Fluid Slosh Behavior for Crashworthiness – A Modeling Approach Validated with Experimental Data

Prabhu Selvaraj, Vikas Sanghavi (Stellantis)
Srikanth Adya, Onkar Akolkar (Ansys)
Vedant Joshi, Dr. Jeremy Seidt (The Ohio State University)

Abstract

FMVSS301 mandates fuel system integrity for vehicles post-crash, so a detailed assessment of the fuel system is needed to validate its integrity. Modeling the fuel tank assembly for vehicle crashworthiness is very challenging due to the fuel slosh phenomenon which occurs in the dynamic crash event. The fuel slosh behavior affects the overall dynamics of the fuel tank and its interaction with surrounding vehicle structures. Extensive studies can be found in the literature to improve fuel tank modeling. However, there is limited information related to modeling fluid's free surface and the pressure profile imparted by the fluid on the tank during the crash event.

A fully coupled Fluid Structure Interaction (FSI) simulation of this event, using an explicit mechanical solver and an incompressible fluid solver (ICFD), is computationally expensive. Smooth Particle Hydrodynamics (SPH) has long been the choice for this application. This study uses the weakly-incompressible SPH formulation (ELFORM 15) to model the fluid behavior. In Phase-I of this two phase detailed calibration effort, the model is calibrated against a series of rigid tank slosh tests at two different speeds and various volume fills with and without baffles. The study shows good correlation of both the pressure time histories and the slosh pattern (fluid free surface) between the experiment and simulation.

1.0 Introduction

FMVSS 301 is a safety regulation that mandates fuel system integrity in vehicles to prevent fuel leaks during collisions [1]. It states the testing criteria and fuel spillage limits for vehicles with GVWR less than or equal to 4,536kg and for school buses with GVWR greater than 4,536kg. This regulation requires vehicles to have robust fuel systems that can withstand crash events and minimize the risk of fuel spillage.

Vehicles that are subjected to crash tests have a fuel volume requirement of 90%. Sloshing phenomena occurs in the partially filled fuel tank due to the sudden acceleration/deceleration experienced during the crash event. As a result, there is significant bulging of the fuel tank due to pressure differences in fuel and vapor in addition to external loading. Fuel system leakage occurs due to pinched/cut fuel lines, fuel tank shell material rupture, joint separation, component detachment, etc. Most of these failures are attributed to fluid-structure interaction caused by the sloshing phenomenon. Fuel slosh is also influenced by the tank shape, presence of internal components like baffles, fuel delivery module, valves, hoses, etc.

To design a fuel system to meet these requirements, a good understanding of the packaging environment, behavior of the fuel inside the tank, as well as its interactions with the tank walls and the internals therein is important.

© 2024 Copyright by Ansys Inc.

To compliment vehicle crash safety evaluations, non-linear dynamic crash codes are routinely used in the automotive industry to simulate vehicle crashes and develop design countermeasures. The simulations allow engineers to predict the vehicles' responses under different impact conditions before prototypes are built. The effect of fuel pressure on the tank boundary in a dynamic impact condition is constantly being studied both numerically and experimentally. However, predictions of fuel system leakage directly using the available non-linear dynamic crash codes are still very challenging.

In past studies [2], pressure profiles obtained from a full vehicle crash were used to validate the modeling approach. This pressure, however, is not constant throughout the tank. As the fuel sloshes within the tank, different surfaces experience different pressure loading. Therefore, there is a need for a more detailed investigation of how fuel slosh influences the fluid free surface and the pressure it generates due to its interaction with the structure.

The detailed experimental fuel slosh studies were conducted in two phases with a goal to acquire critical experimental data than can be used to develop and validate CAE modeling methodologies using LS-DYNA. A transparent tank filled with water, when subjected to an imposed velocity creates a slosh phenomenon. The pressure change in the tank and the fluid free surface due to the slosh are monitored during the experiments. Phase-1 studies were conducted using a rectangular, transparent tank. The tank is thick, assumed to be rigid and concept baffles are used. Phase-2 studies were conducted on a transparent version of an actual vehicle fuel tank. The tank is assumed to be flexible with production intent internals. Only details of Phase-1 studies are discussed in this paper.

In the following section we describe the experimental setup and the choice of instrumentation used to validate the experimental data with simulation. In § 3.0 we discuss the details of the SPH formulation used to model fluid behavior in LS-DYNA. In § 4.0, we describe the details of the CAE modelling and parameter tuning to improve the model accuracy. Finally, we compare the simulation predictions with the experimental data and summarize our findings.

© 2024 Copyright by Ansys Inc.

2.0 Experimental Setup

The experimental studies were conducted at Dynamic Mechanics of Materials Laboratory, Department of Mechanical and Aerospace Engineering, The Ohio State University.

Only Phase-1 of the study (water inside a rectangular tank) is covered in this paper.

A schematic representation of the tank is shown in *Figure 1*. The tank was constructed of 1-inch-thick acrylic, having dimensions of 30 inches (L) x 9 inches (W) x 15 inches (H). The tank was designed such that two baffles can be placed in the center of the tank, spaced about 10 inches apart. The baffles are 6 inches wide and 11 inches tall with 4 horizontal 1-inch-wide slots. The tank is configurable such that tests can be conducted with 1 baffle, 2 baffles, or no baffles, as desired. Leak proof construction of the tank is ensured since it is filled with water to generate slosh.

