

A Comparative Study on 3d Sloshing Dynamics for Negative-G Acceleration

Mustafa Serdar ÇELEBİ

mscelebi@itu.edu.tr

Istanbul Technical University <https://orcid.org/0000-0003-4566-0216>

Utkan Çaliskan

Istanbul Technical University

İbrahim Özküçük

Istanbul Technical University

Research Article

Keywords: Sloshing, Propellant tank, Sloshing dynamics, Center of Mass, CFD Model, Navier-Stokes, Sloshing force, Propellant Management Device, LES, Transient flow, Rocket Dynamics, OpenFOAM, Negative-g sloshing

Posted Date: October 27th, 2026

DOI: <https://doi.org/10.21203/rs.3.rs-11482338/v1>

License: © ⓘ This work is licensed under a Creative Commons Attribution 4.0 International License.

[Read Full License](#)

Additional Declarations: The authors declare no competing interests.

A COMPARATIVE STUDY ON 3D SLOSHING DYNAMICS FOR NEGATIVE-G ACCELERATION

M. Serdar ÇELEBİ¹, Utkan ÇALIŞKAN¹, İbrahim ÖZKÜÇÜK¹

¹Department of Computational Science and Engineering, Informatics Institute,
Istanbul Technical University, 34469, Istanbul, Turkey

Abstract. In this work, we comparatively studied one-dimensional negative-g acceleration of a domed circular cylinder tank with alternative flow models such as Lager Eddy Simulation (LES), kw-SST turbulence and laminar flow models available in *interFoam* solver of OpenFOAM - v1812. Major objective is to validate and compare the accuracy and performance of the flow models in severe sloshing subject to a negative-g maneuvering under various internal pressures. Initially, five different (i.e. *compressibleinterFoam*, *interFoam* Laminar, *interIsoFoam*, *interFoam* kw-SST and *interFoam* LES) flow models are tested with the sloshing cylinder experiment using two fill ratios ($h/2R \approx 0.5$ and $h/2R = 1.0$) under two different excitation (single impulse and periodical) forces. For the negative-g acceleration model, possible impact of internal pressures (0, 50, 300 and 600 bars) on the forces and instant Center of Mass (CoM) locations are studied for 50% fill ratio. Our work contains five critical findings: 1) comparing the capability of solvers for obtaining the sloshing forces and instant CoM's, 2) correctly estimating the duration of propellant discontinuity from tank drain system, 3) potential impact of internal pressures on sloshing dynamics, 4) validating the accuracy of the solvers with comprehensive experimental tests including sloshing damping ratio based on linear theory, 5) comparing the computational cost and robustness of the solvers for 1D negative-g severe sloshing case. Critical nonlinear contributions from high frequency modes during the severe sloshing especially in the lateral directions are observed and comprehensively studied. Advantages and disadvantages of using laminar and turbulent flow models in severe sloshing simulations are analyzed. Finally, during the negative-g maneuvering, accurate prediction of discontinuities in the propellant flow to tank drain system is discussed.

Key words: Sloshing, Propellant tank, Sloshing dynamics, Center of Mass, CFD Model, Navier-Stokes, Sloshing force, Propellant Management Device, LES, Transient flow, Rocket Dynamics, OpenFOAM, Negative-g sloshing.

*Corresponding Author: E-mail: mscelebi@itu.edu.tr
Phone: (+90) 505 424 30 15

1. INTRODUCTION

Liquid sloshing is one of the main disturbances on spacecraft and rocket motions, and affects many different maneuvering missions and launch requirements. Due to its critical importance, numerous researchers have conducted studies on different aspects of sloshing in the literature since the early 50's. These works reveal a series of analytic, experimental, and computational techniques which allow researchers to in depth analyze the characteristics of the sloshing in a confined tank with various shapes. Among these quasi-static methods [1, 2], mechanical analogy methods [3, 4, 5, 6] and computational fluid dynamics (CFD) methods based on the solution of Navier-Stokes equations using linear [7] and non-linear [8, 9, 10] boundary conditions can be mentioned. One of the most critical impacts on the development of these methods is the experimental validation process of the analytical and computational methods, where results and their validity are verified. Recently, various computational models have involved the sloshing with the accompanying scaled experiments [1, 11, 12, 13, 14]. Sloshing, in general, can be classified into two main groups as symmetric and anti-symmetric sloshing. Especially in anti-symmetric sloshing, which defines the most complex type of sloshing, the Center of Mass (CoM) of a sloshing liquid plays a critical role. The effect of CoM displacement on the tank stability is studied extensively using mechanic methods [15, 16, 17, 18] based on the assumptions that liquid displacement is a linear and/or planar motion. However, these methods do not consider both transient flows of the sloshing liquid in a tank and multi-degree of freedom tank movements. Numerical simulations using computational fluid dynamics techniques are the most important and widely used approaches for handling severe sloshing problems having high non-linearity. Based on the maneuvering type and the geometry of a propellant tank, free surface can exhibit a variety of fluid motions such as planar/non-planar, rotational, symmetric/non-symmetric or any combination of these during the sloshing [19]. Severe maneuvering motions or sudden increase in acceleration may lead to the generation of undesirable forces and moments which cause static and dynamic stability problems on the vehicle [20]. Therefore, a complete non-linear computational fluid dynamics model is vital for better capturing the sloshing physics and its critical flow parameters such as the instant location of the propellant mass center, forces, free surface displacements, and the first mode response frequency of sloshing based on the tank geometry and propellant fill depth. If the propellant tank is subject to a large positive-g following micro- or sudden negative-g accelerations, then continuity of liquid propellant to the trust engine(s) through drain the system at the bottom of the tank has an extreme critical importance for obtaining uninterrupted trust. Based on both duration of these micro-g or negative-g accelerations and the level of fill ratio, discontinuity in the propellant flow occurs if an effective propellant management device (PMD) is not utilized. Additionally, high fidelity calculations of the total forces and moments acting on the tank walls due to the severe sloshing have critical importance to prevent the possible divergence from the desired trajectory.

In this work, we comparatively studied a one-dimensional negative-g acceleration of a domed circular cylinder tank with alternative flow models such as Large Eddy Simulation (LES), kw-SST turbulence model, and Laminar flow models available in *interFoam* solver of OpenFOAM - v1812. Our major objective is to validate and compare the accuracy and performance of the flow models in severe sloshing subject to a negative-g maneuvering under various internal pressures implemented within the whole tank volume. Five different (i.e. *compressibleinterFoam*, *interFoam* Laminar, *interIsoFoam*, *interFoam* kw-SST and *interFoam* LES) flow models are tested first with the sloshing cylinder experiment using two fill ratios ($h/2R \approx 0.5$ and $h/2R = 1.0$) under two different excitation (single impulse and periodical) forces [21]. For the negative-g acceleration model, the possible impact of internal pressures (0, 50, 300, and 600 bars) on the forces and instant Center of Mass (CoM) locations are studied for 50% fill ratio.

Our study contains five critical findings: 1) comparing the capability of solvers for obtaining the sloshing forces and instant CoM's, 2) correctly estimating the duration of propellant discontinuity from tank drain system, 3) potential impact of internal pressures on sloshing dynamics, 4) validating the accuracy of the solvers with comprehensive experimental tests including sloshing damping ratio based on linear theory, 5) comparing the computational cost and robustness of the

solvers for 1D negative-g severe sloshing case. The advantages and disadvantages of using either laminar or turbulent flow models in severe sloshing simulation are analyzed. During the negative-g maneuvering, accurate prediction of discontinuities in the propellant flow has a critical importance and requires high fidelity solutions for further PMD design. If the exact duration(s) of discontinuities in the drain system are known, then the amount(s) of propellant that need to be kept by PMD is obtained prior. Accurate prediction of discontinuities in the propellant flow is discussed under various internal tank pressures. Finally, during the negative-g severe sloshing, critical nonlinear contributions from high frequency flow modes especially in the lateral directions are observed and comprehensively studied.

2. MATHEMATICAL MODEL

Sloshing in a moving rigid container is a highly nonlinear two-phase flow exhibiting free surface between fluid and gas. Numerical solution of the complete Navier-Stokes is indispensable and approximation methods for capturing or tracking the free surface interface correctly are necessary and have critical importance. To predict the free surface at each time step, most widely two alternative methods are used. These are Surface Methods and Volume Methods [22]. While Surface Methods track the free surface by a sharp interface or marking it with marker points, Volume Methods define the free surface by a boundary volume. The Volume Fraction Methods are the most commonly used technique in Volume Methods to treat the free surface. Hirt and Nichols' Volume of Fluid (VOF) Method [23] has one critical advantage that volume occupied by one fluid not be able to be occupied by the other and thus continuity is always satisfied. In our sloshing study, we use the VOF within the open-source code OpenFOAM, which provides alternative choices including single-phase, multi-phase, incompressible, compressible, laminar and turbulent flow solvers. The *interFoam* solver of OpenFOAM is a two-phase isothermal flow solver based on interface capturing technique using the VOF phase fraction. In the following, governing equations including continuity, transport of phase-fraction, and momentum are described by following the works of Jasak [24] and Ubbink [22].

2.1 Governing Equations

The *interFoam* solver employs finite volume discretization on collation points of meshes for the solution of the following two phase (liquid and gas) equations of sloshing flow. The flow variables are located at the cell center, however, their interpolated values on the cell faces are used in the solution stage as well. Since in our sloshing study we use compressible and incompressible flow conditions as well as laminar and turbulent flows, fundamental governing equations of the two-phase flow are presented in a general form. The conservation of mass equation is given by

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0, \quad (1)$$

where $\mathbf{U}$ represents the velocity vector. The conservation of momentum is given by

$$\frac{\partial(\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) - \nabla \cdot (\mu \nabla \mathbf{U}) - \nabla \mathbf{U} \cdot \nabla \mu = \sigma \kappa \nabla \alpha - \mathbf{g} \cdot \mathbf{x} \nabla \rho - \nabla p_d, \quad (2)$$

where $\mathbf{g}$ is the gravitational acceleration, $\mathbf{x}$ is the position vector, σ is the surface tension, and $p_d = (p - \rho \mathbf{g} \cdot \mathbf{x})$ is the dynamic pressure and finally $\kappa = -\nabla \cdot (\nabla \alpha / |\nabla \alpha|)$ is the curvature of the free surface interface. Since the VOF method is an Eulerian based interface tracking method, an α step function is used to identify the location of the phases by marking the volume fraction of the tracked phase in a control volume. In other words, while $\alpha = 1$ corresponds to a control volume totally occupied by liquid, $\alpha = 0$ corresponds to a control volume totally occupied by gas. So, it can be observed that the interface between the phases is found in cell members where $0 < \alpha < 1$. Fluid properties are calculated using $\alpha = \frac{V_{cell,liquid}}{V_{cell,liquid} + V_{cell,gas}}$ as volume fraction for liquid phase. The properties of a fluid such as mixture density and dynamic viscosity can be described by

$$\rho = \alpha \rho_{liquid} + (1 - \alpha) \rho_{gas}, \quad (3)$$

$$\mu = \alpha \mu_{liquid} + (1 - \alpha) \mu_{gas}. \quad (4)$$

The VOF method adds one additional governing equation for the transport of the volume fraction α . In this equation, an additional convective term for modeling the velocity based on the volume fraction is introduced into the following transport equations for phase fraction to obtain a sharper free surface resolution. Basic idea is to make use of the two-fluid immiscible Eulerian model for two-phase sloshing flow, where each of the phase fraction equations is solved independently. Thus, the equations for each of the phase fractions are written as

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{U}_{liquid} \alpha) = 0, \quad (5)$$

$$\frac{\partial (1 - \alpha)}{\partial t} + \nabla \cdot [\mathbf{U}_{gas} (1 - \alpha)] = 0. \quad (6)$$

The mixed velocity of sloshing fluid in a VOF model is defined as a weighted average of volume fractions.

$$\mathbf{U} = \alpha \mathbf{U}_{liquid} + (1 - \alpha) \mathbf{U}_{gas}. \quad (7)$$

If we rearrange Equation 6 and use it as an evolution equation for the phase fraction α and use the vector of relative velocity $\mathbf{U}_r = \mathbf{U}_{liquid} - \mathbf{U}_{gas}$ as the compression velocity, we obtain the transport equation for the liquid volume fraction to capture the free surface interface.

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{U} \alpha) + \nabla \cdot [\mathbf{U}_r \alpha (1 - \alpha)] = 0, \quad (8)$$

where $\nabla \cdot [\mathbf{U}_r \alpha (1 - \alpha)]$ is used as an anti-diffusion term to sharpen the free surface interface. Relative (or artificial compression) velocity is modelled at cell faces as

$$\mathbf{U}_{r,f} = \mathbf{n}_f \min \left[C_\alpha \frac{|\phi|}{|\mathbf{A}_f|}, \max \left(\frac{|\phi|}{|\mathbf{A}_f|} \right) \right], \quad (9)$$

where subscript f denotes cells having internal face f and $\mathbf{A}$ represents normal vector points outwards from the computational cell, C_α is a user specified parameter reducing interface smearing, ϕ is the face volume flux and $\mathbf{n}_f$ is the face unit normal flux. This relative velocity is calculated in the sloshing free surface interface based on the phase fraction gradient at cell faces. Face unit normal flux is then given by

$$\mathbf{n}_f = \frac{(\nabla \alpha)_f}{|(\nabla \alpha)_f + \delta_n|} \cdot \mathbf{A}_f, \quad (10)$$

where δ_n is the stabilization parameter, which considers the non-uniformity of the mesh and usually takes the value of 10^{-5} or computed by the following equation

$$\delta_n = \frac{\varepsilon}{\left(\frac{\sum_N V_i}{N} \right)^{1/3}}. \quad (11)$$

Here, N represents the number of computational cells and ε denotes a small parameter usually set to 10^{-8} .

