

General Disclaimer

One or more of the Following Statements may affect this Document

- This document has been reproduced from the best copy furnished by the organizational source. It is being released in the interest of making available as much information as possible.
- This document may contain data, which exceeds the sheet parameters. It was furnished in this condition by the organizational source and is the best copy available.
- This document may contain tone-on-tone or color graphs, charts and/or pictures, which have been reproduced in black and white.
- This document is paginated as submitted by the original source.
- Portions of this document are not fully legible due to the historical nature of some of the material. However, it is the best reproduction available from the original submission.

A CFD Approach to Modeling Spacecraft Fuel Slosh

Brandon Marsell¹, Sathya Gangadharan², Yadira Chatman³
Embry-Riddle Aeronautical University, Daytona Beach, FL 32114

James Sudermann⁴, Keith Schlee⁵, and James Ristow⁶
NASA, Kennedy Space Center, FL 32899

Introduction:

Energy dissipation and resonant coupling from sloshing fuel in spacecraft fuel tanks is a problem that occurs in the design of many spacecraft. In the case of a spin stabilized spacecraft, this energy dissipation can cause a growth in the spacecrafts' nutation (wobble) that may lead to disastrous consequences for the mission. Even in non-spinning spacecraft, coupling between the spacecraft or upper stage flight control system and an unanticipated slosh resonance can result in catastrophe. By using a Computational Fluid Dynamics (CFD) solver such as Fluent, a model for this fuel slosh can be created. The accuracy of the model must be tested by comparing its results to an experimental test case. Such a model will allow for the variation of many different parameters such as fluid viscosity and gravitational field, yielding a deeper understanding of spacecraft slosh dynamics.

Spinning is not only used for spacecraft stabilization, it also helps to moderate the effects of solar heating on the spacecraft and payload [1]. Mostly used for the upper stage of payload launch vehicles, spin stabilization can cause unwanted vibrations and, in turn, nonlinear forces on the liquid propellant tanks. This will cause the propellant to slosh around inside the tank. The sloshing motion of the propellant causes a loss of kinetic energy of the spacecraft. Since the stabilization of these particular vehicles is dependant on their spin (much like a gyroscope), a loss of rotational kinetic energy could prove disastrous for the spacecraft and the payload.

The gyroscopic nature of the spin stabilization implies that the spacecraft precesses about its spin axis at a certain angle. The angle at which it precesses (or nutates) relative to the spin angle is denoted as the nutation angle [2]. For the ideal flight this angle would be relatively small and constant. When the spacecraft is spinning about a minor moment of inertia, the propellant sloshing motion will interact with the nutation causing a growth in the nutation angle (nutation growth rate). If this nutation angle growth is not actively controlled, it may lead to incomplete depletion of the propellant [2]. Even worse, as seen in the ATS-5 mission launched in 1969, it may lead to a complete loss of control of the vehicle [3]. For this reason it is important to fully understand the slosh dynamics of propellants in fuel tanks. The nonlinear, time dependant nature of this complex problem makes it a real challenge.

In order to gain a better understanding of the dynamics behind sloshing fluids, the Launch Services Program (LSP) at the NASA Kennedy Space Center (KSC) is interested in

¹ Graduate Student, Department of Aerospace Engineering, AIAA Student Member

² Professor, Department of Mechanical Engineering, AIAA Associate Fellow

³ Graduate Student, Department of Aerospace Engineering, AIAA Student Member

⁴ Controls Analyst, NASA Launch Services Program

⁵ Propulsion Engineer, Orbital Sciences Corporation

⁶ Loads Analyst, NASA Launch Services Program

finding ways to better model this behavior. Thanks to past research, a state-of-the-art fuel slosh research facility was designed and fabricated at Embry Riddle Aeronautical University (ERAU). This test facility has produced interesting results and a fairly reliable parameter estimation process to predict the necessary values that accurately characterize a mechanical pendulum analog model. The current study at ERAU uses a different approach to model the free surface sloshing of liquid in a spherical tank using Computational Fluid Dynamics (CFD) methods.

Using a software package called Fluent, a model was created to simulate the sloshing motion of the propellant [4]. This finite volume program uses a technique called the Volume of Fluid (VOF) method to model the interaction between two fluids [4]. For the case of free surface slosh, the two fluids are the propellant and air. As the fuel sloshes around in the tank, it naturally displaces the air. Using the conservation of mass, momentum, and energy equations, as well as the VOF equations, one can predict the behavior of the sloshing fluid and calculate the forces, pressure gradients, and velocity field for the entire liquid as a function of time [5].

