Numerical Simulation of liquid Sloshing with baffles in the fuel container

Alireza Javanshir[1], Rasoul Elahi[2], Mohammad Passandideh-Fard[3]
1,2- Graduate Student, Ferdowsi University of Mashhad, Mashhad, Iran
3- Associate Professor, Ferdowsi University of Mashhad, Mashhad, Iran

Abstract
In this paper, a two-dimensional numerical model is developed to study viscous liquid sloshing in a tank with internal baffles. The model is validated by a comparison between the computational and experimental results for time-dependent linear acceleration sloshing scenarios. The governing equations for the 2D incompressible fluid flow are continuity and Navier-Stokes equations along with an equation for the free surface advection. The deformation of the liquid-gas interface is modeled using the Volume-of-Fluid (VOF) method. The fluid flow equations describing the fluid sloshing in the container and the dynamic equation which describes the movement of the container are solved separately in two coupled programs. In each time step of computations, the outputs of the fluid program (forces and torque) are obtained and used as inputs for the dynamic program. The forces and torque are applied to the body of the container resulting in translational accelerations which are then used as inputs to the fluid program. At first this model was validated by theoretical and experimental results for linear acceleration and after that it is used to simulate the effect of vertical baffles to reduce liquid sloshing in a fuel tank that under sinusoidal force and it was found by the total kinetic energy of fluid that baffles have an important role to reduce the liquid sloshing in a container thus it is an effective way. The developed model in this study can be used in the design and optimization of liquid container and the best arrangement for the inside baffles.

Keywords: sloshing- CFD-VOF-baffle- liquid container

Introduction

The sloshing phenomenon occurs in many applications, such as, propellant tank in aerospace devices and LNG (liquid natural gas) cargos in the ship industry. When an external transient or steady force acts on a fluid, the liquid is driven from equilibrium state. In this condition, the free surface of the liquid moves and the liquid splash on the container walls. In many cases, these forces affect the maneuver of the vehicle. The influence of sloshing liquid may hamper critical maneuvers in space, such as the docking of liquid-cargo vehicles or the pointing of observational satellites. Several serious problems with sloshing liquid in a spacecraft have been reported over the years. For example, during the last seconds of the first lunar landing [1], or another example is the NEAR (Near Earth Asteroid Rendezvous) mission to the asteroid Eros in 1998. During an orbital correction, the spacecraft experienced an unexpected motion and Fuel slosh was identified as the probable cause [2].

The Marker-and-Cell (MAC) method is the ‘father’ of all free-surface flow methods [3], and makes the use of mass-less particles to keep track of the liquid region. Accuracy requires a considerable number of particles per grid cell, making the method computationally expensive, especially in 3D. A cheaper way is to apply only surface markers [4], but now splitting and merging of the surface are difficult to handle. The MAC follow-up is the volume-of-fluid (VOF) method introduced by Hirt and Nichols [5]. Here a discrete indicator (or color) function is used that corresponds to the cell volume occupied by fluid. This method improved by young’s method. He modeled free surface by Piecewise Linear Interface Reconstruction (PLIC) Algorithms.

The use of internal baffles is considered to be an effective means on reducing the sloshing amplitude. Armenio and La Rocca [6] analyzed sloshing of water in rectangular open tanks and observed that the presence of a vertical baffle at the middle of the tank dramatically changed the sloshing response. Akyildiz [7] estimated the pressure distribution in a rigid rectangular tank due to large amplitude liquid sloshing. Different cases including baffled and un-baffled tanks with different fill depths were studied near and on resonant frequency. Cho and Lee [8] studied numerically the effects of baffle on liquid sloshing in a tank using a FEM model. Akyildiz and Unal [9] designed an experiment to study non-linear behavior and damping characteristics of liquid sloshing in a tank under pitch excitation. Biswal et al [10] studied the influence of baffles on non-linear sloshing in both cylindrical and rectangular tank using FEM. Jung [11] studied the effect of the vertical baffle height on the liquid sloshing in a rectangular tank. The results indicated that the baffle had a greater influence on the slosh frequencies of liquid when placed near to free surface and the influence was gradually reduced when it was moved towards the bottom of the tank.