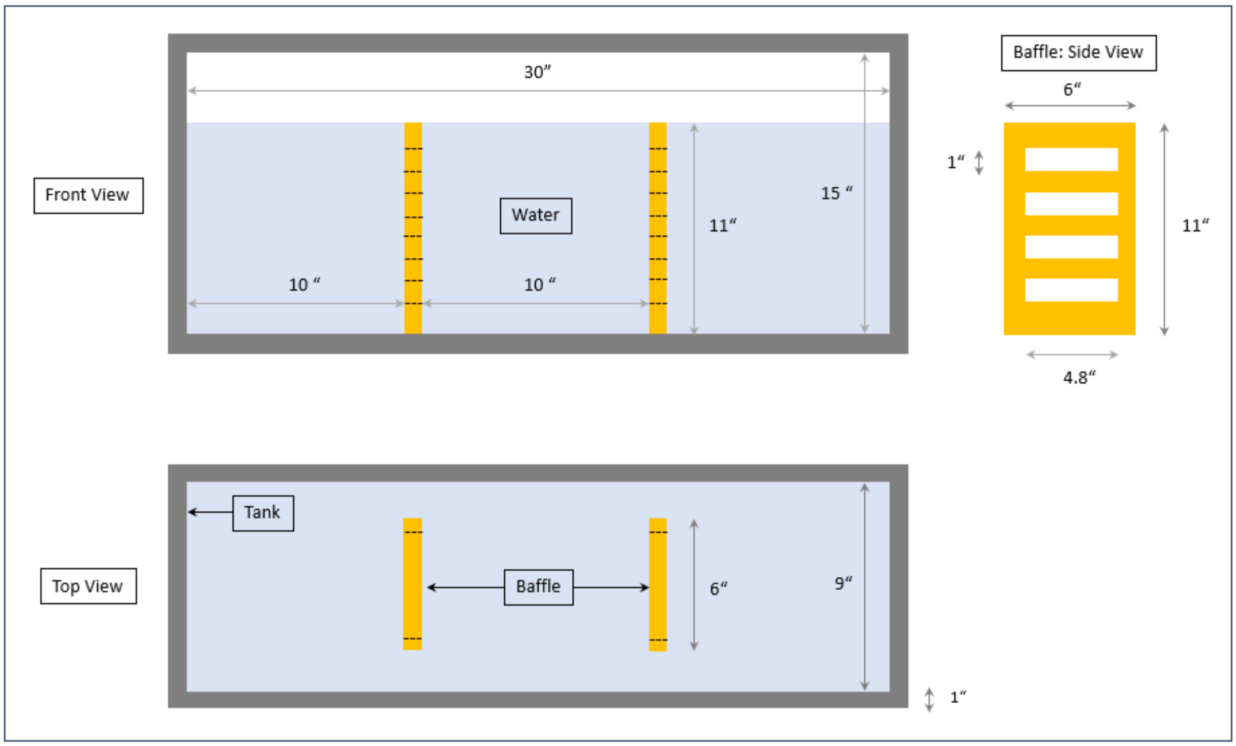

Figure 1: Tank dimensions

During the slosh experiment, the fluid free surface changes shape which creates pressure gradients within the fluid. The tank is instrumented with ten pressure sensors (Endevco model 8507C-5 piezoresistive), two each on the right and left side walls and six in the top cover (*Figure 2*), to monitor pressure change. The sensors have a full-scale measurement capacity of 5psi. The pressure signals from the experiment will be compared with the simulation.

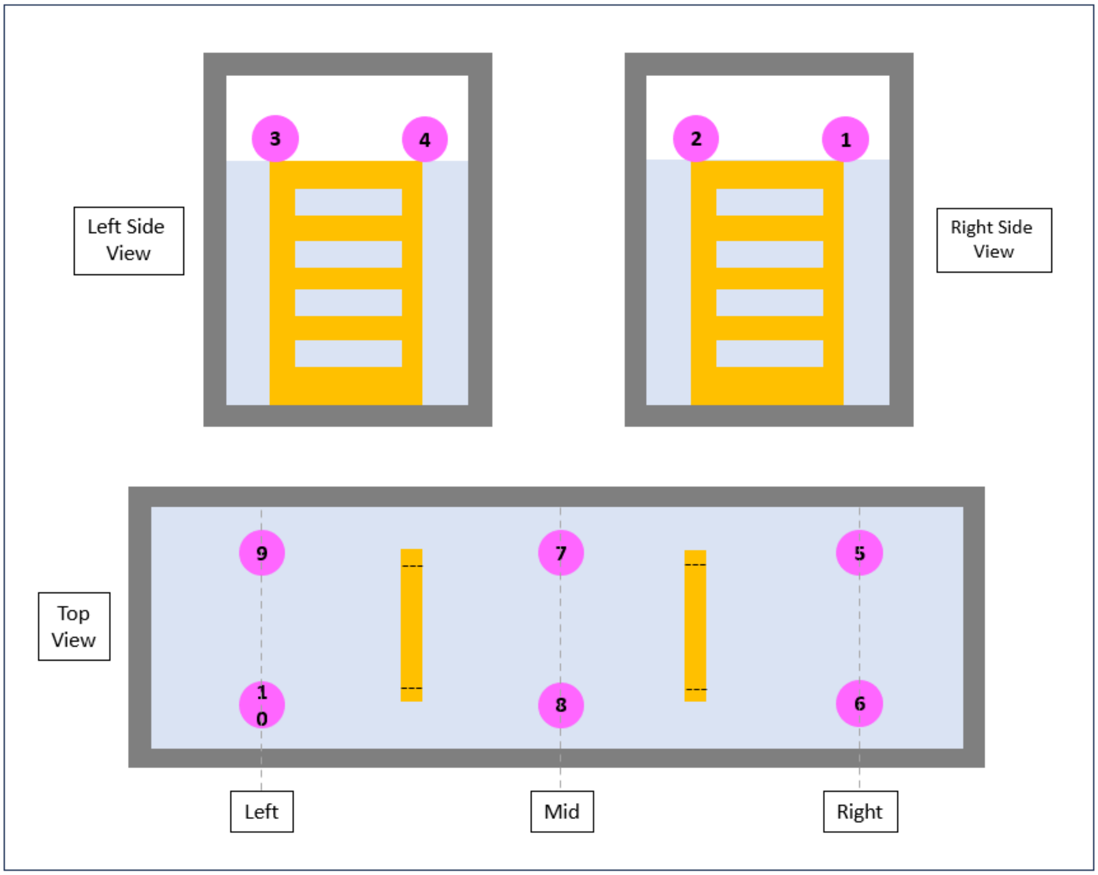

Figure 2: Pressure sensor locations

Refer to *Figure 4* for the complete experimental setup. The tank fixture is connected to two bearing plates that ride on a linear rail fastened to a large I-beam. The rail allows linear motion in the direction of the tank length. The tank fixture is designed such that the tank can be tested flat or inclined at a 20-degree angle. The tank fixture is attached to a hydraulic actuator through a kinematic linkage which amplifies the actuator displacement and velocity by a factor of 2 (tank travels 2 times farther and twice as fast as the linear actuator). A periodic velocity profile was used to accelerate the tank.

© 2024 Copyright by Ansys Inc.

