DOI: https://doi.org/10.36094/sc.v89.2025.CFD_Studies_on_Sloshing_of_Liquid_Fuel.Yadav.117

ARTICLE

Sci. and Cult. 91 (1–2) : 117-125 (2025)

CFD STUDIES ON SLOSHING OF LIQUID FUEL IN FUEL TANK UNDER EXTREME DYNAMIC CRASH CONDITIONS

SHAMBHOO YADAV[1*], MUTHUSELVAN GOVINDARAJ[1], AND VINAY CHIKKANAYAKANAHALLI ANAND[2]

This study presents a detailed computational fluid dynamics (CFD) analysis of sloshing behavior in aircraft wing tanks under various acceleration conditions. The analysis utilizes the Volume of Fluid (VOF) method with the PRESTO algorithm for decoupling pressure-velocity equations, ensuring stability and accuracy, particularly for flows with sharp gradients. A high grid resolution and small time step are employed to accurately capture complex sloshing behavior. The simulation results are validated against published literature, demonstrating good agreement in terms of pressure amplitude and liquid oscillations and confirming the suitability of the method for sloshing analysis in wing tanks. The load factor values of emergency landing condition, as specified in FAR23, are considered in the present study. The study investigates the effects of different load factor values and acceleration directions on fuel movement and pressure variations in the tank. The results provide insights into fuel behavior under acceleration, which is crucial for designing fuel systems that can withstand forces experienced during flight, especially during emergency landing conditions.

Keywords: *Sloshing, fuel tank, VOF, fluent, unsteady simulation*

Introduction

Sloshing of fuel in aircraft wing tanks is a critical phenomenon that can significantly affect the stability, safety, and structural integrity of an aircraft. During flight maneuvers, especially during high acceleration, the fuel inside the tanks experiences accelerations that can cause it to slosh around, leading to changes in the aircraft's center of gravity and potentially impacting its handling characteristics. Therefore, accurate prediction and analysis of sloshing behavior are essential for designing safe and efficient aircraft fuel systems.

The liquid sloshing loads, when subjected to resonant frequency and multiple coupled excitation, can lead to aggressive sloshing and wall pressure can escalated beyond the limits[1]. In the analysis carried out by Hou et al.[1], the volume of fluid (VOF) method with the ANSYS-FLUENT software in the 2-D rectangular tank was used to simulated the liquid sloshing. The tank was subjected to sinusoidal sway, roll, heave and multiple coupled excitation. The dynamic mesh was used to impose the time varing loads.

Experimental and CFD studies had been carried out by Frosina et al.[2] on the 3-D fuel tank to optimise the tank geometry for better fuel suction at all conditions and to reduce additional noise and vibrations due to sloshing. Their test bench was capable of allowing all six Degrees of Freedom (DOFs) with the help of six independent actuators. External excitation of pitch, yaw and roll up to 50 deg were applied on the fuel tank with a peak acceleration of 1 g. During the experiment, liquid surface

1 Propulsion Division, CSIR-National Acrospace Laboratories, Bengaluru 560017, India
2 RNCAC, CSIR-National Acrospace Laboratories, Bengaluru 560017, India
* Corresponding Author: shambhu_yadav@nal.res.in

motion was captured by the camera and tracked using MATLAB. Commercial code PumpLinx was used for carrying out the multiphase CFD studies. They have used $k - \varepsilon$ to model the turbulence inside the tank and 2nd order upwind scheme, which helps stabilise the solution. Their analysis revealed a maximum error percentage of 5.6% in comparing CFD and experimental data in the estimation of free surfaces and the fluid's centres of gravity.

Mougha et al.[3] has studied the dynamics of water in fuel tanks during flight maneuvers. Their study was to understand the local composition and movement of water in the fuel tank due to dynamic motion of the aircraft. They observe that oscillatory motion of the fuel tank creates a complex water-distribution field, which may leads to water entering the fuel line. The taking off and landing of the aircraft can also affects the distribution of water in the fuel tank.

Bernhard et al.[4] carried out studies using a rectangular tank to investigate the effectiveness of two different multiphase approaches for fluid density and viscosity in a commercial CFD code. Their study found that an inhomogeneous multiphase model was more suitable than a homogeneous model, despite the fact that previous research had favored the latter due to its lower computational cost. Their results showed that the homogeneous model underestimated peak pressures by up to 50%, whereas the inhomogeneous model closely matched the experimental data. Further examination of the relative velocity at the fluid interface supported the conclusion that the inhomogeneous model was more appropriate for simulating violent sloshing flows.

Xue et al.[5] conducted study of liquid sloshing in a tank near resonant frequency excitation. They developed a 3-D numerical model NEWTANK using virtual boundary force (VBF) method for two-phase gas-liquid sloshing. They also implimented LES to model the turbulent flow effect. Their model shows good agreement between numerical results and theoretical solutions. Further they used this model to investigate baffle inside the 3-D tank to reduce liquid sloshing under near-resonant excitation for pitch motion. They obseverd significant reduction in oscillation in presence of baffle as compared to no baffle case.

The paper is organized into the following sections. First, the governing equations of CFD approach are briefly introduced followed by a section on the CFD parameter settings applied in the present numerical analysis. Subsequently, the present CFD approach had been validated for a case with a single sinusoidal ex-citation source. Finally, the emergency load factor from various direction, such as forward, sideward, downward and upward, had been analyzed.