2.2 Pressure-Velocity Coupling

The solution of the momentum equation is performed by constructing a predicted velocity field and then correcting it using either the Pressure Implicit with Splitting of Operators (PISO) [25] which uses implicit pressure updating procedure to time iterate the pressure p_d and velocity fields or Semi Implicit Method for Pressure Linked Equations (PIMPLE) for transient flows ensuring strong coupling between fluid and motion. Since the sloshing occurs in a confined rigid container mostly under incompressible flow assumptions, the pressure-velocity coupling algorithm deserves a special attention due to the accuracy and stability requirements of the flow physics. In two-phase flow simulation of sloshing, the following momentum equation is used in [26] after reorganizing the equation (2)

$$\frac{\partial(\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla p + [\nabla \cdot (\mu \nabla \mathbf{U}) + \nabla \mathbf{U} \cdot \nabla \mu] + \rho \mathbf{g} + \int_{\Gamma} \sigma \kappa \delta(\mathbf{x} - \mathbf{x}_s) \mathbf{n} d\Gamma(\mathbf{x}_s), \quad (12)$$

where Γ is the liquid-gas interface and $\delta(\mathbf{x} - \mathbf{x}_s)$ is the three-dimensional Dirac delta function. Using the continuum surface force (CSF) model of Brackbill *et al.* [27] integral form of the last term of equation (12) is transformed from surface (Γ , interface) form to an imbedded volume integral over Ω_i . *InterFoam* solver uses a modified dynamic pressure p_d rather than p . Using these two features (i.e., surface tension term and substitution of p_d), the volume integral of equation (12) becomes

$$\begin{aligned} \int_{\Omega_i} \frac{\partial(\rho \mathbf{U})}{\partial t} dV + \int_{\partial\Omega_i} (\rho \mathbf{U} \mathbf{U}) \cdot \mathbf{n} dS = & - \int_{\Omega_i} \nabla p_d dV - \int_{\Omega_i} \mathbf{g} \cdot \mathbf{x} \nabla \rho dV + \int_{\Omega_i} \sigma \kappa \nabla \alpha dV \\ & + \int_{\partial\Omega_i} (\mu \nabla \mathbf{U}) \cdot \mathbf{n} dS + \int_{\Omega_i} \nabla \mathbf{U} \cdot \nabla \mu dV. \end{aligned} \quad (13)$$

If we use index m for the PISO iteration, n for the present time step t^n and $\mathbf{U}_p^r$ as predicted velocity field at cell P, then explicit discrete expression of equation (13) results in

$$\begin{aligned} \frac{(\rho_p^{n+1} \mathbf{U}_p^r) - (\rho \mathbf{U})_p^n}{\Delta t} |\Omega_P| + \sum_{f \in \partial\Omega_i} (\rho_f \phi_f)^n \mathbf{U}_f^{r'} \\ = \sum_{f \in \partial\Omega_i} \mu_f^{n+1} (\nabla_f^\perp \mathbf{U})^{r'} |\mathbf{S}_f| + \nabla \mathbf{U}_p^n \cdot \nabla \mu_p^{n+1} |\Omega_P|. \end{aligned} \quad (14)$$

Here, μ and ρ represent the mixed viscosity and the mixed fluid density for two-phase flow, respectively. The velocity $\mathbf{U}_f^{r'}$ in equation (14) is defined for a given cell face by

$$\mathbf{U}_f^{r'} = \frac{\mathbf{U}_p^r}{2} [1 + \Theta(f)w] + \frac{\mathbf{U}_N^m}{2} [1 + \Theta(f)w], \quad (15)$$

where $\Theta(f)$ is defined as

$$\Theta(f) = \begin{cases} 1 & \text{if } P \text{ is the owner for face } f \text{ and } N \text{ is the neighbor,} \\ -1 & \text{if } N \text{ is the owner for face } f \text{ and } P \text{ is the neighbor.} \end{cases} \quad (16)$$

and w is given by

$$w = \vartheta(\phi_f)(1 - \lambda_\alpha), \quad (17)$$

where

$$\vartheta(\phi_f) = \begin{cases} 1 & \text{for } \phi_f \geq 0, \\ -1 & \text{for } \phi_f < 0, \end{cases} \quad (18)$$

In equation (17), λ_α represents a limiter function and is determined by the choice of interpolation as well as the distance between cell centers. *InterFoam* offers a variety of interpolation schemes specifically for the divergence (advective) terms. The surface gradient operator $\nabla_f^\perp$ acting on $\mathbf{U}$ in equation (14) is described as

$$(\nabla_f^\perp \mathbf{U})^{r'} = \Theta(f) \frac{\mathbf{U}_N^m - \mathbf{U}_P^r}{|\mathbf{d}|}. \quad (19)$$

Using equations (15), (19) and including the effect of surface and gravity forces, the final form of the equation (14) and the predicted velocity field at cell P becomes

$$\mathbf{U}_P^r = \frac{H(\mathbf{U}^m)}{A_p} + \frac{(\sigma\kappa\nabla\alpha)^{n+1}}{A_p} - \frac{(\mathbf{g} \cdot \mathbf{x} \nabla\rho)^{n+1}}{A_p}. \quad (20)$$

Equation 20 is solved by interpolating $\mathbf{U}_P^r$ to the cell faces considering the high-density ratio flows at free surface. Then, associated volume flux ϕ_f^r is obtained by conducting an inner product with the cell surface area $(\mathbf{U}_P^r)_f \cdot \mathbf{S}_f$.

$$\begin{aligned} \phi_f^r = & \left(\frac{H(\mathbf{U}^m)}{A_p} \right)_f \cdot \mathbf{S}_f + \left(\left(\frac{1}{A_p} \right)_f (\sigma\kappa)^{n+1} (\nabla_f^\perp \alpha)^{n+1} \right) |\mathbf{S}_f| \\ & - \left(\left(\frac{1}{A_p} \right)_f (\mathbf{g} \cdot \mathbf{x})_f^{n+1} (\nabla_f^\perp \rho)^{n+1} |\mathbf{S}_f| \right), \end{aligned} \quad (21)$$

The predictor step of PISO is achieved with equation (21). The dynamic pressure contribution with respect to flux is

$$\left(\frac{-\nabla p_d}{A_p} \right)_f \cdot \mathbf{S}_f = \left(\frac{-1}{A_p} \right)_f (\nabla_f^\perp p_d^{m+1}) |\mathbf{S}_f|, \quad (22)$$

which is then added to equation (21) to obtain next PISO iteration step as

$$\begin{aligned} \phi_f^{m+1} = & \left(\frac{H(\mathbf{U}^m)}{A_p} \right)_f \cdot \mathbf{S}_f + \left(\left(\frac{1}{A_p} \right)_f (\sigma\kappa)^{n+1} (\nabla_f^\perp \alpha)^{n+1} \right) |\mathbf{S}_f| \\ & - \left(\left(\frac{1}{A_p} \right)_f (\mathbf{g} \cdot \mathbf{x})_f^{n+1} (\nabla_f^\perp \rho)^{n+1} |\mathbf{S}_f| \right) - \left(\frac{1}{A_p} \right)_f (\nabla_f^\perp p_d^{m+1}) |\mathbf{S}_f| \end{aligned} \quad (23)$$

The above equation is further simplified in a more compact form as

$$\phi_f^{m+1} = \phi_f^r - \left(\frac{1}{A_p} \right)_f (\nabla_f^\perp p_d^{m+1}) |\mathbf{S}_f|, \quad (24)$$

In order to find the unknown ϕ_f^{m+1} for each cell face, unknown dynamic pressure field p_d^{m+1} must be solved using two steps. First, the conservation of mass in sloshing flow is forced to satisfy $\sum_{f \in \partial\Omega_i} \phi_f^{n+1} = 0$. This condition yields

$$\sum_{f \in \partial\Omega_i} \left(\frac{1}{A_p} \right)_f (\nabla_f^\perp p_d^{m+1}) |\mathbf{S}_f| = \sum_{f \in \partial\Omega_i} \phi_f^r. \quad (25)$$

Equation (25) represents a linear set of equations for the unknown p_d^{m+1} . In OpenFOAM, there are many linear solver options available and based on the symmetry feature of the resulting coefficient matrix, a suitable solver (such as PCG or AMG) with a preconditioner can be selected. Depends on the condition number of the coefficient matrix, which may change during the simulations, it is advised to select a robust preconditioned solver such as PCG (symmetric). The second step is to obtain p_d^{m+1} from equation (25). Once we know the dynamic pressures at cells, finally the velocity fields $\mathbf{U}_p^{m+1}$ at cell centers are solved by reconstructing the face velocity flux using the following equation

$$\mathbf{U}_p^{m+1} = \mathbf{U}_p^r + \left(\frac{1}{A_p} \right) \left(\sum_{f \in \partial \Omega_i} \frac{(\mathbf{S}_f \otimes \mathbf{S}_f)}{|\mathbf{S}_f|} \right)^{-1} \cdot \left(\sum_{f \in \partial \Omega_i} \left(\frac{\phi_f^{m+1} - (\mathbf{U}_p^r)_f \cdot \mathbf{S}_f}{\left(\frac{1}{A_p} \right)_f} \right) \frac{\mathbf{S}_f}{|\mathbf{S}_f|} \right). \quad (26)$$

With this final step, a single PISO iteration loop is completed. Then the PISO iteration is continued by replacing the superscript m to $m+1$ and updating $H(\mathbf{U}^m) = A_p \mathbf{U}_p^r$ with the current iteration velocity obtained from equation (26). So, this iteration procedure is repeated M times to force the velocity vector field and the pressure field in order to satisfy the continuity and momentum equations simultaneously. It should be noted that these iterations have limited numbers (in our simulations M is set to 3) and are repeated within a single time step in the solution of momentum equations. The whole coupled solution procedure for a single time step of *interFoam* solver is summarized in Figure 1.

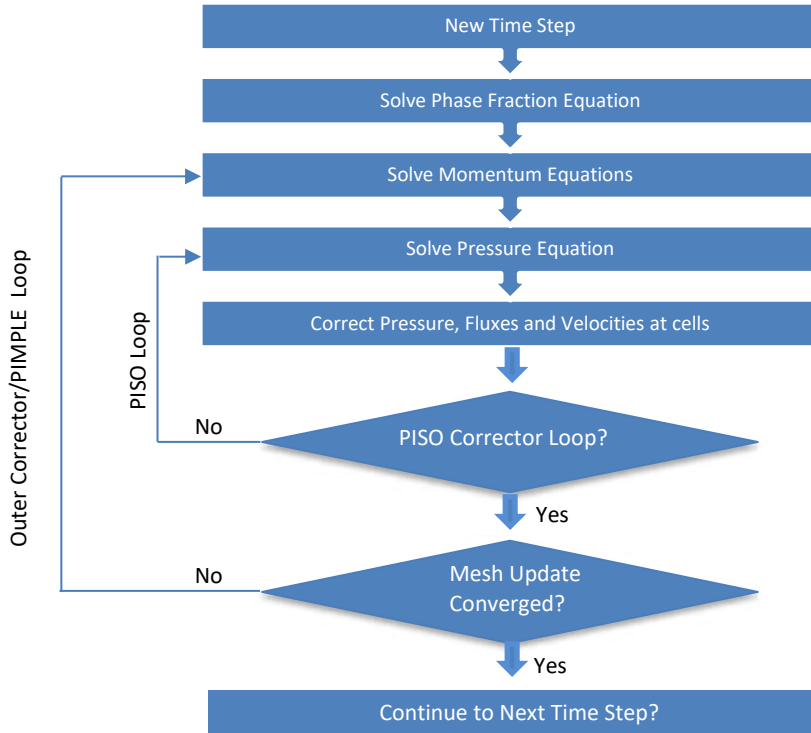


Figure 1. Pressure-velocity coupling procedure of *interFoam* solver

We used PIMPLE algorithm, which is a hybrid algorithm of PISO and SIMPLE. However, with a certain criteria (keeping $n_{OuterCorrectors}$ below 2 as seen in Figure 1) PIMPLE will run as pure PISO. PIMPLE basically allows to model transient simulations at higher courant numbers and under-relaxes the solutions of pressure-velocity coupling at each time step. Another important feature is utilizing outer corrector loops (two and more) to update the dynamic mesh due to rigid body dynamics. By this way, we obtain stronger coupling between sloshing fluid and tank

dynamics. Based on the severity of the tank dynamics, a number of outer loops should be selected to allow fluid meshes updated for a new position of the tank.

2.3 Free Surface Representation in Sloshing

Without a specific free surface capturing method, the surface in the VOF method will be represented as an interface region where α gradually changes from 1 to 0. The cells at the interface including volume fractions of both phases will not have a sharp and clear free surface dividing the phase fractions inside the cell, instead, the whole cell is considered to be filled with a uniform mixture of the two phases as seen in Figure 2. An important point here is that, once the cells where the free surface interface is identified, these cells need to be precisely cut by the surface based on their volume fractions. Therefore, a special attention in the VOF method should be given to surface sharpness to improve the free surface resolution in sloshing. *InterFoam* solver uses MULES (Multi-dimensional Universal Limiter with Explicit Solution) algorithm for this. During the sloshing in the VOF method, the main challenges at the free surface interface are keeping the density field sharp and bounded including artificial velocities at the interface and handling of large density ratios. One important alternative besides MULES is the isoAdvector interface advection method used in *InterisoFoam* solver, which may increase the general performance and accuracy of free surface flow simulations.

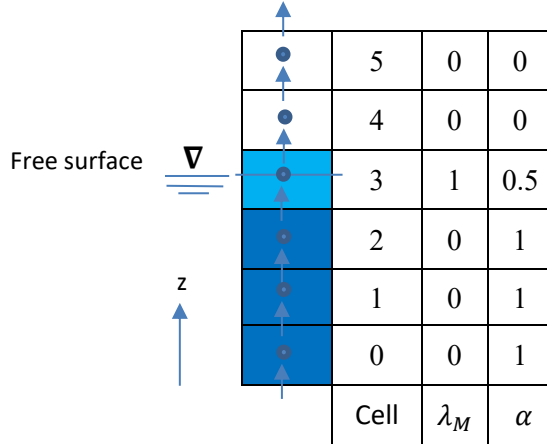


Figure 2. One-dimensional sloshing domain for investigating the behavior of phase fraction.

The phase fraction equation (9) exhibiting a hyperbolic behavior is solved to capture the interface accurately and efficiently, which is given in Figure 2. While the first-order solution of the conservation equation exhibits numerical diffusion, higher-order solutions suffer from the artificial oscillations near the jumps in solutions. *InterFoam*'s phase fraction equation handles this by using the anti-diffusion term in equation (9). In this term, a relative/compression velocity is computed at cell phases based on the volume flux ϕ and face flux gradient $\mathbf{n}_f$. During the updating process of the phase fraction field especially at the interface, the time step size of the equation (9) has a certain limitation by CFL (Courant-Friedrichs-Lewy) number.

