6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11-15 June 2018, Glasgow, UK

A PARTITIONED FSI METHODOLOGY FOR ANALYSIS OF SLOSHING-INDUCED LOADS ON A FUEL TANK STRUCTURE

SAMPANN ARORA, SUDHARSAN VASUDEVAN, SRDJAN SASIC AND SASSAN ETEMAD

Department of Mechanics and Maritime Sciences
Chalmers University of Technology, 412 58 Gothenburg, Sweden
sampann@alumni.chalmers.se, sudvas@chalmers.se, sassan@chalmers.se, srdjan@chalmers.se

Key words: Fuel tanks, Partitioned Fluid Structure Interaction (FSI), Sloshing, Volume of Fluid (VOF)

Abstract. Liquid sloshing is a source of major concern in the structural design of containers. In fuel tanks of heavy duty trucks, with capacities of up to 900 litres, this phenomenon is capable of causing fuel to impact the container tank with high forces, and exposing the vulnerable parts of the tank to heavy dynamic loads. This highly non-linear and transient phenomenon is simulated here using the commercial Computational Fluid Dynamics (CFD) code STAR-CCM+. The two phase problem is solved using the VOF interface capturing approach. Owing to the thin walled structures of the fuel tank, it becomes important to account for the effects of Fluid-Structure Interaction (FSI). To this end, a partitioned FSI methodology is employed by coupling the CFD and Finite Element Analysis (FEA) solvers for this multi-physics problem. One-way and two-way coupled FSI methodologies are compared with experimental results. The one-way coupled simulations yield good agreement of wall deformations with the experiments for low filling levels. While the two-way coupled FSI analysis corroborates well with the experiments for all filling levels, its high computational costs render the one-way coupled methodology a promising tool to analyse sloshing for industrial applications. This coupling strategy could inform a fuel tank design suited to prevent structural damage due to sloshing, thus contributing towards its safety and longevity.

1 INTRODUCTION

Numerical analysis of liquid sloshing is challenging due to complex underlying physics of the process. Simulations of these problems involve an accurate modelling of the two-phase flow problem, as well as accounting for the impacts due to sloshing on the solid structure through coupled fluid-solid interactions [1]. Such simulations have found their place in a variety of industrial applications, for instance, in the directional stability of liquid tank vehicles [2], pressure vessels built to withstand loads due to seismic excitation [3], and fuel tanks in the maritime, aerospace and automotive industries [4]. These simulations can help obtain critical information on the structure and design of fuel tanks at early stages of product design and development. The appearance of novel materials, such as composites and industrial plastics and aimed at weight reduction, has made such analyses an absolute necessity.

The effects of the dynamic sloshing impacts on the fuel tank structures have been analysed quite extensively through various numerical and experimental methods. Investigations show that sloshing loads depend on the frequency of the excitation signals and these loads become maximum when the excitation frequency is close to the natural frequency of the liquid in the tank [5]. The effect of baffles on reduction of sloshing is a significant study within this field. Such studies have been performed on cuboidal tank geometries through experimental and numerical investigations [6, 7]. Simulations on cuboidal tanks with elastic baffles have been the benchmark case for studies involving coupled fluid-solid interaction methodologies [8]. These representative test cases on simplified geometries provide validation data for model formulations using different numerical and analytical techniques.

This work presents a comparison of results from both numerical and experimental studies for the analysis of sloshing-induced stresses on the fuel tank of a heavy-duty truck. The simulation methodology is based on the Volume of Fluid (VOF) model and partitioned fluid structure interaction (FSI). One-way and two-way coupled FSI simulations are carried out and their efficacy based on comparison with experiments and computational costs is analysed. The simulations are performed using the commercial code - STAR-CCM+. Owing to the fact that the experiments were carried out with water, the numerical study is also performed using water as the liquid phase.

2 MODELLING FLUID USING VOF METHOD

The Volume of Fluid (VOF) method is used in analysing immiscible fluids with distinctly defined interface. This model implicitly captures the interface and hence is very efficient in modelling flow problems where the shape of the interface is of interest. The fractional amount of fluid (volume fraction) in a particular cell, that acts as a phase indicator function, is employed. The volume fraction of the i^{th} phase in a cell within a multiphase domain is given by

$$\alpha_i = \frac{V_i}{V} \tag{1}$$

Where, V_i is the volume of the i^{th} phase in the cell and V is the volume of the cell.The model is based on a single fluid formulation with the density and viscosity of the fluid defined in terms α_i.

$$\rho = \alpha_i \rho_i \tag{2}$$

$$\mu = \alpha_i \mu_i \tag{3}$$

In a domain with two phases, the phase indicator function α_1 is defined as, $\alpha_1 = 1$ if the cell is filled with phase-1, $\alpha_1 = 0$ if the cell is filled with phase-2 and $0 < \alpha_1 < 1$ if the cell contains the interface.