The sloshing analysis had been carried out to understand the dynamic oscillation of fuel levels within wing tanks. This involves a detailed examination of the fluctuating patterns of fuel surface movement, particularly in response to emergency landing condition. In addition to that, the wing tank pressure over time was also monitored. The variations of the free liquid surfaces captured by the CFD method are also discussed. The dynamic nature of the fuel movement influences the pressure within the tank; therefore, it is crucial to understand these variations. This insight is essential for optimizing the design and performance of the aircraft, especially in critical situations, such as emergency landings. The present study considers the load factor values specific to emergency situations.

Mathematical Model / Governing Equation

The numerical simulation of liquid sloshing can be carried out using the various fluid interactionn model to estimate the fluid mixture properties. However selection of interaction model should be done by considering numerical stability, accuracy, grid requirement and computational cost. Typically homogeneous model as a fluid interactionn model has been prefered for sloshing analysis[4]. Inhomogenous multiphase model are more general, but highly computationally expensive in comparision with homogeneous model. It require an additional velocity field solution for both fluid with the help of mass and momentum transfer.

Solving Navier–Stokes equations using computational fluid dynamics (CFD) with the help of commercial solver ANSYS-FLUENT has been chosen for the analysis. The present study employs the volume of fluid (VOF) method for numerical analysis. This method is suitable for capturing the motion of the free surface during sloshing inside a liquid container. This method utilizes a characteristic function called F, which acts as a step function to determine the volume of the fluid and locate the position of the free surface. The value of F represents the volume fraction of a cell that is filled with the liquid.

$$F(x, y, t) = \begin{cases} 1 & \text{fluid} \\ 0-1 & \text{free surface} \\ 0 & \text{empty} \end{cases} \tag{1}$$

F=0 or F=1 represents that the cell is away from the liquid interface is fully filled with air or liquid; while

0< F <1 represents the location of the free surface. The advection equation for F is

$$\frac{dF}{dt} = \frac{\partial F}{\partial t} + \left(u.\nabla\right) F = 0 \qquad (2)$$

The advection equation for F involves the velocity (u), and the gradient of F indicates the normal direction of the free surface. The position of the free surface can be obtained approximately by the piecewise linear scheme of geometric reconstruction.

The sloshing behavior of liquid can be represented by an incompressible viscous fluid flow with a free surface. It is governed by the Navier-Stokes equation and the continuity equation. These equations involve the velocity (u), pressure (p), density (ρ), acceleration of gravity (g), body force (F), and viscosity of the mixture.

$$\frac{\partial u}{\partial t} + \nabla \cdot (uu) = -\frac{1}{\rho}\nabla p + \nabla\left[\frac{\mu}{\rho}\left(\nabla u + \nabla u^T\right)\right] + \rho g + F$$

$$\nabla \cdot u = 0 \qquad (3)$$

Numerical / CFD Methodology

Present study utilizes the commercial CFD software ANSYS-FLUENT for the analysis. The volume of fluid (VOF) method has been used for numerical analysis discussed in previous section. The pressure-velocity equations has been decoupled using the PRESTO algorithm. This algorithm, for decoupling the pressurevelocity equations is particularly advantageous due to its stability and accuracy in handling flows with sharp gradients. The volume fraction transport equation is addressed through an explicit time-marching scheme, which ensures robustness and efficiency in capturing transient behavior. The interface utilizes a piecewise linear scheme to compute convective fluxes which significantly enhancing accuracy, particularly in regions characterized by complex flow dynamics. Convergence is reached when residuals for all governing equations fall below 10^{-5}.

A time step of 0.005 s is selected. Non-slip walls define the tank boundaries, and the fuel tank is initially stationary.

CFD Methodology Validation

In order to validate the proposed computational approach for sloshing analysis of fuel tank, unsteady CFD simulations are carried out in the model 2-D tank. Fig. 1 show the details of reference geometry, taken from Hou et al.[1], used for the validation study. CFD simulations are carried out in 2-D tank with perturbing form of acceleration applied along the x direction. The sinusodial form of acceleration used in the study is given by eq. 4. The acceleration amplitude variation with respect to time is shown in Fig. 2. The oscillating form of pressure at the point P_2(as shown in fig. 1) in the 2-D tank due to the perturbing acceleration is captured from the results of unsteady CFD simulations.

$$\ddot{x} = -\omega^2 x_0 \sin(\omega t) \qquad (4)$$

where,

$\ddot{x}$ = Horizontal translational acceleration is angular frequency

$\omega = 2\,\pi\, f_n$ is angular frequency

x_0 = 0.015, Horizontal displacement amplitude

f_n = 0.692 Hz

The oscillating form of pressure at point P_2 in the 2-D tank captured from the analysis is shown in Fig. 3. The results obtained from the present work are comparable to the results available in reference documents in terms of pressure amplitude and liquid oscillations. So, the proposed computational approach using Volume of Fluid (VOF) method can be applied for the actual sloshing analysis of wing tank model.

Emergency Landing Condition (FAR 23.561) simulation

The developed computational approach is applied to the sloshing analysis of the generic aircraft wing tank. Emergency landing condition of FAR 23 [add reference] is considered for realizing the boundary condition of computational simulations.

