Numerical Analysis of a Propellant Tank Slosh Baffle

S. Chintalapati¹, D. R. Kirk²

Abstract — A current problem that severely affects the performance of spacecraft is related to slosh dynamics in liquid propellant tanks under microgravity conditions. Accurate prediction of the slosh dynamics is critical for successful mission planning and may impact vehicle control and positioning during rendezvous, docking, and reorientation maneuvers. The purpose of this work is to assess the performance of various slosh-mitigating baffle designs and configurations using computational fluid dynamics. This work develops metrics, including wall wetting, peak slosh amplitude, and bulk fluid motion, to assess the relevance of a particular baffle geometry and placement within the tank for a prescribed bulk fluid motion over a range of acceleration levels. The two- and three-dimensional studies are used to assess the slosh model’s sensitivity to grid resolution, laminar versus turbulent flow models, and Bond number scaling. The results are used to develop a foundation on which to build a full six-degree-of-freedom dynamic mesh model, allowing for fluid-force interaction with a propellant tank, which will be benchmarked against low-gravity slosh flight data. Copyright © 2008 Praise Worthy Prize S.r.l. - All rights reserved.

Keywords: Liquid Propellant Rockets, Slosh, Slosh Baffles, CFD, Volume of Fluid Methods

I. Introduction

Slosh is a pressing problem for spacecraft stability and control. Propellant remaining inside a tank may be excited by the motion of the vehicle, and reaction forces and moments caused by slosh can degrade the pointing accuracy of the system, [1]-[2]. For example, in preparation for orbital insertion of the payload, the upper-stage of a rocket undergoes a series of maneuvers which may lead to large amplitude sloshing motion of the propellants. Liquid propellant reaching the relief and orbital control vents may result in a significant increase in expelled mass which may cause mission failure due to loss of the proper orbital attitude. As another example delicate docking maneuvers between spacecraft and space stations may also be impacted by liquid slosh motion. Although baffles add to the weight of the tank, they play a vital role in mitigating undesired slosh induced motion. Slosh baffles located within the Space Shuttle external tank are shown in Fig. 1, and Fig. 2 shows an example of a baffle used by Armadillo Aerospace to damp oscillations observed during test flights on a vehicle under development, [3].

![Fig. 1. Space Shuttle External Tank Slosh Baffle, [4]](image)

This paper utilizes computational fluid dynamics (CFD) to assess the performance of various slosh-mitigating baffle configurations for spacecraft design. Depending on the type of disturbance and tank shape, the liquid propellant can experience different types of motion including simple planar, non-planar, rotational, symmetric, asymmetric, quasi-periodic and chaotic. In low-gravity, surface tension and capillary action may dominate even in large booster size tanks and the liquid may be oriented randomly within the tank depending upon the wetting characteristics of the tank wall, [5]. Theoretical research dates back decades, but only now have CFD models been capable of handling complex sloshing motion. This work forms a foundation for the development of a six-degree-of-freedom (6-DOF) dynamic mesh model that will be benchmarked against low-gravity experimental data. Before such a model can be developed it is important to have an understanding of grid sensitivity, turbulence modeling, and Bond number scaling during low-gravity conditions so that computation resources can be optimized. Metrics are also developed for characterizing slosh baffle...
effectiveness and guidelines for comparing various slosh scenarios.

early work in slosh dynamics has been mostly theoretical, and treats the liquid as a variable inertia only, i.e. the viscosity, surface tension and other important effects are not considered, [6]-[7]-[8]-[9]. Early research includes comparison of experimental data to theory by Abramson, et al., [10], Babicki, et al. developed a mathematical theory for behavior of liquid under total or partial weightlessness, [11], and Narimanov, et al. developed the equations of motion for the nonlinear dynamics of liquid-containing flight vehicles, [12]. Analytical solutions of fluid motion based on first principles has been developed for simple tank geometries, excitations, and boundary conditions using potential theory, [13]-[16]-[17]-[18], and experimental verification of simple, low-amplitude slosh has been performed, [13]. When liquid excursions remain small, a linear second-order oscillator provides a useful representation of the slosh dynamics, which can be integrated with the state equations of the complete vehicle system. More complicated plants result from nonlinear slosh models, which have been shown to yield reliable predictions over larger ranges, or by including additional oscillators and an increased state dimension [8].