The measured velocity profile of the tank (*Figure 3*) has two cycles. Each cycle comprises of four events (two peaks and two zero crossings) with right to left motion. The right and left position of the tank is represented by the positive and negative peaks.

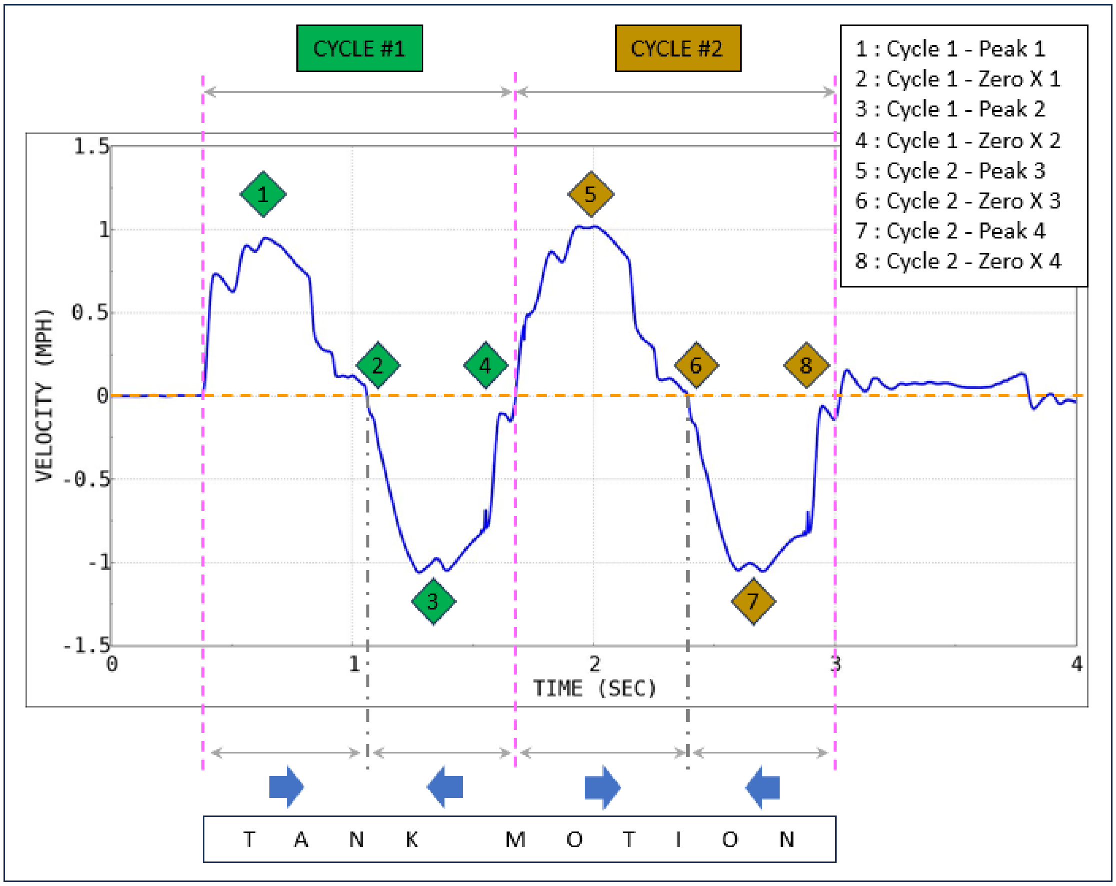

Figure 3: Actuator velocity profile at 1mph

The fluid free surface during slosh was recorded using a Photron SA1.1 monochrome high-speed camera oriented normal to the side of the tank. The fluid was dyed a dark blue color to facilitate contrast discrimination between the fluid-air interface. Images (1024 x 1024 resolution) from the camera were recorded at 250 frames per second.

The fluid free surface images from the experiment will be compared with the simulation for the eight events shown in *Figure 3*.

In addition to monitoring the fluid free surface and pressure on tank walls, the tank displacement and velocity was measured using 2D Digital Image Correlation (DIC). DIC speckle patterns were added to the tank fixture without obstructing the view of fluid *(Figure 4)*.

© 2024 Copyright by Ansys Inc.