FAR 23.561(b)(2): The ultimate static load factors experienced by the occupants for emergency landing conditions are the following

(i) Upward, 3.0 g for normal, utility, and commuter category airplane

(ii) Forward, 9.0 g

(iii) Sideward, 1.5 g

(iv) Downward 6.0 g

These test loads must be applied for three seconds in each direction of the load.

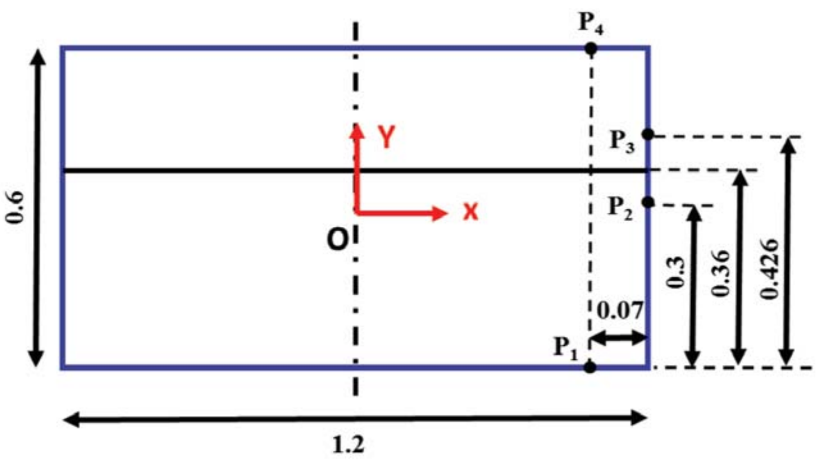

Fig. 1: Geometry of 2-D tank used for validation1

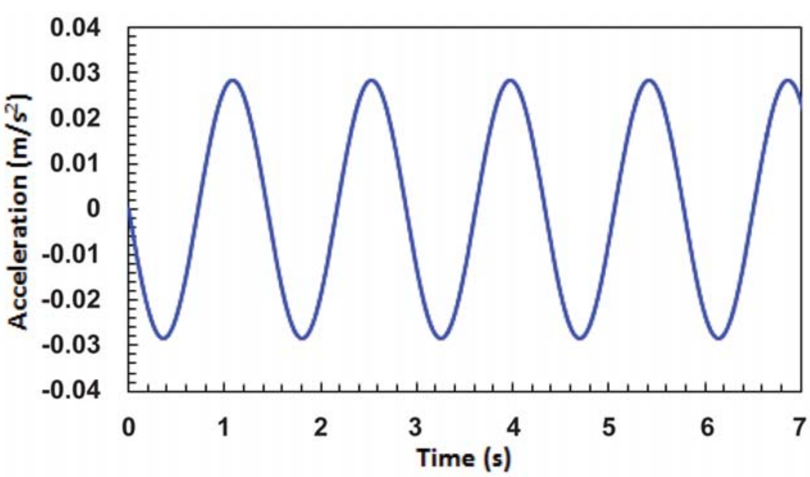

Fig. 2: Acceleration profile used for validation study[1]

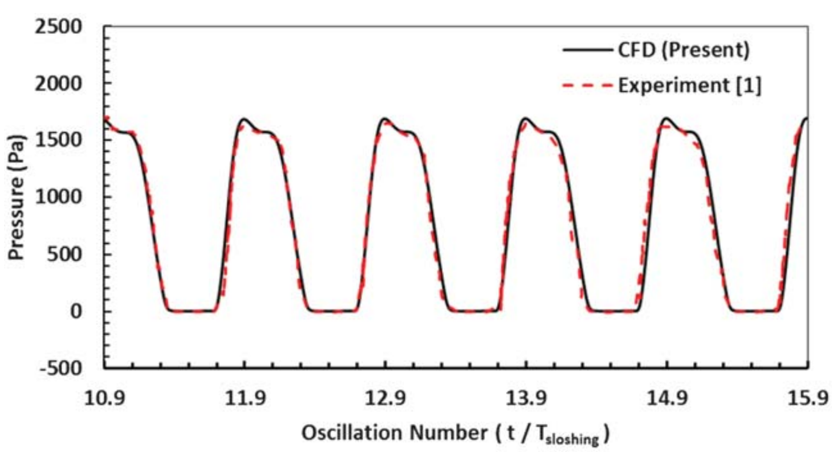

Fig. 3: Variation of tank wall pressure for validation case at point P_2

The CAD model of the generic aircraft fuel tank used in the analysis is shown in Fig. 4. The geometry of the wing tank is discretized (with 3 mm size) into 1102500 elements using ANSYS software as shown in Fig. 5. A high grid resolution is required because when simulating violent sloshing using a viscous fluid model, it is crucial to ensure the conservation of fluid mass over each iterations and time steps. This can be quite challenging, especially when dealing with postimpact fluid fragmentation, wave overturning, breaking, and the attachment of a thin layer of fluid on tank walls. To tackle these issues, two solutions are typically used: increasing mesh resolution and/or using a higher-order time marching scheme, both of which come at the cost of increased computational expenses per time step. Alternatively, one can formulate the governing equations to enforce global conservation of mass. Finer grid, smaller time step, and higher number of iteration per time step has been chosen instead of tinkering with governing equation. In the present analysis, 50% of tank volume is filled inside the tank before the start of the simulation. 1 milli second has been defined as the time step for the analysis. 20 iterations has been considered for each time step. Total duration of 4 seconds is distributed in the form of trapezoidal profile for simulation of the loading conditions in each direction as shown in Fig. 6.