To show the importance of the flux limiter at free surface let us consider one dimensional sloshing system as in Figure 2. Half of the tank is fully filled with liquid where α is 1. Free surface cell has a mixture of liquid and gas having 0.5α and above cells are fully filled with gas where α is 0. For simplicity, mesh sizes are considered equal and flow velocity in the z direction has a constant value of unity. Flow direction in the domain is in the z direction where the liquid is flowing into domain from the bottom face and gas flows out through cell' upper face. Under these assumptions, the flow in Figure 2 is driven by convection initially because the gradients of pressure and the velocity does not exist at the beginning and thus resulting momentum equation is simplified as

$$\frac{\partial(\rho \mathbf{U})}{\partial t} = -\nabla \cdot (\rho \mathbf{U} \mathbf{U}) - g z \nabla \rho. \quad (28)$$

Assuming the gravity is zero, the flow only accelerates if the density changes across a cell. For example, flow velocity at cell 4 in Figure 2 can be approximated as

$$U_4^{n+1} = \Delta t \frac{\Delta \rho_{f,4}^n}{\rho_4^{n+1}} + \frac{\rho_4^n}{\rho_4^{n+1}} U_4^n. \quad (29)$$

Here, U shows sloshing flow velocity in z direction, $\Delta \rho_{f,4}^n = \rho_{f,34}^n - \rho_{f,45}^n$ is the density difference along the cell 4 and subscripts $f, 34$ and $f, 45$ represent the densities on the mutual bottom face between cell 3 and 4 and upper face between cell 4 and cell 5. Since the density of liquid is significantly larger than gas, $\rho_{f,45}^n$ term causes a large difference in $\Delta \rho_{f,4}^n$ due to its very small value. As a result of this, U may get a very large value near the free surface. In order to prevent these unwanted jumps on the free surface interface, a flux limiter is used with the assumption that density on the faces of a mixed phase at the interface is set identical to the phase of the corresponding single-phase cell (either fully wet or dry). This idea is implemented in *interFoam* with MULES solver. MULES numerical scheme where the advection term in transport equation (9) for liquid volume fraction is modified to compress the surface. Discrete advection term is given by

$$\frac{\alpha_i^{n+1} - \alpha_i^n}{\Delta t} = -\frac{1}{|\Omega_i|} \sum_{f \in \partial \Omega_i} (F_u + \lambda_M F_c)^n, \quad (30)$$

where left hand side is advanced by implicit Euler scheme (alternatives are forward Euler, Crank-Nicholson and backward differencing), and the advection term has a summation over the cell faces (represented by f) of Ω_i . Advective fluxes F_u and F_c are described as

$$F_u = \phi_f \alpha_{f,upwind}, \quad (31)$$

$$F_c = \phi_f \alpha_f + \phi_{rf} \alpha_{rf} (1 - \alpha)_{rf} - F_u. \quad (32)$$

Here, volume flux ϕ_f is equal to $\mathbf{U}_f \cdot \mathbf{S}_f$. The flow limiter λ_M has a role in the treatment of the liquid fraction advection in equation (30) and is equal to one at interface and zero elsewhere as can be seen in Figure 2. The critical problem occurs at interface when $\lambda_M = 1$, and then the advection term becomes

$$(F_u + \lambda_M F_c) = \phi_f \alpha_f + \phi_{rf} \alpha_{rf} (1 - \alpha)_{rf}. \quad (33)$$

Due to the last compression flux term in equation 33, it is aimed to reduce both the numerical diffusion and computational cost at the interface. In order to solve this equation, unknowns on the right-hand side ($\phi_f, \alpha_f, \phi_{rf}, \alpha_{rf}$) need to be defined at the interface through $\mathbf{U}_f$ and $\mathbf{S}_f$.

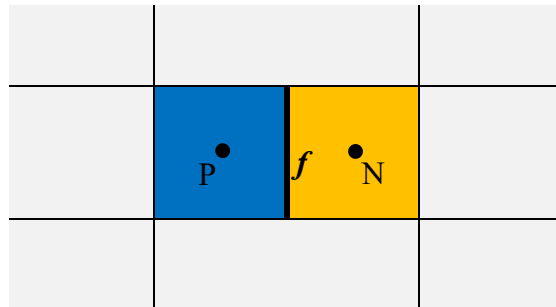


Figure 3. The cell face f whose owner is P and its neighbor N.

Fluid particle velocities at the interface are computed in general form using a point-to-point interpolation given by

$$\mathbf{U}_f = \lambda_{f,1} \mathbf{U}_P + \lambda_{f,2} \mathbf{U}_N, \quad (34)$$

where $\lambda_{f,1}$ and $\lambda_{f,2}$ are the face weights at the free surface for P and N cells, respectively. *InterFoam* provides different interpolation alternatives for this. On the other hand, liquid fraction phase values can be calculated using the combination of central and upwind differencing schemes as

$$\alpha_f = (1 - \lambda_\alpha)(\alpha_f)_{UD} + \lambda_\alpha(\alpha_f)_{CD}, \quad (35)$$

where

$$(\alpha_f)_{CD} = \frac{\alpha_P + \alpha_N}{2}, \quad \text{and} \quad (\alpha_f)_{UD} = \begin{cases} \alpha_P & \text{for } \phi_f \geq 0 \\ \alpha_N & \text{for } \phi_f < 0 \end{cases}. \quad (36)$$

Then, using equations (35) and (36) the following form for the face centered α field is obtained as

$$\alpha_f = \alpha_P + \frac{\alpha_N - \alpha_P}{2} [1 - \vartheta(\phi_f)(1 - \lambda_\alpha)], \quad (37)$$

where ϑ is a step function given in equation (19) taking the value 1 if volumetric face flux ϕ is positive and -1 if it is negative, λ_α is a limiter at the interface cell. There are variety of limiters available in OpenFOAM such as vanLeer, SuperBee, Minmod, QUICK. ϕ_{rf} on the other hand, is obtained by multiplying equation (10) with $\mathbf{S}_f$. α_{rf} in equation (32) is also computed using the *interfaceCompression* scheme which is very similar to equation (37). Finally, in the computation of α_{rf} , the limiter $\lambda_{\alpha r}$ is used and its form is given by

$$\lambda_{\alpha r} = \min \left\{ \max \left(1 - \max \left[\left(1 - (4\alpha_P(1 - \alpha_P)) \right)^2, \left(1 - (4\alpha_N(1 - \alpha_N)) \right)^2 \right], 0 \right), 1 \right\}. \quad (38)$$

3. NUMERICAL MODELS

This section consists of two main sub-sections. First part includes model validation and comprehensive test comparisons for a domed circular cylinder tank with two different fill ratios. Five different flow models are tested mostly with *interFoam* solver and compared with the measured values under two different 1D tank motions including impulsive and periodical loads. In the second part, sloshing simulation under negative-g acceleration is modelled with three different flow types using hydrazine type propellant. Solver performances are evaluated in terms of accuracy, convergence, stability and computational cost.

3.1 Model Validation and Experimental Test

All ground tests used conducted under 1-g gravity conditions. OpenFOAM® -v1812 version is used for all CFD model validation studies. During the sloshing simulations with multiphase *interFoam* solver, five different flow simulations i.e. *compressibleinterFoam*, *interFoam Laminar*, *interIsoFoam*, *interFoam kw-SST* and *interFoam LES* flow models are considered. *interFoam* LES model uses *Dynamic k-equation* model [32] based on new local dynamic formulation using one equation sub-grid scale (SGS) closure. The model solves turbulent kinetic energy equation by two coefficients derived from local flow properties. In the following tests, LES model will be referred as *interFoam*. The second turbulence model used is kw-SST two equation Reynolds Averaged Navier Stokes (RANS) model [33] which overcomes the deficiencies of the standard kw model with respect to dependency on the freestream values of k and w. It uses a linear eddy viscosity model (kw-Shear Stress Transport) as an isotropic turbulence model. *interIsoFoam* solver is derived from *interFoam* which assumes two incompressible

isothermal immiscible fluids. Free surface capturing is handled by using isoAdvector phase-fraction based interface capturing. Finally, *compressibleinterFoam* solver assumes two compressible non-isothermal immiscible fluids *where both fluids* are taken as isothermal during the simulations. This solver uses phase fraction based free surface interface capturing algorithm.

Comparisons with cylinder experiment: We use cylinder experiments conducted at German Space Center (DLR) using six-DOF Hexapod system [21]. Water filled cylindrical tank is made of transparent Plexiglass hard plastic material, and subjected to lateral sinusoidal and impulsive forces. The details of Cassini type cylindrical tank and the experimental setup is given in Figure 4 which can be obtained from Konopka *et.al.* [21].

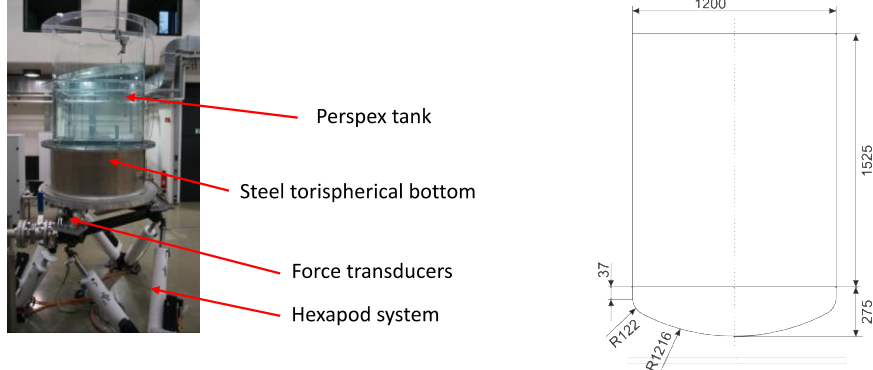


Figure 4. Perspex cylinder tank with 6-DOF Hexapod system and its geometry [21].

The tank geometry and its mesh structure are constructed with FreeCAD® and ICEM CFD® software, respectively, and presented in Figure 5. For the grid independence tests, four different mesh structures are considered and their mesh parameters are given in Table 1. Cylinder is subjected to linear acceleration in lateral -x direction as

$$\ddot{x}(t) = -w_c^2 x_a \sin(w_c t) \quad (39)$$

where $w_c = 2\pi f_c$ is the excitation frequency. Two experimental scenarios are set up and the total duration of the experiments is divided into two phases: the first is excitation phase derived by hexapod and the second is natural damping phase observed right after the hexapod stopped. For the periodic loading, excitation frequency ($w_c = 0.74 \text{ Hz}$) is selected below the resonance frequency ($w_c = 0.87 \text{ Hz}$) of the tank and two different fill ratios are defined in Table 2.

Cylinder Tank Experiment 1 (fill ratio 635/1800 mm, $h/2R \approx 0.5$): The acceleration measured on the hexapod system in the -x direction at 26. second is presented in Figure 6a, and the corresponding shifted acceleration data used in CFD simulations is given in Figure 6b.

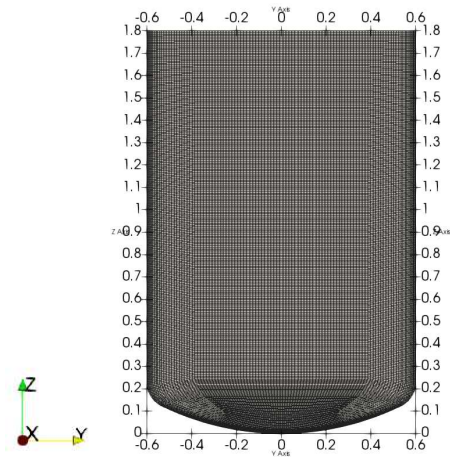


Figure 5. Tank mesh structure at -zy plane for 2.4 million mesh.

Mesh	Total Mesh Numbers	Minimum Mesh Spacing (m)	Maximum Mesh Spacing (m)
1	$N_x \cdot N_y \cdot N_z = 698490$	0.0052785	0.017003
2	$N_x \cdot N_y \cdot N_z = 1548666$	0.0044145	0.013059
3	$N_x \cdot N_y \cdot N_z = 2415519$	0.0040030	0.010932
4	$N_x \cdot N_y \cdot N_z = 3647509$	0.0032933	0.009563

Table 1. Mesh parameters for different resolutions.

Experiment No	Excitation Frequency (f_c)	Excitation Amplitude (x_a)	Maximum Excitation Amplitude ($\ddot{x}_{max}$)	Fill Ratio
1	-	-	1.0 m/sn ²	635 mm
2	0.74 Hz	20 mm	0.43 m/sn ²	1200 mm

Table 2. Selected experiments [21] and their parameters used in CFD simulations.

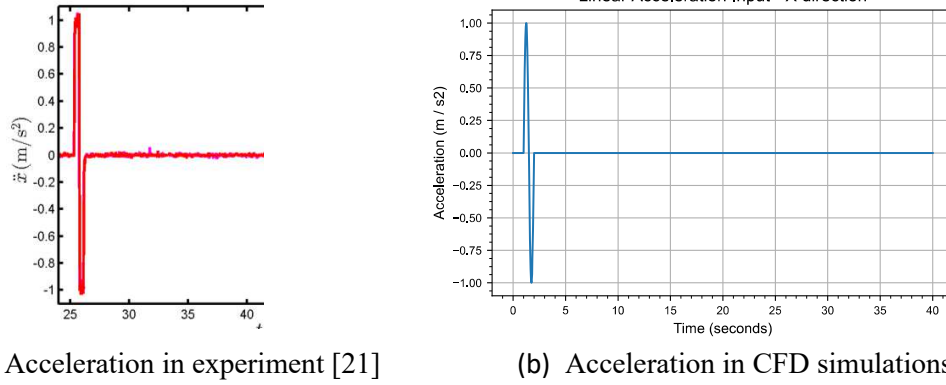
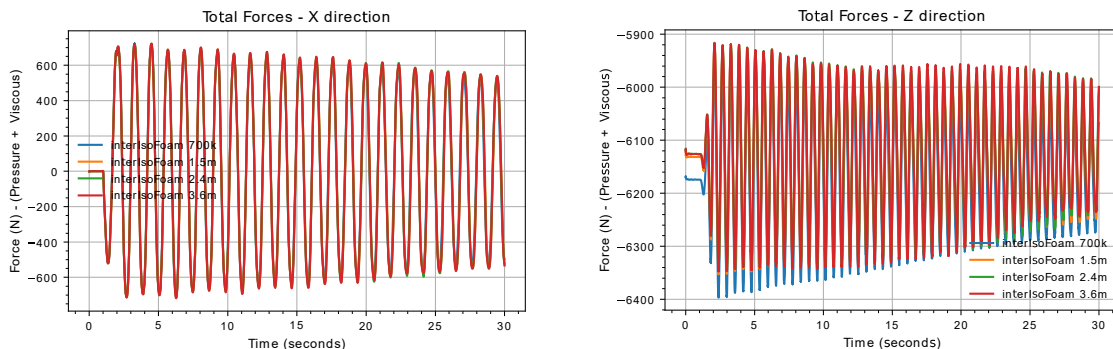


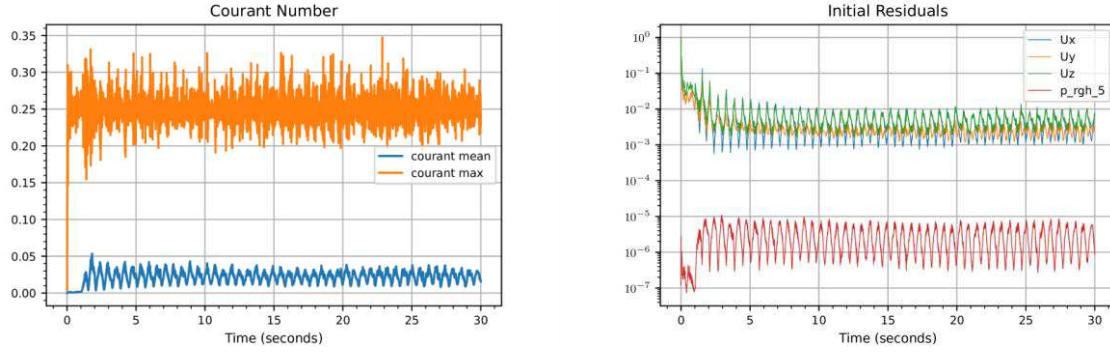
Figure 6. Pulsatile acceleration used in Experiment and CFD simulations for fill ratio 635 mm.