The information acquired from the CFD model will be compared to several test scenarios that have been studied in the laboratory in order to verify the CFD results. Once the results have been experimentally verified and there is significant confidence in the values calculated by the CFD technique, it will be applied to various other scenarios such as variable gravitational conditions, larger tanks, different shaped tanks, and more viscous liquids. Ultimately, the model will be modified to include tanks with propellant management devices such as diaphragms and baffles.

Figure 1 illustrates the integration of the CFD model into the current fuel slosh research being conducted at ERAU. Currently, the parameter estimation process requires data from experimental results and other simple pendulum analog models. This parameter estimation process will produce values to be used in the initial spacecraft design for the determination of the Nutation Time Constant (NTC). The new CFD model will serve as source of data that may be used to verify the experimental results. Ultimately, the CFD model will be used to obtain “experimental type” data for scenarios that are not easily reproduced in the laboratory.

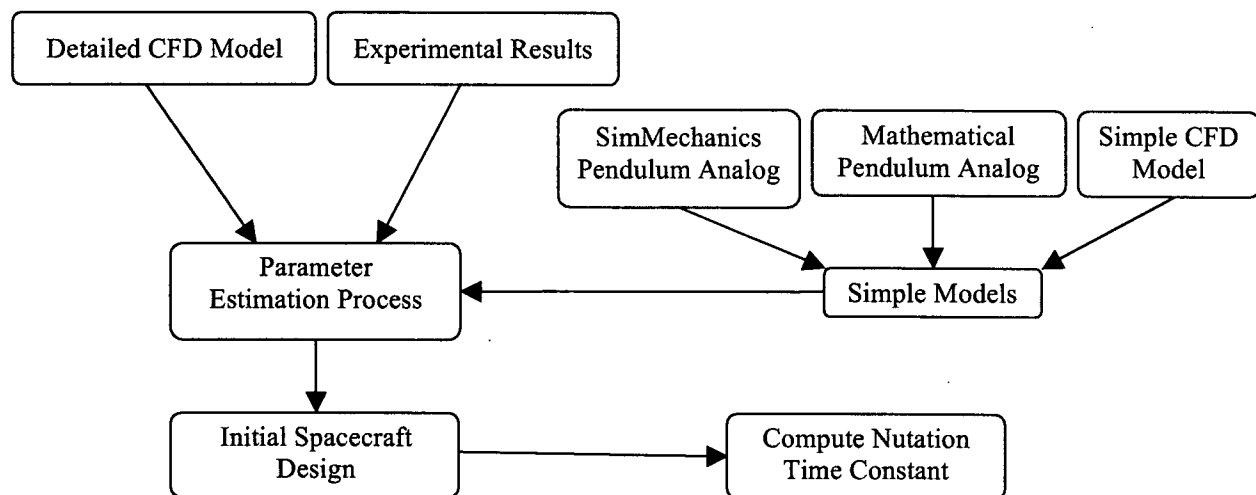


Figure 1. Fuel Slosh Research Flow Chart

Method of Approach:

Experimental Methodology

Experimental data for this research was acquired at ERAU. The fuel slosh research laboratory is equipped with a state-of-the-art linear actuator and data acquisition system. An eight inch in diameter spherical fuel tank is suspended from a frame by cables and attached to a linear actuator as seen in Figure 2. A force transducer placed at the interface between the linear actuator and the fuel tank will measure the forces induced by the fuel slosh and transmit the data to a computer for analysis.