In addition to equations for conservation of mass and momentum, evolution of the interface is governed by

$$\frac{\partial \alpha_1}{\partial t} + \frac{\partial(\alpha_1 v_i)}{\partial x_i} + \frac{\partial(v_{r,i}\alpha_1(1-\alpha_1))}{\partial x_i} = s_{\alpha_1} \tag{4}$$

The first two terms on the left hand side of Equation 4 are the unsteady and convection terms, respectively. The third term on the left hand side is known as the surface compression term that contributes only at the interface ($0 < \alpha_1 < 1$). The term on the right hand side is a source term.

The SIMPLE algorithm, in combination with a collocated variable arrangement and Rhie-Chow interpolation, is used to solve the pressure-velocity coupling. The discretization scheme used for the convection term in Equation 4 has a significant impact on the accuracy of the results. A blend of the second order accurate High Resolution Interface Capturing (HRIC) scheme and the first order accurate Upwind Differencing (UD) scheme is used [10]. The realizable $k-\epsilon$ model is used for modelling the turbulence.

The transport equation for the phase indicator function determines the fractional volume of the fluids in the control volume, which is reflected in the Navier-stokes equation through the equivalent density and viscosity. The velocity obtained from the Navier-Stokes and continuity equations is reflected in the convective term of the transport equation for the phase indicator function. Thus, the three governing equations are solved together to capture the interface implicitly.

3 FLUID-STRUCTURE INTERACTION

Problems involving FSI can be solved through a monolithic or a partitioned approach. Governing equations for the fluid and structure domains are solved simultaneously in the monolithic approach within one system of equations [11]. In the partitioned approach, the fluid and solid domains are solved for separately in their respective solvers, and the exchange of field variables across the domains takes place through an interface. Thus, the coupled problem can be solved by means of separate grids, dedicated solvers and algorithms for each domain.The finite element (FE) formulation of the conservation equation for linear momentum in the solid domain is given by

$$\mathbf{M}\ddot{\boldsymbol{u}} + \mathbf{C}\dot{\mathbf{u}} = \mathbf{f_{ext}} - \mathbf{f_{int}} \tag{5}$$

where $\mathbf{M}$ is the mass matrix, $\mathbf{u}$ is nodal displacement vector and $\mathbf{C}$ is the damping matrix. $\mathbf{f_{ext}}$ denotes the externally applied loads and the stiffness matrix $\mathbf{K}$ is related to the internal force $\mathbf{f_{int}}$ through the displacement term as

$$\mathbf{K} = \frac{\partial \mathbf{f_{int}}}{\partial \mathbf{u}} \tag{6}$$

A second-order accurate Newmark method is used for expressing the velocity and acceleration in terms of displacement, which is in turn sought for as the solution of this dynamic problem. The velocity and displacement at time step n are given by

$$\mathbf{\dot{u}^n} = \mathbf{\dot{u}^{n-1}} + (\delta \boldsymbol{\ddot{u}^n} + (1-\delta)\boldsymbol{\ddot{u}^{n-1}})\Delta t \tag{7}$$

$$\mathbf{u^n} = \mathbf{u^{n-1}} + \mathbf{\dot{u}^{n-1}}\Delta t + \big(\gamma \boldsymbol{\ddot{u}^n} + (\frac{1}{2} - \gamma)\boldsymbol{\ddot{u}^{n-1}}\big)\Delta t^2 \tag{8}$$

where δ and γ take values 0.5 and 0.25, respectively.

In addition to the governing equations, the Dirichlet and Neumann boundary conditions are specified at the fluid-solid interface Γ_s to maintain the no-slip condition

$$v_i^s = v_i^f \quad on \quad \Gamma_s, \tag{9}$$

$$\sigma_{ij}^s n_i = \sigma_{ij}^f n_i \quad on \quad \Gamma_s, \tag{10}$$

The fluid and solid domains are meshed with different topologies in partitioned FSI. The pressure and wall shear stress from the fluid side of the interface are mapped as forces and moments at the solid side of the interface, which act as the loads on the deformable solid domain. The displacement is mapped from the solid side to the fluid side, altering the flow field by modifying its volume mesh. Data mapping with interpolation schemes that are accurate, continuous, bounded and conservative are preferred. This modification is performed by a mesh morpher in STAR-CCM+ [10]. Since the solid domain is solved in the Lagrangian formulation and the fluid domain in the Eulerian, an efficient way of accounting for the effect of the solid displacement on the fluid domain is required. This is done by using grid velocities in the fluid transport equations arising from the solid displacements through a technique known as the Arbitrary Eulerian Lagrangian (ALE) formulation [12].