In the previous studies some assumptions such as two phase flow and violent sloshing weren't considered but in this study the VOF method is used to find the free surface and those assumptions are considered.

1. Graduate Student, Ferdowsi University of Mashhad, ajavanshir19@yahoo.com
2. Graduate Student, Ferdowsi University of Mashhad, r.elahi.msc@gmail.com
3. Associate Professor, Ferdowsi University of Mashhad, mpfard@um.ac.ir

Mathematical Model

Fluid dynamics

The governing equations are the unsteady, incompressible, 2D continuity and Navier–Stokes equations. The fluid motion is described by means of the conservation of mass [12]:

$$\nabla \cdot u = 0 \tag{1}$$

and conservation of momentum, for a moving tank-fixed given by:

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\frac{1}{\rho}\left(\nabla p - \mu(\nabla \cdot \nabla)u\right) + F_B + F_V \tag{2}$$

Where u denotes the velocity of the fluid relative to the tank, p the pressure, ρ and μ the fluid density and viscosity, respectively. The vectors F_B and F_V represent body force and a virtual body force induced by the motion of the tank.

The free surface is found by:

$$\frac{df}{dt} = \frac{\partial f}{\partial t} + \vec{V} \cdot \vec{\nabla} f = 0 \tag{3}$$

Equation 3 is used to track the location of the interface and is solved according to Youngs' PLIC algorithm [12]. For boundary conditions, the usual no-slip condition for viscous flow is applied at the tank wall.

Solid-body dynamics

The model for solid-body motion consists of an equation for linear momentum [17].

$$m_s \vec{\ddot{q}} + \vec{\dot{\omega}} \times m_s \vec{r_s} + \vec{\omega} \times \left(\vec{\omega} \times m_s \vec{r_s}\right) = \vec{R} + m_s \overrightarrow{F_B} \tag{4}$$

and an equation for angular momentum

$$m_s \vec{r_s} \times \vec{\ddot{q}} + I_s \vec{\dot{\omega}} + \vec{\omega} \times \left(I_s \vec{\omega}\right) = \vec{T} + m_s \vec{r_s} \times \overrightarrow{F_B} \tag{5}$$

In these equations, $\vec{\ddot{q}}$ (linear acceleration) and $\vec{\dot{\omega}}$ (angular acceleration) are unknown variables. The mass of the solid body is denoted by m_s; further, I_s is the moment of inertia tensor and $\vec{r}$ is the center of mass of the solid. The last terms in Equations 4 and 5 represent the force and torque due to an external body force such as gravity. Finally, $\vec{R}$ and $\vec{T}$ are, respectively, the force and torque that the fluid, via pressure and viscous effects, exerts on the boundaries of the solid body. $\vec{R}$ and $\vec{T}$ are defined as:

$$\vec{R} = \oint_{\partial V} (pI_3 - \mu \nabla \vec{V}).nds \tag{6}$$

$$\vec{T} = \oint_{\partial V} (r \times (pI_3 - \mu \nabla \vec{V})).nds \tag{7}$$

Here, I_3 is the 3×3 identity matrix and n the outward pointing normal to the boundary of volume ∂V of the solid body.

Virtual body force method

In order to couple the fluid dynamics and solid-body dynamics, the motion of the fluid in presence of the solid body movement has to be modeled. Here, the fluid velocity is considered in two reference frames: the velocity $\overrightarrow{V^*}$ of a fluid particle with respect to an inertial reference frame and the velocity $\vec{V}$ of the same liquid particle with respect to a moving reference frame. The relation between $\overrightarrow{V^*}$ and $\vec{V}$ is given by

$$\frac{D\overrightarrow{V^*}}{Dt} = \vec{\ddot{q}} + \vec{\dot{\omega}} \times \vec{r} + \vec{\omega} \times \left(\vec{\omega} \times \vec{r}\right) + \frac{D\vec{V}}{Dt} + 2\vec{\omega} \times \vec{V} \tag{8}$$

where $\vec{\ddot{q}} = \frac{d\vec{\dot{q}}}{dt} + \vec{\omega} \times \vec{\dot{q}}$ is the acceleration of the moving origin with respect to the origin of the inertial reference frame; $\vec{\dot{\omega}}$, $\vec{\omega}$ and $\vec{r}$ are the angular acceleration ,angular velocity and the position vector of the liquid particle respectively and they are in the moving reference frame. The third and fifth terms in the right-hand side of Equation 8 represent the centrifugal and Coriolis accelerations respectively. Now, the Navier-stokes equations become:

$$\frac{D\overrightarrow{V^*}}{Dt} = -\frac{1}{\rho}\left(\nabla p - \mu(\nabla \cdot \nabla)\vec{V}\right) + \overrightarrow{F_B} \tag{9}$$