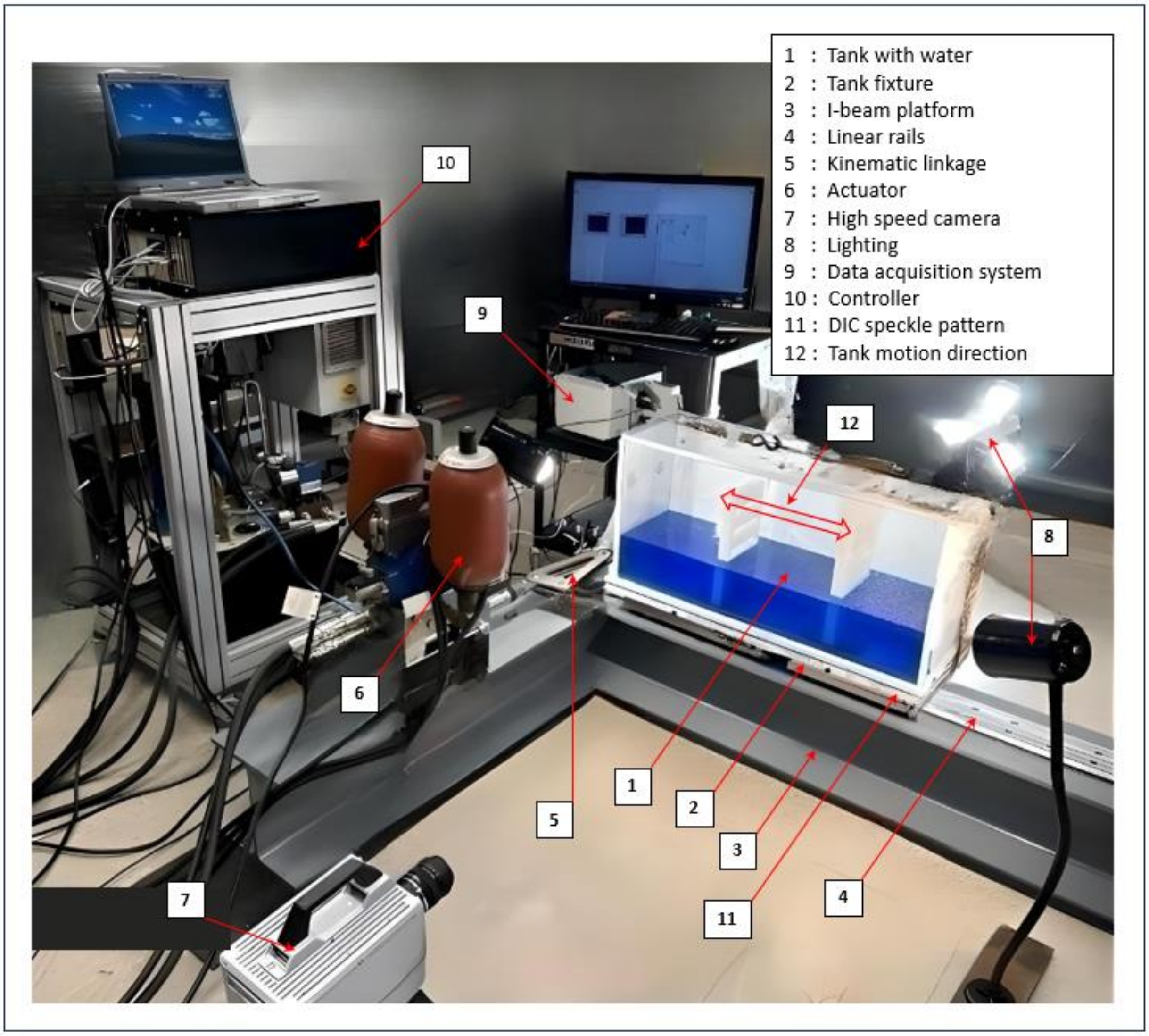

Figure 4: Experimental setup

The experiments were conducted by varying the parameters shown in the table below *(Table 1).* Each configuration was tested thrice to check for repeatability in results.

Actuator Velocity Profile	1mph	2mph	
Fuel Tank Volume	50%	70%	90%
Baffles	None	One	Two
Tank Orientation	Horizontal	Angled	

Table 1: Text matrix

© 2024 Copyright by Ansys Inc.

3.0 Weakly Compressible SPH Formulation

The SPH solver in LSDYNA was traditionally used for modelling transient events involving very large deformations of solid structures under impact loads. Because of its ability to model large deformations, SPH has become the chosen method to model violent free-surface flows. However, the traditional SPH formulation is also plagued with the well-known pressure oscillation issue [3]. To overcome this issue, a new SPH formulation ELFORM=15 was introduced in LSDYNA. This element formulation uses a density smoothing scheme using a Shepard Filter which produces a smooth density field. This in turn reduces the oscillations in the pressure field when combined with an equation of state [4].

The Murnaghan Equation of State expressed the pressure at any point in the field as:

$$p = k_0 \left[\frac{\rho}{\rho_0}^{\gamma} - 1 \right] \quad (1)$$

Where ρ_0 is the density of the fluid at rest, γ is usually set to 7. k_0 is chosen such that:

$$c_0 = \sqrt{\frac{\gamma k_0}{\rho_0}} \geq 10 v_{max} \quad (2)$$

Where v_{max} is the expected maximum fluid velocity. This ensures minimum variation in density making it suitable for modeling the quasi-incompressible nature of fluids.

Additionally, an artificial viscosity term is added to the momentum equation. Two parameters Q_1 and Q_2 are chosen such that the material is not too dissipative and behaves more “fluid-like”. These parameters are listed in § 4.0.

4.0 CAE Model Setup

The tank with two baffles was modeled with solid elements, and SPH particles were used to represent 70% volume fill with water *(Figure 5)*.

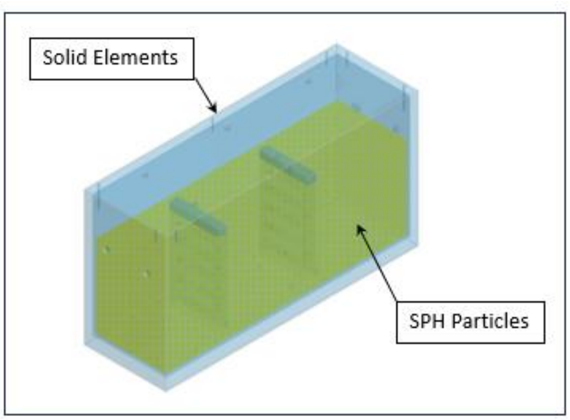

Figure 5: CAE model of tank with 2 baffles and 70% volume fill with water

© 2024 Copyright by Ansys Inc.

LS-PREPOST was used to generate the SPH particles in the closed volume of the tank. The internal void of the tank is first defined as a shell volume in order to generate the SPH particles. The segments that line the inner surface of the tank and the outer surface of the baffles are selected, and a shell part is created using the segment set. With this shell Part, SPH particles are generated using the mesh generation tool in LS-PREPOST.

