

International Engineering Research Journal

Comparative Study of Sloshing Phenomenon in a Tank Using CFD

Vilas P. Ingle[†], Dattatraya Nalawade[‡] and Mahesh Jagadale[†]

[†]PG Student, Mechanical Engineering Department, GSMCOE Balewadi, Pune (M.S.) India

[‡]Assistant Professor, Mechanical Engineering Department, VIIT, Kondwa (BK), Pune (M.S.) India

[†]Assistant Professor, Mechanical Engineering Department, GSMCOE Balewadi, Pune (M.S.) India

Abstract

Sloshing is a phenomenon which has gained importance where liquid and air interaction happens within a closed volume. This is predominantly observed in the field of Automotive, Aeronautics, and Petrochemical storage to name a few. Fundamentally enclosed fluid in a defined volume is a relative motion with respect to time when there is acceleration or de-acceleration which will change the direction of fluid within the enclosed volume leading to damage the enclosure itself.

Here in this case the study revolves around different levels of fluid in the fuel tank, tank without baffle and tank with baffle in order to understand the Fluid Structure Interaction (FSI). During the course of our study it helps us to understand the design, position of baffle and opening on the baffle in co-relation with the different level of fluid in the tank. To carry out the study we have taken the help of ANSYS FLUENT 15.0.7 and User Defined Function (UDF) to understand different parameters and evolve a viable design.

Keywords: Sloshing; CFD; FSI; Baffle; UDF

1. Introduction

Liquid is one of the phases of a material which does not have a definite shape and geometry. Due to this arbitrary nature it takes the shape of the enclosure in which it is stored. The enclosed liquid is characterized by its own properties like density, viscosity and surface tension which defines the interactive relationship into Fluid Structure Interaction (FSI). Atmospheric operating conditions are taken into account while studying the sloshing phenomenon. Volume of Fluid (VOF) multiphase model in ANSYS FLUENT is used to simulate the sloshing phenomenon; though this approach gives a good result for sloshing phenomenon but it is unable to give the forces exerted on different parts of the fuel tank. So in order to capture those forces we developed a User Defined Function (UDF) which gives the magnitude of forces acting normally on each cell of the particular part of the fuel tank at each instant.

In a tank, fluid by itself is laminar in nature at stationary condition but when excited by an acceleration or de-acceleration of defined value, liquid gets displaced from its equilibrium position into a phenomenon called sloshing. Momentum and inertia which comes with the sloshing leads to a force on baffles and tank structure. These developed forces by sloshing are captured with the help of UDF. The importance of sloshing is not only limited in catering the turbulence but also in curbing Noise, Vibration, Harshness (NVH) and the stability of the system.

2. CFD slosh modeling

Initially laminar flow is assumed and the flow is considered to comprise of two fluids: air as a gas phase fluid and water as liquid phase fluid. In the model air is treated as primary phase and water as secondary phase. The liquid flow is governed by unsteady incompressible continuity and Navier-Stokes (N.S) momentum equation.

$$\nabla \cdot \vec{u} = 0 \quad (1)$$

$$\rho \left(\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla) \vec{u} \right) = -\nabla p + \mu \nabla^2 \vec{u} + \vec{f} \quad (2)$$

Where,

∇ is the gradient operator

$\nabla \cdot$ is the divergence operator

∇^2 is the Laplace operator

p is the pressure

ρ is the density of fluid

μ is the fluid viscosity

\vec{u} is the velocity vector and

\vec{f} is the body force vector representing the forces arising from accelerations due to gravity and external disturbances, such as vehicle maneuvers, such that

$$\vec{f} = \rho \vec{g} + \rho \vec{a} \quad (3)$$

Where, \vec{g} is the acceleration due to gravity and \vec{a} is the acceleration attributed to a vehicle maneuver. The air-water interface is tracked by solving the volume fraction equation:

$$\frac{\partial F}{\partial t} + \nabla \cdot (\vec{u}F) = 0 \quad (4)$$

Where, F is the volume fraction function. In the computation domain, the unity value of F is assigned for all grid cells which are completely occupied by the liquid, while zero value is assigned for all grid cells completely occupied by the gas. The grid cells partially occupied by liquid assume values of F ranging from zero to one, depending on the fractions of both fluids in the cells. The gas-liquid interface is thus determined by the cells with F values lying between zero and unity. Although the flow is treated as a two phase flow, only one set of the governing equations, Equations (1) and (2), coupled with the volume fraction equation (Equation (4)), are solved. The fluid properties, such as density and viscosity, for different phase fluids in the Navier-Stokes (N.S.) equations are determined by the evolution of volume fraction for a grid cell. For example, since the liquid phase is the interest of tracking, the liquid density in each cell can be determined by a general expression:

$$\begin{aligned} \phi &= F_2 \phi_2 + (1 - F_2)\phi_1, \\ (\phi &= \rho \text{ or } \mu) \end{aligned} \quad (5)$$

Where ϕ_1 and ϕ_2 indicate the densities or viscosities of air and water, respectively. F_2 is the value of volume fraction of the liquid phase for a cell. The above modeling was realized using the ANSYS FLUENT software package. The software uses the finite volume method to discretize the integral form of the governing equations, Equations (1) and (2). In the present modeling, the segregated solver was activated, so the governing equations could be solved sequentially. The flow equations are solved based on the pressure correction and pressure-velocity coupling technique, Pressure-Implicit with Splitting of Operators (PISO). This scheme requires dramatically less number of iterations to reach convergence, particularly for the transient flow. The second order upwind scheme was chosen for the momentum discretization. Considering the large body forces involved in the two phase flow, the body-force weighted scheme was adopted for the pressure interpolation. The implicit body force treatment was used to achieve the partial equilibrium of pressure gradient and body forces, and also to obtain a good convergence. The first order implicit scheme was chosen for the transient formulation with non iterative time advancement. No slip condition was selected for all the tank walls.

The geometric reconstruction (Geo-Reconstruct) scheme was used for the calculation of face fluxes near the interface between two fluids for the VOF model. The geometric reconstruction scheme represents the interface between fluids using a piecewise-linear

approach. A smooth interface shape could be formed using this interpolation scheme, even for an unstructured grid. Since the FLUENT software uses the gauge pressure in the calculations, the location of reference pressure is very important. It should be kept constant, the reference pressure is located in the lighter air phase within the tank ceiling. Since the FLUENT software does not give the output parameters of interest, slosh forces were calculated at each instant using the user defined functions (UDF).

The slosh force components, F_x (lateral), F_y (longitudinal) and F_z (vertical), can be derived from the integration of the pressure over the liquid wetted wall and baffle boundaries, such that:

$$F_y(t) = \sum_{\partial\Omega} (P_i \vec{A}_{yi}) \quad (6)$$

$$F_x(t) = \sum_{\partial\Omega} (P_i \vec{A}_{xi}) \quad (7)$$

$$F_z(t) = \sum_{\partial\Omega} (P_i \vec{A}_{zi}) \quad (8)$$

Where P_i is the pressure at the i^{th} face centroid in the wall zones \vec{A}_i is the area vector of the i^{th} face in the wall zones, and $\delta\Omega$ is the domain of liquid wetted faces on the wall boundary.

3. CFD slosh analysis

3.1 Geometry

The geometry of the tank is generated by using CATIA V5. Following are the different tank geometries used for the simulation (Diameter 300 mm; Length 410 mm).