Emergency Condition FAR Profiles: Fig. 6, show the acceleration trapezoidal profile for the *Upward*, *Downward*, *Forward* and *Sideward* direction, respectively. For the upward and downward directions, the acceleration rises from 0 to full scale in 0.5 seconds and stays at full scale for 3 seconds. Then, the acceleration falls from full scale to zero in 0.5 seconds. The simulation result for these two directions shows that the maximum sloshing of the liquid surface occurs during the phase when the acceleration is rising to its peak value. Once the acceleration reaches its peak value, there is no significant oscillation or increase in pressure on the wall of the tank. Therefore, continuing the simulation beyond this point would not provide any additional meaningful insights and would only add to the simulation time. Hence, for the remaining two directions

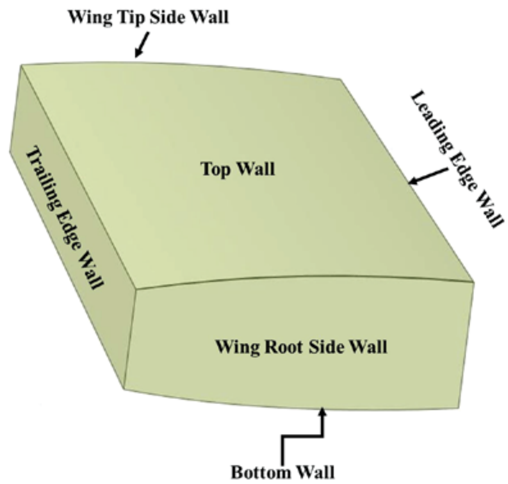

Fig. 4: CAD model of generic fuel tank used in the sloshing analysis

Fig. 5: CFD mesh of generic fuel tank used in the sloshing analysis

(forward and sideward), the simulation was stopped after 1 second once the acceleration reached its peak value. This observation helped in reducing the total simulation time from 4 sec to 1 sec.

Results and Discussion: In the following paragraphs, the behaviour of liquid fuel in an aircraft wing tank is described under different load factor values and acceleration directions. The liquid surface level and pressure variation on different walls of the wing tank over time have been shown and discussed. It has been shown how the fuel tends to move towards a specific wall of the tank due to the direction of acceleration and causes an increase in pressure on that wall. After some oscillations, the fuel settles near that wall and remains there. The thrashing phenomenon of fuel on the wall of the wing tank, which causes an increase in pressure, has also been shown and discussed.

Upward Acceleration: The instantaneous images in Fig. 7 shows the liquid level in the wing tank at various moments when the load factor value of 3.0 g is applied in the upward direction. These images are demonstrated in a cut section plane that runs transversely across the wing tank. Fig. 8 shows the variation of maximum pressure on the six different walls of the wing tank over time. Due to the upward acceleration component, all the fuel within the tank tends to move towards the top wall of the wing tank, as

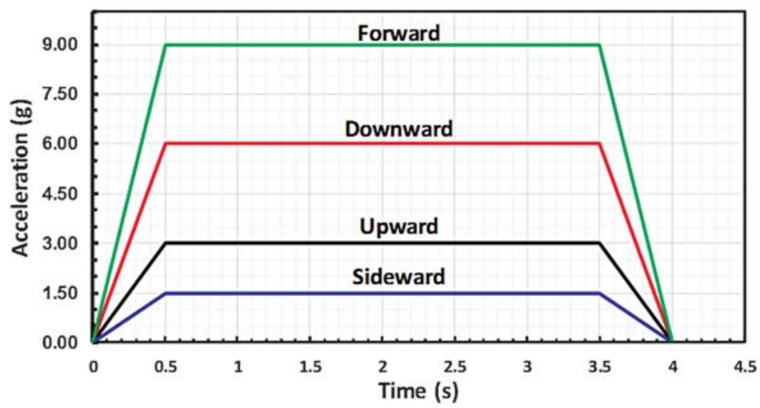

Fig. 6: FAR emergency landing condition acceleration profiles for forward, downward, upward, and sideward direction

