World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:16, No:2, 2022

Study of the Sloshing Phenomenon in a Tank Filled Partially with Liquid Using CFD Simulation

Amit Kumar, Jaikumar V., Pradeep A. G., Shivakumar Bhavi

***Abstract*—**Reducing sloshing is one of the major challenges in industries where transporting of liquid is involved. The present study investigates the sloshing effect for different liquid levels of 50% of the tank capacity. CFD simulation for two different baffle configurations has been carried out using a time-based multiphase Volume of fluid (VOF) scheme. Baffles were introduced to examine the sloshing effect inside the tank. Results were compared against the baseline case to assess the effectiveness of baffles; maximum liquid height over the period of the simulation was considered as the parameter for measuring the sloshing effect inside the tank. It was found that the addition of baffles reduced the sloshing effect inside the tank as compared to the baseline model.

***Keywords*—**3D effect of sloshing, multiphase volume of fluid, CFD, baffles.

I. Introduction

SLOSHING occurs when there is a movement of tank filled with liquid. There will be dynamic interaction between the walls of the container and the fluid, sometimes fluid can very well deform the container as well. The free liquid surface in a road tanker might experience large disruptions for even very small motions of the container leading to stability problems. A tank undergoes wave motions when fluid is partially occupying in a moving tank [1], [2].

Therefore, it is essential while designing a tank which is intended to be in dynamic motion while carrying full or partially filled with fluid to consider amount of sloshing as a design criterion [3]. But it would be difficult to test this criterion as the prototype has to completely assembled, filled with fluid, and then moved around to simulate the dynamic movement that it might undergo. We decided to take a numerical approach to study if we are meeting the sloshing criteria of the tank design [4].

The boundary conditions of the motion at the interface of the fluid tank and the nonlinear motion of the free surface are the two main problems arise in a computational approach. The main objective of this work is to reduce the sloshing in a 50% filled tank using different baffle configurations through the CFD simulations.

There are several studies conducted on controlling sloshing using experiments and numerical methods. The damping effect of the baffles was clearly demonstrated by Vaibhav et al. [5]. The relationship between the sloshing and vertical height of the baffle has been well investigated by J.H. Jung et al. [6], they were able to stop the liquid going over the baffle and restrict the sloshing within half of the tank. Hakan Akyildiz et al. [7] investigated the pressure distributions at different locations and 3D effects on liquid sloshing and conducted systematic studies by changing excitation parameters to assess the sensitivity of sloshing loads. They concluded that baffles significantly reduce fluid motion through their experiments.

The dynamic response of baffled liquid storage tank was studied extensively by Gurinder Singh Brar et al. [8], they examined baffle configurations and found that role of combination of horizontal and vertical baffles were significant in controlling the sloshing.

II. Materials and Methodology

In this work, the fluid used is water. The elliptical fluid tank was configured with 50% filled water tank without baffles (Baseline model), with vertical baffles at the center of the tank (Modified 1) and three baffles placed at the bottom of the tank (Modified 2) Fig. 1 shows the comparison study made for all the three domains using CFD analysis.

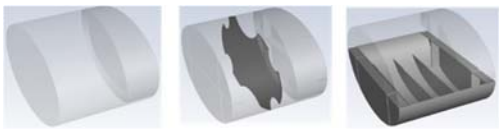
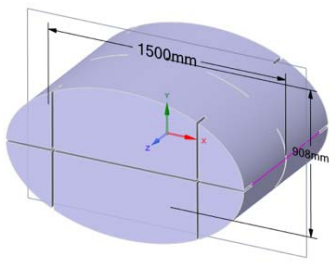


Fig. 1 Fluid domain used for CFD analysis

III. Numerical Modeling

The 3D tank geometry was prepared in Space claim tool. The measurements of the fluid tank are given in Fig. 2. The baffles are shown in Fig. 3. Forward acceleration translation motion was set for all the configurations of 9.81 m/s^2 in the direction of negative y axis. The tank movement is given in negative X direction with an acceleration for 4 seconds.



Amit Kumar is with the HCL Technologies Ltd, India (e-mail: amitkuma1@hcl.com).

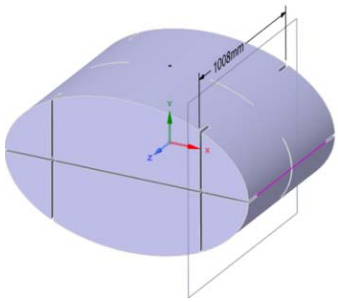


Fig. 2 CAD model dimensions

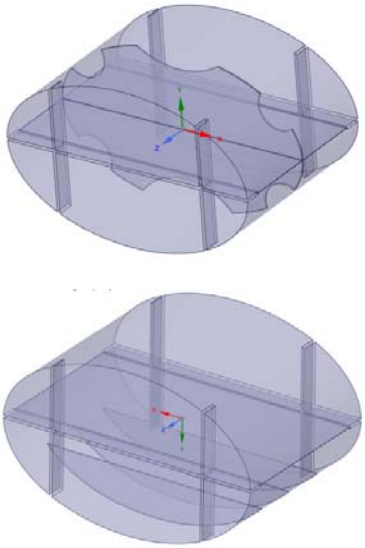


Fig. 3 CAD of differnts baffles used in the current study

A. Governing Equations

The predication of fluid movement and interface was done using Volume of Fluid (VOF) multiphase model in ANSYS FLUENT. The incompressible Reynolds-Averaged Navier–Stokes equations are adopted in this paper to investigate the flow.