Numerical solution to slosh has been emerging in recent times, owing to the major advances in computational capabilities. CFD models to make slosh predictions during the high acceleration ascent phases of a rocket have been used, although very little work has been done in cases of very-low accelerations when the vehicle is in space, [18]-[2]. When available, predictions can be used to validate the performance of simpler models, and a direct comparison with CFD-predicted damping is of significant interest, however, these models dynamic mesh simulations, which allow the container to move in 6-DOF with feedback from the liquid acting on the tank walls.

Another approach for predicting slosh motion is to use scaled model testing, such as that done at Southwest Research Institute, but thus far the results are largely qualitative and there has not yet been direct data comparison with detailed CFD models. Other studies have focused on analyzing available flight data to identify conditions leading to mission failure. The FLEVO project, under the direction of the National Aerospace Laboratory (The Netherlands) has been the most substantial effort devoted to fill the gap between numerical simulations, [16]-[18], and the development of an experimental framework to measure and characterize slosh under microgravity, [2].

II. Computational Setup and Numerical Modeling Overview

CFD studies were performed utilizing the transient Volume of the Fluid (VOF) method, which is well suited to multiphase flows involving two immiscible fluids. An 1st order explicit VOF unsteady scheme was used with absolute velocity formulation and with body forces enabled. The default iterative time advancement scheme was used, where a global iteration on the momentum and pressure (and other) equations is performed many times within each time step. The method relies on the basis that the fluid state is described in each cell by one value; the method introduces a function F whose value F=1 correspond to a cell full of fluid, and F=0 to an empty cell, and a geo-reconstruct scheme is used to track the fluid-air interface, [19]. A generic tank was modeled and meshed with several grids to check the dependency of solution on the grid density. The tank has a cylindrical section and is capped by two elliptical domes. The diameter of the cylindrical section is 4 m and tank height is 3.5 m. Fig. 3 shows the geometry and various meshes with increasing grid density used. Since computational run time increases as grid density is increased, there is motivation to find a grid density which gives an accurate approximation of the bulk fluid behavior in low-gravity conditions with slosh motions of interest.

![Fig. 3. Tank geometry and mesh details](image)

Fig. 4 shows a 2D transient simulation from 0 to 5 seconds at 2 different gravity levels. The three different series are for three different grid densities shown in Fig. 3. The domain is patched with water and free flow is induced by gravity, with surface tension (σ = 0.073 N/m) and the contact angle for water and the tank wall is set to 55 degrees. Initially water occupies the lower left quadrant of the domain while air occupies the remainder. Computational run time depends on cell density and run times increase from grid 1 to grid 3. The figures also shows the qualitative difference between gravity levels on Earth (g~9.81 m/s²) and gravity levels of 1/1,000 of Earth levels (g/1000), as might be experienced by a spacecraft coasting with a low settling thrust.

Two non-dimensional numbers that are typically used to characterize low-gravity flows are the Bond number, which is a ratio of body to surface tension forces, $\rho g R^3 / \sigma$, and the Weber number based on $V_{max}$, $W_{max}$, which is a ratio of inertial to surface tension forces, $\rho V^2 R / \sigma$ are identified for each simulation.
These numbers give a qualitative assessment of whether the flows are dominated by body forces or inertial forces relative to surface tension. This study also provides an assessment of solution dependence on grid density is important to minimize computational time, which can be significant especially under low-gravity conditions.