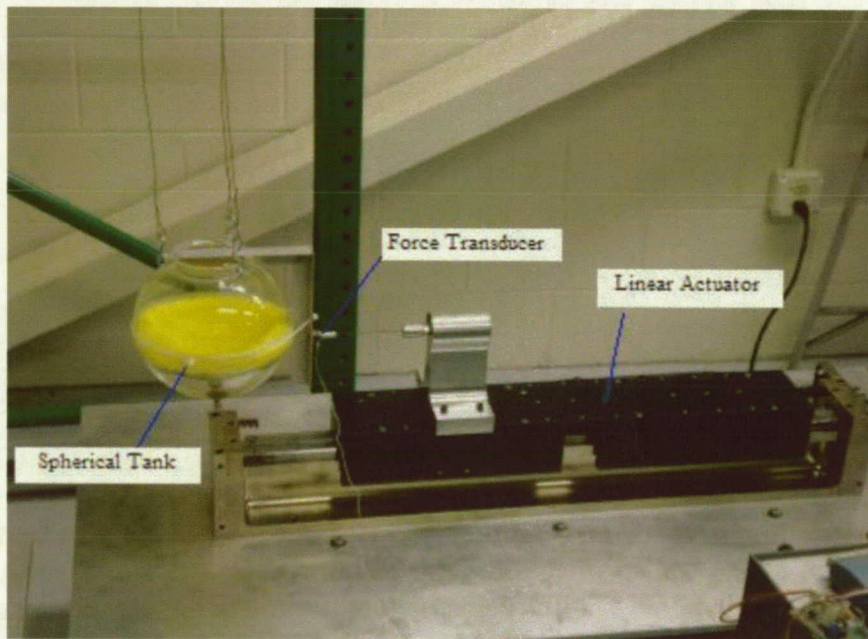


Figure 2. Experimental Set-up [6]

The effect of energy dissipation in sloshing fuel is best illustrated by exciting the tank with a “sudden push” and quickly bringing it to a stop. This causes the fluid to begin sloshing and slowly damp out the oscillations. The damping effect of the fluid is caused by the fact that the fluid in the tank is not inviscid. The viscous nature of the fluid causes an energy dissipation that will eventually bring the fluid to rest. The maximum amplitude of the reaction force decreases over time. This damping is one of the most important effects that the CFD model must replicate in order to validate the fuel slosh behavior. Another quantity of interest is the frequency at which the slosh oscillations are occurring. This is useful information that may be used to prevent frequency coupling and inducing resonance in the fuel tank system. An example of experimental slosh data is shown in Figure 3.

The eight inch sphere at a 60% fill level holds about 2.6 liters of fluid. As seen in Figure 3, the amplitude of these oscillations is on the order of 0.2 lbs. The force transducer used has an accuracy to within 0.001 lbs. This is more than enough resolution to clearly resolve both the

damping effect and the natural frequency of the sloshing liquid. Both of these quantities must be accurately predicted by the CFD model.

