Simulation of Fluid Sloshing in High-Acceleration Environments for Aerospace Applications

Hesha Amin[1] *and Connor Perrine*[2]
University of Georgia, Athens, Georgia, 30602

The University of Georgia's International Rocketry Engineering Competition (IREC) Team developed a fluid sloshing payload experiment to test and measure the movement of water during a level 3-certified high powered rocket's flight to 10,000 feet. In order to accurately predict the sloshing of the fluid, a computational fluid dynamics (CFD) approach was taken to model this movement. This research delves into utilizing the altitude, velocity, and acceleration data from the UGA IREC previous launch to create a detailed simulation of the payload in ANSYS. This simulation provides a scaled model of how liquid fuel behaves during a rocket's liftoff phase, which allows engineers to better understand the issue of sloshing fluids and improve solutions to mitigate their effects.

I. Nomenclature

CFD - Computational Fluid Dynamics
CoM - Center of Mass
IREC - International Rocketry Engineering Competition
RANS - Reynolds-Averaged Navier-Stokes
SPH - Smoothed Particle Hydrodynamics
UGA - University of Georgia
VoF - Volume of Fluid

II. Introduction

Fluid sloshing refers to the oscillation of a liquid's free surface within a container due to acceleration forces, a phenomenon particularly critical in aerospace applications [1]. Fluid sloshing in containers is a serious concern for engineers, manufacturers, and consumers alike, causing multiple problems in an aircraft's structural stability, vibrations, and acoustics. Sloshing has caused many failures in the aerospace industry, one notable being SpaceX's Falcon One failing to reach orbit due to unforeseen oscillations in the upper stage caused by sloshing in the liquid oxygen tank [2]. With the addition of baffles and pressurized tanks, sloshing is mitigated, but as the aerospace industry continues to rapidly develop, these solutions will become outdated. For this reason, further research into how fluid sloshing occurs and how to best predict sloshing is needed to prevent the dangerous consequences.

The University of Georgia's (UGA) Rocketry Team competes in the annual International Rocketry Engineering Competition (IREC), where students design and manufacture a level 3-equivalent high powered rocket to compete against hundreds of other university teams. The goal for the UGA IREC team is to launch a rocket to 10,000 feet carrying a payload experiment and successfully recover both the rocket and the data collected. For the 2025 competition, UGA rocketry decided to conduct a fluid sloshing experiment. In this experiment, the sloshing of water is measured to demonstrate the impacts of fluid motion during high acceleration situations, similar to those encountered in fuel tanks during a rocket's flight. In order to predict the sloshing effect, a simulation using Computational Fluid Dynamics (CFD) software is used to create an accurate model of the payload. Utilizing data collected from a previous test launch, ANSYS is employed to input the varying accelerations experienced in flight and simulate the sloshing payload.

III. Fundamentals of Fluid Sloshing

Under high acceleration environments, such as those experienced in rockets, a fluid's motion is highly sporadic, leading to high free surface deformation and oscillations. The sloshing behavior is particularly pronounced in partially filled tanks, where the liquid remains unconstrained along the free surface, making it highly susceptible

---

[1] Undergraduate Student, College of Engineering, AIAA Student Member (ID: 1810351)
[2] Undergraduate Student, College of Engineering, AIAA Student Member (ID: 1740846)

to external excitations. The motion of the fluid is primarily governed by the interplay of three key forces: gravity, inertia, and surface tension. Gravity acts as the restoring force to return the liquid to its equilibrium state, while inertia dictates the fluid's resistance to changes in motion. The surface tension influences the formation of waves at the interface between the fluid and the gas inside the tank. The complex interaction between these three forces determines the amplitude, frequency, and damping characteristics of sloshing [1].