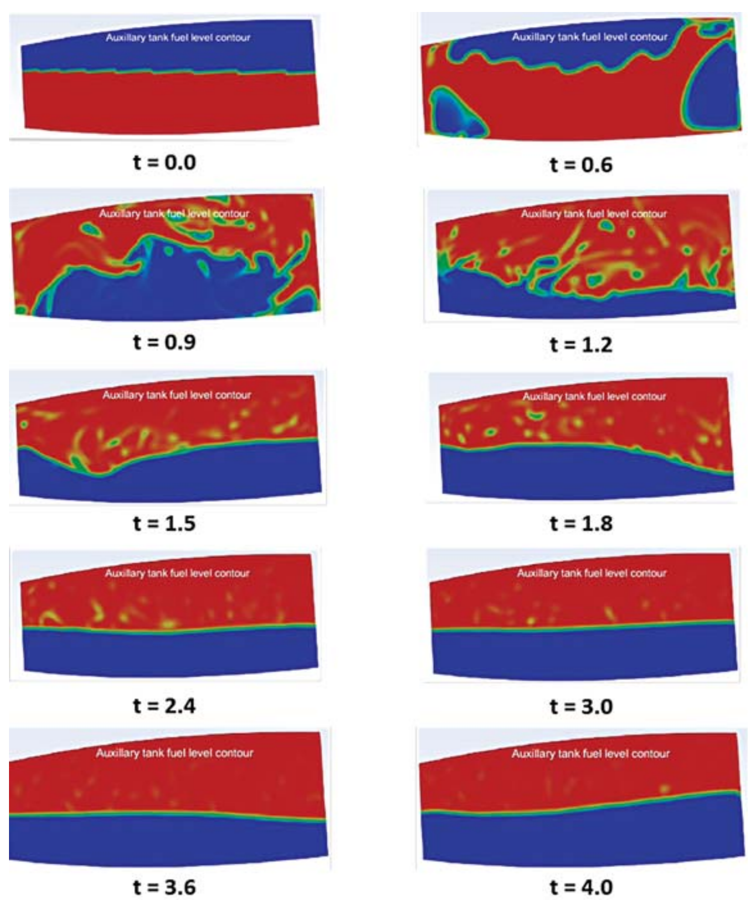

Fig. 7: Instantaneous image of liquid level in wing tank for load factors of upward conditions