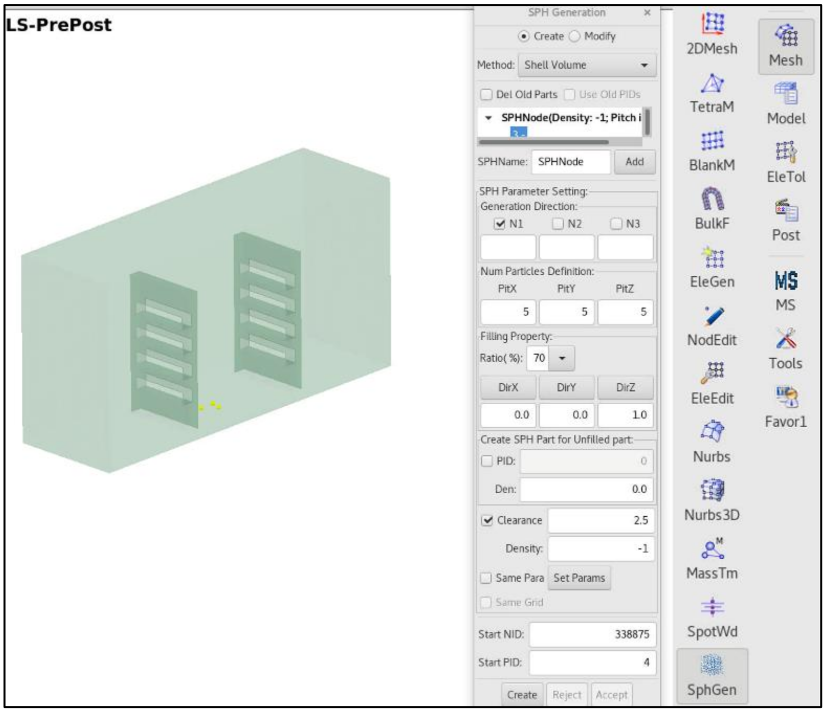

Figure 6. LS-PREPOST interface for SPH mesh generation

While generating the SPH particles, the parameter "Density" in this interface was set to -1. This replaces the mass field in *ELEMENT_SPH with a negative number defining the volume of each particle as shown in *Figure 7*. This provides the flexibility to change the density of the fluid without having to regenerate the SPH particles. Also note in *Figure 6*. that the "Filling Property" was set to the desired percentage volume fill (in this case 70%) of the tank. The weakly compressible enhanced fluid formulation was used by setting FORM=15 in *CONTROL_SPH.

```
*ELEMENT_SPH
$#    nid      pid             mass
   338875        4           -125.0
   338876        4           -125.0
   338877        4           -125.0
   338878        4           -125.0
   338879        4           -125.0
```

*Figure 7: *ELEMENT_SPH card - has a negative mass number*

© 2024 Copyright by Ansys Inc.

*MAT_NULL was used to define the density and viscosity of the fluid (*Figure 8*). The pressure-density relationship was defined using the *EOS_MURNAGHAN equation of state model (*Figure 8*). There are two parameters to be defined in this EOS model. *Gamma* is often set to 7 for fluids, and the value of k_0 was computed using the Murnaghan equation (*Equation 2*) such that quasi-incompressibility is achieved. Initially, v_{max} is assumed equal to the tank velocity. The fluid velocity predicted by this value is then used to correct the k_0 value. The viscosity coefficients Q_1 and Q_2 are assigned to the SPH part by defining them in the *HOURGLASS card (*Figure 8*). The values for these two parameters were obtained from [4].

```
*MAT_NULL_TITLE
Fluid
$#     mid        ro        pc        mu     terod     cerod        ym        pr

*HOURGLASS
$#    hgid       hiq        qm       ibq        q1        q2        qb        qw

*EOS_MURNAGHAN
$#   eosid     gamma        k0
```

Figure 8: Material properties, EOS definition and coefficients Q_1, Q_2 used to model water

The interaction between the SPH particles (fluid) and the structure (acrylic) was modeled using *CONTACT_AUTOMATIC_NODES_TO_SURFACE. Since the densities of the tank and the fluid are dissimilar, SOFT=1 is used in the contact card. To ensure SOFT=1 is always used in the contact, the default Surface A and Surface B penalties were scaled down. Without penalty scaling, it was noticed that SPH particles would leak out of the tank.

The rigid tank was prescribed a velocity using *BOUNDARY_PRESCRIBED_MOTION_RIGID. The velocity profile was obtained from DIC measurements in the corresponding test. Refer to *Figure 3* for the velocity profile used in the model.

Gravity load is ramped from zero to max value during the initial flat portion of the velocity profile (about 300 milliseconds). Instantaneous application of gravity from time=0 caused the particles to slosh aggressively, while ramping up the gravity allowed particles to settle down uniformly prior to velocity application.

The pressure in the simulation is measured on segments whose coordinates match with the pressure sensors in the experimental property. The segments measure the force resulting from the impact of SPH particles on the walls of the tank. The force measurement is activated using *CONTACT_FORCE_TRANSDUCER and assigning the respective segment sets to the contact. The 'RCFORC' file provides the measure of forces experienced by each segment. Pressure at each segment is obtained by dividing the segment force by the area of segment.

5.0 Results

Two configurations (two baffle and no baffle) are chosen to compare the experimental and simulation results for fluid free surface and pressure signals. The selected configurations are subjected to 1mph velocity profile, 70% volume fill with horizontal tank orientation.

A. Fluid free surface:

1) Two baffle configuration: *Figure 9*
   The presence of baffles reduces and controls the fluid slosh. Due to the reduced slosh, most of the fluid volume interacts as a single body with the tank during both Cycle #1 and Cycle #2 events. The simulated slosh behavior compares well with experiment for all Cycle #1 events and most of Cycle #2 events.

2) No baffle configuration: *Figure 11*
   The slosh of fluid is aggressive due to the absence of baffles. The aggressive slosh splits the fluid volume into multiple bodies particularly during the Cycle #2 events. During Cycle #1, the fluid interacts as a single body with the tank and the simulated slosh behavior compares well with the experiment. During Cycle #2, the simulated slosh behavior deviates from the experiment.

B. Pressure signals:
   Experimentally measured pressures are represented by black curves while simulated pressures are represented by magenta curves.

1) Two baffle configuration: *Figure 10*
   For side wall pressure sensors (S1, S2, S3, S4), the simulation has a good match with the experiment for the positive pressure signals. Negative pressure signals caused by the pressure drop from ambient were not predicted by the simulation. For the top cover pressure sensors (right - S5, S6 and left - S9, S10), the simulation compares well for the positive signals. The model response deviates from experimental data for sensors located in the middle of the top cover (S7, S8).

2) No baffle configuration: *Figure 12*
   For side wall pressure sensors (S1, S2, S3, S4) and top cover pressure sensors (right - S5, S6 and left - S9, S10), the simulation is in good agreement with the experiment for the positive pressure signals. Negative pressure signals were again not predicted by the model. The model response deviates from experimental data for sensors located in the middle of top cover (S7, S8).

For both configurations, side wall:

a. Pressure history trends and magnitudes are well predicted (when positive)
b. Pressure magnitudes are 2 to 2.5 times higher than those measured on the top cover of the tank

© 2024 Copyright by Ansys Inc.