The above description pertains to a *two-way coupled* FSI methodology, wherein the effects of both the domains are accounted for. In the *one-way coupled* FSI methodology, the fluid field variables are supplied as boundary loads on the solid domain but the effect of the resultant displacements from the solid solver on fluid domain is disregarded. Thus, there is no mesh morphing involved. This holds correct when the solid deformations are too small to affect any changes in the flow field, for example, in cases of high solid stiffness and large solid-fluid density ratios.

4 NUMERICAL MODELLING AND EXPERIMENTAL SETUP

4.1 Benchmark test for the VOF method

The experimental results from a simplified tank geometry are chosen for this validation based on one of the test cases from the work of Rhee [13]. The dimensions of the cuboidal tank are 1200 *mm*, 600 *mm*, 300 *mm* in the x, y and z directions, respectively. 20% of the tank volume is filled with water and is subjected to a sinusoidal translation motion with an amplitude of 0.06 *m* and time period of 1.94 *s* along the x-direction. Pressure is monitored at three locations on the bounding walls of the tank, as indicated in Figure 1.

In the simulation setup, the convection terms are discretized using the second order upwind scheme, the diffusion terms using second order central differencing scheme and the temporal term using second order implicit scheme. The time step is chosen as 0.001 s and the simulation is run for 12 s, corresponding to 5 sloshing impacts on the right wall. A finite volume mesh with hexahedral cells was chosen for the simulations. With this time step and mesh configuration, an average Courant number, in both the interface and the entire domain, is always less than 0.5, ensuring the predominant implementation of the second order HRIC scheme. For simulating the liquid motion, the accelerations resulting from the sinusoidal motion of the tank is provided as a momentum source term.

Figure 1 shows the comparison of the sloshing pressures on the tank walls at one of the three probes, namely probe P3. It is to be noted that similar corroboration with experimental results were obtained at other two probe locations. The sharp spikes observed at certain sloshing impacts in the pressure plots occur for a very small duration of 0.01 s, corresponding to 0.5 per cent of the excitation time period. These extremely high frequency pressure signals are purely numerical and are considered noise. Such spikes are also observed in similar studies dealing with sloshing analysis [8, 15, 16]. Moving average filters which retain the overall dynamic response are therefore used to obtain physically-relevant continuous signals, as illustrated in Figure 1.

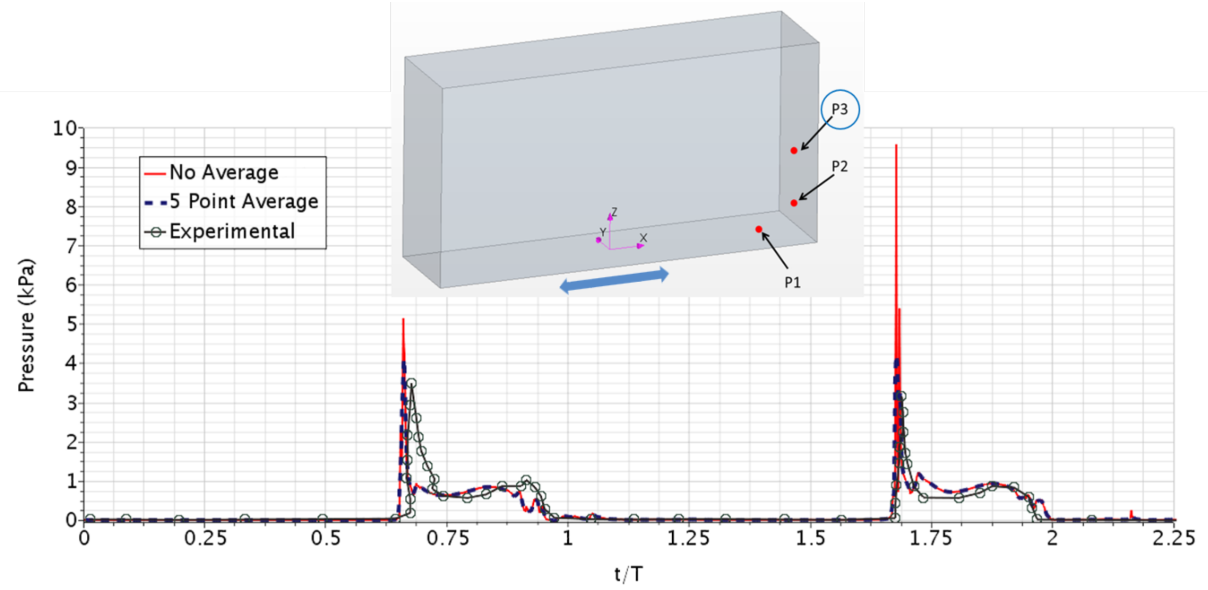


Figure 1: Comparison of pressure signals with experimental and filtered pressure signals.

The model parameters from this validation are carried forward to the coupled CFD-FEA methodology in all following simulations.