The nature and severity of the sloshing is dependent on multiple factors, including the external accelerations enacted on the fluid, the tank geometry, and the fill level. A larger surface area and lower fill levels generally promote more pronounced sloshing, whereas the presence of internal structures, such as baffles, can suppress fluid motion by disrupting the wave propagation. The disruption of sloshing is vital for the successful operations of rockets since the movement of fuel inside a tank introduces unwanted dynamic loads, shifts in CoM, and induces structural stresses. Without proper settling of the fluid, the sloshing behavior is amplified and causes these problems. In microgravity situations, the absence of a dominant gravitational force leads to unique sloshing behaviors, where capillary effects become more pronounced [3].

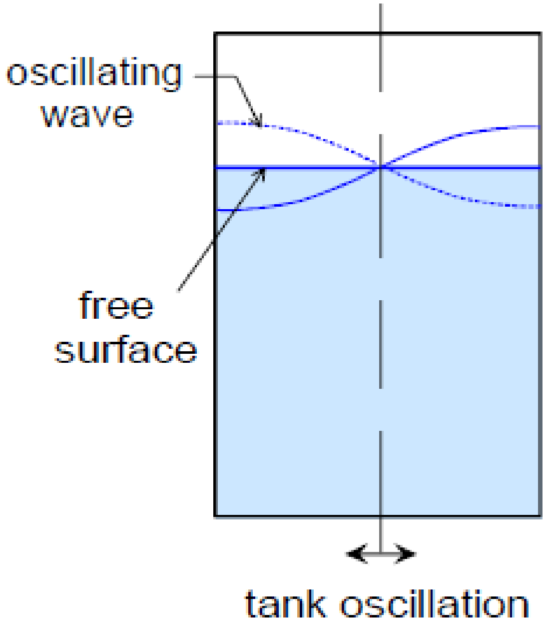

Fig. 1 Fluid Sloshing Induced by Tank Movement

Fluid sloshing can be understood through mathematical modeling and fluid dynamics formulae, such as the Navier-Stokes equation. The most common mathematical model for sloshing behavior is the mass pendulum model, where the fluid's motion is represented by the oscillation of a swinging pendulum [1]. This model is then applied with the Navier-Stokes equation, where density, velocity, pressure, and viscosity are related together to calculate the oscillation motion from an external force.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u} + \mathbf{F} \tag{1}$$

Solving these equations for sloshing dynamics by hand requires a lot of time and intensive calculations, so computer modeling systems, such as CFD, were developed to model fluid dynamics. Various simulation techniques, including the Volume of Fluid (VoF) method, Smoothed Particle Hydrodynamics (SPH), and Reynolds-Averaged Navier-Stokes (RANS) models, were developed to capture the complex interplay between liquid and gas phases. These simulations allow engineers to predict how sloshing evolves over time and develop mitigation strategies, like baffles, to minimize its adverse effects.

Over the past decades, researchers extensively studied sloshing effects using CFD simulations. The VoF method is most widely employed in aerospace applications due to its ability to track the interface between liquid and gas phases accurately. For this reason, the VoF method and the CFD software ANSYS are used to model this experiment's sloshing effect while under the influence of a rocket's thrust.

IV. Experiment and Simulation

The experiment consists of a 15 cm tall, 8 cm diameter cylindrical tank. The tank is filled with 60% water and 40% air at atmospheric pressure. This tank would be inside the UGA IREC rocket and flown. This experiment is for the 2025 IREC competition, but UGA only has data from the 2024 IREC competition. Since the primary goal of the CFD simulations is to predict the sloshing of the fluid, the simulation used the previous year's data of flight to 8,971 feet to test the validity and feasibility of modeling fluid sloshing during liftoff. During this flight, the tank would experience various accelerations and vibrations due to the M1939W solid rocket motor and atmospheric forces. The onboard altimeter provided data of the altitude and velocity at intervals of 0.05 seconds. From this, the acceleration for each time step can be derived. As the rocket reaches apogee, the drogue parachute is deployed, then, around 1,500 feet, the main parachute is deployed. The acceleration of the entire 321.5 second flight is shown in Figure 2.

