, 2004.