4.2 Benchmark test for FSI solver

The coupled analysis method is validated with numerical and experimental results from the benchmark studies on flexible baffles subjected to sloshing loads by Idelsohn et al[14]. These experiments were conducted on a cuboidal tank with clamped elastic beam (made of dielectric polyurethane), filled with sunflower oil. One of the three test cases of the aforementioned work is chosen for this validation study, where the oil is filled in the tank up to the height of the beam. The dimensions of the tank are 609 mm, 344.5 mm and 39 mm in the x, y and z directions, respectively. The flexible beam clamped to the bottom of the tank has dimensions of 4 mm, 33.2 mm and 57.4 mm in x, y and z directions. The tank is subjected to sinusoidal oscillatory motion about the y-axis, with an amplitude of 4 degrees and frequency of 0.61 Hz.

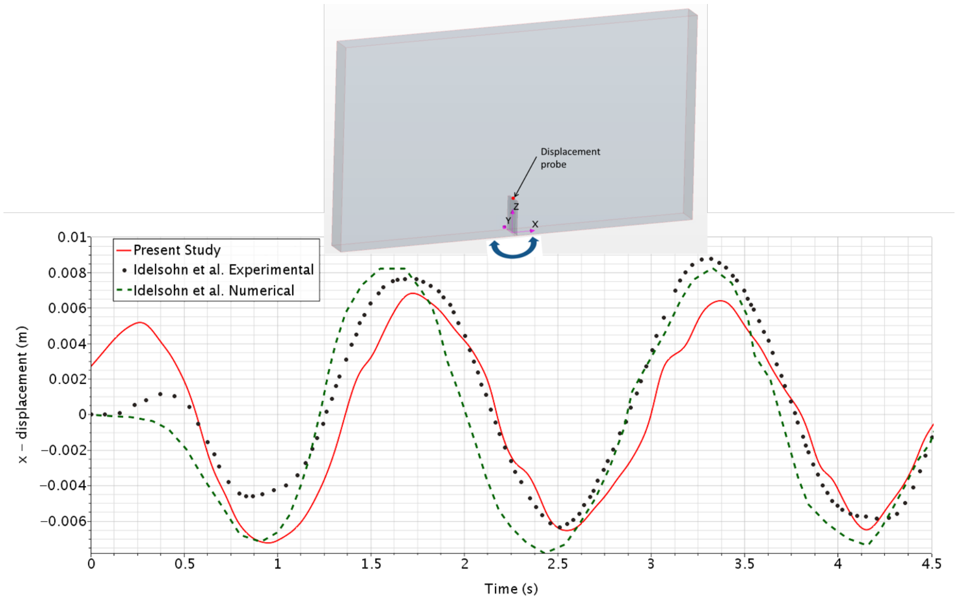


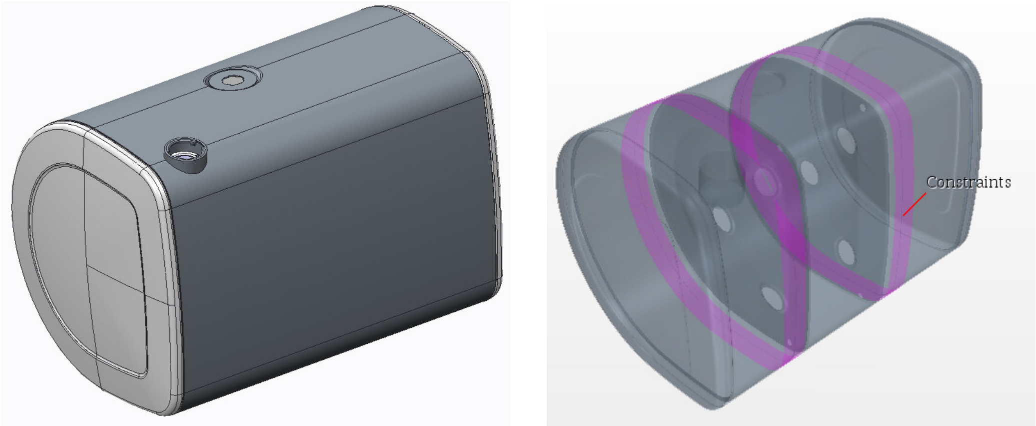
Figure 2: Comparison of displacements obtained in the present study with experimental and numerical results in [14]

The simulations are performed using the FSI module within STAR-CCM+. Both solid and fluid domains are solved using the second order accurate implicit unsteady solver. The solid domain is solved using the linear elastic isotropic solid stress model. It is meshed using linear hexahedral elements with 8 nodes, with its bottom constrained to prevent rigid body motions. The displacement field of the solid region is mapped at the fluid-solid interface; the fluid domain is then morphed in accordance with this displacement. The mesh morphing is performed at every inner iteration, since it is a strongly coupled problem. A common time step of 0.001 s for the fluid and solid domains is used in solving the problem.