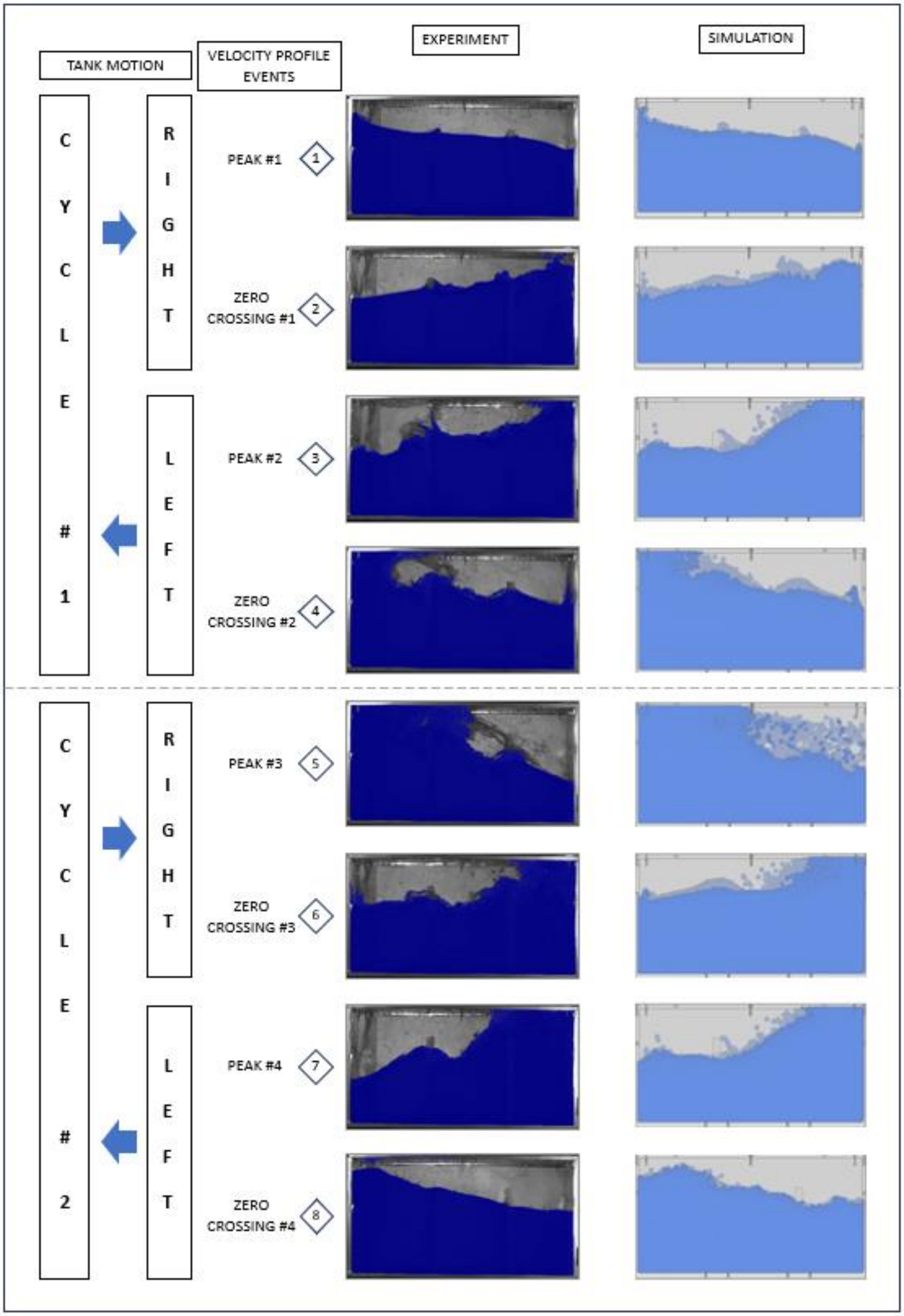

Figure 9: Fluid free surface – Two baffles, 1mph velocity, 70% volume fill, horizontal tank

©2024 Copyright by Ansys Inc.