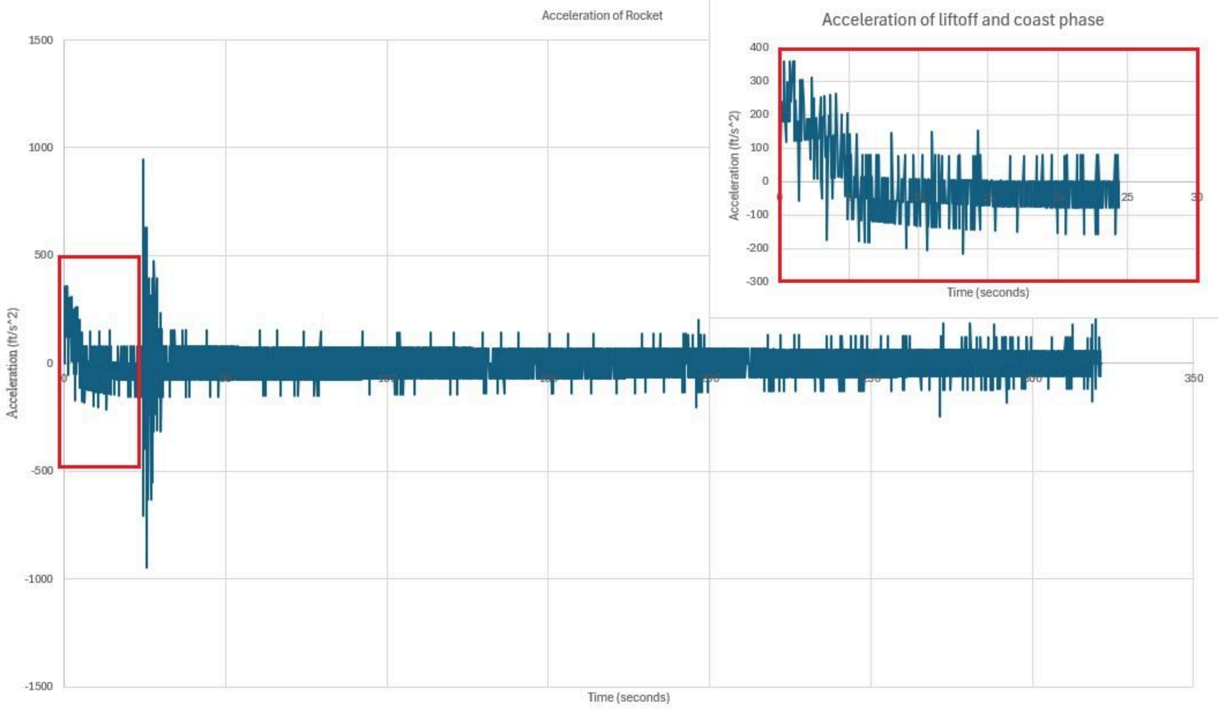

Fig. 2 Acceleration of Rocket

The acceleration has four main stages: liftoff, coast, parachute deployment, and descent. Since the descent stage has relatively constant acceleration from gravity, and the parachute deployment has a near instantaneous change in acceleration, the initial liftoff and coast phase was considered for the sloshing data. During the liftoff stage, the motor burns for 6.2 seconds, propelling the rocket with 500 pounds of thrust and 2,302 pound-seconds of impulse. This high thrust produces a large acceleration profile during the liftoff stage, as shown in Figure 2. After the motor burns out, the rocket travels around 730 feet per second, and decelerates due to gravity until apogee, where velocity equals zero. This coast phase lasts for 18.2 seconds until the drogue parachute is deployed, around T+ 24.4 seconds.

The data collected during this time period consists of 490 acceleration data points corresponding to the trajectory from liftoff to apogee. We concluded that simulating all 490 accelerations would be challenging and unreasonable, so the dataset was condensed into 20 representative acceleration segments using an averaging technique. By narrowing our focus to these 20 iterations, we aimed to streamline our analysis, concentrating on capturing the most critical dynamics of fluid behavior under the influence of motor-induced thrust. Each segment captures the key characteristics of the acceleration profile while significantly reducing the sporadic errors and computation needed. This is shown in Figure 3, with a logarithmic best fit line to represent a function of average acceleration.