Figure 2 shows the displacements at the tip of the baffle obtained in the present study compared with those in Idelsohn et al. It is seen that the results compare fairly well with the existing experimental and numerical data. It is to be noted that there is an initial 0.3 s mismatch in the excitation signal of the current study and that in Idelsohn et al. The displacement plots are therefore appropriately offset and the initial deviation is attributed to this offset.

4.3 Simulation methodology for the fuel tank of a heavy-duty truck

The geometry of a D-shaped fuel tank of heavy duty trucks, considered in the present work is shown in Figure 3. The tank is 1.03 *m* long and 0.71 *m* high. The walls and baffles are made of 2.5 *mm* thick aluminum plates. It is supported with belts that wrap around the welds close to the baffles. For the simulations, these are taken as constraints (see Figure 3(b)), to prevent any rigid body motion.



(a) D-shaped tank structure

(b) Model showing the baffles and highlighting the constrained surfaces

Figure 3: CAD model of the fuel tank of heavy-duty trucks

Polyhedral cells are used for volume meshing of the fluid domain and the solid region is meshed with tetrahedral elements (see [17] for further details). The model setup for the fluid and solid domains is derived from the previously described validation studies. A mesh sensitivity analysis yields 8.28 million cells for the fluid domain and 0.56 million cells in the solid domain. A time step of 0.0005 s is used to ensure the implementation of the 2nd order accurate HRIC scheme. Analyses for varying fill levels (20%, 40%, 60% and 80% by volume) are conducted with one-way and two-way coupled FSI simulations.

4.4 Experimental Setup

Liquid sloshing is simulated with the input acceleration signals from the test track runs. The acceleration signals are used, corresponding to each fill level from one of the sensors positioned at the end plates of the tank. These signals are recorded during the test track lap runs through obstacles of varying topology. The same signals are replicated at the shake-tests. The test rig consists of actuating pistons driven by the acceleration signals recorded at the test track as shown in Figure 4. There are two acceleration sensors on the front and rear of tank and four strain gauges – three on the front end wall and one on the rear. Figure 5 shows the acceleration signals used for the analyses of 40% filled tank. These correspond to the most severe (low frequency and high amplitude) excitation signals extracted from the entire test track acceleration data.

