

Investigation and Development of Fuel Slosh CAE Methodologies

Dhaval Vaishnav, Mike Dong, Mayur Shah, Francisco Gomez, and Mohammad Usman
Ford Motor Co.

ABSTRACT

When a vehicle with a partially filled fuel tank undergoes sudden acceleration, braking, turning or pitching motion, fuel sloshing is experienced. It is important to establish a CAE methodology to accurately predict slosh phenomenon. Fuel slosh can lead to many failure modes such as noise, erroneous fuel indication, irregular fuel supply at low fuel level and durability issues caused by high impact forces on tank surface and internal parts. This paper summarizes activities carried out by the fuel system team at Ford Motor Company to develop and validate such CAE methodology. In particular two methods are discussed here. The first method is Volume Of Fluid (VOF) based incompressible multiphase Eulerian transient CAE method. The CFD solvers used here are Star CD and Star CCM+. The second method incorporates Fluid-Structure interaction (FSI) using Arbitrary Lagrangian-Eulerian (ALE) formulation. While Eulerian domain predicts motion and forces of fluid inside the tank, Lagrangian domain models tank shell and predicts its vibration under these forces. Solver used here is LS-DYNA. While details of second method are covered in another publication [1] by co-author of this paper, Dr. Usman, first method is focused on in greater detail due to its successful integration in to design verification (DV) process. Many engineering quantities such as tank surface pressure vs. time plot, surface integral force, momentum, mean kinetic energy, free surface shape etc. are monitored. However, surface pressure time history, surface integral force and free surface shape are found to be the most useful information to characterize slosh behavior and correlate with the test. Simulation results are compared with bench test. Issues related to effect of pressure reference location, wall Y+, courant number, mesh size and inner iteration are also discussed. Current CFD method is fully developed and integrated in the product design process to evaluate tank and baffle design early in the process to avoid costly tool changes or extensive testing. This method can be combined with other analytical tools to further include effect of aero/fluid acoustic and vehicle architecture.

CITATION: Vaishnav, D., Dong, M., Shah, M., Gomez, F. et al., "Investigation and Development of Fuel Slosh CAE Methodologies," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(1):2014, doi:10.4271/2014-01-1632.

1. INTRODUCTION

The sloshing phenomenon is highly non-linear movement of the free-surface of liquid inside the tank under the effect of continuous or momentarily excitation forces. For automobile tanks, this excitation force can result from sudden acceleration, braking, sharp turning or pitching motions. Due to liquid inertia, fuel waves are generated inside tank. Such waves can create high pressure gradient on tank surface and cause tank vibration. This phenomenon can produce audible noise known as slosh noise. Literature [2, 3] describes three mechanism of fuel slosh. First mechanism is 'hit' which is described as liquid waves hitting the tank surface and creating high local pressure gradient due to its momentum. This phenomenon creates maximum pressure gradient on tank surface and is dominant for long prismatic tanks commonly found in heavy duty vehicles. Second mechanism is 'Splash' which is described as two liquid waves colliding with each other. This can entrain some air with it below liquid free-surface. This air again escapes from liquid in the form of small bubbles. This is more commonly seen in L-shaped tanks of passenger cars. Third

mechanism is 'Clonk' which is described as noise produced by air or vapor compression. Liquid waves can entrap and compress air in sharp corners or a vapor dome. This compression can also contribute to tank skin vibration and cause slosh noise.

Noise generated inside tank is transmitted to passenger compartment as air borne and structural borne noise. While air borne noise is transmitted through air, structural borne noise can be transmitted via support straps, floor pan, cross-members and frame rails. Depending on vehicle configuration, contribution of each component may vary significantly. As per tests conducted at Ford, for unibody configurations structural borne component can be as high as 70%. However, for body-on-frame design, this contribution may be relatively small because of presence of flexible mounts between body and frame. Human's sensitivity to slosh noise further depends on other surrounding masking noise such as engine noise, transmission noise, wind noise and tire noise. When a vehicle is operated at very low speed, all of these masking noises are absent. Hence, slosh noise is noticed by customer very easily.

for low speed driving such as approaching a stop light or coming out of drive-way. Moreover, in vehicles equipped with hybrid and start-stop technologies, engine and other auxiliary systems stop running when vehicle stops at signal or stop-sign. Hence, it becomes critical to reduce slosh noise.

