

# Crash Performance Simulation of a Multilayer Thermoplastic Fuel Tank with Manufacturing and Assembly Consideration

2011-01-0009

Published  
04/12/2011

Ryan Craig and Tony (YI) Qu  
Ford Motor Co.

Ligong Pan  
ETA Inc. (Engineering Tech. Assoc)

Tau Tyan  
Ford Motor Co.

Jiamaw Doong  
Ford Light Truck Div.

Syed Ahmad and Yi Zhang  
Ford Motor Co.

Copyright © 2011 SAE International  
doi:[10.4271/2011-01-0009](https://doi.org/10.4271/2011-01-0009)

## ABSTRACT

The modeling of plastic fuel tank systems for crash safety applications has been very challenging. The major challenges include the prediction of fuel sloshing in high speed impact conditions, the modeling of multilayer thermoplastic fuel tanks with post-forming (non-uniform) material properties, and the modeling of tank straps with pre-tensions. Extensive studies can be found in the literature to improve the prediction of fuel sloshing. However, little research had been conducted to model the post-forming fuel tank and to address the tension between the fuel tank and the tank straps for crash safety simulations. Hoping to help improve the modeling of fuel systems, the authors made the first attempt to tackle these major challenges all at once in this study by dividing the modeling of the fuel tank into eight stages. An ALE (Arbitrary Lagrangian-Eulerian) method was adopted to simulate the interaction between the fuel and the tank. The ALE method enables the sloshing phenomena to be simulated and allows vortices and cavitation to be captured. Blow molding simulation is conducted in the study to predict the thickness changes, as well as the residual stress and strain

distributions in a multilayer thermoplastic fuel tank due to manufacturing processes. The information obtained was used as the initial condition in the crash simulation. An approach also has been developed to model the pre-tension of the fuel tank straps that was introduced in the fuel tank assembly process. The weight of the entire fuel system including the fuel is considered in this study to improve the modeling of the contact tightness between the fuel tank and tank straps. Validated by fuel tank sled tests, the proposed fuel tank modeling approach, incorporating the latest ALE method, manufacturing and assembly processes, and gravity effect, helps improve the correlation between CAE and physical tests significantly.

## INTRODUCTION

To compliment vehicle crash safety evaluations, non-linear dynamic crash codes have been used in the automotive industry routinely to simulate vehicle crashes and develop design counter-measures. The simulations allow engineers to better predict the vehicles' responses under different impact conditions before prototypes are built. Design iterations can

also be conducted to evaluate modifications or alternative designs and test modes. Crash simulation techniques to predict structural responses have advanced year after year and resulted in reduction of physical tests and cycle time needed to develop new vehicle programs. However, predictions of fuel system leakage directly using the available non-linear dynamic crash codes are still very challenging. The fuel system leakage not only involves fluid-structure interaction but also other factors such as seal breakage, material rupture, joint separation, component detachment, forming and manufacturing considerations, gravity settling, and so on. Among the mentioned factors, the fluid-structure interaction has attracted most attention with many studies being conducted to predict phenomena related to fuel flow and sloshing. As a result, advanced modeling approaches such as ALE and SPH (Smooth Particle Hydrodynamic) and many CFD (Computational Fluid Dynamics) codes have been developed and made available commercially. However, few attempts have been made in the past to tackle the other factors mentioned above.

Before finite element codes were made available for fluid-structure interaction applications, simplified mechanical models had been used to analyze fuel (or liquid-filled) tank truck braking performance. One of the examples is to use a two dimensional kineto-static roll plan model to determine the influence of partition location on the braking performance of a partially-filled liquid cargo [1]. The model, developed in 1995, leads to the optimization of compartment sizes to minimize the magnitude of load transfer for braking performance. Later in 2000 [2], the model was expanded to analyze a liquid filled cargo truck under three dimensional and quasi-dynamic maneuvers. As a result, the yaw and roll characteristics and braking responses of a partially-filled tank truck subjected to braking in a turn can be predicted.

Using a boundary element method, Dutta and Laha [3] estimated the natural sloshing frequencies and forces developed due to an inviscid, incompressible fluid oscillating in a rigid tank. Both free and forced oscillation cases were studied and compared. The method was found to be more economical compared to the traditional finite element approach.

In 2001, Hayashi, et al.[4] used a simplified finite element model to help determine the material and design beads and embossments for a fuel tank under development. In their ABAQUS analyses, only a portion of the tank was modeled and the internal pressure was assumed uniform on each element. Based on the predicted stress and strain for different materials and bead configurations, a multiple layer tank made of PE (polyethylene) and low permeability PA (polyamide) materials was chosen and wavelike convex beads were designed for the final fuel tank. Banerjee, et al., [5] used a commercially available CFD code, FLUENT, to understand the fluid flow inside an automotive gas tank filler pipe. In this

finite element analysis, the unsteady stage was studied with gasoline and air chosen as the working fluids. The fuel tank model included a tank, a filler pipe and a vent pipe. The phenomenon of well-back, the process of fuel flooding the filler pipe and flowing backwards at the filler pipe mouth, and the pressure transients inside the tank leading to premature nozzle shut-off were predicted. In addition, Banerjee, et al., developed an algorithm [6] to simulate evaporation over a liquid free surface based on the concept of continuous thermodynamics.