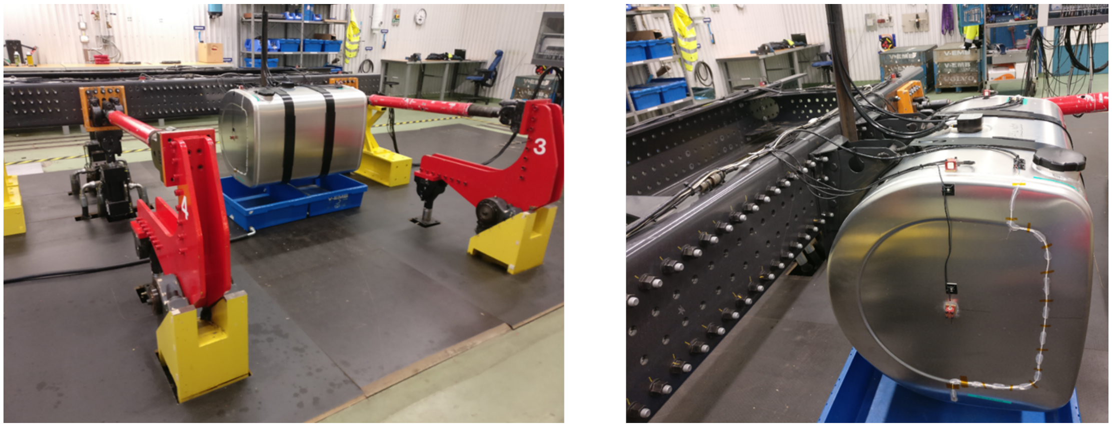


Figure 4: Test rig for experimental investigation of fuel tanks

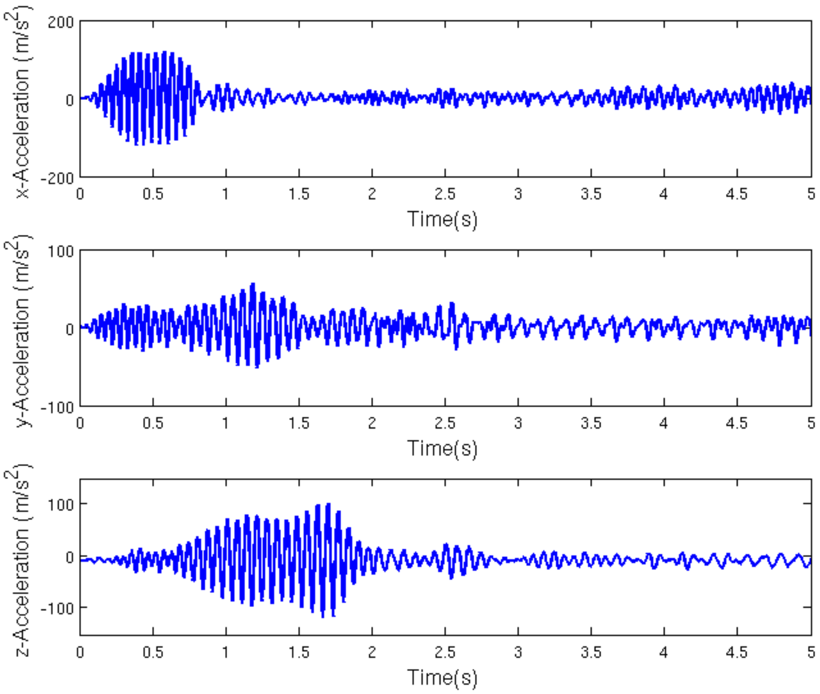


Figure 5: Input acceleration signal for a 40% filled tank

5 RESULTS AND DISCUSSION

Strain data obtained from the experiments and simulations are used for comparison of the *one-way* and *two-way* coupled FSI methodologies. Figure 6 shows the percentage deviation of the average impact strains of the simulations from the experiments for the one-way coupled FSI simulations. It is observed that, except for strain gauge $Stg2$, the difference between the experiments and the simulations increases with higher fill levels.

This is due to the fact that the one-way coupled method is unable to accurately predict the wall displacements for higher fill levels since the impact stresses are proportional to the fill levels.