The bottom of Fig. 4 shows the difference between solutions at 1 and 5 seconds. The dark region is an area of no difference, and light region shows a difference near the liquid-gas interface. Grid 1 is in relatively good agreement with the finer grids even for the low gravity case. The results of this study show that for a tank on the order of 4 m diameter, relatively coarse grid densities can be used to capture the bulk features of slosh motion, and can be effectively used when details such as wave breakup are not the critical elements under investigation.

Fig. 4. Free flow of fluid at $\gamma = 1$ and $\gamma = 0.001$, three different grids, and solution grid dependence
III. Two-Dimensional Preliminary Sloss Baffle Effectiveness Studies

Studies were performed to determine the effectiveness of various baffles geometries. Fig. 5 shows an example result with 3 different baffle sizes at two different gravity levels \((g/\rho g=1, Bo=536,000\) and \((g/\rho g=0.001, Bo=536)\). Water is patched in 25% of the domain and free flow is induced by gravity at \(t=0\). The fluid is then allowed to flow until the fluid motion completely settles.

Three metrics are used to qualitatively assess the performance of the slosh baffle in suppressing slosh motion: (1) the peak height of wall wetting, (2) the peak height of bulk flow, which is defined as that on the inner side of the slosh baffle, or the height of any flow that does not pass between the small gap between the slosh baffle and the tank wall, and (3) the maximum fluid velocity in the domain, which also sets the maximum Weber number. For three-dimensional studies a fourth metric is introduced, (4) percentage of wall coverage. A methodology was developed which allowed the CFD results to be imported into analysis program to determine the peak location of any cell that contains the fluid phase at any time step.

Fig. 6 shows an example of how the CFD result is interrogated.

Fig. 7 shows the velocity vectors colored by velocity magnitude in a non-baffled tank at 1 second into the simulation. The picture also shows the direction of slosh and the velocity with which it is travelling. Such data is useful for optimizing the placement of the slosh baffle and for estimating its required structural integrity based on the momentum of the incident fluid.

Fig. 8 shows the velocity vectors colored by velocity magnitude in a baffled tank at 1 second.
The difference in slosh behavior can be seen clearly; the baffles help in mitigating the amplitude of the slosh wave and the peak location of the wall wetting. The fluid motion which tends to move towards the upper region of tank is mitigated by the baffle and directed towards the lower middle region of tank. The high velocity regions are in the middle region of the tank, which is major difference from a non-baffle case.

An important aspect of this investigation is to compare the differences laminar and turbulent flow models. Although the dynamics of low-gravity slosh take longer to develop, the resulting slosh can be just as violent as in high gravity cases, see Fig. 5. Fig. 9 shows a sequence of images, from 1 to 5 seconds, comparing the laminar and turbulent models, applied at Earth gravity conditions (g/g0=1) using grid 2.

A subtraction of the two images to highlight the differences in the solution is shown in the bottom row. Comparisons between the two models were performed and in all cases good qualitative agreement was observed. These observations are consistent with slosh results employing laminar and turbulent models discussed in [20], which are also conducted at Earth gravity. The results of this study demonstrate that similar agreement occurs between laminar and turbulent models at low-gravity (g/g0=0.001) conditions, suggesting that if bulk motion is being sought, the laminar model, which requires less computational time, is adequate.

The details of droplet breakup and free surface structure do show some differences between the two models, and for the conditions shown in Fig. 9 the maximum velocity of fluid, and thus Weber number, is higher for the laminar flow model.

### IV. Two-Dimensional Slosh Baffle Performance

Three different baffle sizes (large, medium, and small) at two different tank locations, both with and without a gap between the baffle and the tank side wall were examined in this study. The large baffle has a width of 0.5 m, the medium baffle 0.25 m, and the small baffle 0.125 m. The vertical location of the baffle is either at the middle of the cylindrical section or at the bottom at the intersection with the elliptical end cap. The gap is 0.1 m, and the baffle size does not change if a gap is introduced. Fig. 10 shows several configurations considered in this study.