Alternatively, using Equation 9:

$$\frac{D\vec{V}}{Dt} = -\frac{1}{\rho}\left(\nabla p - \mu(\nabla \cdot \nabla)\vec{V}\right) + \overrightarrow{F_B} + \overrightarrow{F_v} \tag{10}$$

where

$$\overrightarrow{F_V} = -\vec{\ddot{q}} - \vec{\dot{\omega}} \times \vec{r} - \vec{\omega} \times \left(\vec{\omega} \times \vec{r}\right) - 2\vec{\omega} \times \vec{V} \tag{11}$$

Eq.10 has a form similar to Eq. 2 (meaning that in the numerical model for the liquid dynamics only small changes are required). Using Newton's third law, the extra term $\overrightarrow{F_v}$ in Equation 10 can be seen as acceleration due to a virtual body force. Instead of actually moving the solid body in the numerical model, the fluid is subjected to an acceleration (equal in magnitude and opposite in sign) to account for the solid-body motion.

Numerical Method

VOF algorithm

The current numerical algorithm is based on a modified version of the RIPPLE computer code (Kothe et al. 1991). Bussman et al. (1999) used Youngs' algorithm in place of the original Hirt–Nichols and optimized the continuous surface force (CSF) model (Brackbill et al. 1992) of RIPPLE. The finite difference discretisation of the governing equation follows the Euler forward scheme in a staggered grid [13].

$$\frac{\vec{V}^{n+1} - \vec{V}^n}{\delta t} = -\left(\vec{V} \cdot \vec{\nabla} \vec{V}\right)^n - \frac{1}{\rho^n}\vec{\nabla} p^{n+1} + \frac{1}{\rho^n}\vec{\nabla} \cdot \vec{\tau}^n + \frac{1}{\rho^n}\vec{F}_B^n \tag{12}$$

In the above equation, all terms except for pressure are computed explicitly. The velocity field is calculated according to a two-step projection method as follows. First, an intermediate velocity ($\vec{\tilde{V}}$) is obtained,

$$\frac{\vec{\tilde{V}}^n - \vec{V}^n}{\delta t} = -\left(\vec{V} \cdot \vec{\nabla} \vec{V}\right)^n + \frac{1}{\rho^n}\vec{\nabla} \cdot \vec{\tau}^n + \frac{1}{\rho^n}\vec{F}_B^n \tag{13}$$

The CSF method is used to model surface tension as a body force ($\vec{F}_b$) that acts only on interfacial cells (Brackbill et al. 1992). In the second step, the intermediate velocity is projected to a divergence free velocity field:

$$\frac{\vec{V}^{n+1}-\tilde{\vec{V}}}{\delta t}=-\frac{1}{\rho^{n}}\vec{\nabla}p^{n+1} \quad (14)$$

The continuity equation is also satisfied for the velocity field at the new time step:

$$\vec{\nabla}\cdot\vec{V}^{n+1}=0 \quad (15)$$

Results and Discussion

Container under linear acceleration

In order to evaluate the performance of the model in simulating free surface flows, a test case was considered and the results were compared with available theoretical data. A 25×25 centimeters rectangular container was considered and this allows for approximately 62.5 liters fuel. The container was almost 25% (15.6 liters) filled with fuel and It was assumed that the vehicle was at rest at time equal zero then it started to move along the x-direction with a constant acceleration equal to $4.5\,m/s^2$. The fuel inside the container was assumed to have the property of water. The liquid was deviated from equilibrium state and moved and clashed with the wall of the container. After a while, the liquid neared a stable condition at a certain angel. In this study, we compare the results of this simulation with the available theoretical data. Based on the theory, the free surface of liquid must be perpendicular to the pressure gradient and is thus tilted at a downward angle θ such that:

$$\theta = Arc\tan\left(\frac{a_x}{a_y+g}\right) \quad (16)$$

Where a_x, a_y and g are uniform acceleration in x and y-direction and gravity acceleration respectively. In this study the free surface tilts at an angle of 24.64 deg, regardless of the shape of the container. In Fig. 1, the free surface deformation is shown at different times. In this figure, the theoretical tilt angle is represented with a dash line. As can be seen, computed free surface shapes at the steady state condition (t=10s) are in good agreement with those obtained from Eq. (16) (dash line).

Oscillating tank

As a second test case In order to evaluate the performance of the model in simulating free surface flows under time dependant acceleration , a test case was considered and the results were compared with available experimental data.

A rectangular tank of 0.4 m wide and 0.2 m high, filled with 60 percent of water (0.12 m high) , was forced to oscillate from left to right. The water began to move in an oscillatory manner before impacting the top wall. Experimental free surface shapes [14] are available prior to the first impact on the top wall (Fig. 2). The tank was moved in a horizontal plane as follows:

$$X(t)=A_0\left[\sin(2\pi f_1 t)-\sin(2\pi f_2 t)\right] \quad (17)$$

$$with\ A_0=0.0075m,\ f_1=1.598Hz\ \ and\ \ f_2=1.307Hz$$

As can be seen in Fig. 2 computed free surface shapes are in good agreement with those obtained from experiments (black dots) [14]. However, at t=1.23s it can be seen small differences between numerical results and experimental data for the free surface shape. This difference can be attributed to the 2D modeling instead of the real 3D nature of sloshing phenomenon. Furthermore, for this simulation it was considered an equilibrium contact angle on the container walls whereas in reality the contact angle must be dynamic.

Tank with installed baffels

As a second test case, we consider a 0.4×0.2 meters rectangle container. The container is almost 50% (40 liters) filled with fuel. The tank is moved in a horizontal plane as follows:

$$X(t)=0.0105\times\left[\sin(2\pi\times 1.037t)\right] \quad (18)$$

Because of the small size of the container, the assumption of two dimensional sloshing is valid and for this reason we only apply the rotational acceleration to account for the moving system of reference in the fluid governing equations. The liquid inside the container was assumed to have the property of water. Fig. 3 shows the liquid behavior inside the container at different times.

As can be seen from the figures, the liquid undergoes a harsh sloshing with breakup. In figure 4, the simulation results for the same case, but this time with installed four equal baffles at the bottom of the container are shown. The height and width of the baffles are 6.0 and 1.0 centimeter respectively. As seen clearly, installing baffles will damp the liquid waves quickly and this will reduce the undesirable consequences of the sloshing. The liquid sloshing can be characterized with the variation of the liquid kinetic energy in time. In Fig. 4 the total kinetic energy of the liquid calculated for two cases without baffle and with baffle are shown. It was found that installing baffles decreases the effects of liquid sloshing significantly.

Conclusion

In this study, we have developed a computational model that can simulate liquid sloshing in 2D containers and fuel tanks with angular and translational movement including constant or time dependent linear/angular acceleration. The model was validated by a comparison between the computational and expremental results and timed dependent acceleration. The deformation of the liquid-gas interface is modeled using the Volume-of-Fluid (VOF) method. Finally, the effects of the baffels in the liquid containers is investigated, and the results show that, the baffles installation in the containers can decrease the undesirable consequences of sloshing waves significantly. Therefore, the code can be used in the design and optimization of internal baffels in the liquid tanks to reduce the sloshnig effects that may cause undesirable forces on the carrying tankers and vehicle.