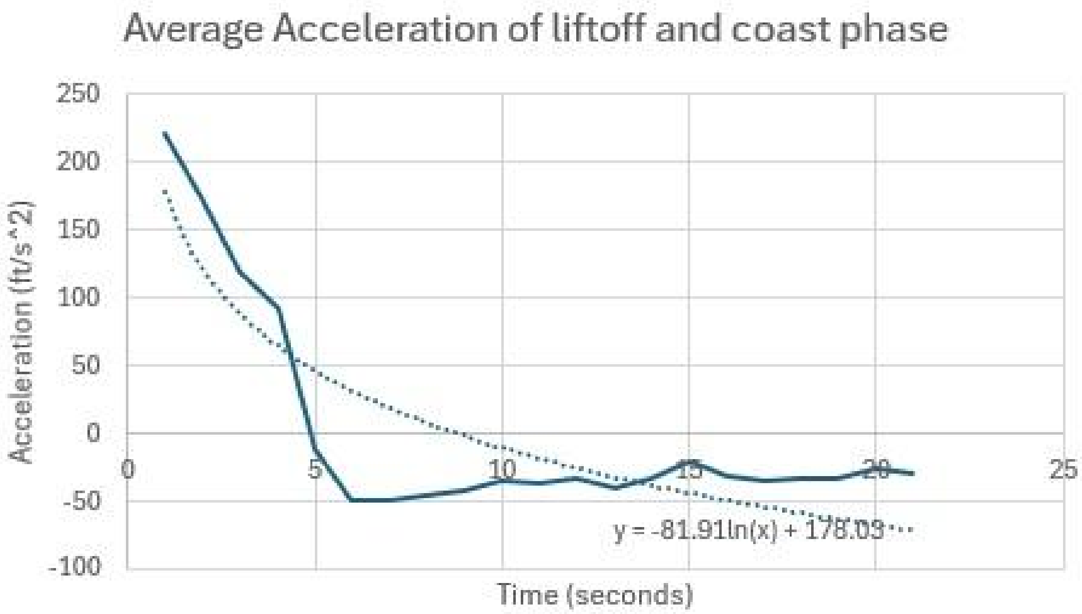

Fig. 3 Average Acceleration of Liftoff and Coastal Phase

Using ANSYS Fluent, a commonly used CFD software in industry, the tank geometry, boundary conditions, and accelerations are all inputted to create a simulation of the sloshing. The VoF method is applied to track the liquid-gas interface, ensuring accurate representation of wave motion within the tank. The acceleration profile is applied in the y direction, and the simulation is run with a time step of 0.005 seconds. The simulation is run for each average acceleration, totaling in 18 iterations of the simulation at different accelerations. Figure 4 shows the simulated tank at time zero, when the water is settled and stationary. The red represents the water, while blue represents air, and the green-yellow represents the boundary interface between the water and air.

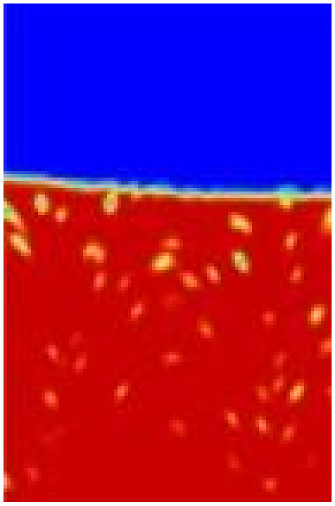

Fig. 4 Settled Water in Tank

Each iteration of the simulation is conducted under both steady-state and transient conditions, enabling the evaluation of sloshing wave patterns, pressure variations, and structural loading. The results provide insights into turbulence intensity, oscillation frequencies, and the regions of high kinetic energy transfer within the fluid. The rapid changes in acceleration caused the water in the tank to have sporadic sloshing. This caused some problems with the simulations due to floating point errors, so the simulation was run with smaller time steps to be more accurate. After the simulation was run for the 18 iterations, the final predicted sloshing of the tank during the liftoff and coast phase was evaluated. By analyzing the data, key trends are identified, and potential strategies for mitigating sloshing effects can be explored.