Qualifying a fuel tank for slosh noise performance based on testing is a time consuming and costly process. Also, such tests only provide pass/fail decision. They provide very limited insight about slosh characteristics and baffle design. Moreover, tests need availability of all components which is often very late in design cycle. At this stage, changing blow molding tool is usually not feasible. For this reason, it was highly desired to establish a CAE methodology for slosh noise prediction. It was also of interest to develop a component level (in this case, fuel tank) CAE method instead of system level method. In this way CAE can be used early in the design process when full vehicle CAD is not available and also tank suppliers can be tasked to conduct this analysis. This paper describes successful implementation of the CAE method into product design cycle.

2. LITERATURE STUDY

Several analytical and experimental approaches can be found in literature to characterize automobile tank slosh noise issue; a brief overview of which is provided next.

Our team members Jean Ma and Mohammad Usman developed CAE methodology [1] using Arbitrary Lagrangian-Eulerian (ALE) formulation. Eulerian domain includes both air and fuel inside the tank and the space around the tank. Lagrangian domain includes the tank shell and baffle structures. Correlation between test and CAE results has been demonstrated.

Kamiya and Yamaguchi [4] from Hori Metal Co. also describe fluid-structure interaction (FSI) approach using ALE formulation within MSC.Dytran environment. MSC Software Corporation has further developed this capability with fully coupled co-simulation of MSC.Dytran (an explicit CFD code) with an MSC.ADAMS (an implicit mechanical systems simulation code). Particular interest is paid to baffled fluid tanks mounted on heavy trucks [5].

Another approach is to include aeroacoustic along with FSI. The motivation behind including acoustic is ability to predict sound level in more familiar units such as sound pressure level (SPL, db). Fan Li [6] demonstrates combining of CFD, FE and aeroacoustic simulation methods to predict radiated fuel tank slosh noise. The predicted SPL values with de-coupled fluid-structure interaction (FSI) method have been validated with test results.

However, it should be noted that Including FSI and aeroacoustic increases computational cost. In order to make CAE method efficient, many researchers have tried modeling only fluid domain using eulerian approach with Volume Of Fluid (VOF) or Marker and Cell (MAC) algorithm. In general, VOF

method is found to be more commonly used compared to MAC due to its efficiency. Hoi Sum, W. L. Cleghorn and J.K. Mills [7] present VOF approach to study different baffle concepts using a commercial CFD software FLOW-3D. Their simulation results showed that the best baffle configuration can reduce kinetic energy as much as 70% compared to the no-baffle configuration. At high fluid level, the best baffle configuration can reduce the sound level by approximately 15 decibels. Hyundai-Kia Motor Company analyzes Noise from fuel tanks by Marker and Cell (MAC) algorithm. Their simulation result of gross sloshing flow pattern is verified by experiments with transparent fuel tanks [8]. Luigi Di Matteo and Francesco Fortunato use VOF method but combine it with moving mesh capability to use tank oscillations as driving mechanism instead of longitudinal or lateral accelerations [9]. Their research aims at closely reproducing road bends in a bench test using triangular and sinusoidal variation in acceleration.

It is also important to study effects of vehicle architecture and noise transfer path to study slosh noise completely and develop mitigation techniques other than baffles. However, capturing all such effects in CAE will be resource intensive. As a result, significant work has been done to better understand these factors using bench and vehicle level tests. Mahesh Balthi and Chetan Prakash [10] has presented detail investigation on relation between fill level and slosh noise with focus on experiments. However, CAE results (Eulerian VOF approach) were used to decide positions of sensors (accelerometers and microphones) to ensure that slosh is measured at most severe conditions.