The following three conditions are possible, depending upon the volume fraction values, the qth fluid's volume fraction in the cell is denoted as α_q

α_q=0 : The cell is empty (of the qth fluid).

α_q=1 : The cell is full (of the qth fluid).

0<α_q<1 : The cell contains the interface between qth theluid and one or more other fluids

The appropriate properties and variables will be assigned to each control volume within the domain, based on the local value of α_q [9].

The continuity equation for the qth phase is:

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

where,

$\dot{m}_{qp}$ - mass transfer from phase q to phase p

$\dot{m}_{pq}$ - mass transfer from phase p to phase q

$S_{\alpha q}$- mass source for each phase

The implicit formulation used is:

$$\frac{\alpha_q^{n+1}\rho_q^{n+1} - \alpha_q^n\rho_q^n}{\Delta t}V + \sum_f\left(\rho_q^{n+1}U_f^{n+1}\alpha_{q,f}^{n+1}\right) = \left[S_{a_q} + \sum_{p=1}^{n}\left(\dot{m}_{qp} - \dot{m}_{pq}\right)\right]V \quad (2)$$

where:

n+1 -index for current time step

n - index for previous time step

α_q^{n+1} - cell value of volume fraction at time step n+1

α_q^n - cell value of volume fraction at time step n

$\alpha_{q,f}^{n+1}$- face value of the qth volume fraction at time step n+1

U_f^{n+1}- volume flux through the face at time step n+1

V - cell volume.

Momentum Equation:

$$\frac{\partial}{\partial t}\nabla.(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.[\mu(\nabla\vec{v} + \nabla\vec{v}^T)] + \rho\vec{g} + \vec{F} \quad (3)$$

where,

p- static pressure

$\overrightarrow{\rho g}$ - gravitational body force

$\vec{F}$ - external body forces

μ-molecular viscosity

The polyhedral grid generation was done using ANSYS FLUENT the section view of the mesh is show in the Fig. 4, and the details of grid information is given below Table I.

TABLE I
SUMMARY OF MESH DETAILS OF ALL THE MODELS

	Baseline	Model 1	Model 2
Cell count (in million)	1.51	1.52	1.48
Skewness	0.85	0.88	0.87

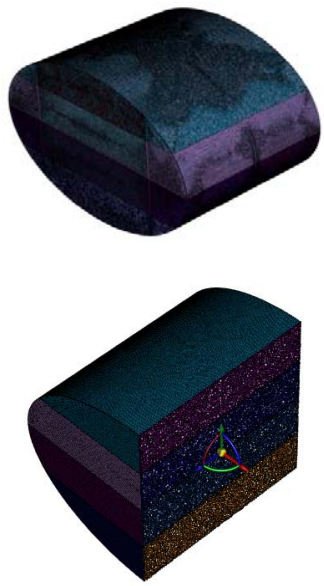


Fig. 4 Section view of polyhedral grid

The CFD simulation is carried out to study the sloshing in a 3D fluid tank using multiphase VOF method in ANSYS FLUENT solver. The Transient pressure-based solver is used for the current study. Fractional step algorithm is used as Pressure-Velocity Coupling Method as the flow is time dependant. For spatial discretization scheme compressive method was used under volume fraction.The VOF model was used to capture the interface between the two immiscible fluids. The 50% of the total volume of tank was occupied by water for all the three models in the present case. The acceleration profile was given as an input for the movement of tank domain. The properties of the fluid are shown in Table II below.

TABLE II
FLUID THERMO PHYSICAL PROPERTIES

Property	Water	Air
Density (kg/m^3)	998.2	1.1768
Specific heat (J/kg-K)	4182	1006.43
Viscosity (kg/m-s)	0.001003	1.7894e-05
Thermal conductivity (W/m-K)	0.6	0.0242
Molecular Weight (J/kgmol)	18.01	28.97

IV. RESULTS AND DISCUSSIONS

The simulations results captured from CFD analysis is shown from Figs. 5 to 7 with movement of tank against time.

From Fig 5, higher fluid sloshing is observed in the tank as the time progress.With the addition of single baffle at the mid of the tank results in higher sloshing at the ends of tank as shown in Fig 6. The addition of three baffles in modified 2 shown that sloshing at the beginning of the tank movement and later resulted in reduced sloshing as shown in Fig 7.

The results depict that sloshing has been reduced using three baffles at the bottom of tank as compared to the baseline model. The three baffles have shown fluctuations in sloshing during initial movement of tank from 0 seconds to 1.5 seconds. This could be due to the baffles provided near the side of the tanks. The baseline and the center baffle model have reached maximum height of 0.45 m at 3 seconds of time with tank movement as shown Fig. 8.