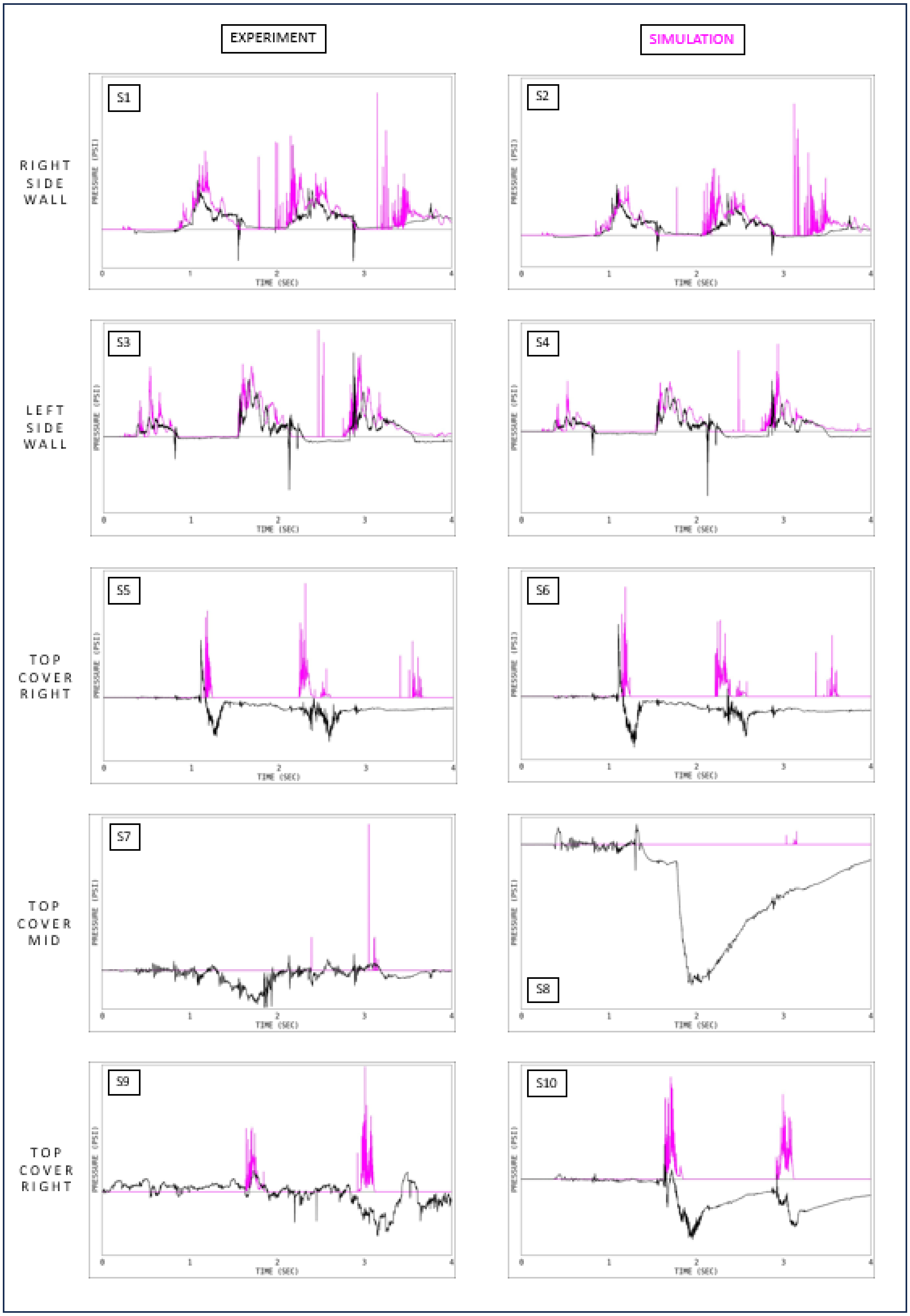

Figure 10: Pressure sensor - Two baffles, 1mph velocity, 70% volume fill, horizontal tank

©2024 Copyright by Ansys Inc.

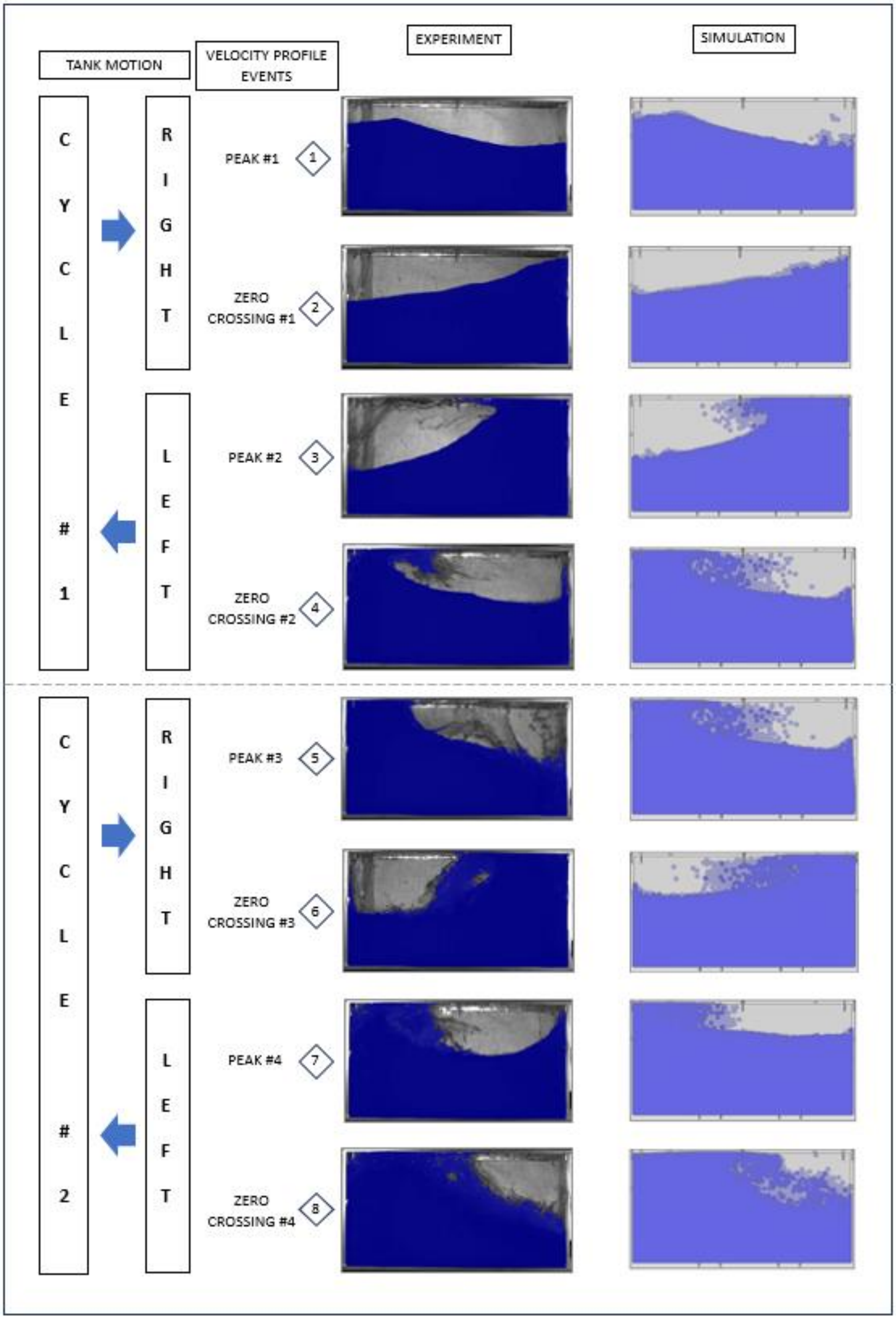

Figure 11: Fluid free surface – No baffles, 1mph velocity, 70% volume fill, horizontal tank

©2024 Copyright by Ansys Inc.