C.Wachowski [3] presents a comprehensive investigation of noise source location on the tank surface using array of microphones and acoustic camera in bench test. CFD simulations (Eulerian VOF approach) are performed in parallel, to gain a deeper understanding of the slosh waves. Effect of tank structure is analyzed in isolated fashion. Tank is excited at single point using electro-dynamic shaker at specified frequency and its response at other locations on surface is measured. Masashi Kamei and Junichi Hanai [11] also take experimental approach to establish relation between slosh noise and design factors related with tank shape, tank mounting and vehicle architecture.

Pierre De Man [12] conducted an experimental validation of the source-path-receiver approach for slosh noise to establish relation between component and vehicle level performance. Motivation behind this investigation is to define component level slosh noise specification from known vehicle level target. The study validates source-path-receiver model.

Prof. Raouf A. Ibrahim [13] has systematically summarized almost every aspect of liquid sloshing dynamics. His book covers theory of linear slosh dynamics, non-linear slosh dynamics, and interaction of sloshing with elastic container as well as sloshing in microgravity. Another valuable resource for slosh dynamic theory is a monograph NASA SP-106 edited by Norman Abranson [14].

3. CAE METHODOLOGIES FOR SLOSH SIMULATION

Two approaches have been pursued by the Fuel System team at Ford Motor Company to model fuel tank sloshing. The first method is Volume Of Fluid (VOF) based Eulerian CAE method using Star CCM+. The second method incorporates fluid-structure interaction (FSI) with Arbitrary Lagrangian-Eulerian (ALE) formulation in LS-DYNA. While good correlations have been demonstrated for both methods, choice of the preferred method is many times governed by available resources and required turn-around time. ALE method would facilitate inclusion of aero-acoustic effects but it is also relatively time intensive, has slow turn-around time and requires structural as well as fluid dynamic expertise. This makes VOF method more widely used. Consequently, this paper focuses on Eulerian VOF method. However before dwelling into details of this method, a brief overview of ALE method is provided below.

3.1. Slosh Modeling with Arbitrary Lagrangian-Eulerian (ALE) Formulation [1]

ALE method is a fluid-structure interaction (FSI) method that uses Eulerian domain to model fluids (liquid and air) and Lagrangian domain to model tank structure.

Lagrangian algorithms are mainly used in structure mechanics in which each node of the computational mesh follows the associated material particle during motion and also allows an easy tracking of free surfaces and interfaces between different materials. However, for large deformations such as folding and braking of liquid free surface, Lagrangian algorithm is less effective due to the excessive distortions of the computational mesh. For fluid modeling, Eulerian algorithm is better suited. Here mesh is fixed and the continuum moves and deforms with respect to the grid, this formulation does not capture the effects of the relative motion between the deforming material and the fluid mesh.

In current FSI method, Lagrangian and Eulerian algorithms are coupled using Arbitrary Lagrangian-Eulerian (ALE) formulation available in LS-DYNA. In ALE formulation, Eulerian domain predicts fuel slosh and forces on tank surface and Lagrangian domain then predicts displacement and deformation of the tank structure.

In ALE formulation nodes of the computational mesh may be moved with the continuum in normal Lagrangian fashion, or be held fixed in Eulerian manner, or be moved in some arbitrarily specified way to give a continuous rezoning capability. Because of this freedom, greater distortions of the continuum can be handled than would be allowed by a purely Lagrangian method, with more resolution than that afforded by a purely Eulerian approach [21] (Figure 1).