Corresponding lateral total force in -x direction is measured in the experiment by Konopka *et al.* [21], and the measured data is extracted using digitizer WebPlotDigitizer®. In this cylinder experiment, our main objective is to compare the different *interFoam* solvers, which basically model the sloshing as two-phase immiscible flow (i.e. liquid and gas), and discuss their effectiveness and usability in severe sloshing based on the experimental data. We compare the measured F_x and free surface elevations directly obtained from the experiments, and the first mode frequency analysis based on either force or center of mass results indirectly.

Mesh Independent Analysis: The effect of mesh resolution on the solutions is investigated for three different solvers at mesh numbers (700k, 1.5m, 2.4m, 3.6m). Comparisons are made for both force and Center of Mass (CoM) calculations. Three solvers used in the mesh independent analysis are *compressibleInterFoam*, *interIsoFoam* and *interFoam*. Analysis shows in Figure 7a that there are well-overlapped results in the directions of both excitation (-x) and vertical (-z) directions for force calculations. We used 1.5m mesh numbers in our simulations based on the Grid Convergence Index (GCI) results obtained.

Residual Analysis: Results show in Figure 7b that the flow parameters in sloshing simulations (such as velocities and pressure) are converged at the desired accuracy, where mean and maximum Courant numbers (Co) oscillate and keep their values around 0.025 and 0.25 respectively.

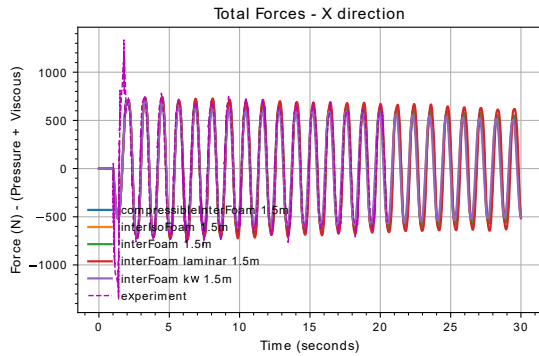




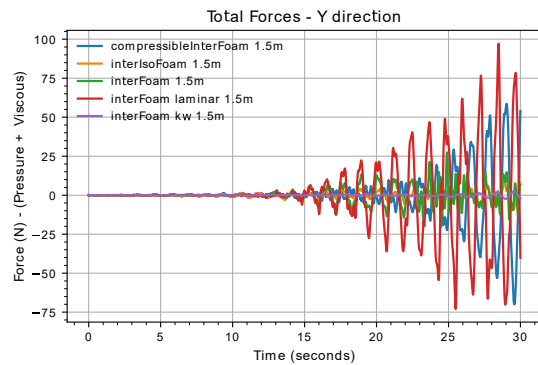
(b) Courant and Residual comparisons for *interIsoFoam* solver.

Figure 7. Courant number, Residual and Force results at 1.5m mesh size.

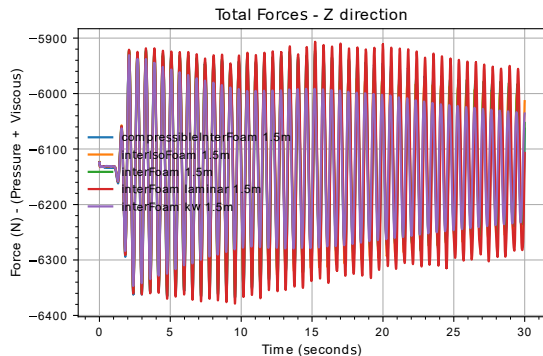
Forces on the tank wall: Forces are computed under single impulse acceleration given in Figure 6b at tank fill ratio $h/2R \approx 0.5$ and results are presented in Figure 8. Since the force in the x-direction (F_x) is the only measured data presented in the experiment [28], it is compared with five different solvers. Results show that, after the first impulse force resulting from the acceleration which has a spike due to the measurement and the test equipment, F_x values in Figure 8a demonstrates similar amplitudes and phase angles regardless of solver types. Slight damping effect on the amplitudes of F_x force is observed after initial excitation impulse. This effect is larger in the *interFoam* kw-SST turbulent solver while smaller in the *interFoam* Laminar solver. In other words, Laminar solver over estimates the forces compared to experimental results and other solvers. Similarly, Laminar solver has a greater estimate for force amplitudes as in Figures 8b-8c and phase shifts of force oscillations in y-direction as shown in Figure 8b. All solvers except laminar solver have a similar force approximation in z-direction. Figures 8a-8c show that phase angles of the force oscillations are well aligned and there is no significant phase shift. F_y force in the lateral direction has small perturbations around zero up to the first 15 seconds then amplitudes nonlinearly increase. The reason of why F_z force amplitudes oscillate around small amplitudes is that this force is mostly dominated by hydrostatic forces and there is a limited contribution coming from dynamics of the oscillating fluid perpendicular to excitation direction.



(a) F_x under single impulsive acceleration.



(b) F_y under single impulsive acceleration.



(c) F_z under single impulsive acceleration.

Figure 8. Force comparisons for various solvers.

Center of Mass (CoM) calculations of a sloshing fluid: During the simulations, high fidelity calculation of an instant center of sloshing fluid mass has a critical importance to obtain the correct moment acting on a propellant tank (i.e. on a spacecraft). Simple formulation is given below:

$$\begin{aligned}
 CoM_{x_i} &= \frac{\sum(\text{density of fluid} \times \text{volume of cell} \times \text{cell location}_{x_i})}{\sum(\text{density of fluid} \times \text{volume of cell})} \\
 CoM_{y_i} &= \frac{\sum(\text{density of fluid} \times \text{volume of cell} \times \text{cell location}_{y_i})}{\sum(\text{density of fluid} \times \text{volume of cell})} \\
 CoM_{z_i} &= \frac{\sum(\text{density of fluid} \times \text{volume of cell} \times \text{cell location}_{z_i})}{\sum(\text{density of fluid} \times \text{volume of cell})}
 \end{aligned} \tag{40}$$

Embedded CoM calculation is not available in OpenFOAM®-v1812 and we compute CoM's at each time step using swak4foam. A swak4foam script coding of Equation 40 is added to system/controlDict file to compute CoM's at each time step.

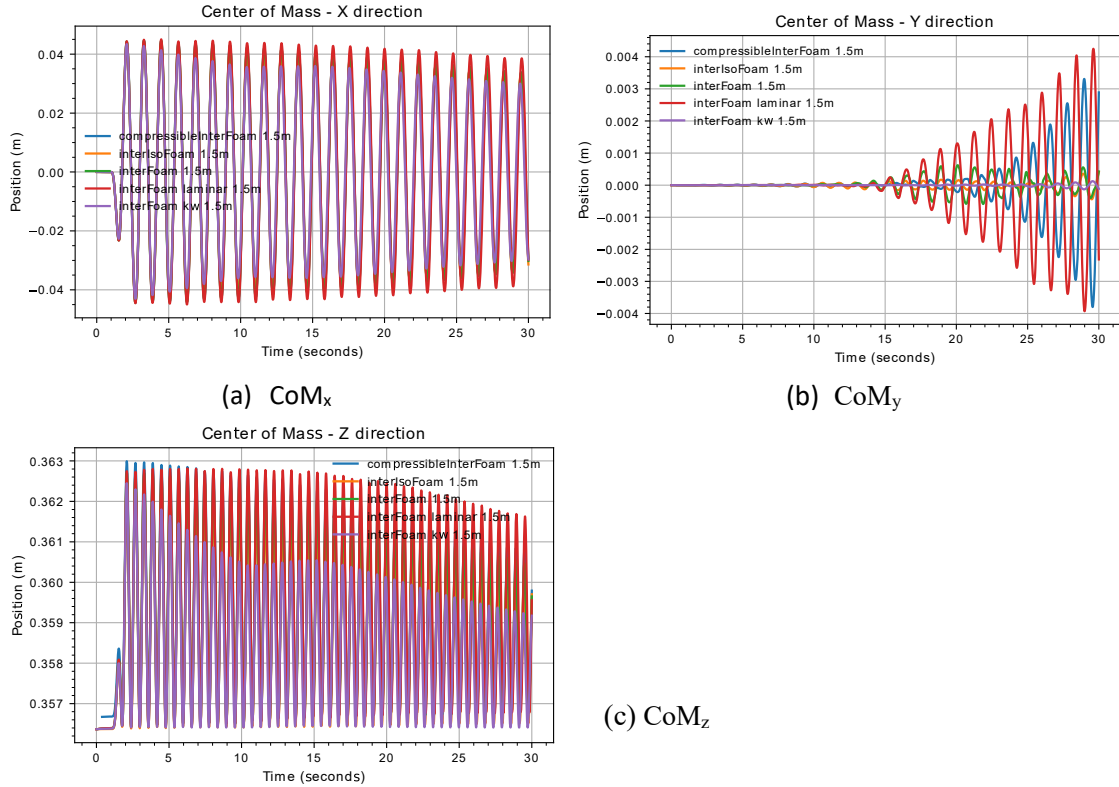


Figure 9. CoM comparisons for various solvers under single impulsive acceleration.

Instant center of propellant mass under the excitation of a single impulsive force in -x direction shown in Figure 9a presents relatively good agreement for all solvers in terms of phase angle and CoM displacements. However, *interFoam* kw-SST and *interIsoFoam* solvers show notable underestimate CoM results in all directions. In -y direction, all solvers have different amplitudes and phase angle results. On the other hand, a limited damping effect after single excitation is observed in both -x and -z directions. In -y direction, all solvers show almost zero center of mass oscillations up to 15 seconds and then each solver gives different CoM amplitudes with shifted phase angles. Even if in general, all solvers give compatible CoM amplitudes and aligned phase

angles except in -y lateral direction, it should be noted that these differences may increase with the increasing excitation amplitudes and nonlinearity originated from geometry.

Cylinder Tank Experiment 2 (fill ratio 1200/1800 mm, $h/2R=1.0$): Periodic acceleration measured by Konopka *et al.* [21] on the hexapod system in -x direction at 265. second is presented in Figure 10a and the corresponding shifted acceleration data used in our CFD simulation is given in Figure 10b.

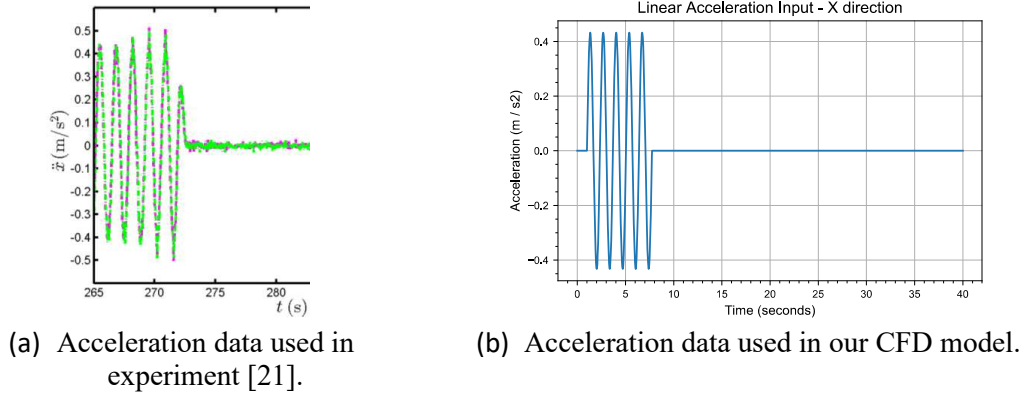


Figure 10. Acceleration data used in sinusoidal excitation for fill ratio $h/2R=1.0$.

Mesh Independent Analysis: The effect of mesh resolution on the solutions is investigated for three different solvers (*compressibleInterFoam*, *interIsoFoam* and *interFoam*) at mesh numbers (700k, 1.5m, 2.4m). Grid Convergence Index (GCI) analysis shows that 1.5m mesh generates overlapped results.

Residual Analysis: Flow parameters such as velocities and pressure are converged at the desired accuracy where mean and maximum Courant numbers (Co) oscillate and keep their values around 0.025 and 0.25 respectively.

Forces and CoM calculations for higher fill ratio: Forces and instant centers of sloshing fluid mass are computed under five periodic accelerations given in Figure 10b at tank fill ratio $h/2R = 1.0$. Simulated F_x forces for three different solvers are compared with the experimental data. Results show that F_x values in Figure 11a have similarities in terms of amplitudes and phase angles of force oscillations regardless of solver types. The damping effect on the amplitudes of F_x forces is not clearly observed after five excitation impulses even if there is a small damping tendency in both experiment and simulation results except *interFoam* laminar solver. In other words, laminar solver slightly overestimates the force compared to the experimental result. Similarly, the laminar solver has a greater estimate for force amplitudes and the phase shifts in y-direction as shown in Figure 10b. All solvers have similar force approximation in z-direction shown in Figure 10c where low order modes, gravity and hydrostatic forces dominate the flow in the vertical direction.

CoM results in Figure 11d, show good agreement in the excitation direction for all solvers. However, *interFoam* laminar solver shows notable overestimation and phase shift in -y direction presented in Figure 11e. No damping effect is observed in all directions due to the limited simulation time. For all solvers, center of mass oscillations in -y direction show almost zero up to 10 seconds and then each solver generates different CoM amplitudes with phase shifts. In vertical direction, *interIsoFoam* solver in Figure 11f gives under estimate CoM results compared to other two solvers.