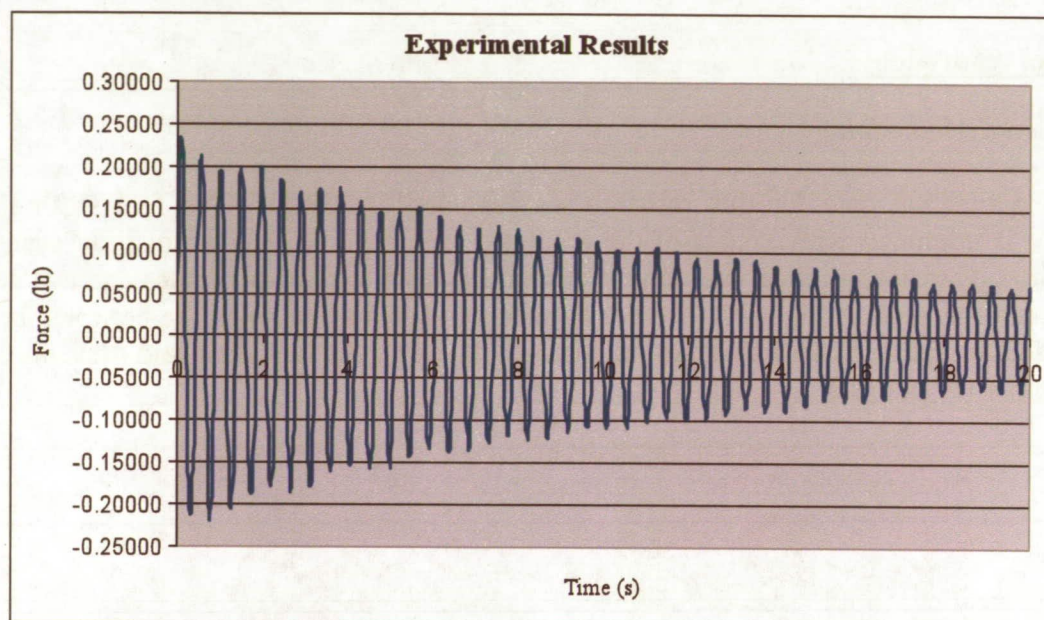


Figure 3. Experimental Results

CFD Model

- Computational Domain

The computational domain used in this study consists of the experimental tank complete with the opening at the top. The tank used is an eight inch spherical tank with a three inch hole at the top that is used for filling. As seen in Figure 4, most of the cells are located at the tank walls. This high resolution at the walls is necessary for capturing the viscous effects that cause the damping of the fluid oscillations. The fully unstructured grid was generated using Gridgen and later modified by Fluent's polyhedral mesh conversion [7]. This polyhedral mesh conversion takes a fully tetrahedral mesh and combines adjoining tetrahedrals to form polyhedrals. The technique reduces the total number of cells without adversely affecting the solution. In this particular problem, the cell number was reduced from 1.2 million to 431,000 therefore greatly reducing the computation time.

A no slip boundary condition was used for the tank walls and a pressure inlet was used for the top. This pressure inlet boundary condition is used to enforce the constant atmospheric pressure at the opening of the tank. Though it may seem trivial, the pressure inlet boundary condition here must allow air to move in and out of the container as the fluid displaces the air over time, but still maintain a constant pressure. Fluent's pressure inlet boundary condition does just this. A standard atmospheric pressure of 101325 Pa was used.

The fully unstructured grid was created using an extrusion method. The first few cells may seem structured but are actually isotropic tetrahedrals. These first few closely spaced cells allow for a y^+ value of less than 2. This puts the first cell well within the viscous sublayer, a desirable property when wall viscous effects are important.

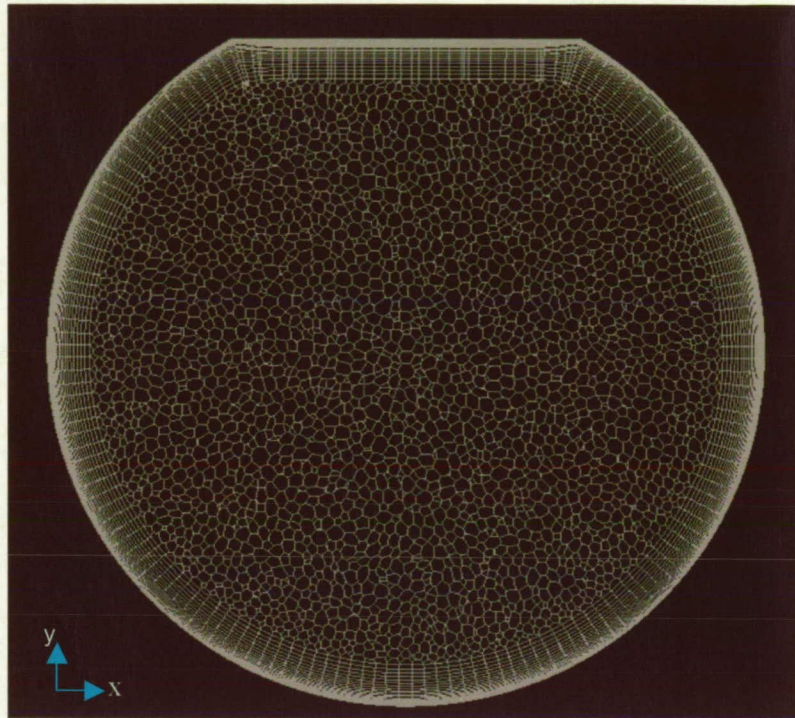


Figure 4. Fully Meshed Tank

- Initial conditions

It was found through experiments that the largest forces generated by the sloshing fluid occurred at a 60% fill level. For this reason, all CFD runs were conducted at this fill level. Since most propellants such as hydrazine are extremely toxic, a liquid with similar properties was used in the experiments. In this particular case, hydrazine is modeled by water which has a very similar density and viscosity. This is one of the problems that can be solved by using a CFD model. Since it is purely computational, there is no problem testing with even the most toxic of chemicals. In order to fully replicate experimental conditions, the CFD computations discussed here also used water.

To simulate the initial push given to the tank in the experimental setup, the liquid in the tank is initialized with a small horizontal (x) velocity of 0.1 m/s. This will allow the liquid in the tank to begin oscillating and eventually die out. The expected frequency is to be around 2 Hz. so the models were set to run for 10 seconds real time at a time-step size of 0.01 s.