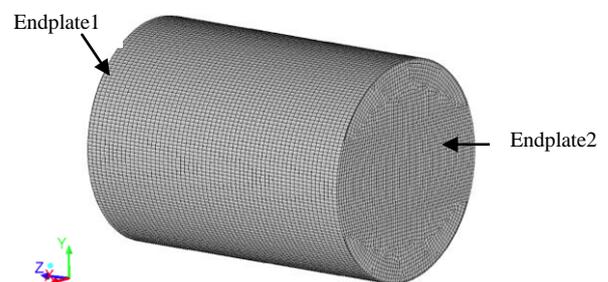


Fig.1 Tank without baffle (mesh)

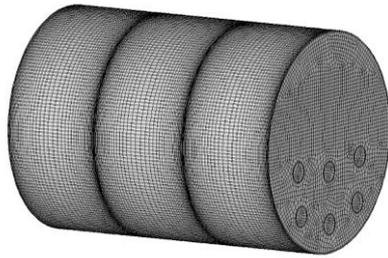


Fig.2 Tank with baffle (mesh)

Mesh generation process is carried out using GAMBIT. Hex mesh is preferred in order to reduce the cell count and ultimately the solution time. The meshing details are as below.

Element type – Hexahedral

Total elements – 0.4 to 0.8 million cells (Depends on configuration)

Mesh quality (Skewness) < 0.8 (0-best quality, 1-worst quality)

3.2 Simulation setup

The simulation is done with the help of ANSYS FLUENT by using Volume of Fluid (VOF) multiphase model. The simulation was done for 4 seconds of sloshing, from 0 to 2 seconds with an acceleration of 4.5 m/s² and for the next two seconds i.e., after breaking 2 to 4 seconds with a de-acceleration of 4.5 m/s² (assumed). As the process is isothermal, the energy equation is turned off because there is no exchange of heat energy.

The two phases used here are air and water. The primary phase is air and secondary phase is water.

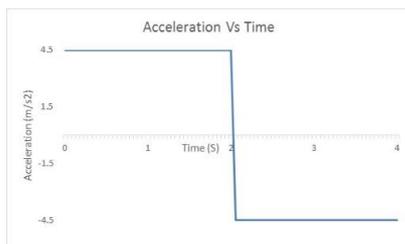


Fig.3 Input data

3.3 CFD Result

To study the liquid sloshing phenomenon, interface between the air and fluid has to be simulated and the sloshing behavior has to be found out on the interface. The contours of the sloshing at t = 0.5 s, 1.0 s, 1.5 s, 2.0 s, 2.5 s, 3.0 s, 3.5 s, 4.0 s are shown below for both the cases i.e., without baffle and with baffle.

Simulation is carried out for 25%, 50% and 75% tank fill condition for given tank geometries. Comparison of contours for tank 25% fill before and after breaking is done as shown below:

Before Breaking

Without Baffle

With Baffle

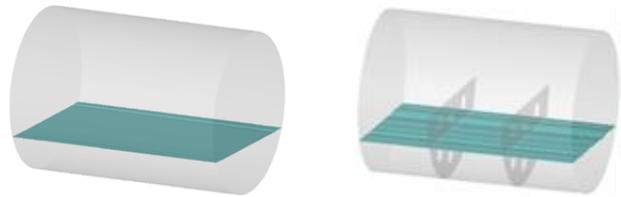


Fig.4 Time=0.5 sec



Fig.5 Time=1 sec



Fig.6 Time=1.5 sec

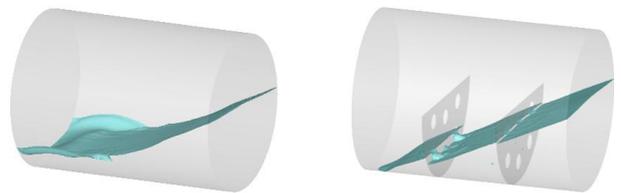


Fig.7 Time=2 sec

After Breaking

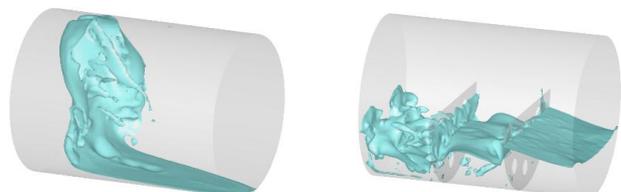


Fig.8 Time=2.5sec

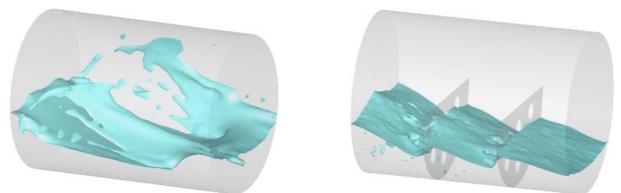


Fig.9 Time=3sec

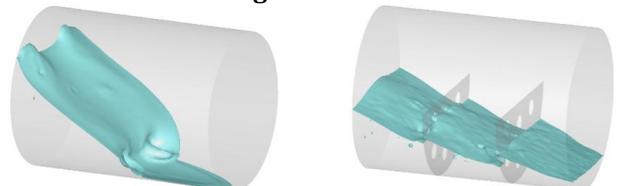


Fig.10 Time=3.5sec

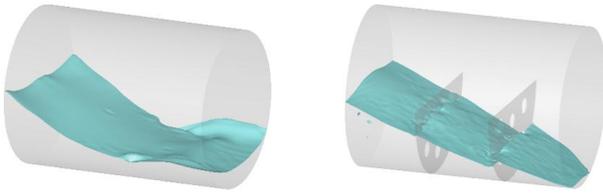


Fig.11 Time=4sec

3.4 Force variation on tank

To evaluate strength of sloshing force acting on different parts of tank, the force variation on different parts of tank with respect to time is plotted at different tank fill condition. When there is more turbulence inside the tank more force is acting on the tank surface. With the help of UDF the force extracted on tank surfaces is computed. The plotted force is the force acting normally on the tank surface. It is observed from the plot that sloshing is more in lower and moderate tank fill condition.

The plot suggest that the presence of baffle could help to reduce the slosh force developed under the application of longitudinal acceleration, reduction in the peak is observed as compared with and without baffle case. The results shows that the fluid motion yields lower magnitude of longitudinal slosh force, particularly under the higher fill level. The presence of baffles not only diminishes the peak but also yields lower steady values due to portions of fluid being trapped in lower section of the tank between two consecutive baffles or between the baffle and the endplates.

3.4.1 Tank 25% fill

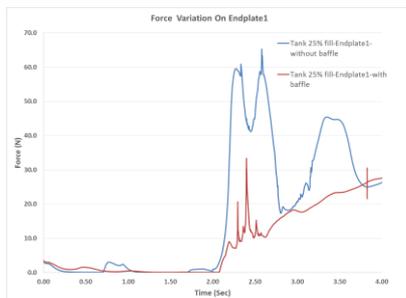


Fig.12 Force variation on endplate1 (tank 25% fill)

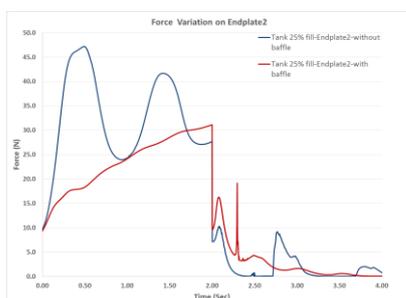


Fig.13 Force variation on endplate2 (tank 25% fill)

3.4.2 Tank 50% fill

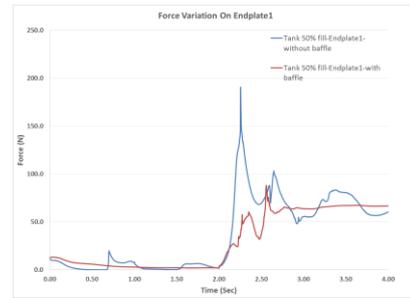


Fig.14 Force variation on endplate1 (tank 50% fill)