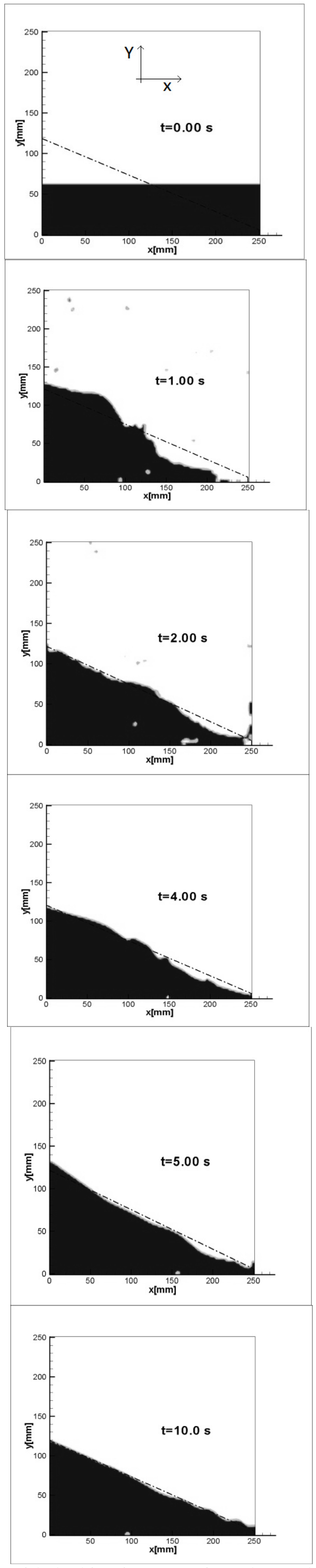


Fig. 1: Liquid sloshing in a container that undergoes a linear acceleration equal to 4.5m/s^2 in the x direction.

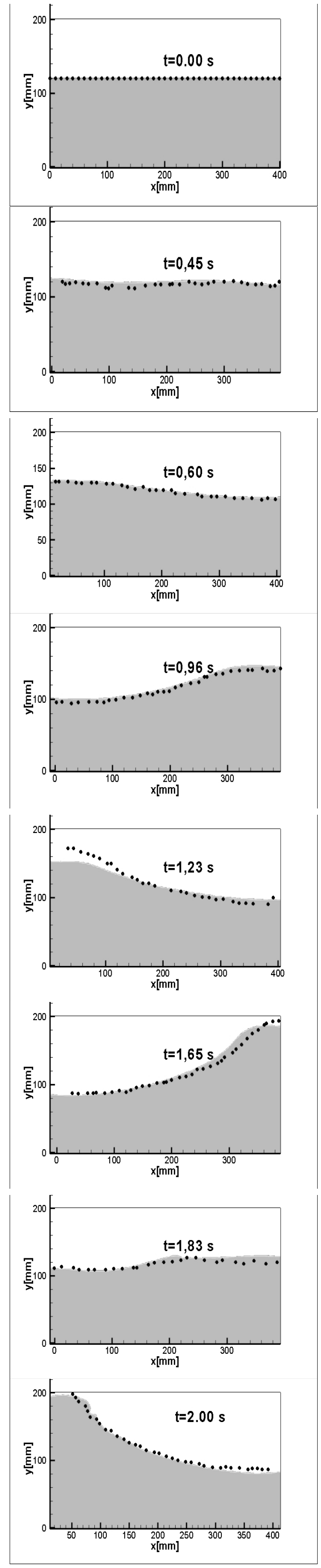
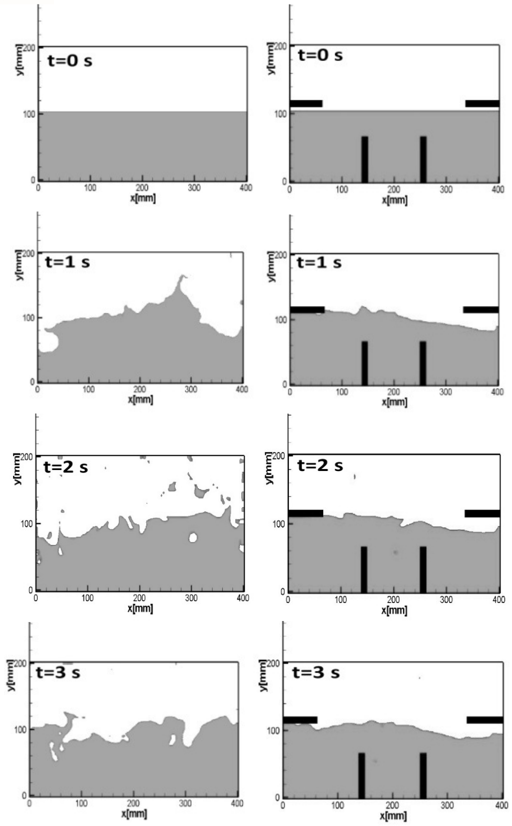


Fig. 2: A comparison between numerical (grey area) and experimental (black dots) free surface shape [14].