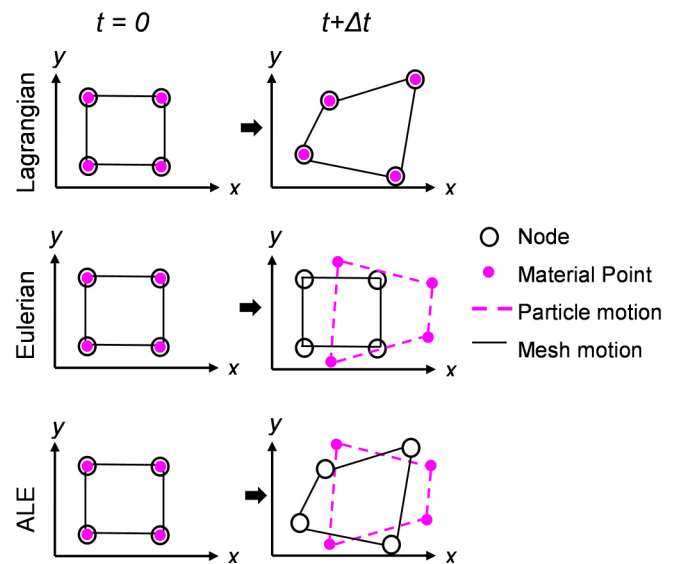


Figure 1. One-dimensional example of Lagrangian, Eulerian and ALE mesh and particle motion [21]

In this study, the Eulerian domain for air and fuel portions was discretized by solid elements and the structure of the tank which defines the Lagrangian domain, is discretized by shell elements (Figure 2).

This research was able to validate that ALE formulation in LSDYNA can model the coupled FSI problem such as liquid sloshing in fuel tank with reasonable accuracy. Validation was established by comparing prediction of waves with physical sloshing test. In the test, a section of tank material was replaced by a transparent window and liquid motion was recorded by camera.

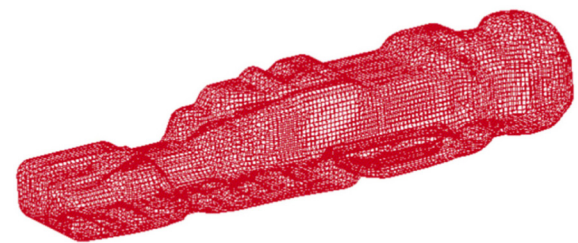


Figure 2a. Lagrangian domain (coupling surface)



Figure 2b. Eulerian domain (fuel and air domain)

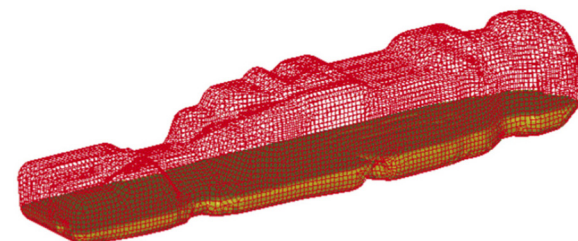


Figure 2c. Final model (coupling surface)

Figure 2. CAE model set up for ALE formulation

Next section details slosh modeling using Eulerian VOF formulation which is the focus of this paper.

3.2. Slosh Modeling with Eulerian VOF Formulation

This approach uses multiphase eulerian Volume Of Fluid (VOF) analysis with incompressible fluids. Analysis is done using Star CCM+. K-E with 'all Y+' wall treatment is used to model turbulent flow. Transient behavior is modeled with implicit unsteady solver. 'No-slip' wall boundary condition is specified for all solid surfaces. The typical fuel tank for a midsize passenger car is used in this study (Fig. 3). Internal surface of the tank along with all internal parts are considered in the analysis. For volume meshing, trimmer mesh is preferred over polyhedral or tetrahedral meshes for such free surface flows since it gives better cell quality with smaller cell count. Distribution of eulerian phases (fuel and air) are specified for time $t=0$ s (Fig. 4). Such analysis is typically run at different fill-levels since slosh noise strongly depends on fill-level. Furthermore, g-load or acceleration is specified as a boundary condition. Fig. 5 shows a typical g-load profile for the analysis. The g-load shown is applied along longitudinal axis and is calculated based on a vehicle traveling at 2-3 mph and applying sudden brake (over a time of 0.5 s). In addition to longitudinal and/or lateral g-load, -1 g (-9.81 m/s²) is applied in vertical direction to replicate earth's gravity. Analysis is run for 6-8 s after applying g-load to capture at least first four waves. Analysis is run with water to do correlation study with bench test in which water was used as working fluid. Based on the studies [15, 16], it has been found that density and viscosity are the important fluid properties for slosh problem. Higher density fluid gives higher impact pressure and higher viscosity causes faster wave decay. Since water's density is higher than commonly used fuels (diesel, gasoline), it represents worst case scenario. Solution time-step and inner iterations are carefully chosen based on sensitivity studies and courant number considerations.