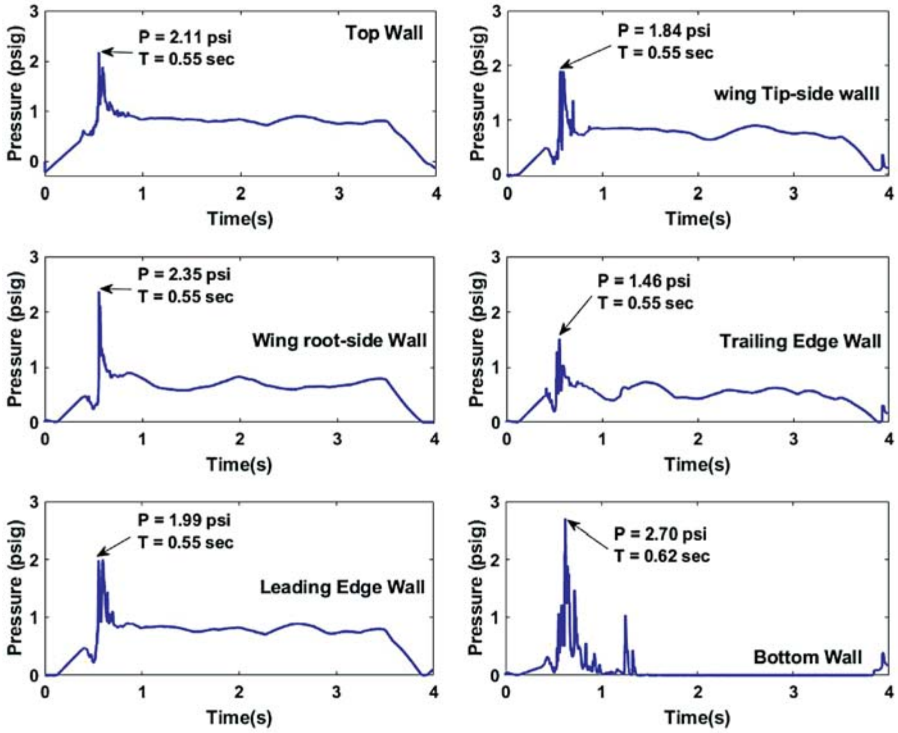

Fig. 8: Variation of wall pressure under FAR upward acceleration Profile

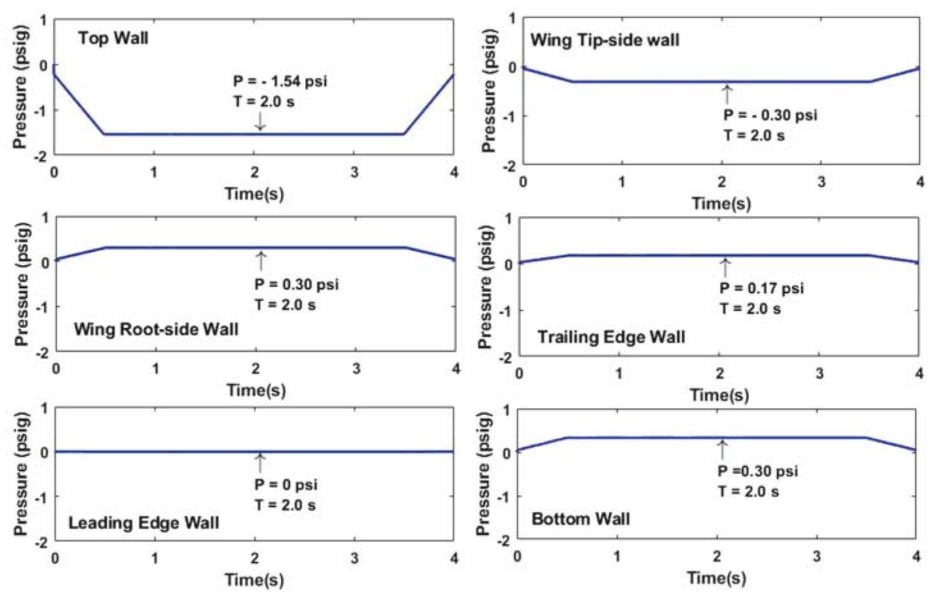

Fig. 9: Variation of wall pressure under FAR downward acceleration Profile

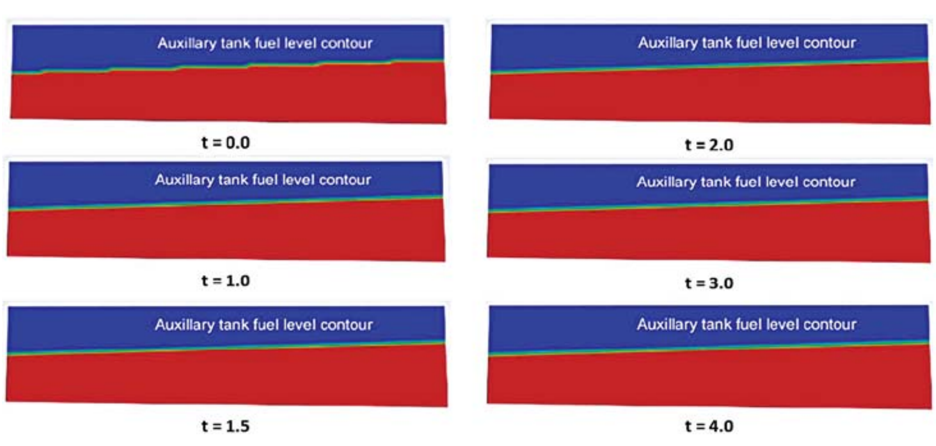

Fig. 10: Instantaneous image of liquid level in wing tank for load factors of downward conditions

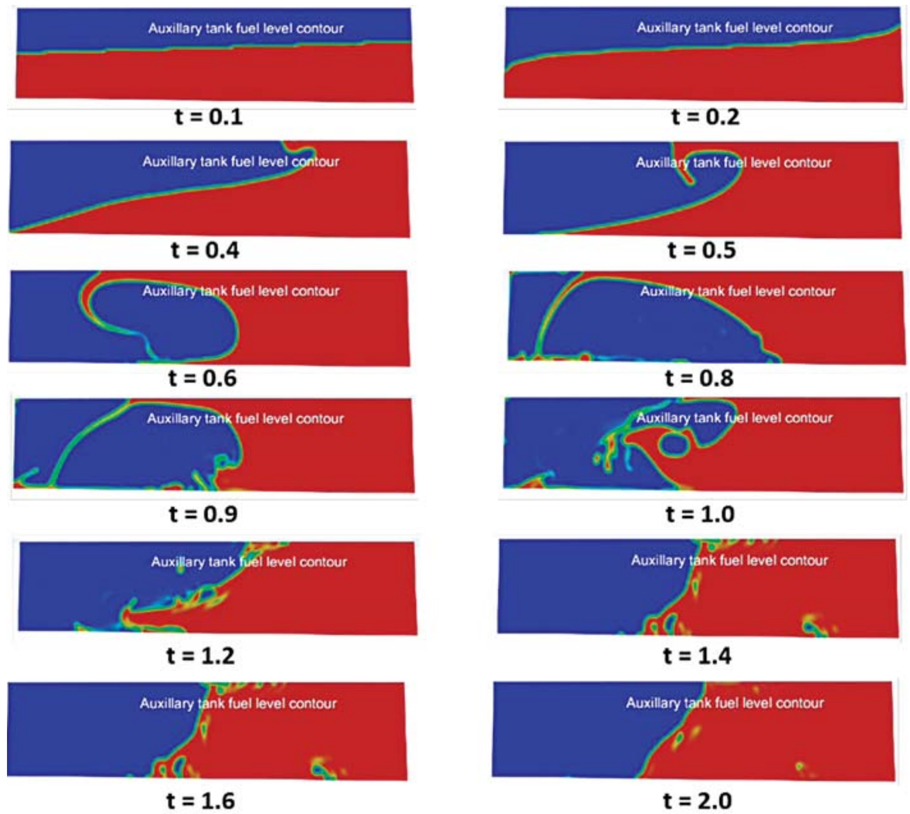

Fig. 11: Instantaneous image of liquid level in wing tank for load factors of sideward conditions