The comparison of instantaneous free surface wave elevations is shown in Figure 12. *interIsoFoam* solver captures the similar wave profiles at 2.0 and 4.9 seconds except some minor differences at the free surface slopes near to tank walls.

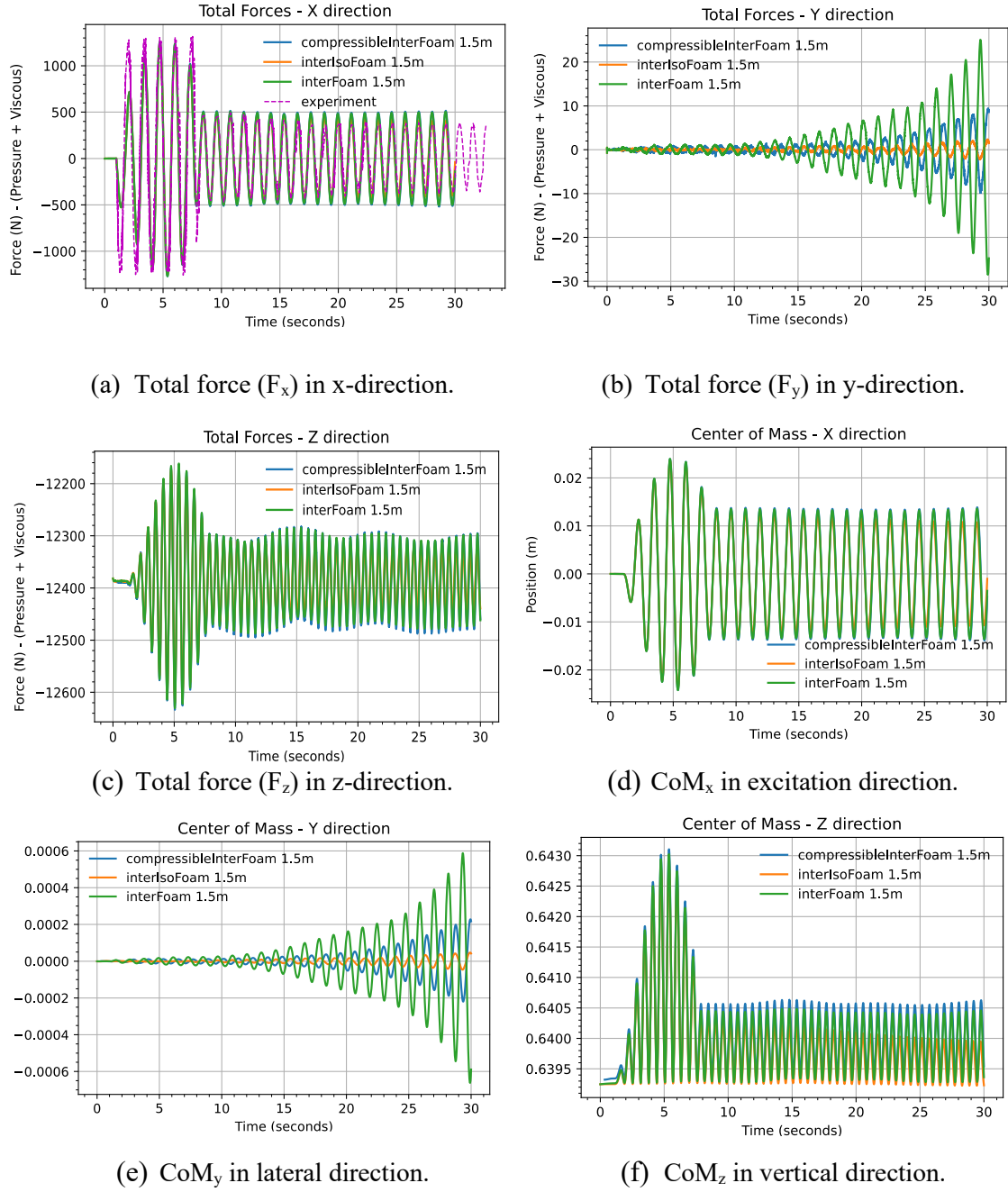
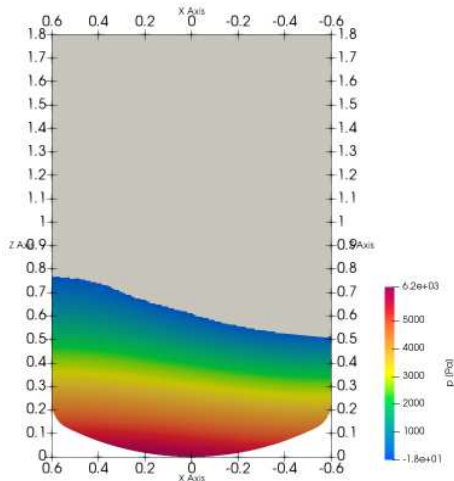
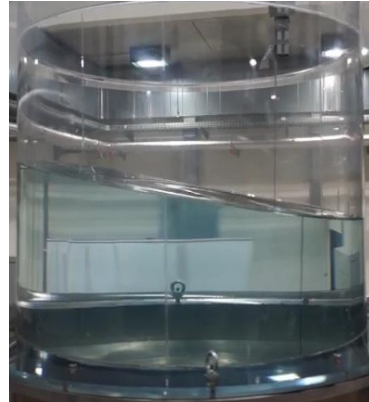
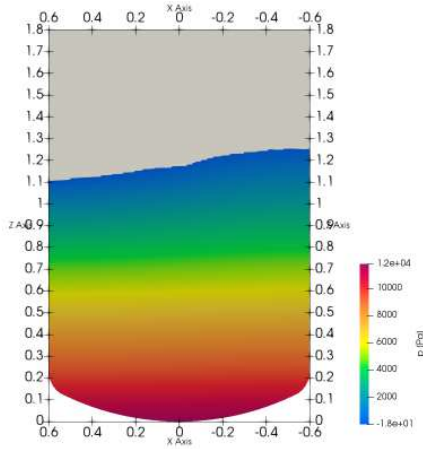


Figure 11. Force and Center of Mass comparisons for different *interFoam* sloshing solvers.



a) Our *interIsoFoam* sloshing model @2.0 s. b) Experiment by Konopka *et al.* [21] @2.0 s.



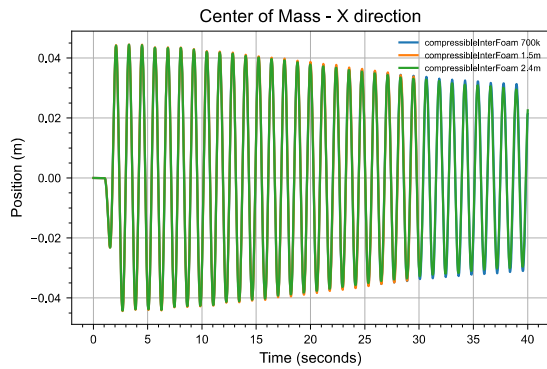
c) Our *interIsoFoam* sloshing model @4.9 s. d) Experiment by Konopka *et al.* [21] @4.9 s.

Figure 12. Free surface instantaneous wave profile comparisons at x-z plane.

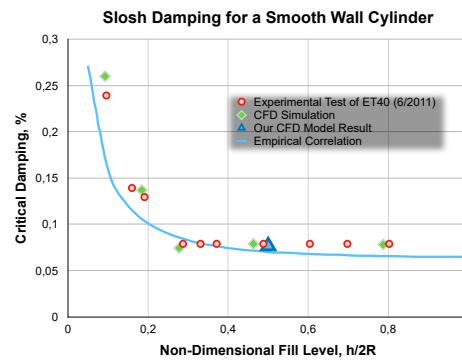
Sloshing Damping: Correctly estimating the sloshing damping of a periodically or impulsively excited tank has a critical importance to observe the sloshing damping ratio for different tank fill levels. In a real dynamic system, sloshing wave (or Force/CoM) amplitudes decay with time when exciting force removed. Characterization of the sloshing damping for a given tank geometry still not a straight forward task and existing approaches are based on the empirical formulations using different experiments such as [28, 29, 30, 31] conducted for a circular cylinder to measure the viscous damping. There are different methods in the literature estimating the sloshing damping using different data types. These methods are classified as *ring force*, *drive force*, *wave amplitude response*, *wave amplitude decay* and *wave force decay methods*. The following formulation is used for obtaining a critical damping ratio of an ideal system with a constant damping.

$$\zeta = \frac{1}{2\pi n} \ln \frac{A_0}{A_n} \quad (41)$$

where A_0 , and A_n are the first and n^{th} amplitudes of the oscillating force. Using CoM_x oscillations in Figure 13a for a fill ratio $h/2R = 0.5$, damping ratio through 10 cycle becomes % 0.076. Our critical damping is correlated with the CFD and the experimental results of Konopka *et al.* [21] presented in Figure 13b.



(a) CoM_x oscillations during free damping in a cylindrical tank.



(b) Critical damping comparisons as a function of fill level [21].

Figure 13. Critical slosh damping in a cylindrical tank using center of mass oscillations.

Evaluation of the solvers: Sloshing experiments under 1D excitation force show that solvers having different flow assumptions mostly predict the F_x and CoM_x correctly in the excitation direction. However, forces and CoMs in the lateral and vertical directions are obtained in different magnitudes and phase angles (in some cases) for each solver and flow types. These differences are expected to increase in cases where external excitations to the propellant tank are severe and

adverse. It is also observed that, when the tank fill ratio reduces, the sloshing and its impact loads on tank wall increase. One of the most interesting findings in this validation process is to observe a highly nonlinear secondary vorticity flow with higher frequency modes in -y direction, which is perpendicular to the excitation direction. For fill ratio $h/2R = 1.0$, maximum lateral force (F_y) oscillates around 20N and following force magnitudes are around 10N. On the other hand, for fill ratio $h/2R = 0.5$, maximum lateral force (F_y) oscillates around 75N and following force magnitudes estimated by the same solver is around 50N. When the fill ratio reduces half, F_y force increases 3-5 times based on the solver type and F_x force increases around %30 due to sloshing impact load. However, F_z force reduces by half due to the dominant hydrostatic effect as expected. The curvature of the tank wall also plays an important role on generating high-energy eddies together with secondary flow near to tank wall. Turbulent models used in these simulations are expected to capture these small eddies. But this requires both high order schemes to adequately resolve the high-energy eddies and large enough mesh size (which is related to mesh scale $Re_{Lx}^{2/5}$) to resolve the viscous sub-layers.

3.2 Negative-g Acceleration

Capturing the physics of extreme propellant sloshing under sudden maneuvering of a rocket/space vehicle has a critical importance. During the maneuverings, keeping the vehicle on its desired trajectory and providing uninterrupted trust is one of the most vital mission critical conditions need to be satisfied. Correct estimation of the disconnection time and the duration of this disconnection from the propellant drain system is important for further PMD design. High precision calculation of the total impact loads (forces and moments) acting on propellant tank walls and thus vehicle is also important to prevent the possible divergence from its desired trajectory and estimate the trust vector correctly. Propellant tank geometry shown in Figure 14a is selected as a domed circular cylinder where its diameter and length are 0.8m and 1.5m respectively. 1D negative-g acceleration motion in the vertical direction shown in Figure 14b is considered as a sudden maneuvering scenario. Initially tank subjects to a positive acceleration for three seconds then drops to a negative-g value for 18 seconds with a 50% fill ratio. The propellant is selected as incompressible hydrazine type liquid.

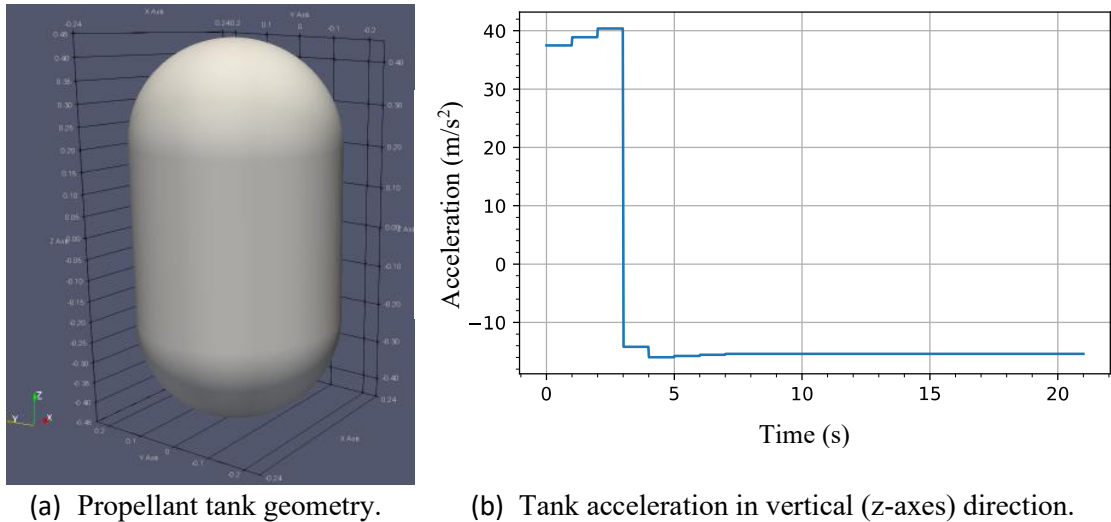


Figure 14. Tank geometry and its 1D acceleration in z-axes.

Three different flow models (LES, kw-SST and Laminar flow) are studied using *interFoam* solver under four different internal pressure conditions (0, 50, 300 and 600 bar). Total forces (F_x , F_y , F_z) and instant centers of propellant mass (CoM_x , CoM_y , CoM_z) are compared for each solver models. Additionally, their stability, convergence rates, computation costs and precision are discussed.

Numerical validation of the model is performed by residual analysis and grid independence tests. Results show that the mean Courant oscillates under 0.05 and max Courant number steadily oscillates around 0.25. Mean interface Courant number oscillates around 0.19. The residual

analysis proves that the flow parameters such as velocities, pressure and continuity are converged with acceptable error rates. The mesh independence analysis is conducted for 1.12m, 4.73m and 9 million meshes. Close results showed that 1.12m mesh is acceptable for engineering simulations even if turbulence models may require higher spatial mesh resolutions for sub-grid scale and kinematic energy levels.

Forces on the domed cylinder tank wall: Forces illustrated in Figure 15 are computed under single impulse acceleration at tank fill ratio $h/2R \approx 0.5$ and Courant number ($Co = 0.25$). Two extreme internal pressures are compared (0 and 600 bar) for three solvers.