V. Results and Discussion

The varying acceleration throughout the rocket's liftoff and coast phase induced a large sloshing effect. In our computational fluid dynamics (CFD) simulation, we conducted an extensive analysis of the behavior of water within the rocket's fuel tank during the critical phase of liftoff. A total of 20 iterations, with a focus on the dynamic response of the water to varying generalized accelerations as the rocket ascended. The first two iterations were dedicated to capturing the fluid at rest, allowing baseline conditions to observe its subtle vibrations before the ignition of the motor. This preliminary data was essential for understanding the initial state of the fluid, as it set the stage for the subsequent interactions that would occur during launch. As the motor ignited, the water within the tank experienced instantaneous upward acceleration, a direct consequence of the thrust generated. This rapid acceleration caused the water to crest and form distinct wave patterns, visually depicting the fluid's dynamic response to the rapidly changing forces acting upon it. The interaction between this upward acceleration and the downward force of gravity further complicated the scenario, resulting in a dynamic sloshing effect characterized by oscillations and complex wave interactions.

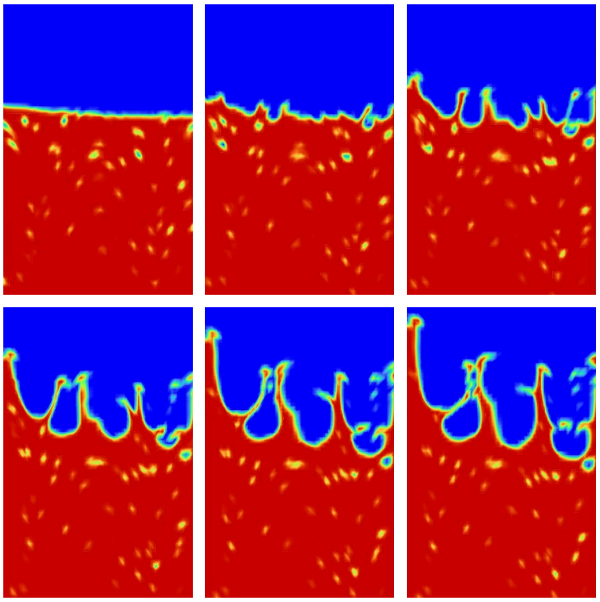

Fig. 5 First Six Iterations of Sloshing Simulation

In order to achieve the 0.005 second time step for each simulation, a linearized refinement of the acceleration between each known value was employed, allowing for ANSYS to accurately model the sloshing motion with smaller time steps. This refinement precisely captures the sloshing behavior of the fluid, enabling the simulation of rapid changes in the water's movement. Realistically, simulating drastic changes in accelerations—from 0 to 176 ft/s² and then to 239 ft/s²—presents significant challenges, as high accelerations induce turbulent fluid motion that complicates the accuracy of CFD results. By focusing on the 20 iterations and implementing smaller time steps, the intense dynamics of sloshing behavior are visualized and the subtleties of fluid movement during these critical changes are explored.

Throughout the iterations, close attention was paid towards the effect of harmonic motion in the x-direction. The deliberate introduction of simple harmonic motion was aimed at investigating how vibrations induced by the motor during launch would impact the sloshing dynamics within the tank. This harmonic motion added a layer of complexity to the fluid behavior, with oscillations influencing the distribution of the fluid and generating conditions that could potentially exacerbate sloshing. The simulations revealed a notable asymmetry in the distribution of water within the tank, with a significant accumulation on the left side, contrasting with the right side, which exhibited comparatively less fluid. This uneven distribution can be attributed to the complex interactions between the upward thrust and the vibrational effects introduced by the harmonic motion. The investigation into the impact of these vibrations was particularly vital for understanding the potential challenges associated with sloshing. The induced vibrations could lead to undesirable conditions that compromise the rocket's stability during ascent. The fluid's response to both the thrust generated by the motor and the harmonic vibrations illustrated the intricate interplay between these factors, emphasizing the importance of considering their combined effects when analyzing fluid behavior in rocket applications.