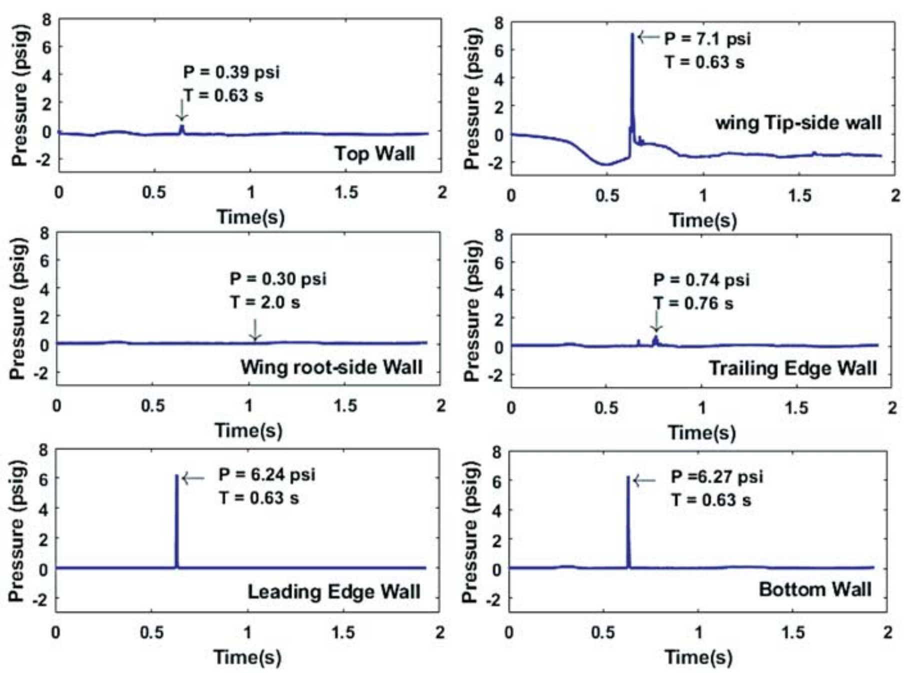

Fig. 12: Variation of wall pressure under FAR sideward acceleration profile

shown in the first four images of Fig. 7. After the fuel has moved to the top wall of the tank, the liquid level oscillate between t = 1.2 to 1.8 seconds. Following this oscillation, all the fuel remains near the top wall of the wing tank, as depicted in Fig. 7. The peak pressure of 2.70 psi(g) was observed around t = 0.6 seconds on the bottom wall.

Downward Acceleration Fig. 15 shows the instantaneous images of the liquid level in the wing tank for the load factor value of 6.0 g in the downward direction. These images are shown in the cut section plane along the longitudinal direction of the wing tank. Meanwhile, Fig. 16 shows the variation of maximum pressure with respect to time in six different walls of the wing tank. Due to the increased downward acceleration component, as shown in Fig. 8, all the fuel collects at the bottom wall of the wing tank. As a result, after a period of 0.5 sec, the pressure in the bottom wall of the wing tank increases, as shown in Fig. 16.

Sideward Acceleration Fig.11, shows the instantaneous images of the liquid level in the wing tank when the load factor value is 1.5 g in the sideward direction. The images are shown in the cut section plane of the longitudinal direction of the wing tank. Fig. 12 shows the variation of maximum pressure on the six different walls of the wing tank with respect to time. Due to the sideward acceleration, all the fuel in the tank begins to shift towards the wing root side wall. This is shown in the first three images of Fig.11. The fuel that accumulates near the wing root wall starts to circulate along the longitudinal direction of the wing at t = 0.5 sec. At t = 0.6 sec, the circulated fuel falls on both the wing tip side and bottom wall of the wing tank, as shown in Fig.11. This falling phenomenon causes an increase in the pressure of the wing tank, which can be observed as the presence of peak pressure in the wing tank at t = 0.63 sec, as shown in Fig. 12. After this