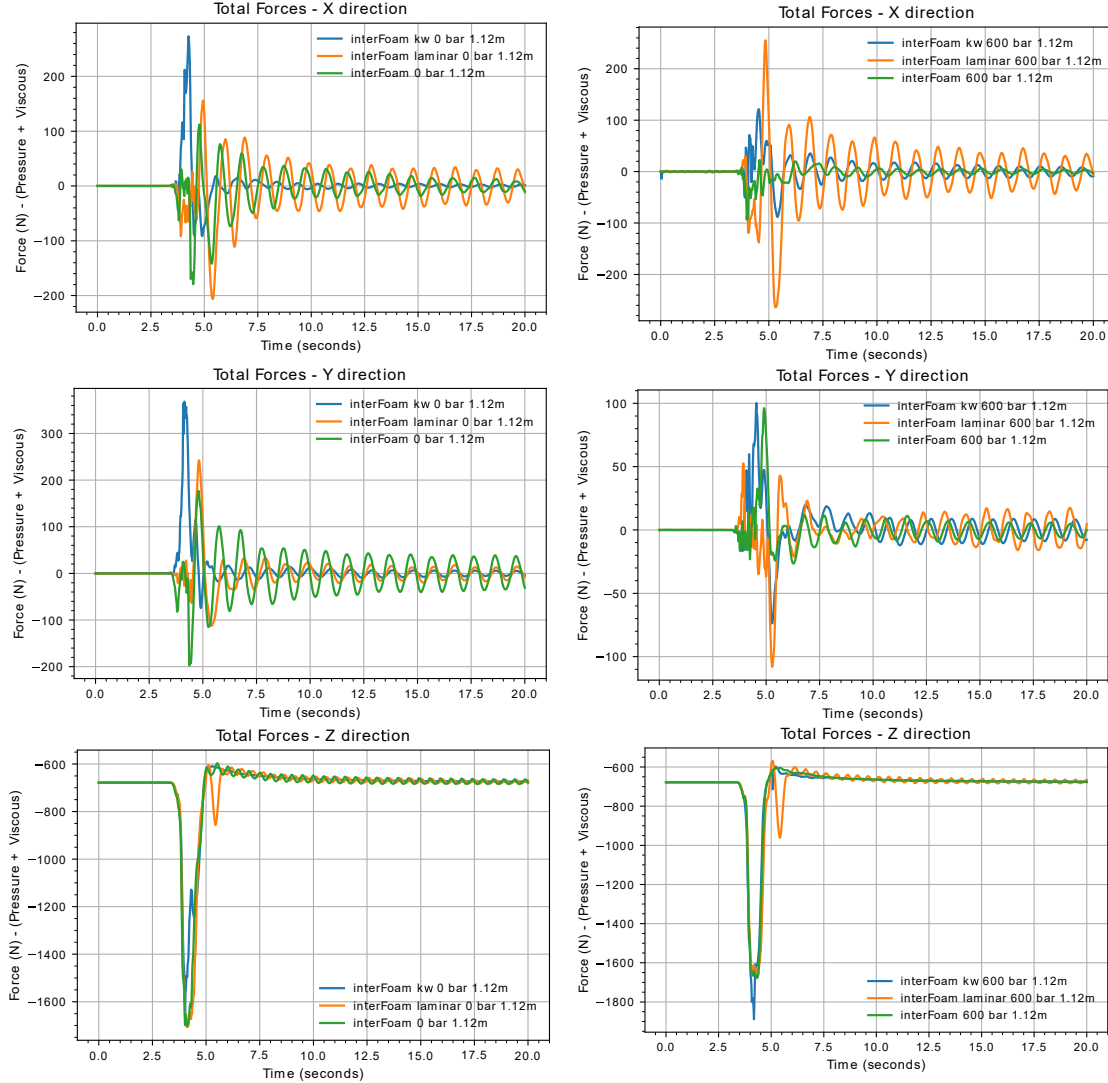


Figure 15. Force comparisons of three different flow models for 0 bar (left column) and 600 bar (right column) internal pressure.

Figure 15 shows that, when tank pressure is increased in two phases (propellant and gas) simultaneously, there is no significant impact on the sloshing damping. However, pressure-velocity coupling and its convergence rates are improved with the increasing pressure. Nonlinear effects due to the doomed ceiling of the tank are observed mostly at the lateral directions where higher frequency contributions on free surface oscillations are dominated rather than low order terms. The most significant differences occur in -y direction due to these short-lived contributions and higher internal pressures have a limited damping effect on free surface oscillations. These results support that effective free surface damping based on the higher internal pressure may only occur if the gas phase is pressurized alone.

In order to observe whether a simplified laminar flow solution is mature enough to capture the basic severe sloshing dynamics in obtaining correct forces and instantaneous CoM's, we

compared in Figure 16 forces obtained by two flow models (LES and Laminar) under various internal tank pressures.

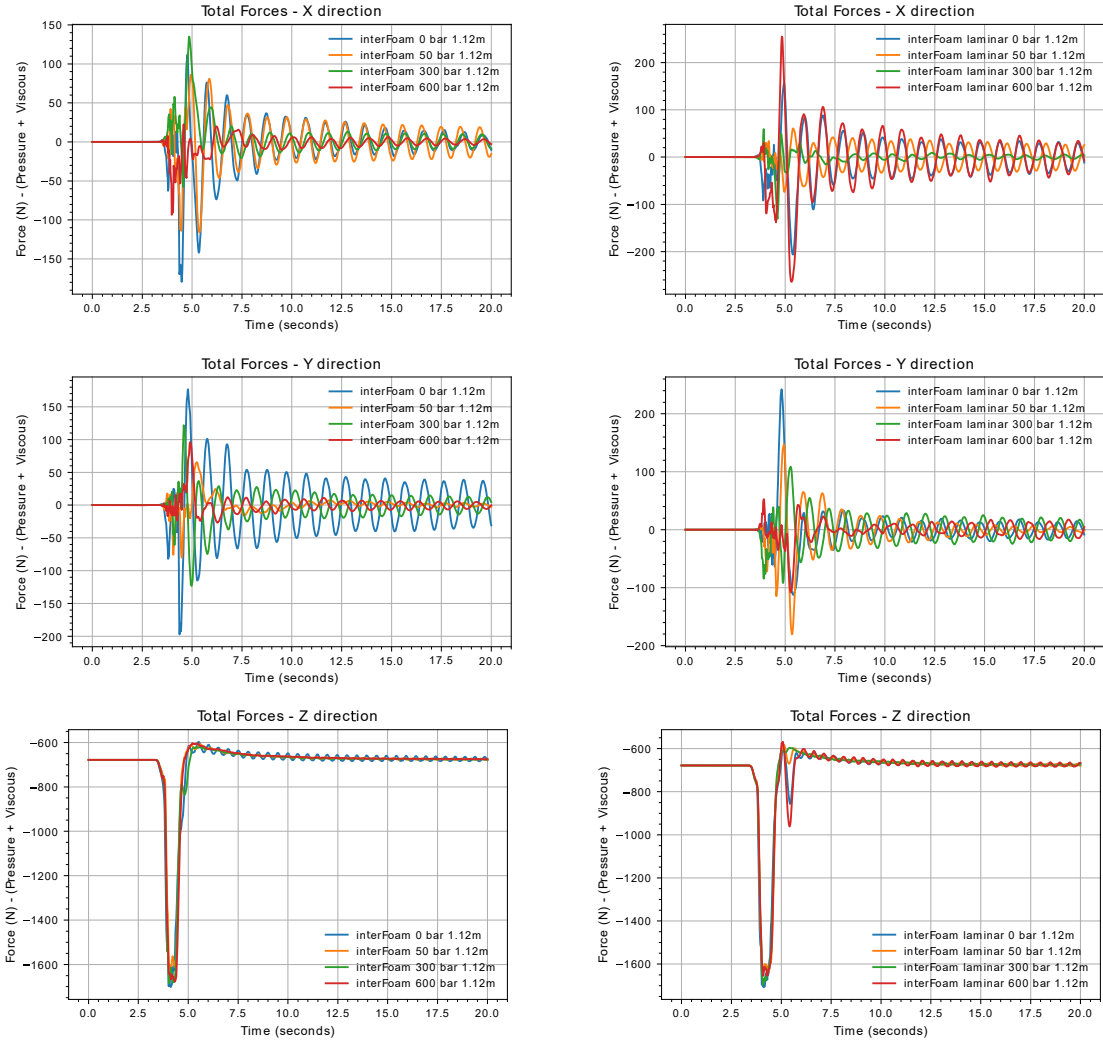


Figure 16. Comparisons of forces for different internal pressures using *interFoam* LES (left column) and Laminar (right column) flow models.

During the negative-g acceleration (between 3-6 seconds) laminar flow solver gives higher force amplitudes in -x and -y directions where momentum effect is significantly less compared to -z direction. Both solvers compute the force amplitudes in -x and -y directions with some phase shifts for each internal pressure. In F_z calculation, mostly dominated by hydrostatic and momentum forces, laminar flow solver gives a second jump after the first impact load which is not observed in LES results. Magnitudes of the forces in -x and -y directions tend to reduce with the increasing internal pressure. But this is not always true and mostly attributed to the numerical coupling algorithm subject to incompressible assumption in a confined environment.

Similarly Figure 17 displays the center of mass (CoM) comparisons obtained by three solvers for two different internal pressures (0 and 600 bar). When tank pressure is increased, the amplitudes of CoM's are slightly reduced in the lateral directions (-x and -y) where the high frequency terms are dominated and no significant impact of pressure in the acceleration direction. Three solvers give similar instant center of mass displacements in -z direction, whereas significant differences occur in the other directions where hydrostatic (i.e., gravity force) and momentum forces are negligible. Among the solvers laminar flow solver mostly gives overestimated CoM results (between 5-110%) in -x direction, where short lived nonlinear sloshing effects are dominated. Phase shifts of CoM oscillations in -x and -y directions are observed for each solver. Differences

in the results are due to high frequency contributions, and thus higher-order schemes are necessary to capture the nonlinearity at the free surface especially in the lateral directions.

Figure 18 compares the CoM's for four different internal pressures using *interFoam* LES and Laminar flow models. Results for two solvers with varying internal pressures are in good agreement in the excitation direction (-z) where the flow is heavily dominated by the momentum and gravity forces.

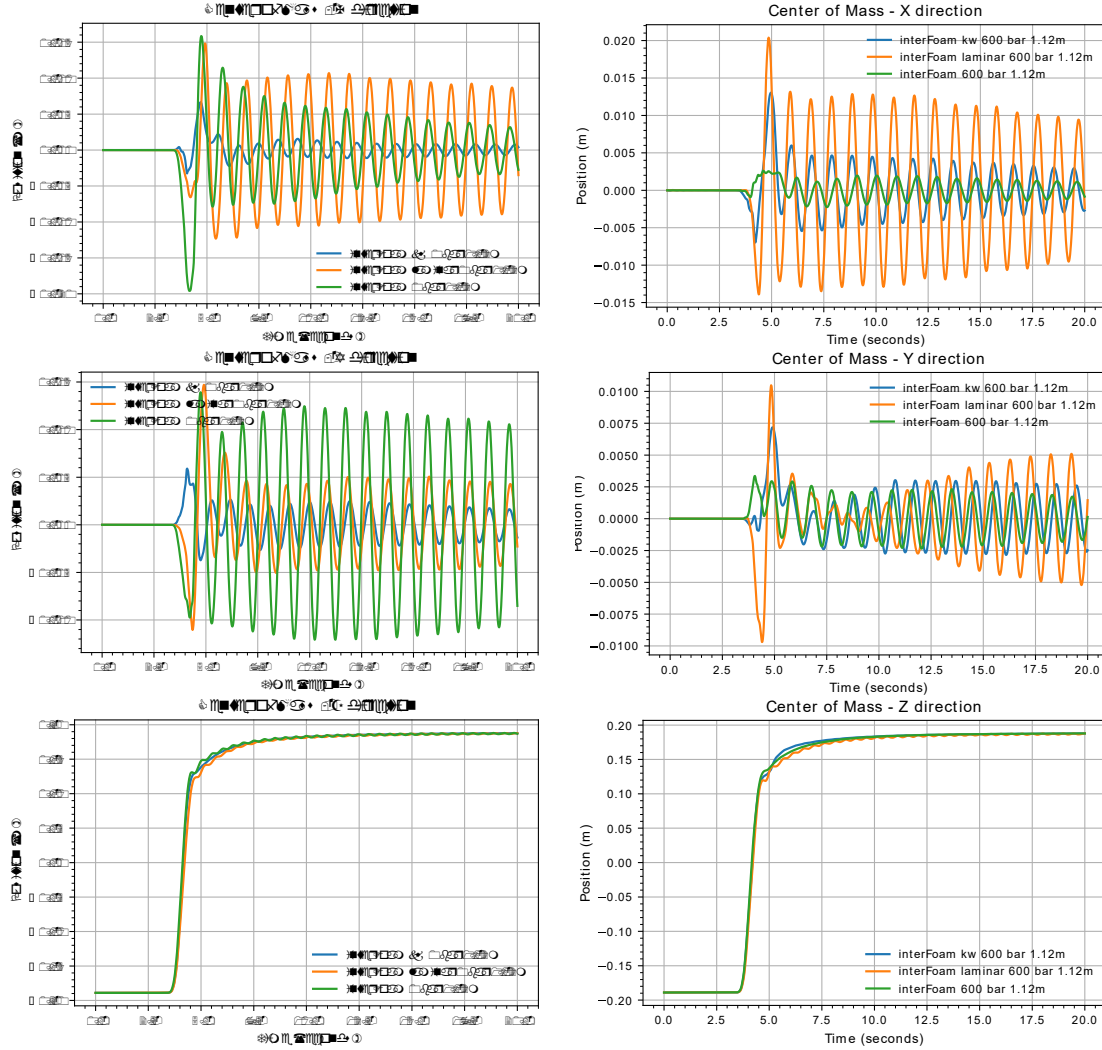


Figure 17. CoM comparisons of three different flow models for 0 bar (left column) and 600 bar (right column) internal pressure.

These results confirm that, if flow is dominated by strong momentum and gravity forces, laminar flow model works fine and its force and CoM results are comparable with the turbulent flow models. On the other hand, the CoM displacements show strong disagreement in the lateral directions (-x and -y), where geometric nonlinearity and high frequency flow oscillations are dominated. There is a need for more intensive numerical study to validate the usability and effectiveness of higher order schemes in laminar flow model to capture the nonlinearity and short-lived high frequency contributions such as secondary flow in the lateral directions. First order time stepping scheme is used in our simulations and this part is open to more numerical investigation and testing.