- Solver

The CFD model was created using Fluent's 3D, pressure based, unsteady, implicit, cell based solver. Since the problem involves the interaction of two fluids (water and air), the volume of fluid multiphase model was used to keep track of the interface. The viscosity at the wall was modeled using the laminar flow equations. Though several different turbulence models were tested, the laminar model was determined to be the most accurate. Due to the small excitation amplitude the free surface of the sloshing fuel in the experimental setup was fairly smooth and did not display breaking waves or large discontinuities. For this reason there was to be no need to

run a turbulence model. The Discretization method used was second order accurate in space and first order accurate in time (limited by the Volume of Fluid Method).

Results

- Damping Rate

The damping rate is defined as the rate at which successive peaks in the oscillations diminish in an exponential fit to the data over time. Plotted in Figure 5, are the peaks of the positive oscillations as recorded by the force transducer in the experimental setup. Recall that the tank was given an initial “push” and then allowed to dissipate the oscillations. This peak data, is normalized (to cross the axis at $y=1$) and then fitted to an exponential curve. This yields a damping rate of 0.0656 for the experimental data shown in Figure 5.

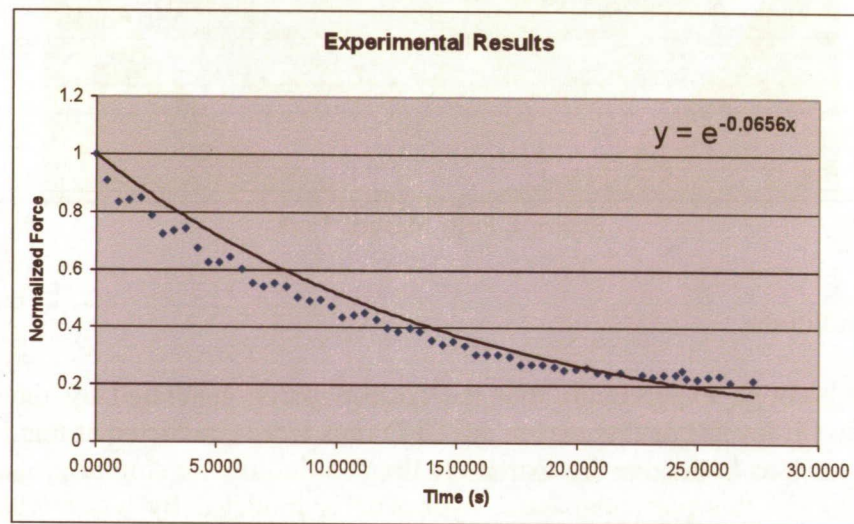


Figure 5. Experimental Damping Results

The force measured by the transducer in the experimental setup, is a force in the direction of excitation (x). In order to calculate this force in the CFD model, a nondimensional force coefficient in the x direction is plotted. This force coefficient is then submitted to the same analysis as the experimental results in order to calculate a damping rate. Table 1 compares the experimental and CFD model damping results.

Table 1. Damping Rate

	Damping Rate	% error
Experimental Results	0.06564	N/A
CFD Model	0.06557	0.11

- Frequency

When the tank is given an initial excitation, the liquid in the tank begins to oscillate at its natural frequency. This natural frequency is defined by the gravitational force, fluid fill level, and tank shape. It is very important to be able to predict this natural frequency. If the spacecraft

begins to oscillate at this frequency, the forces caused by the sloshing liquid can resonate and greatly amplify causing problems with the control system. Table 2 compares the frequency predicted by the CFD model with the measured frequency of the experimental setup.

Table 2. Frequency

	Frequency	% error
Experimental Results	2.148	N/A
CFD Model	2.081	3.12

- Slosh Dynamics

Besides accurately predicting both the damping rate and natural frequency, the CFD model can yield valuable information about the fluid as it sloshes. As seen in Figure 6, this model can predict characteristics such as the shape of the free surface and the velocity field of the fluid as a function of time. This valuable information helps to visualize the fluid motion, and gain a deeper understanding of the physics behind spacecraft propellant slosh.