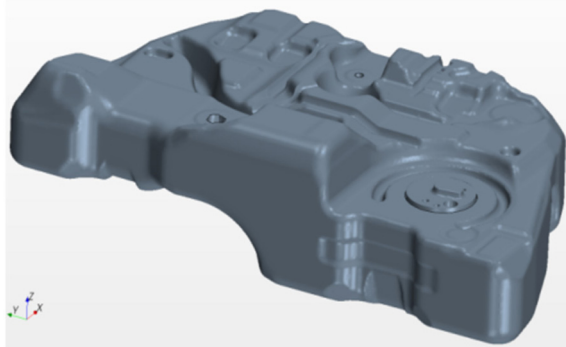


Figure 3. Fuel Tank

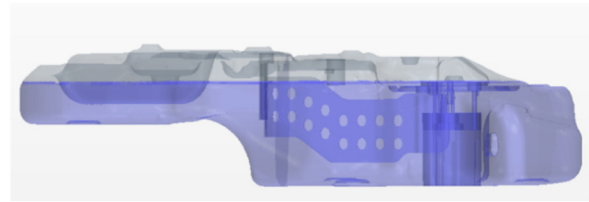


Figure 4. Initial distribution of fuel and air

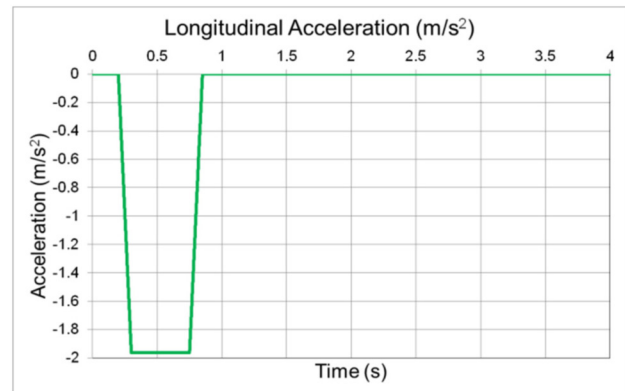


Figure 5. Longitudinal acceleration (m/s²) for sudden braking at low speed driving

3.2.1. Volume Of Fluid (VOF) formulation

Volume Of Fluid (VOF) is an efficient multiphase method suited for simulating flows of immiscible fluids [17]. Here, each phase constitutes a large volume with a relatively small contact area (free surface) between them. Free surface is considered to be surfaces on which discontinuities exist in dependent variables. There are three main problems involved with free surface modeling such as fuel slosh (1) clearly defining shape and location of free surface (2) its evolution with time and (3) defining boundary conditions on the free surface. While detail description of formulation is covered under other technical papers [18, 19], brief discussion of how each difficulties are addressed is provided here. In multiphase VOF formulation, a function F is defined such that its value is unity for a cell fully occupied by heavier fluid (fuel) and zero for a cell fully occupied by lighter fluid (air). For each cell, the volume fraction (F) is defined as,

$$F = \frac{V_l}{V_t} \quad (1)$$

Where V_l and V_t are volume of liquid in the cell and total volume of cell respectively. In that case, mesh cells with VOF value between zero and one must contain free surface. A free surface cell is defined as a cell containing a nonzero value of F and having at least one neighboring cell with non-zero value of F . This is the primary reason why VOF method cannot capture air bubbles with diameter less than a cell size. Once cells with free surface are identified, free surface shape can be defined. In the VOF method it is assumed that the free surface can be approximated by a straight line cutting through the cell. This indicates importance of having small enough mesh so that