Similarly, results in Figure 18 clearly show that there is no sloshing damping effect on the momentum and gravity force dominated flow regimes where momentum effect in this direction is much higher than the pressure applied. Of course this finding is valid under the assumption that internal pressure is occupied to both fluid and gas phases simultaneously. However, there is a

sloshing damping effect of increased internal pressure in the lateral directions (as observed in other cases) resulting from numerical aspects (such as increased pressure gradients). Two phases are assumed isothermal and there is no phase change from fluid to gas. Because the density ratio between fluid and gas is fixed. Since the damping forces in sloshing specifically at lateral directions are generally not physical based in higher internal pressures, even if there is a tendency to observe higher damping on force and CoM values with the increasing internal pressures, this is not observed for all pressure cases. Numerically, increased pressure gradients at the interface especially with the low order schemes may lead to a suppression of high frequency oscillations.

During the negative-g maneuvering, accurate prediction of discontinuities in the propellant flow to drain system is critical and this is computed in CoM_z for 50% fill ratio in Figures 17 and 18. All solvers are computed the same mass displacement in vertical direction (roughly 36 cm) which is critical to drain and PMD system. Results show that, after 3.85 seconds, propellant mass starts to disconnect from the bottom of the tank system. Additionally, higher internal pressures do not affect the mass displacement due to strong momentum force as expected.

Stability, Accuracy and Performance comparisons of the solver models: In OpenFOAM CFD model, critical parameters related to the flow stability, convergence rate and accuracy are mostly defined in `fvSchemes` and `fvSolution`.

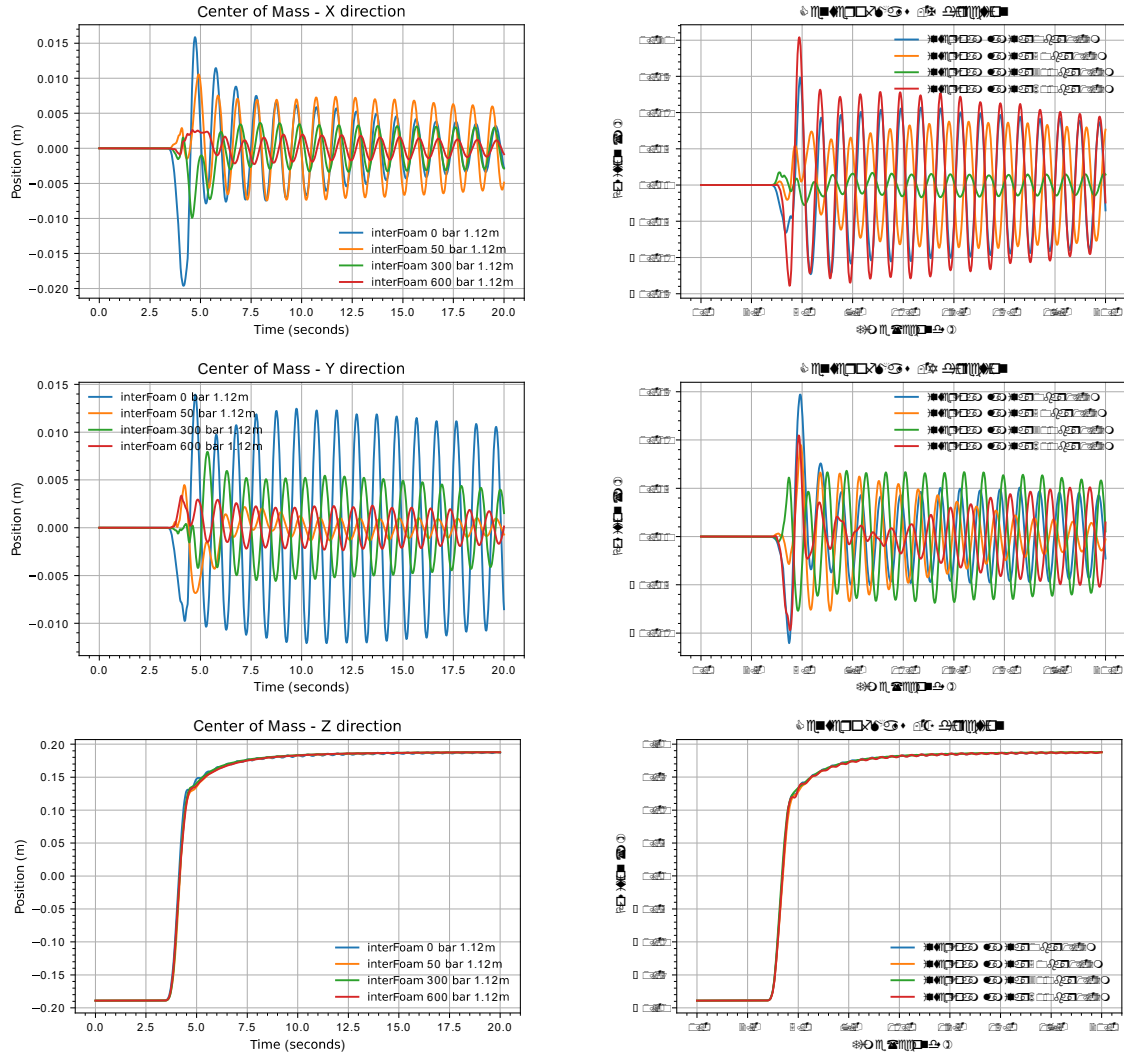


Figure 18. Comparisons of center of mass (CoM) for different internal pressures using *interFoam* LES (left column) and Laminar (right column) flow models.

PIMPLE algorithm used usually converges with `Outer Corrector` parameter 2 but for higher converging rates and low residual values, one may take 3 or more iterations with relatively

less computational cost. Because, due to rigid body dynamics of the fuel tank, mesh update occurs after this iteration. Outer Corrector 3 and pressure-velocity coupling iteration 6 generate residual convergence rates around 10^{-7} .

All models are run on the Intel processors (cases that can be divisible to 40, run on Intel(R) Xeon(R) Gold 6148 CPU @ 2.40GHz, and cases that can be divisible to 28, run on Intel(R) Xeon(R) CPU E5-2690 v4 @ 2.60GHz) for 20 seconds simulation time. Following Tables 3-5 show simulation time, core numbers and wall clock time for each model cases.

Case	Wall Clock Time (s)	Core Numbers	Simulated Time (s)
Hydrazine0barkwSST	70324	80	20
Hydrazine50barkwSST	76458	84	20
Hydrazine300barkwSST	92193	84	20
Hydrazine600barkwSST	81577	84	20

Table 3. Performance comparisons for kwSST.

Case	Wall Clock Time (sec.)	Core Numbers	Simulated Time (s)
Hydrazine0barLaminar	83620	80	20
Hydrazine50barLaminar	75382	84	20
Hydrazine300barLaminar	67541	168	20
Hydrazine600barLaminar	99033	84	20

Table 4. Performance comparisons for Laminar.

Case	Wall Clock Time (s)	Core Numbers	Simulated Time (s)
Hydrazine0barLES	86400	80	17.74
Hydrazine50barLES	86968	84	20
Hydrazine300barLES	82391	84	20
Hydrazine600barLES	123378	84	20

Table 5. Performance comparisons for LES.

Case	Wall Clock Time (s)	Core Numbers	Simulated Time (s)
Hydrazine50barLaminarCoarse	75382	80	20
Hydrazine50barLaminarMid	249210	200	20
Hydrazine50barLaminarFine	344890	280	20
Hydrazine50barLEScoarse	86968	84	20
Hydrazine50barLESMid	426327	120	20
Hydrazine50barLESFine	518396	240	15.04

Table 6. Wall clock time comparisons of the models for different mesh numbers.

The simulations are just repeated one or two times due to the limited computational resources, therefore, above presented wall clock time results may not be reliable in a sense that they may be effected by the network contingency or any other external factors. Another important parameter effecting on the performance is the local cache sizes. We run on two different node types having 2 and 4 GB cache sizes. It should be noted that we are comparing the computational performance and cost of the flow models (i.e. solvers) not scalability (speedup) and their efficiency. Thus, results only show the average wall clock times for flow models. In Table 6, three mesh sizes (Coarse = 1.12 million, Mid = 4.73 million and Fine = 9 million) are tested, and their computational costs are listed for various core numbers. Wall clock time and corresponding core numbers show that the lowest computational cost is obtained in the kw-SST model, the highest is in the LES, and the laminar model has the medium cost. Similarly, the computational cost of the LES model is 15% higher than the coarser case of the Laminar model, 70% in mid and 50% in the fine mesh cases. It is again worthwhile to mention that these are rough estimates of computational costs and require multiple runs to get more precise wall clock time numbers.

4. DISCUSSION

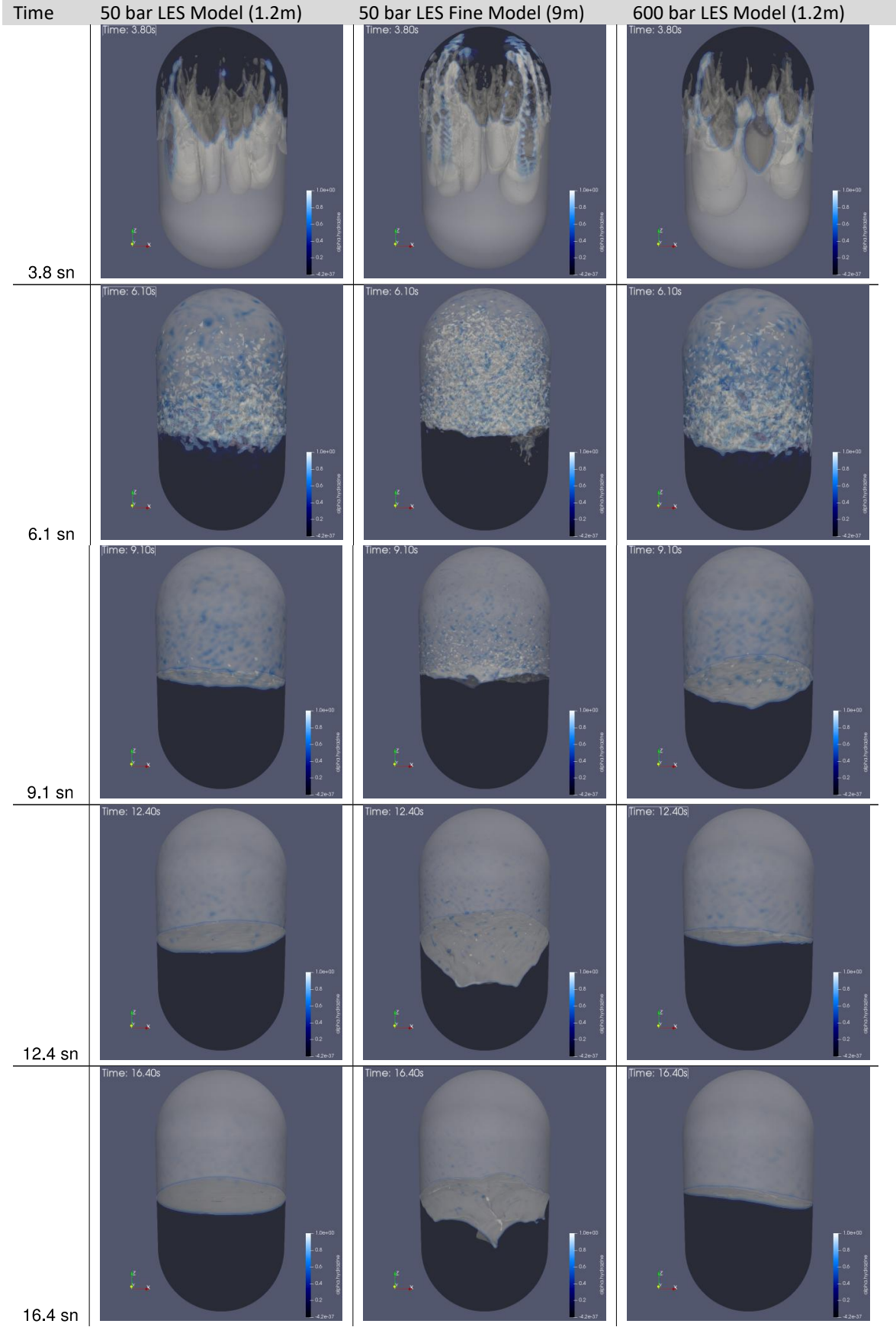
Selected *interFoam* solver from OpenFOAM® -v1812 is run for five different flow models and then reduced to three solvers (LES, kw-SST and Laminar) under the same initial and BC's to further analyze the impact forces and CoM's in the severe sloshing dynamics exposed to 1D negative-g acceleration motion. All models are converged with the acceptable error rates based on residual and mesh independence analysis. Residual and mesh independence analysis of each model reveal slightly different rates due to their flow assumptions.

In general results show that LES model comes forward and captures the severe sloshing dynamics better and gives acceptable performance for the most cases. Inherently, this performance of the LES model was expecting, and in most cases, it captures the severe sloshing dynamics in three directions better than the other flow models, even if we used coarser mesh numbers for the SGS model for a given Re number. But there is a need for running the LES model with higher mesh numbers to reduce the mesh size close to the turbulence length scale in SGS [34]. The main reason for this is that the LES model uses time and space (or only space) averaging instead of time-averaging of the velocities as assumed in the kw-SST model. In the negative-g scenario, kw-SST model initially did not converge due to the diverging effect of kinetic energy. This problem is resolved by defining relative tolerance (relTol) which uses residual values from previous and current iteration steps. During severe sloshing in negative-g maneuvering after three seconds, we did not observe any numerical diffusion in the hydrazine phase, which normally shows itself in volume rate as diverging from one. One critical point is that the liquid-gas interface generates unstable solutions due to the numerical diffusion in the solution of turbulence kinetic energy (k) equation. Kinetic energy has a very limited effect on the fluid part at the free surface interface due to the significant density difference between gas and liquid and it is expected to diffuse. To do that there are some solutions in the literature including additional damping for the turbulence parameters on the free surface interface. But this approach may generate adverse effects on the results.

Turbulence flow models such as *interFoam* LES have a close relation to mesh scale ($Re_{Lx}^{2/5}$) and their precision is highly sensitive to mesh resolution defining the resolved space. To show this close relation and sensitiveness, two mesh resolution cases (1.2 million and 9 million) of the LES model are compared in Figure 19 with two internal pressures. Snapshots taken from different δt times show that there is a significant difference in the capturing of sloshing flow field and free surface interface for two mesh resolutions as expected. Lower scale effects (SGS) are better resolved and high frequency contributions better trucked and captured. The main characteristic of the flow field in 9 million mesh case is that it contains high frequency flow modes resulting from high kinetic energy zones. But, the life spans of these higher frequency contributions are short and have a negligible effect on the direction where momentum and gravity effects are dominant. On the other hand, they have a significant effect to forces and CoM's in the lateral directions and our results support this effect. High-order numerical schemes are suggested to capture these effects better with the existing mesh sizes.