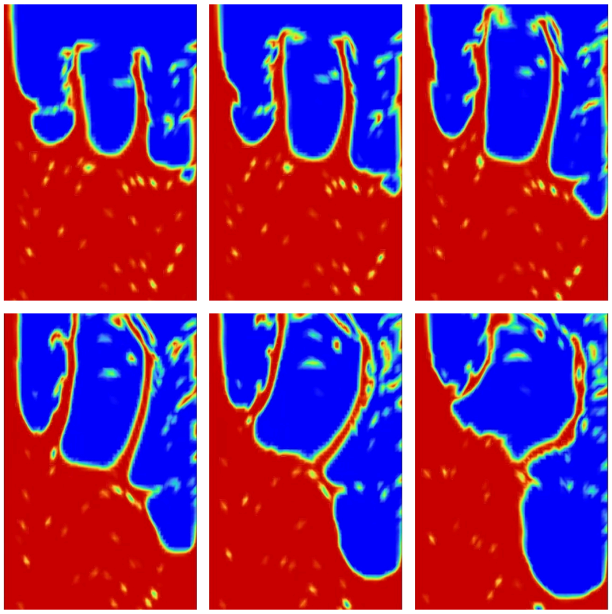

Fig. 6 Iteration Seven to Twelve of Sloshing Simulation

From the next six iterations of the simulation, as shown in Figure 6, the fluid began to move to one side of the container. This oscillation happens in a small amount of time, which would cause extreme loads on the tank walls. The movement of fluid to one side also shifts the center of mass, which would cause the rocket to lose stability and attitude control.