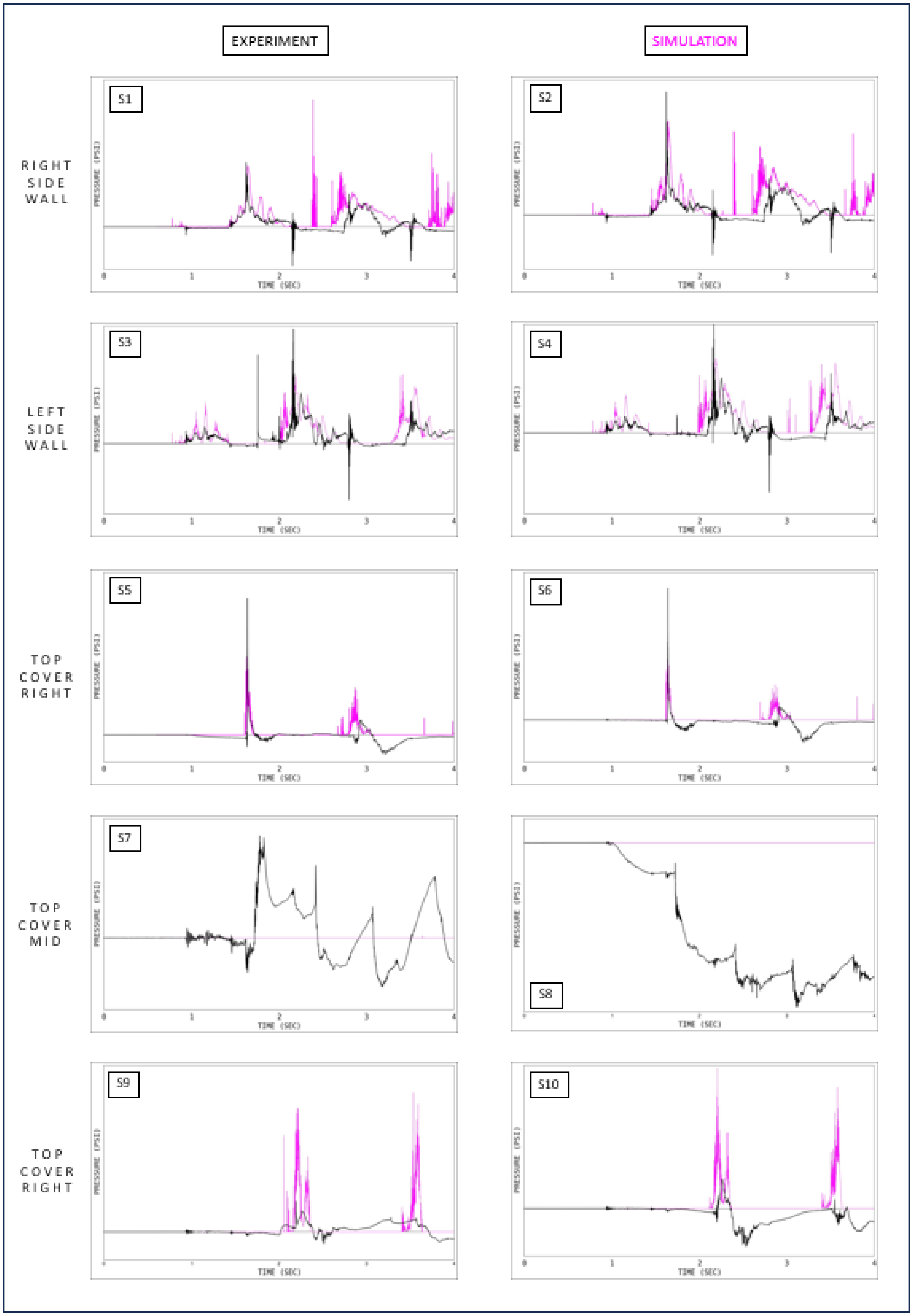

Figure 12: Pressure sensor – No baffles, 1mph velocity, 70% volume fill, horizontal tank

©2024 Copyright by Ansys Inc.

6.0 Conclusion

An experimental and numerical effort to validate fluid sloshing phenomena for automotive applications has achieved good correlation between experimental and simulation results comparing fluid free surface and pressure signals.

An extensive experimental dataset (side wall pressure histories and high-speed camera footage) was collected using the experimental setup at several initial conditions (actuator velocity profile, baffle configuration, tank volume fill percentage, and inclination angle). Numerically the fluid was modeled as SPH particles with element formulation 15 which helped in achieving a smooth pressure field. The viscous dissipation (Q_1, Q_2), *Gamma,* and k_0 terms in the Equation of State models were carefully chosen to predict the slosh behavior with reasonable accuracy.

The simulation predicts the fluid free surface extremely well for the baffle case and slightly less accurately for the no baffle case with aggressive slosh. In both cases, better fluid free surface agreement between the simulation and experiment was noted during the first cycle. During the second cycle, the fluid free surface match is somewhat imprecise, yet still predicts the general shape trends.

The model predicts only positive pressure signals and cannot predict negative pressure signals. The pressure sensors are modeled as segments and pressure is calculated using the force measured resulting from the impact of SPH particles (which is always positive). With this sensor modeling approach, the negative forces cannot be measured when the SPH particles move away from the segment.

For both cases (baffle and no baffle), the simulated side wall pressure history trends and magnitudes are well predicted (when positive). The top cover right and left pressure sensors show agreement for positive signals. However, the top cover middle sensor response is deficient.

Side wall sensor pressure magnitudes are 2 to 2.5 times higher than those measured on the top cover of the tank. During the slosh event, the pressure on the side walls in the slosh direction will be highest and the primary contributor of the fluid force transfer to the tank.

In addition to the structural loading predicted in full vehicle crash simulations, accurate simulation of the pressure exerted by the fuel on the tank walls will help improve the accuracy of numerical fuel tank integrity evaluations.

©2024 Copyright by Ansys Inc.

References

1) NHTSA “Laboratory test procedure for FMVSS 301 – Fuel System Integrity”
2) Tushar et.al., “Modeling of vehicle fuel with smooth particle hydrodynamics (SPH) method in LSDYNA for vehicle crash virtual simulation”
3) Hong-Guan Lyu et al., “On removing the numerical instability induced by negative pressures in SPH simulations of typical fluid – structure interaction problems in ocean engineering”
4) Edouard Yreux “Fluid flow modeling with SPH in LSDYNA”
5) Emma Frosina et al., “Experimental and numerical analysis of the sloshing in a fuel tank”

Acknowledgements

The authors would like to thank:

1) Suri Bala (d3VIEW Inc.) for project conceptualization
2) Edouard Yreux (Principal R&D Engineer LS-DYNA, Ansys) for SPH development
3) Hamid Keshtkar (ex-Stellantis, Virtual Engineering) for project support
4) Russell Sarquis (YAPP USA Automotive Systems, Inc.) for project support
5) Jerome Klein (Dynamic Mechanics of Materials Laboratory, The Ohio State University) for instrumentation and fabrication support
6) Chad Bivens (Department of Mechanical and Aerospace Engineering, The Ohio State University) for custom fixture machining
7) Kevin Wolf (Department of Mechanical and Aerospace Engineering, The Ohio State University) for custom fixture machining

©2024 Copyright by Ansys Inc.