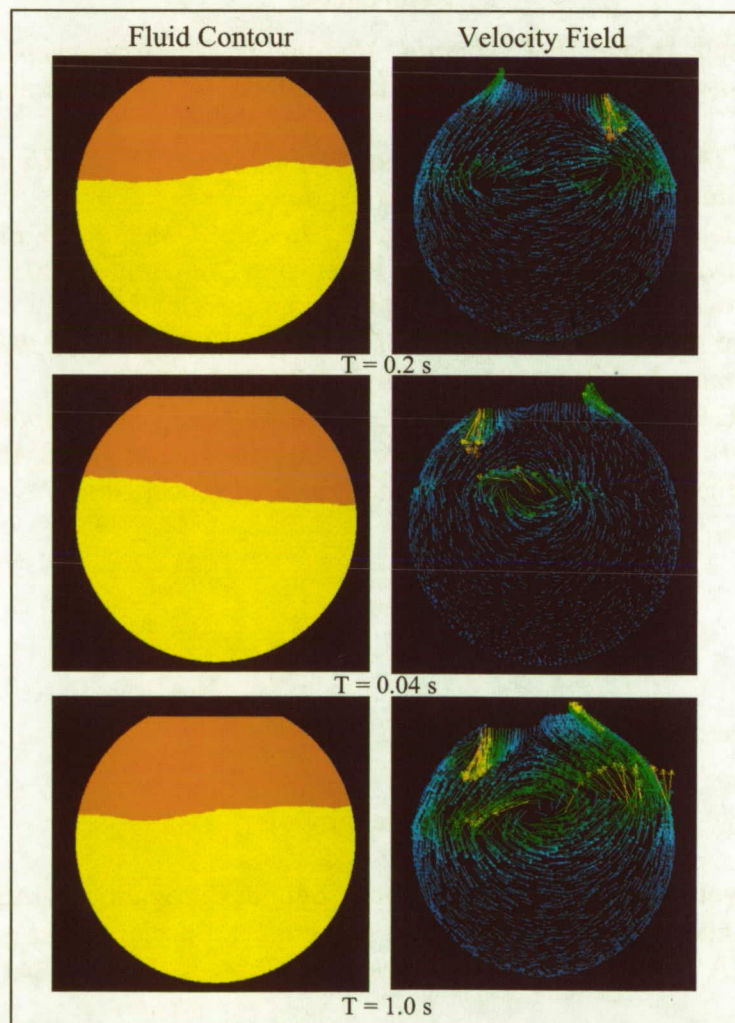


Figure 6. Fluid Contour and Velocity Vector Field at 0.2, 0.4, and 1 s.

Conclusion

This CFD model was successful for simulating the small amplitude slosh of free surface tanks. Further research will be aimed at creating models for more complex environments such as tanks fitted with propellant management devices (PMD's) like baffles and diaphragms. These models will eventually allow the simulation of full scale flight tanks.

Thanks to the power of Computational Fluid Dynamics, an accurate model of spacecraft fuel slosh can be created. This model allows for the prediction of many different parameters that are useful for the completion of any space mission. Besides being an accurate source of data, it allows for a deeper look into the dynamics of sloshing fuel. All of this can be achieved without the cost of building an expensive experimental setup. CFD models like these are the future of fluid studies.

References

- [1] Weihs, D., and Dodge, F.T., *Liquid Motions in Nonaxisymmetric, Partially Filled Containers Rotating at Zero Gravity*, AIAA Journal of Spacecraft and Rockets, Vol. 28 1991.
- [2] Dodge, F.T., *The New Dynamic Behavior of Liquids in Moving Containers*, Southwest Research Institute, Technical Report, San Antonio, Texas.
- [3] Schlee, K. Gangadharan, S.N., Ristow, J., *Advanced Method to Estimate Fuel Slosh Simulation Parameters*, 41st AIAA Joint Propulsion Conference, 2005.
- [4] FLUENT Software Version 6.2.16, © 2004 Fluent Inc.
- [5] Hirt, C.W. and Nichols, B.D., *Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries*, Journal of Computational Physics 39, 201, 1981.
- [6] Chatman, Y. Gangadharan, S.N. Marsell, B., *Mechanical Analog Approach to Parameter Estimation of Lateral Spacecraft Fuel Slosh*, 49th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, 2008.
- [7] Gridgen 2007-2008 Pointwise, Inc

Acknowledgments

The authors would like to thank the Launch Services Program at NASA Kennedy Space Center for their continuous help and support for this project. Special thanks go to the fellowship offered by the NASA Graduate Student Researchers Project (GSRP) that makes this study possible.