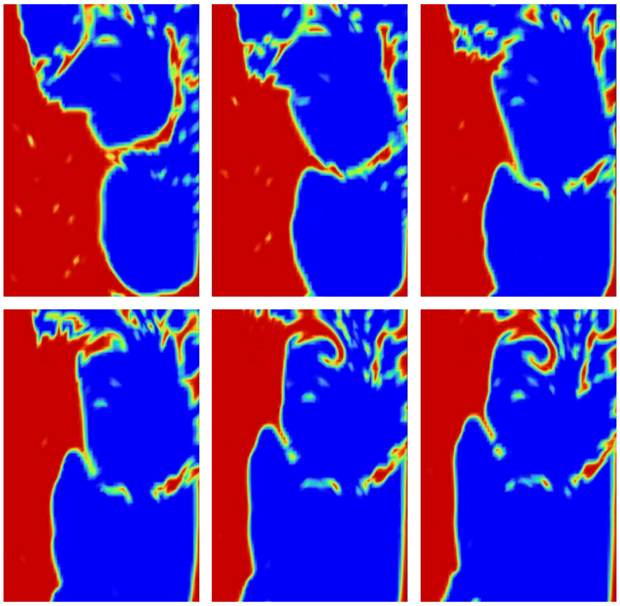

Fig. 7 Iteration Thirteen to Eighteen of Sloshing Simulation

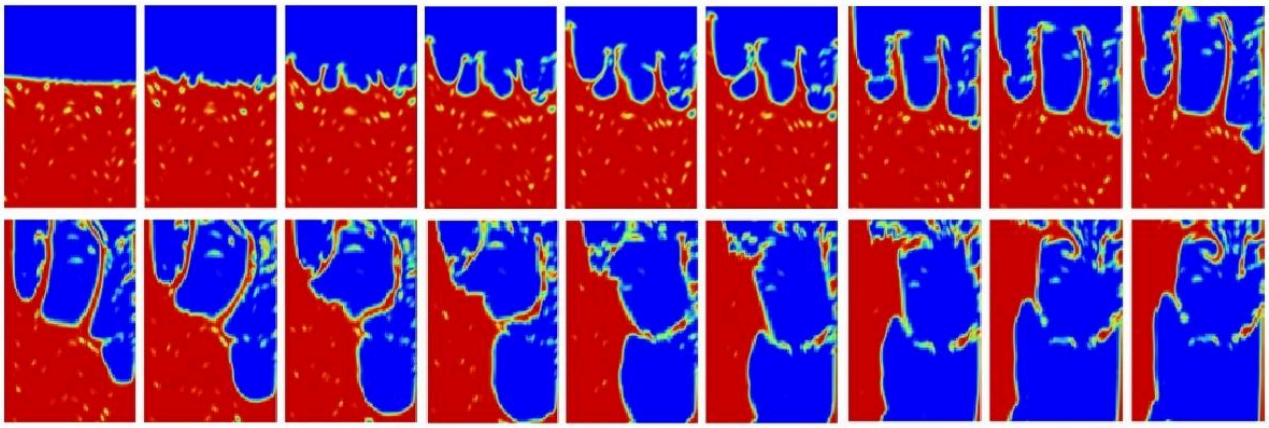

Fig. 8 All Eighteen Iterations of Sloshing Simulation

Following the completion of the 18 iterations focused on fluid dynamics during liftoff, the final predicted sloshing behavior was evaluated. The analysis of the data revealed the correlation between the frequency of applied vibrations and the resultant sloshing amplitude.

The insights gained from this CFD simulation will significantly inform the design and engineering of future rocket systems. By recognizing the complex interplay between the thrust generated by the motor, gravitational forces, and harmonic vibrations, engineers can enhance the understanding of fluid dynamics in aerospace applications. This knowledge will be instrumental in optimizing tank designs, implementing internal structures to dampen slosh, and improving operational procedures to ensure the stability and performance of the vehicle during critical phases of flight.

VI. Conclusion

The UGA IREC Team made significant strides in understanding fluid sloshing dynamics through dedicated experimental and computational efforts. By leveraging advanced CFD simulations, models of water within a rocket's fuel tank during the critical phases of liftoff and coast were visualized. This research not only addresses a pressing concern in aerospace engineering—namely the unpredictable nature of fluid sloshing—but also provides valuable insights that can inform future rocket design and operational strategies.

The decision to concentrate on a consolidated data set of 20 iterations was crucial. This strategic approach focused on the most critical phases of the rocket's ascent, thereby enhancing the accuracy of our simulations while keeping computational demands manageable. By condensing the time step for simulations, the rapid changes in fluid behavior that occur under high acceleration were captured, revealing the intricate dynamics at play. The findings highlighted the pronounced effects of both thrust-induced acceleration and harmonic vibrations on fluid distribution within the tank. The asymmetrical pooling of water observed during the simulations underscores the importance of considering these factors when designing fuel tanks for high-powered rockets. Furthermore, the exploration of sloshing dynamics in both steady-state and transient conditions has illuminated the relationships between various influencing factors, providing a foundation for developing targeted mitigation strategies.

The implications of this research extend beyond the immediate context of the IREC competition. As the aerospace industry continues to evolve, the insights gained from these simulations can contribute to the ongoing effort to improve the safety and reliability of rocket systems. Understanding the complexities of fluid motion under high-stress conditions will enable engineers to design more effective solutions, such as the implementation of baffles and internal structures, that can dampen slosh-induced effects. The groundwork laid by this project sets the stage for further investigations into more complex fluid dynamics scenarios. Future studies can explore varying payload configurations, different fluid types, and the effects of microgravity conditions to enhance our knowledge of sloshing behavior in diverse environments.

VII. References

[1] Dodge, F, *The New "Dynamic Behavior of Liquids in Moving Containers"* San Antonio, Texas, 2000, Chap 2.
[2] Cowing, K. (2008, August 6). "SpaceX determines cause of Falcon 1 launch failure". *SpaceRef.* URL: https://spaceref.com/uncategorized/spacex-determines-cause-of-falcon-1-launch-failure/
[3] Zheng, Z., et. al., "Simulation of sloshing and settling behavior of liquid hydrogen in an insulated tank during coast period", *International Journal of Hydrogen Energy,* published 30 Nov. 2024