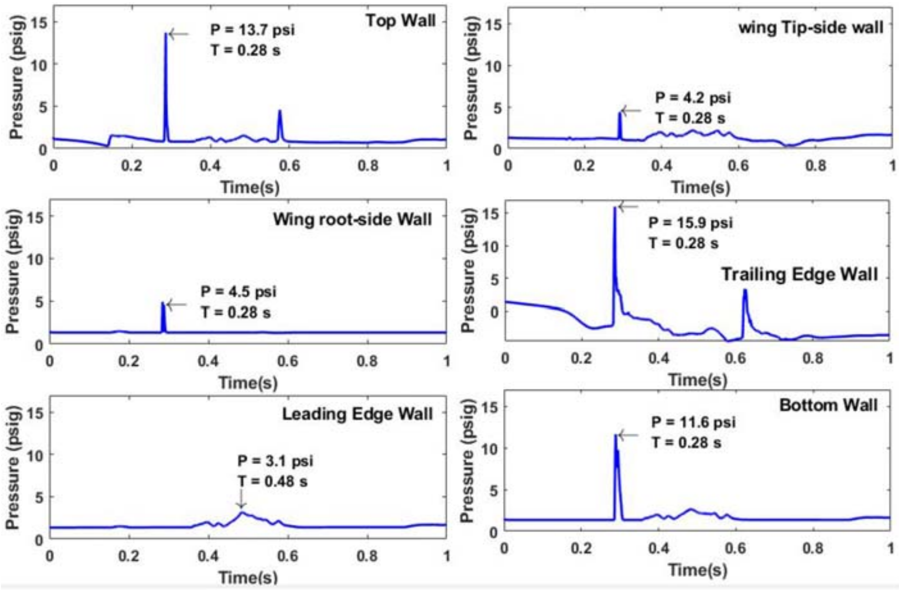

Fig. 13: Variation of wall pressure under FAR forward acceleration profile

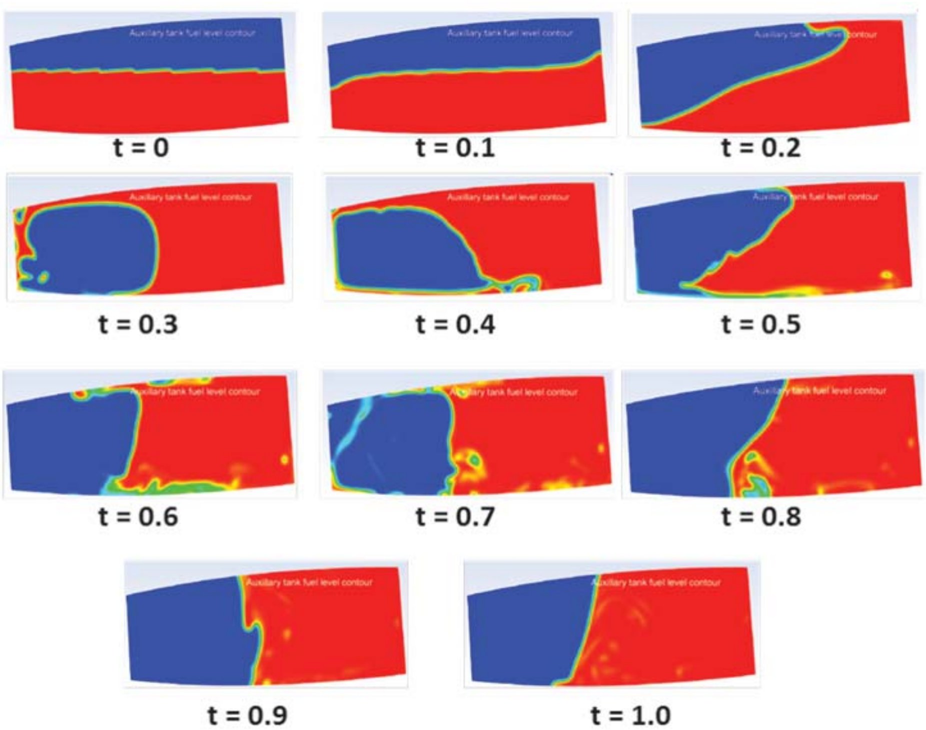

Fig. 14: Instantaneous image of liquid level in wing tank for load factors of forward conditions

oscillation, all the fuel stays near the wing root side wall of the wing tank, as shown in the last image of Fig. 11.

Forward Acceleration: Fig. 14 shows the instantaneous images of the dynamic oscillation of liquid level inside the tank in response to the acceleration profile defined in fig. 6 (load factor value of 9.0 g in the forward direction). These images are shown in the cut section plane along the transverse direction of wing tank. The variation of maximum pressure with respect to time on the six different walls of the wing tank are shown in fig. 13. Due to the forward acceleration component, all the fuel starts to shift towards the leading edge of the wing tank as shown in the first three images of fig. 13. The accumulated fuel near the leading edge starts to circulate at t = 0.2 sec. The circulated fuel is thrashing on the trailing edge wall of the wing tank at t = 0.3 sec. This thrashing phenomena of fuel is causing an increase in the pressure on all the walls of wing tank except on the leading edge wall as shown in fig. 12. This is observed due to the presence of peak pressure in the wing tank at t = 0.3 sec. Similar to this, second cycle of thrashing phenomena is observed around t = 0.6 sec as shown in fig. 13. The second thrashing is relatively weaker with respect to the first one. Therefore, the increase in wing tank pressure is lower than the first one. The second peak of wing tank is observed only in the top and trailing edge walls at t = 0.6 sec. After this initial oscillation, all the fuel stays near the leading edge region of the wing tank as shown in the last image of fig. 13. Table 1 summarizes the peak pressure on the six different walls of the tank for various load conditions.

Conclusions

Sloshing analysis of fuel tanks has been carried out using the Volume of Fluid (VOF) method. This modelling approach had been validated through unsteady CFD simulations in the 2-D tank and the results obtained were comparable to the results published in literature. The approach was then applied to the sloshing analysis of the generic aircraft wing tank, considering the emergency

Table 1. Comparison of peak pressure (in psig) at six different walls of the tank for various load conditions

Tank Wall	Forward	Sideward	Upward	Downward
Top Wall	13.7	0.39	2.18	-1.54
Bottom Wall	11.6	6.27	2.70	0.30
Leading Edge Wall	3.1	6.24	1.99	0
Trailing Wall	15.9	0.74	1.50	0.17
Wing Tip	4.2	7.1	1.90	-0.30
Wing Root	4.5	0.3	2.35	0.30

landing condition of FAR 23. The high grid resolution and smaller time step were chosen to capturing the complex sloshing behavior, especially in regions with violent sloshing and post-impact fluid fragmentation. The acceleration trapezoidal profile for the upward, downward, forward, and sideward directions was simulated. Various sloshing phenomenoa such as overturning, wave formation, and forceful collisions with the tank walls was observed when subjected to extreme accleration disturbances. It was observed that under different load factor values and acceleration directions, the liquid fuel tends to move towards specific walls of the tank. The oscillation and thrashing of fuel against the tank walls caused the pressure to increases on the walls. The fuel movement and pressure variations observed in the simulations provide insights into how the fuel behaves under different acceleration conditions. This understanding is crucial for designing fuel systems that can withstand the forces experienced during flight, especially during emergency landing conditions. ❒

References

1. L. Hou, F. Li and C. A. Wu, *J. Marine. Sci. Appl* **11**, 305–310 (2012).
2. E. Frosina, A. Senatore, A. Andreozzi, F. Fortunato and P. Giliberti, *Energies*, **13**, 682 (2018).
3. S. Z. Moughal, *North Carolina Agricultural and Technical State University*, Theses, 290 (2013).
4. B. Godderidge, S. Turnock, M. Tan, C. Ear, *Computers & Fluids* **38**, (2009) 183–193
5. M.-A. Xue, P. Lin, *Computers & Fluids*, **52**, (2011) 116–129