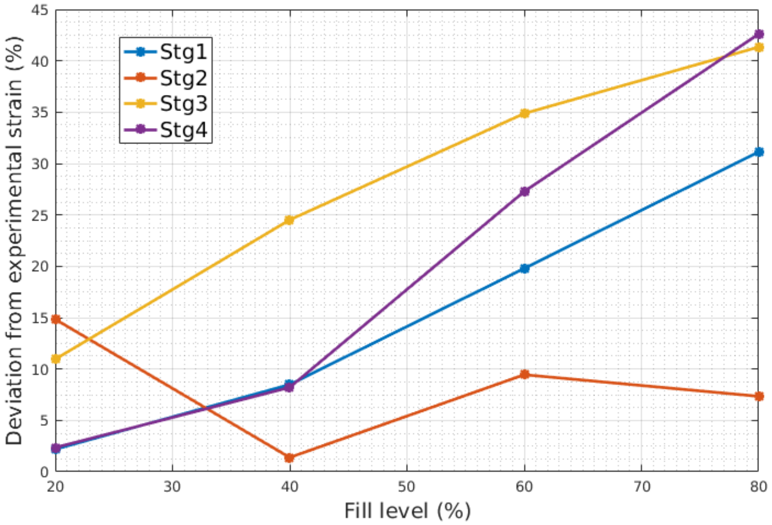
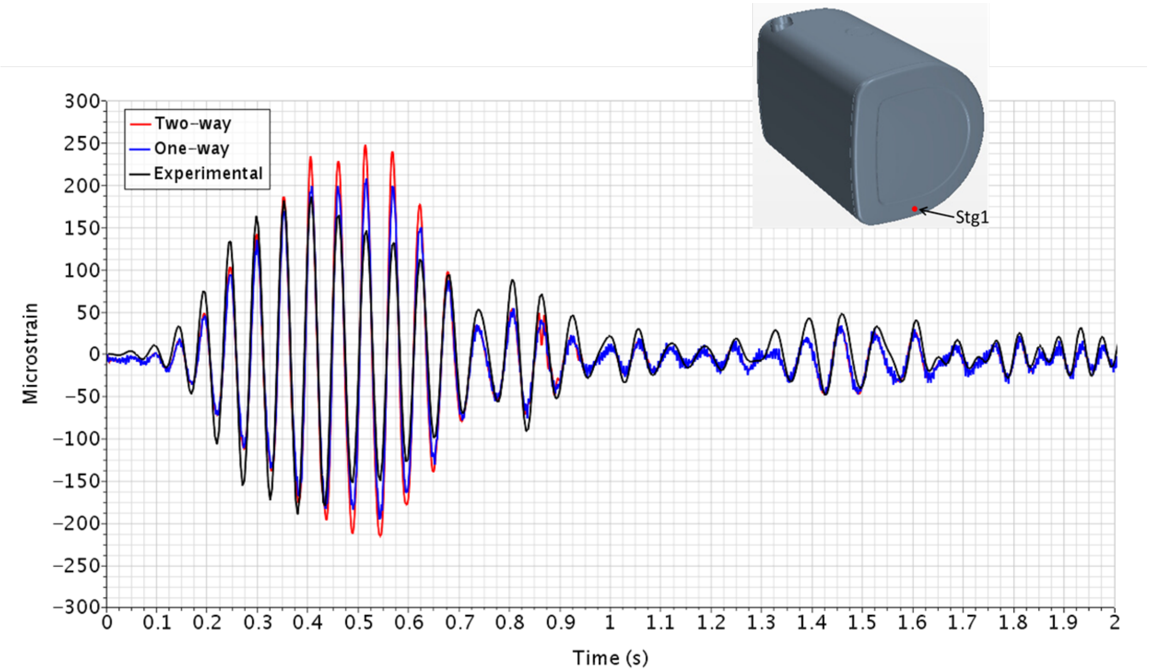
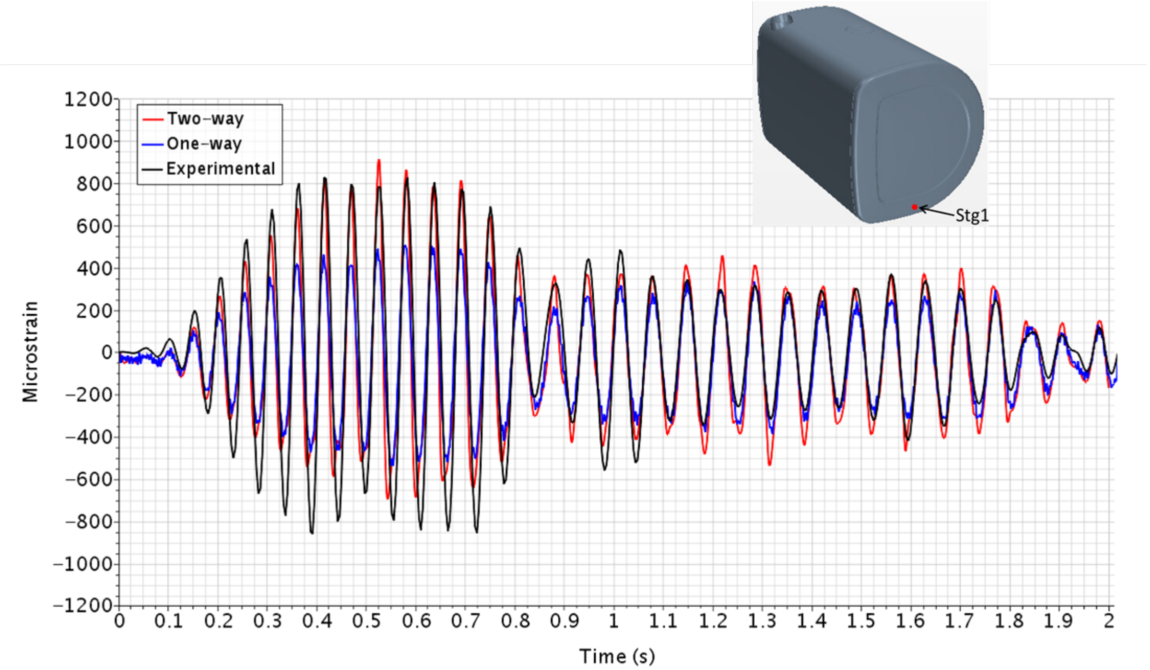


Figure 6: Deviation of the one-way coupled simulations from experimental results. See [17] for the transient strain plots at the four strain gauges for the different fill levels.

From Figure 7a, it can be observed that for a 20% fill level, the one-way and two-way coupled simulations yield nearly the same strain levels. For a 40% fill level, the two-way coupled simulations are able to capture the high sloshing-induced strains better than the one-way coupled simulations for the same mesh configuration, resulting in close comparison with the experimental signals as shown in Figure 7b. Thus, the two-way coupled FSI methodology yields better comparison with experiments for higher fill-levels. However, this comes at very high costs – the two-way coupled simulations are ten times more computationally expensive than the one-way coupled simulations. For a one-way coupled simulation of $2s$, it takes approximately 2 days of computational time on 4 CPU's (Xeon E5-2680v2, 2.80 GHz) with 10 cores each. The high computational cost is due to the fact that the calculations and field-exchange between the solid and fluid domains are done at every inner-iteration of a time-step for a two-way coupled simulation, which also leads to scaling issues. For a one-way coupled simulation, the field exchange takes place in only one-direction (from the fluid to the solid domain), and that too, only once per time step.