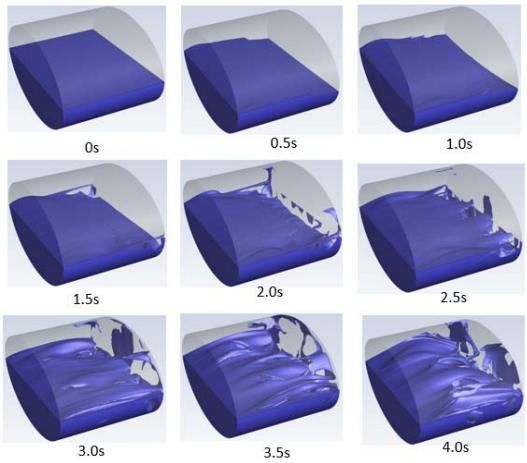


Fig. 5 Scenes of sloshing at different time duration of Baseline simulation

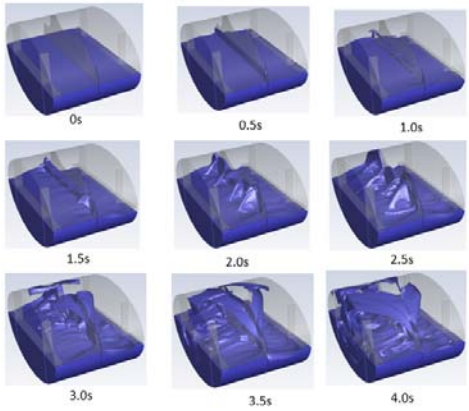


Fig. 6 Scenes of sloshing at different time duration of Modified 1 simulation

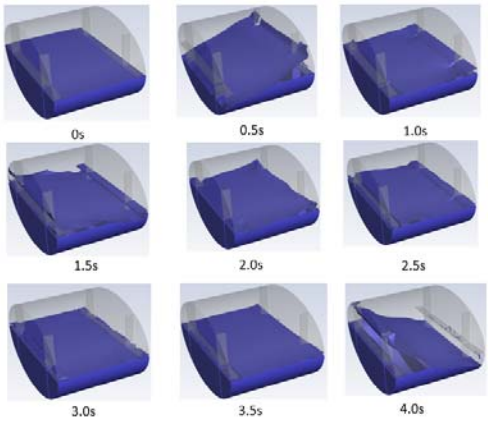


Fig. 7 Scenes of sloshing at different time duration of Modified 2 simulation

It was observed that from CFD simulations the modified 2 model has reduced 24% of sloshing effect in terms of sloshing height as compared with the other two models.

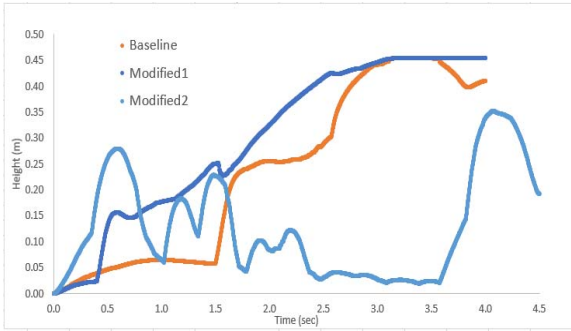


Fig. 8 Plot of Maximum sloshing height vs. time

V. CONCLUSION

In the present study, the transient simulations of liquid sloshing were carried out using 3D simulation approach. The liquid interface of the tank configurations without and with baffles was examined. The liquid interface well captured using Volume of Fluid methodology in all the three models. The sloshing damping effect of baffles is clearly visible in the

second case of analysis. It was observed that the addition of three baffles at the bottom of the tank in modified 2 model has reduced 24% of sloshing effect in terms of sloshing height as compared with the other two models.

REFERENCES

[1] B. Molin, F. Remy, Experimental and numerical study of the sloshing motion in a rectangular tank with a perforated screen
[2] Hamn-Ching Chen,CFD simulation of compressible two-phase sloshing flow in a LNG tank
[3] H. Q. Yang Propellant Sloshing Parameter Extraction from CFD Analysis
[4] C. G. Koh,M.Luo , M. Gao,W. Bai Modelling of liquid sloshing with constrained floating baffle
[5] Vaibhav Singala, Jash Bajajb, Nimish Awalgaonkara & Sarthak Tibdewalc on CFD Analysis of a Kerosene Fuel Tank to Reduce Liquid Sloshing
[6] J. H. Jung a, H. S. Yoon b, C. Y. Lee c & S. C. Shin on Effect of the vertical baffle height on the liquid sloshing in a three-dimensional rectangular tank
[7] Hakan Akyildiz & Erdem Unal on Experimental investigation of pressure distribution on a rectangular tank due to the liquid sloshing
[8] Gurinder Singh Brar & Simranjit Singh on An Experimental and CFD Analysis of Sloshing in a Tanker
[9] ANSYS fluent theory guide version 2021