In hoping to reduce the noise produced by fuel tanks and meet required performance specification in service, sloshing simulations with different commercial CFD codes have been conducted by different researchers. Kamiya, et al. [7] adopted ALE available in MSC.Dytran to simulate the fuel sloshing inside a tank. The fuel sloshing and pressure generated were used as indicators for the noise source. With the simulation information, baffles could be designed to reduce the noise in different fuel tanks. Further investigations of fuel sloshing simulation in fuel tanks and baffle designs to reduce noise by decreasing fluid velocity and impact pressure on the tank can be found in [8, 9, and 10]. IU, et al., [8] used FLOW-3D to predict mean kinetic energy and average turbulent kinetic energy of fluid. The peak mean kinetic energy was found to correlate with the peak noise level for an individual slosh situation. Ma and Usman [9] used ALE available in LS-DYNA to design a baffle during the development of a fuel tank. In the study, the tank was assumed rigid and coupled solid-fluid computation was employed to simulate the sloshing phenomenon in a partially filled fuel tank. Roh, et al., [10] applied another general purpose CFD code, SCRYU/Tetra, to analyze fuel sloshing problems. Parametric study with different dimensions and shapes was performed to reduce impact pressure in fuel tanks. The simulation results were also compared to those of component sloshing tests with transparent fuel tanks. Using a statistical approach, Kamel, et al., [11] confirmed the correlation between the noise performance and fuel sloshing. As a result, the computational methods can be used to design fuel tanks for different vehicles.

Finite element methods coupled with test data also have been used to predict fatigue life of fuel tanks and tank straps. Song, et al., [12] used ALE available in Radioss to simulate the fuel tank and tank strap movement under proving ground conditions. A commercial fatigue code, FE-Fatigue, was used to predict the fatigue life of the fuel tank. The approach was found to accurately predict the crack initiation sequence and locations in the fuel tank straps. An MFLUID method, also called a "Virtual Fluid Mass" method, was used by Qin and D'Souza [13] to represent the interaction between the fuel and fuel tank. This method assumes that the fluid is incompressible and the free surface has zero pressure with no sloshing effects or rotational flow. The MFLUID method was found to correlate better with tests than the "lumped mass

method". Fatigue tests were conducted by Himeki, et al., [14] on fuel tanks made of high-density polyethylene materials. The results indicated that the fatigue properties of the materials can be expressed as a function of stress, frequency and temperature. The function can be coupled with the stresses obtained from finite element analyses to assess fuel tank durability.

In an attempt to simulate fuel leakage, van der Veen [15] developed a method called "Multiple Adaptive Euler Domains" in MSC.Dytran to model the fuel inside the tank (fuel domain) and the air surrounding the tank (ambient domain). An example with a partially filled fuel tank being crushed between a wall and a moving rigid barrier was presented. Crash simulations for tanks with and without baffles were conducted to show the effect of baffles on fuel leakage. It was found that fuel leakage was significantly less for the case with baffles. The developed technology was also found by Ding, et al., [16] to be very useful to simulate a fuel tank filled with air and fuel. Similarly, Tang, et al., [17] adopted ALE in LS-DYNA to study the sloshing of fuel and ballooning of fuel tanks in high speed impacts. A sub-system test with a fuel tank system mounted onto a vehicle frame was conducted for comparison with the simulation. The predicted deceleration using the ALE method with deformable fuel tank system and vehicle frame correlated with the data obtained from the test. Simulations of tanks with different lids and fuel levels were performed to understand the effect of fuel sloshing on the fuel tanks. The tank stresses and deformations obtained from simulations were analyzed and used to optimize the tank design for high speed impacts.

To help design the right location for the vapor valves on a tank shell, Matteo, et al., [18] adopted a commercial CFD code, Star-CD, to predict the dynamic behavior of the fuel inside a vehicle tank in standard transient driving conditions. The developed modeling techniques and the simulation tool were found to be good enough for early stage tank designs. A finite element model has also been developed by Benrabah et al., [19] to predict fuel hydrocarbon permeation for multilayer thermoplastic fuel tanks. The diffusion mode 110 is based on the Fick's second law and the continuum approach with steady state permeation regime across the multilayer polymeric wall. The numerical results correlate very well with those of the analytical results. The barrier layer percentage was found to be the primary factor to reduce the evaporation fuel emission for the fuel tank studied.