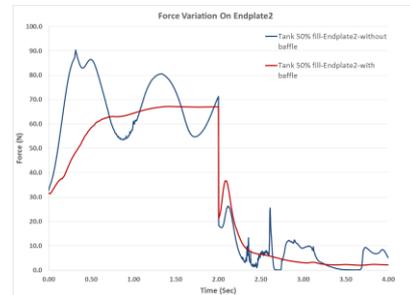


Fig.15 Force variation on endplate2 (tank 50% fill)

3.4.3 Tank 75% fill

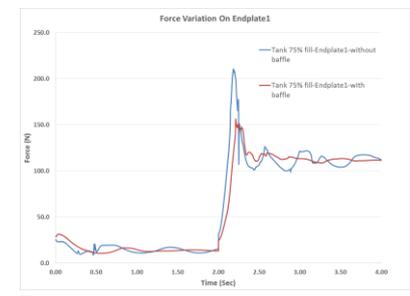


Fig.17 Force variation on endplate1 (tank 75% fill)

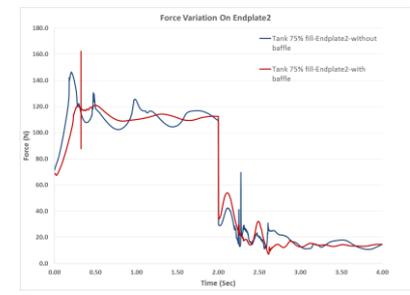


Fig.18 Force variation on endplate2 (tank 75% fill)

Conclusions

It is observed that sloshing force fluctuations are higher in lower and moderate tank fill condition than that of at higher fill level. At intermediate fill level, baffles are attributable to greater containment of fluid between the successive baffle or between the baffles and the tank endplates. In order to reduce the sloshing excitation, baffles are used as anti-sloshing device. Number of holes (baffle opening area) on the baffle plays important role in limiting the sloshing force.

References

- L. Khezzar, A. C. Seibi & A. Goharzadeh "Water Sloshing in Rectangular Tanks – An Experimental Investigation & Numerical Simulation" International Journal of Engineering (IJE), Volume (3): Issue (2).
- O. Faltinsen, O. Rognebakke, N. Alexander, and A.N. Timokha. (2003) "Resonant three dimensional nonlinear sloshing in a square-base basin", Journal of Fluid Mechanics, Volume. 487, pp. 1–42.
- O. Faltinsen, O. Rognebakke, N. Alexander, and A.N. Timokha. (2005) "Resonant three dimensional nonlinear sloshing in a square-base basin, part 2. Effect of higher modes", Journal of Fluid Mechanics, Volume. 523, pp. 199–218.
- Guorong Yan, (2008) "Liquid slosh and its influence on breaking and roll responses of partly filled tank vehicles" Ph.D thesis, Concordia university.
- Roshan Ambade, Rajesh kale "CFD analysis of sloshing with in tank" International Journal of Innovative Research in Technology and Science, Volume 2, Number 3.
- S.Rakheja, T.kandasamy, A.K.W. Ahemad. (2010) "An analysis of baffle design for limiting fluid slosh in partly filled tank trucks." The open transportation journal, pp.23-32.
- Mr. G. C.Koli, Prof V. V. Kulkarni (2010) " Simulation of Fluid Sloshing in Tank" proceedings of world congress on engineering, Volume 2.
- Brandon Marsell, Sathya Gangadharan, Yadira Chatman (2009) "A CFD approach to modeling spacecraft fuel slosh" 47th AIAA Aerospace science meeting including the new horizons forum and aerospace exposition, Orlando, Florida.
- Fu-Kun Gui, Sheng Chao Jiang "Numerical simulation of liquid sloshing problem under resonant excitation" Hindawi Publication Corporation, Advances in Mechanical Engineering.
- Fluent 15.0.7 Documentation, user's guide, Chapter 25 "Modeling Multiphase Flow".