(a) 20% Filling



(b) 40% Filling

Figure 7: Comparison of strains obtained from one-way and two-way coupled simulations with experimental results at strain gauge $Stg1$ for 20% and 40% fill levels

6 CONCLUSIONS

Analysis of sloshing-induced loads on the fuel tank structure is carried out using the partitioned FSI approach. Validations of the VOF and the coupled CFD-FEA methodologies with numerical and experimental results from the literature on simplified representative tank geometries lay the foundation for the analysis of the truck fuel tank. Low-frequency and high-amplitude acceleration signals from the test track are considered for the analyses, corresponding to the worst case scenario for sloshing. A fair agreement is obtained between the results of the one-way coupled FSI simulations and the experiments. It is seen that the deviations from the experimental results increase with increasing fill levels, attributed to the increased loads and the resulting structural displacements that are not accurately captured in the one-way coupled simulations. This is further reinforced by the results from the two-way coupled simulations that compare well with the experiments. The two-way coupled simulations are, however, highly computationally expensive and can often be impractical for running simulations on the industrial scale for long-duration acceleration signals. A comparative analysis of the two-way coupled simulations with experiments for 60% and 80% filling levels still needs to be made in the future simulations. The simulation methodology developed in this work does not include effects of fatigue loads and a further investigation is therefore necessary to make a comprehensive structural analysis of the fuel tanks.

REFERENCES

[1] Eswaran, M., Saha, U.K. and Maity, D., 2009. Effect of baffles on a partially filled cubic tank: Numerical simulation and experimental validation. Computers & Structures, 87(3), pp.198-205.

[2] Sankar, S., Ranganathan, R. and Rakheja, S., 1992. Impact of dynamic fluid slosh loads on the directional response of tank vehicles. Vehicle System Dynamics, 21(1), pp.385-404.

[3] Nicolici, S. and Bilegan, R.M., 2013. Fluid structure interaction modeling of liquid sloshing phenomena in flexible tanks. Nuclear Engineering and Design, 258, pp.51-56.

[4] Zhou, W., Wu, Z. and Mevel, L., 2010. Vibration-based damage detection to the composite tank filled with fluid. Structural Health Monitoring, 9(5), pp.433-445.

[5] Jung, J.H., Yoon, H.S. and Lee, C.Y., 2015. Effect of natural frequency modes on sloshing phenomenon in a rectangular tank. International Journal of Naval Architecture and Ocean Engineering, 7(3), pp.580-594.

[6] Akyildiz, H. and nal, E., 2005. Experimental investigation of pressure distribution on a rectangular tank due to the liquid sloshing. Ocean Engineering, 32(11), pp.1503-1516.

[7] Liu, D. and Lin, P., 2009. Three-dimensional liquid sloshing in a tank with baffles. Ocean engineering, 36(2), pp.202-212.

[8] Hwang, S.C., Park, J.C., Gotoh, H., Khayyer, A. and Kang, K.J., 2016. Numerical simulations of sloshing flows with elastic baffles by using a particle-based fluidstructure interaction analysis method. Ocean Engineering, 118, pp.227-241.

[9] Klostermann, J., Schaake, K. and Schwarze, R., 2013. Numerical simulation of a single rising bubble by VOF with surface compression. International Journal for Numerical Methods in Fluids, 71(8), pp.960-982.

[10] CD-adapco. *STAR-CCM+ Version 11.06 User Guide*, 2016.

[11] Hron, J. and Turek, S. A Monolithic FEM Solver for ALE Formulation of Fluid Structure Interaction with Configurations for Numerical Benchmarking. In *Computational Methods for Coupled Problems in Science and Engineering* (1st edn), M. Papadrakakis, E. Onate, and B. Schrefler, editors, International Center for Numerical Methods in Engineering: Barcelona, 2005; 148. Konferenzband *First International Conference on Computational Methods for Coupled Problems in Science and Engineering*, Santorini, 2527 May 2005.

[12] Quaini, A., 2009. Algorithms for fluid-structure interaction problems arising in hemodynamics. PhD thesis, Ecole Polytechnique Federale de Lausanne, Switzerland, 2008.

[13] Rhee, S.H., 2005. Unstructured grid based Reynolds-averaged Navier-Stokes method for liquid tank sloshing. Journal of fluids engineering, 127(3), pp.572-582.

[14] Idelsohn, S.R., Marti, J., Souto-Iglesias, A. and Onate, E., 2008. Interaction between an elastic structure and free-surface flows: experimental versus numerical comparisons using the PFEM. Computational Mechanics, 43(1), pp.125-132.

[15] Kim, Y., 2001. Numerical simulation of sloshing flows with impact load. Applied Ocean Research, 23(1), pp.53-62.

[16] Hsiao, S.C. and Lin, T.C., 2010. Tsunami-like solitary waves impinging and overtopping an impermeable seawall: Experiment and RANS modeling. Coastal Engineering, 57(1), pp.1-18.

[17] Arora, S. and Vasudevan, S. (2017) Analysis of sloshing-induced loads on the fuel tank structure. Gteborg : Chalmers University of Technology (Diploma work - Department of Applied Mechanics, Chalmers University of Technology, Gteborg, Sweden, no: 2017:77