The modeling of plastic fuel tank systems for crash safety applications has been very challenging. During a high speed impact condition, a fuel tank can be deformed due to the interactions of the tank with the internal fuel and surrounding components including heat shield, tank straps, isolators, floor, side sill, rear axle, suspensions, cross members and so on depending on vehicle designs. As has been discussed

previously [17], sloshing of the internal fuel can cause ballooning or bulging of the fuel tank when the fuel is nearly full and impact speed is high, while turbulence, vortices and cavitation may also form with partially filled tanks. As a result, the internal fuel and vapor pressure distributions can change; thus, affecting the distribution of stress on the tank. Thermoplastic fuel tanks manufactured using a blow molding process do not have uniform thickness and material property throughout the entire tank as commonly used in the fuel tank modeling practices. To predict accurate stress distribution on a post-formed fuel tank will require additional efforts. In addition, the surrounding components can move relative to the fuel tank and cause elastic and/or plastic deformation of the tank depending on the contact conditions. In high speed impact conditions, the tank straps constrain the movement of the tank in different degrees and can contribute to the final deformation of the fuel tank.

Moreover, tension is applied to the fuel tank straps during tank assembly processes to keep the fuel tank in position. The weight of the tank and fuel also affect the tightness of the fuel tank straps. These phenomena present additional challenges to the modeling of the fuel tank systems.

In this study, the authors made an attempt to consider all different aspects in the modeling of a fuel tank for crash simulations. The goal is to help improve simulations of fuel tank with the best possible CAE techniques. The modeling of fuel system was divided into eight stages including fuel tank blow molding simulation, forming result mapping, fuel tank and heat shield translation, fuel tank and heat shield placement, tank strap fitting, fuel filling, fuel tank system gravity loading, and finally the crash simulation. The approach also utilizes both explicit and implicit calculations to save computation time. The eight-stage simulation replicated the key steps required to manufacture and install a fuel tank into a vehicle. To the authors' knowledge, this is the first published attempt in the industry to incorporate the influences of both manufacturing and assembly processes into crash safety simulations. The same effort can be applied to the modeling of other vehicle components.

## SUB-SYSTEM TESTING

Sub-system tests were chosen in the study to prove out the proposed modeling methodology due to the flexibility of test set up and data acquisition. A side objective for this study is to design a generic frame test fixture which can be used for different fuel tanks and different impact conditions. The vehicle being simulated has different wheel bases and several fuel tank options including gasoline and diesel tanks of different sizes. As a result, the longest frame is chosen as the test fixture while the fuel tank, the cross-members from which the tank is suspended, and the attaching straps are inter-changeable to accommodate different fuel system configurations. Although sled tests and simulations for

different fuel tank configurations have been conducted in the project, only one configuration, for a 35 gallon gasoline tank, is discussed in this paper to avoid repetition. The generic sub-system test fixture with the 35 gallon gasoline development tank is shown in [Figure 1](#).



**Figure 1. Illustration of a sub-system fuel tank test used in this study.**

To capture the conditions that occurred in full vehicle tests, the pulses (decelerations) obtained from full vehicle tests were then used as the inputs in the sub-system tests. The sub-system tests can be conducted by using servo-hydraulic sled test (acceleration sled) or VIA sled test (deceleration sled) facilities. When the servo-hydraulic sled test facility is used, the pulses obtained directly from full vehicle tests including vehicle pitch can be used exactly as the inputs in the sub-system tests. To use the VIA sled test facility, the pulses obtained from full vehicle tests need to be approximated by choosing an appropriate initial velocity and then coupled with adjustable shock absorbers and honeycomb blocks to decelerate the frame. In this study, the sub-system tests were conducted by using the VIA sled test facility due to the sled availability. Accelerometers were instrumented at different locations of the tank and frame to record the decelerations. With the data obtained from the accelerometers, the relative movement between the tank and the frame can also be calculated and compared to the results obtained from CAE simulations. The deceleration recorded from the frame is also checked against that of the full vehicle test to ensure the sub-system test can approximate the corresponding full vehicle test as accurately as possible. The tank shown in [Figure 1](#) is a 35 gallon development tank made with multilayer thermoplastic material. The tank is partially filled with 80% liquid the same as the full vehicle test. For safety concerns, water instead of gasoline was used in each of the sub-system

test. The water also makes it easier for CAE modeling since the density and other properties of water are well known and can be obtained and incorporated readily into the model.

## SUB-SYSTEM MODELING

In order to capture the effect of fuel sloshing as reported in a previous study [17], the ALE method in LS-DYNA was again used in this study. The sub-system fuel system model, without ALE elements, used in this study is shown in [Figure 2](#). The finite element model has 57 parts, 75732 nodes, 74610 shell elements, and 680 beam elements, representing spot welds and other joints.



**Figure 2. The sub-system fuel tank model used in this study.**

The fuel tank elements measured about 4 to 9 mm on each side and about 27,000 of these shell elements were required to create the tank shell without internal components. Smaller elements were needed in some areas to capture the geometry of the tank in detail. Larger elements were used in areas where the surface is flat. Different part IDs were used in the fuel tank model to reflect the non-uniform thickness and residual strain in the tank after the blow molding process. The elements used for the heat shield were slightly larger than those of the tank. The size of the elements varied from 7 to 10 mm with about 6,000 elements total used for the entire tank shield. Tank straps were modeled by using smaller shell elements, with 2 to 5 mm, due to the curved edges of the straps. All fuel system components, fuel tank, tank straps, tank shield, internal parts, fuel and vapor are considered deformable in this study.