(a) (b)

Fig. 3: Liquid sloshing in a container a) without baffles and b) with installed baffles.

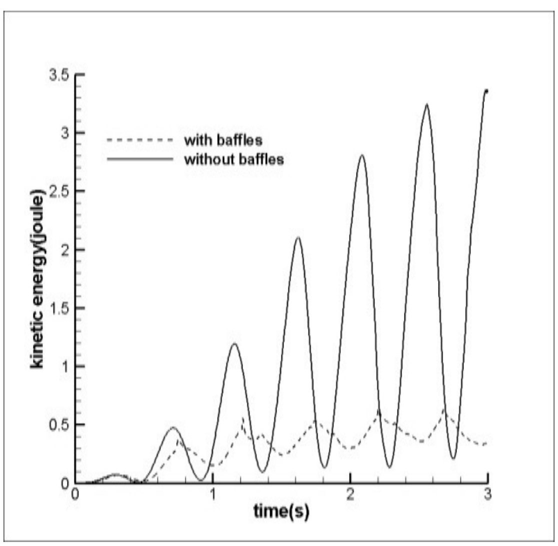


Fig. 4: A Comparison between total kinetic energy in two cases.

References

1- Apollo 11 Lunar Surface Journal, The first lunar landing mission time, On the WWW,at http://www.hq.nasa.gov/office/pao/History/alsj/a11/a11.landing.html.

2- NEAR Anomaly Review Board, The NEAR Rendezvous Burn Anomaly of December 1998, Johns Hopkins University, Applied Physics Laboratory, 1999.

3- Harlow F.H., Welch J.E., Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface, Phys. Fluids, 2182–2189, 1967.

4- Juric D., Tryggvason G., A front tracking method for dendritic solidification, J. Comput. Phys, 127–148, 1996.

5- Hirt C.W., Nichols B.D., Volume of fluid (VOF) method for the dynamics of free boundaries, J. Comput. Phys. 201–225, 1981.

6- Armenio V., La, Rocca M., on the analysis of sloshing of water in rectangular containers numerical study and experimental validation, OceanEng, 23:705–39, 1996.

7- Akyildiz H., elebi M.S., Numerical computation of pressure in a rigid rectangular tank due to large amplitude liquid sloshing, Turkish J Eng Environ Sci; 25:659–74,2001.

8- Cho J.R, Lee H.W. ,Numerical study on liquid sloshing in baffled tank by nonlinear finite element method, Comput Methods Appl Mech Eng, 193:2581–98,2004.

9- Akyildiz H., Unal E., Experimental investigation of pressure distribution on a rectangular tank due to the liquid sloshing, Ocean Eng, 32:1503–16,2005.

10- Biswal K.C., Bhattacharyya S.K., Sinha P.K., Non linear sloshing in partially liquid filled containers with baffles, Int J Numer Methods Eng, 68:317–37, 2006.

11- Jung J.H., Yoon H.S., Lee C.Y., Shin S.C., Effect of the vertical baffle height on the liquid sloshing in a three-dimensional rectangular tank, Ocean Engineering 44 (2012) 79–89,2012.

12- Veldman A.E.P., Gerrits J., Luppes R., Helder J.A., Vreeburg J.P.B., The numerical simulation of liquid sloshing on board spacecraft, J. Comput. Phys., Vol. 224, 82-99, 2007.

13- Passandideh-Fard M., Roohi E., Transient simulations of cavitating flows using amodified volume-of-fluid (VOF) technique, J. Computational Fluid Dynamics vol22:1, 97 – 114.

14- Corrignan P., Analyse Physique des Phénoménes Associés au Ballotement de Liquide dans des Réservoirs (Sloshing), Ph.D. thesis, Ecole Centrale de Nantes, 1994.