---

Fig. 10 Example of various baffle geometries
Table I summarizes the performance of various baffle configurations using the metrics discussed above and shown in Fig. 6. In the table OD and ID are the outer and inner diameter of the slosh baffle, gap is the space between the baffle and tank wall, and H is the vertical location of the baffle along the cylindrical section of the tank. The large baffle geometry ensures that none of the bulk fluid rises above the centerline of the tank — or none of the bulk fluid rises above the initial condition of the fluid. This is not true for either the medium or small slosh baffles, and in these cases the bulk fluid reaches above the initial fluid height at the centerline of the tank. This is an important result because it provides criteria for the size of the slosh baffle that will ensure that the bulk of the fluid does not hit the upper dome of the tank. For the cases of no baffle, the bulk fluid reaches a height of 1.66 m, so all baffles provide some mitigation of peak slosh height, with the large size being most effective for this simple scenario. For the baffles with a gap between the baffle and tank wall, irrespective of baffle size, the fluid tends to flow to the top of the dome and forms a thin layer along the inside of the tank. Results between the laminar and turbulent models are consistent and show minimal differences. For example, for the large slosh baffle with and without a gap, the laminar results show 10-25% higher peak velocities and high peak slosh amplitude (when gap is present), but the bulk fluid height is identical. Similar results are observed for the other baffle sizes with the laminar model always exhibiting larger heights and velocities of slosh. Placing the slosh baffle in the lower portion of the cylindrical section leads to lower velocities within the tank (the fluid travels a shorter distance before encountering the baffle), however lower placement of the baffle is not as effective in minimizing the height of the bulk fluid. From this study, assuming that the slosh baffles are rigid enough to withstand the fluid momentum, the placing of the baffle in the center of the cylindrical section more effectively suppresses the bulk slosh motion.

V. Three-Dimensional, Laminar vs. Turbulent Models, and Dynamic Mesh Modeling

Another important component of this study is to determine how to extend the two-dimensional studies shown above to a true three-dimensional simulation. Fig. 11 provides an example of the 3D analog to the 2D simulation of Fig. 4. The metrics discussed above were used to assess baffle effectiveness along with percentage wall wetting for the 3D cases. Baffle configurations were identified to minimize overall percentage of wall wetting.

![Fig. 11. Example 3D tank flow inside with an annular slosh baffle, $g/B_o=1$, Bo = 536,000, $V_{max} = 437,030$](image-url)
An additional three-dimensional example is shown in Fig. 12. The tank is a cylinder, which contains 3 slosh baffles. Multiple baffle configurations have been examined but are not discussed in this paper.

![Simulation of flow inside a tank with 3 annular slosh baffles](image)

Fig. 12. Simulation of flow inside a tank with 3 annular slosh baffles, \( g/g_w = 1, Bo = 1095,793 \)

Another important aspect of this investigation was to determine grid sensitivity, baffle performance, and Bond number scaling when different fluids are used. The most important fluid to examine from a rocket propulsion standpoint is cryogenic hydrogen, LH₂, which has a density of 75.2 kg/m³ at 16 K, surface tension of 0.0026 N/m, and a wall contact angle of 2 degrees. Fig. 13 compares slosh behavior between water and LH₂, over a 5 second interval. The first row shows slosh of liquid water and the second row shows slosh of LH₂ in a tank without baffles at \( g = 9.81 \text{ m/s}^2 \), and the behavior can be seen to be qualitatively different, which is due to the different density and surface tension of the two fluids. In the third row of Fig. 13 the gravity level was set to match the Bond number of water from the first row, and as can be seen the results still show differences. It is likely that the physics of slosh in these scenarios also depends on other parameters like viscosity and wall contact angle, and simple Bond number scaling is not accurate between different fluids.

In the models developed above, there is no feedback of liquid forces on the motion of the tank; the tank is kept rigidly still. Liquid sloshing behaviors in a moving fuel tank can be simulated by applying Dynamic Mesh model. The dynamic mesh model is used to model flows where the shape of the domain is changing with time due to motion on the domain boundaries. In certain cases, the tank motion can be defined as a prescribed, e.g., the tank is driven by strong external forces such that the liquid motion feedback can be ignored. In a dynamic mesh model, this case can be simulated by defining the linear and angular velocities about the center of gravity of the tank with time, and liquid forces do not impact the tank’s trajectory. When fluid force feedback is enabled the tank’s motion is non-prescribed, and subsequent liquid motion pushed back on the tank walls. For example, in the space, the motion of the propellant contained in a vehicle’s tank may have a significant effect on the vehicle’s orientation. In the dynamic mesh model the linear and angular velocities are determined from a force balance on the tank which is calculated by 6-DOF solver. An example of a tank whose motion is impacted by internal fluid forces is shown in Fig. 14.

![Comparison between water and liquid hydrogen at similar Bond numbers](image)

Fig. 13. Comparison between water and liquid hydrogen at similar Bond numbers

As the diagram shows, a 2D box filled with half of water is moving in the free space. Initially this box has a horizontal acceleration and an external force applied to the bottom. Since there is no constrained force acting on the other surfaces, the box is free to translate and rotate in space. The box’s motion is calculated by combining the acceleration and external force of the box as well as the water motion feedback.
VI. Conclusion

This paper utilizes CFD to assess the performance of various slosh baffle configurations for spacecraft design. In low-gravity fields, surface tension and capillary action may dominate even in large booster size tanks, and simple Bond number scaling is not adequate. Key results from this investigation assessed the importance of grid resolution, laminar versus turbulent flow models, and Bond number scaling between water and liquid hydrogen during low-gravity conditions so that computation resources can be optimized. Metrics associated with maximum wall wetting and bulk flow motion were developed for characterizing slosh baffle effectiveness and guidelines for comparing various slosh scenarios. This work indicated that relatively coarse grids and laminar flow models can be used with relatively good accuracy if a qualitative assessment of bulk fluid motion during slosh events is sought at both Earth and low-gravity conditions. This result is fortuitous as computational times are reduced. This study also showed that the models can be used to help determine a baffle geometry and location within the tank that should be employed to minimize bulk fluid motion, as well as the effect of having a gap between the tank wall and the baffle. Three-dimensional studies were performed and percentage internal tank wall wetting was used as an additional metric to assess the performance of various baffle configurations.

Future work will take into account the effect of the ribbed weight-saving isogrid located on the internal surface of many modern tanks. Other analyses will also include the effects of wall heating from solar radiation, as well as the effect of tank rotation about its longitudinal axis for thermal conditioning. Rotation at low gravity conditions caused the bulk fluid inside the tank to assume a parabolic free-surface shape. Wall heating is critical because when cold propellant comes in contact with a hot surface, boil off occurs tank pressure is increased and venting of useful propellant occurs.

References


Author's information

Srinivas Chintalapati, BS Aerospace Engineering, Florida Institute of Technology, Melbourne, Florida 32901, USA. Current research interests include Tank dynamics, isogrid, slosh, slosh baffle, CFD. Mr. Chintalapati is affiliated with the AIAA and ASME.

Dr. Daniel R. Kirk is an Assistant Professor of Aerospace Engineering at Florida Tech. Dr. Kirk received his PhD from the Massachusetts Institute of Technology in Aeronautics and Astronautics in 2002. Dr. Kirk’s research interests include air-breathing and rocket propulsion, modeling of thermal behavior of liquid rocket propellants, micro-rockets and transient shock tunnel experimental techniques. Dr. Kirk is a member Senior Member of the AIAA and ASME. He is a recent recipient of the SAe Ralph A. Teetor Award.

International Review of Aerospace Engineering, Vol. 1, N. 2