It was noted before that forces and CoM values are not significantly affected when the flow is dominated by the strong momentum and gravity forces. During the severe sloshing where high frequency mode contributions are most critical at time steps between 3.8-12.4 s, the effect of pressure is numerically observed at lateral directions. At time steps 16.4 and 19.1 s, high-frequency modes disappear and momentum and gravity forces start to dominate again, and thus flow similarity increases.

Increasing the internal pressure in both phases simultaneously have no effect on low frequency contributions coming from momentum and gravity forces. Thus, pressurization of two phases has no significant impact on the sloshing damping. However, pressure-velocity coupling and its convergence rates are improved numerically with the increasing pressure. These results also support the fact that effective damping based on the higher internal pressure may only occur if the gas phase is pressurized alone.



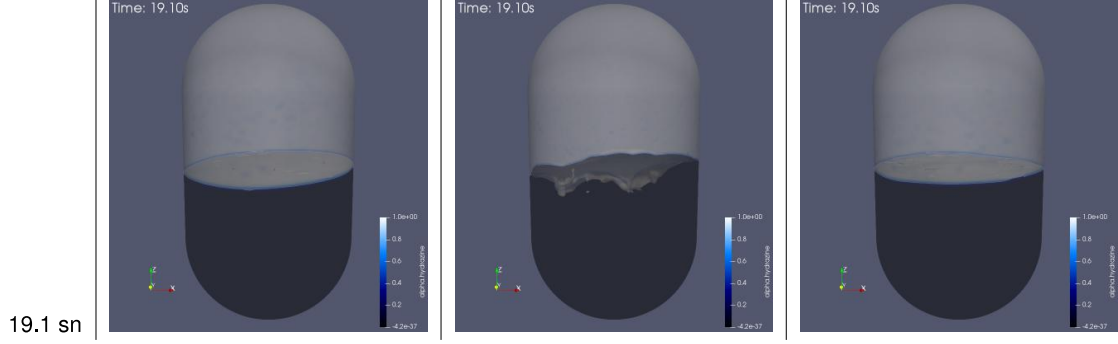


Figure 19. Impact of fine mesh and internal pressure on *interFoam* LES solver.

To further analyze the lateral nonlinearity effects in different cases, we studied the experimental sloshing cases in close. Figure 20 illustrates free surface streamlines of velocity magnitudes in two different fill ratios under impulsive and sinusoidal excitation loads. Under the same excitation force amplitude we observe higher velocity magnitudes in $h/2R = 0.5$ fill ratio as expected. The most important observation is that, for both cases, there are significant velocity components occur in the lateral direction which is perpendicular to excitation direction resulting from the shape of circular geometry. But these lateral velocity components have a short life span and not dominating the flow as velocities in the main direction of excitation. Their life span in impulse load case is observed shorter than the sinusoidal excitation case which gives the continuous kinetic energy and excitation force to the tank.

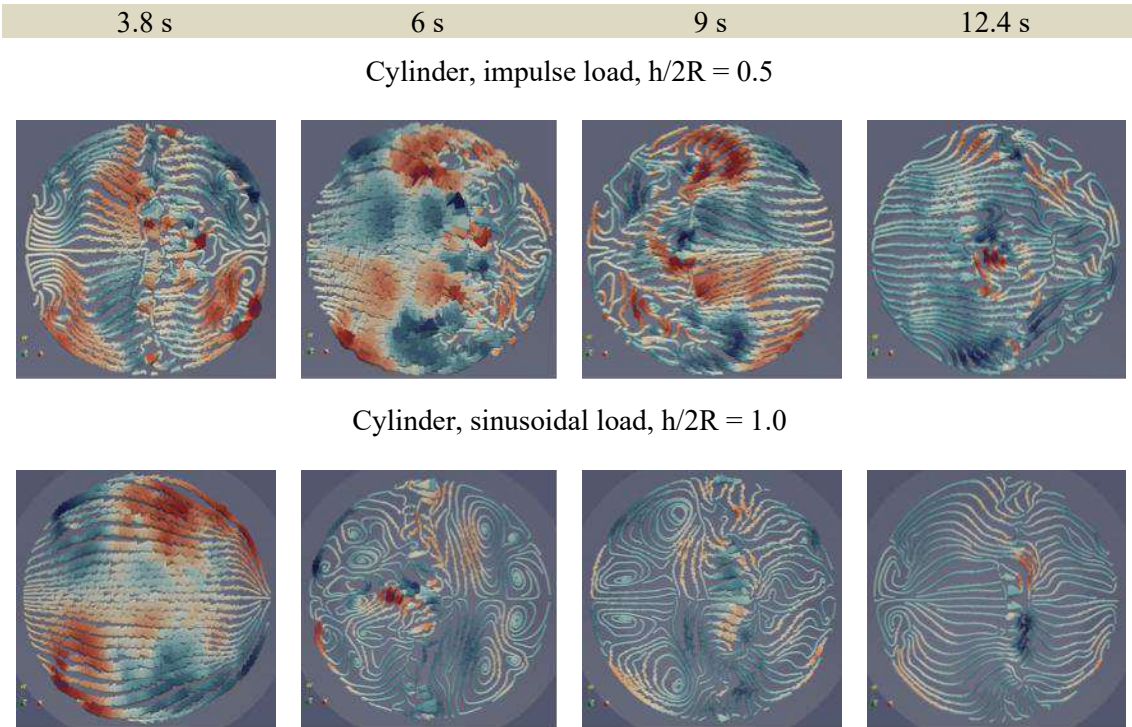


Figure 20. Velocity magnitude vectors near to free surface.

We observed similar short lived lateral velocity components in negative-g acceleration case shown in Figure 21, even if there exist a highly momentum dominant flow in $-z$ direction and corresponding U_z velocities in negative direction. Fluid mass hits to the upper dome of the cylindrical tank at the negative acceleration moment with maximum $Re \approx 21,000$ and consequently generates multi-directional distribution of the fluid particles. Most of the dispersed streams oriented in the negative excitation direction and some of these streams join to circular and rotational trajectories within the same fluid mass. In Figure 21, we see positive-negative

velocity fluctuations around zero velocity in both lateral directions (U_x and U_y) while the main flow velocity (U_z) is heavily in the negative direction. After 5 s lateral velocities oscillate around zero but not disappear.

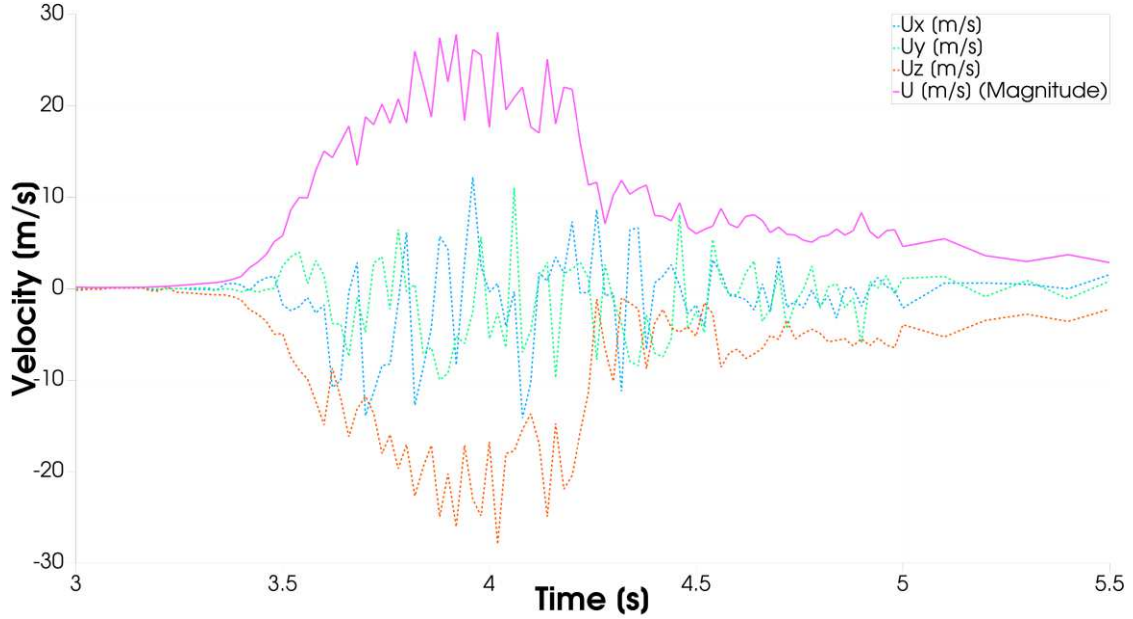


Figure 21. Total velocities between 3.0 – 5.5 seconds in three directions during 1D negative-g acceleration.

In general sloshing simulation results indicate that all solvers calculate the same force values at the excitation direction in case of tank is excited in small amplitudes. In the case of severe excitation, solvers generate different impact force values with the shifted phase angles. All solvers in negative-g sloshing are computed the same mass displacement in vertical direction due to the strong momentum and inertia forces as expected and after 4 seconds propellant mass starts to disconnect from the bottom of tank drain system. However, in the lateral directions there may be important differences in the Force and CoM calculations regardless of the solver types.

5. CONCLUDING REMARKS

The results of the current study show that the choice of flow model for propellant sloshing in a negative-g accelerated cylinder tank may cause a substantial difference in the prediction of forces and CoM's (thus moments) especially in two directions lying on the plane which is perpendicular to the dominant flow direction. For the momentum and inertia-dominated flows, all solvers generate close results in the major excitation direction but, there is a strong need to use higher order schemes in numerical algorithms to capture short-lived strong nonlinear effects.

It is observed as expected that increased internal pressure did not affect the sloshing dynamics as a whole when it is implemented on both phases (liquid and gas) simultaneously. All solvers predict the location of instantaneous mass displacement in vertical direction correctly and thus the total duration of propellant disconnection time from the tank drain system.

Further study is necessary to investigate the nonlinear effects of lateral flow and higher frequency mode contributions.

6. ACKNOWLEDGEMENT

This work is funded by the ROKETSAN (Roket Sanayii ve Ticaret) A.Ş. under the contract numbers 138298/154139.

REFERENCES

- [1] Rakheja S., Yan G., Siddiqui K., Experimental study of liquid slosh dynamics in a partially filled tank, *Trans. ASME, Journal of Fluids Engineering*, Vol. 13, Issue 1, 2009.
- [2] Ranganathan R., Rakheja S., Sankar S., Kinetostatic roll plane analysis of articulated tank vehicles with arbitrary tank geometry, *International Journal of Vehicle Design*, Vol. 10, No. 1, pp. 89-111, Concordia University, Montreal, Canada, 1989.
- [3] Rattayya J. V., Natural frequencies and modes of oscillation of liquids in partially filled axisymmetric ellipsoidal tanks calculated using variational principle, *American Institute Of Aeronautics and Astronautics*, pp. 65-114, 1965.
- [4] Abramson H., Bauer H., Brooks G., and Chu W.H., *The Dynamic Behavior of Liquids in Moving Containers, With Applications to Space Vehicle Technology (NASA-SP-106)*. Technical report, 1966.
- [5] Ibrahim R. A., *Liquid Sloshing Dynamics: Theory and Applications*, Cambridge University Press, Cambridge, UK 2005.
- [6] Dodge, F. T. et. al., *The new dynamic behavior of liquids in moving containers*, Southwest Research Inst. San Antonio, TX, 2000.
- [7] Hasheminejad S. M., Aghabeigi M., Liquid sloshing in half-full horizontal elliptical tanks, *Journal of Sound and Vibration* 324, pp. 332-349, 2009.
- [8] Liu D., Lin P., A numerical study of three-dimensional liquid sloshing in tanks, *Journal of Computational Physics*, Vol. 227, pp. 3921-3939, 2008.
- [9] Ramaswamy B., Kawahara M., Lagrangian finite element analysis to viscous free surface fluid flow, *International Journal for Numerical Methods in Engineering*, vol. 7, pp.953-984, 1987.
- [10] Celebi M.S. and Akyildiz H., Nonlinear Modelling of Liquid Sloshing in a Moving Rectangular Tank, *Journal of Ocean Engineering*, Vol 29, No.12, pp. 1527-1553, June 2002.
- [11] Brizzolara S., Savio L., Viviani M., Chen Y., Temarel P., Couty N., Hoflack S., Diebold L., Moirod N., Iglesias A., Comparison of experimental and numerical sloshing loads in partially filled tanks, *Analysis and Design of Marine Structures*, Taylor & Francis Group, 2009.
- [12] D'Alessandro V., *Modeling of tank vehicle dynamics by fluid sloshing coupled simulation*, doctoral dissertation, Politecnico di Milano, 2011.
- [13] Mi-An Xue, Jinhai Zheng, Pengzhi Lin, Numerical Simulation of Sloshing Phenomena in Cubic Tank with Multiple Baffles, *Journal of Applied Mathematics*, Vol. 2012, 21 pages, 2012.
- [14] Singh G. B, Singhb S., An Experimental and CFD Analysis of Sloshing in a Tanker, *Procedia Technology*, Vol. 14, pp. 490–496, 2014.
- [15] Graham E. W. and Rodriguez A. M., The characteristics of fuel motion which affect airplane dynamics, *ASME J.Appl.Mech.* 19(3), 381–388, 1952.

- [16] Pinson L. D., Longitudinal spring constants for liquid propellant tanks with ellipsoidal tanks, NASA TN D-2220, November 1964.
- [17] Ewart D. G., Fuel oscillations in cylindrical tanks and the forces produced thereby, De Havilland Propellers Ltd, G. W. Dynamics Dept, Tech. Note No 2050, 8 November 1956.
- [18] Bauer H. F., Mechanical model for the description of the liquid motion in a rectangular container, Lockheed Company, RN ER-8559, June 1960.
- [19] Hutton R. E., An Investigation of Resonant, Nonlinear, Non-planar Free-Surface Oscillations of a Fluid, NASA TN D-1870, 1963.
- [20] Abramson H. N., Amazing Motions of Liquid Propellants, *Astronautics*, 6, pp. 35-37, 1961.
- [21] Konopka M., De Rose F., Strauch H., Jetzschmann C., Darkow N., and Gerstmann J., Active slosh control and damping-Simulation and experiment, *Acta Astronautica*, Vol. 158, 2019, pp. 89–102.
- [22] Ubbink O., Numerical prediction of two fluid systems with sharp interfaces. *PhD Thesis*. Imperial College of Science, UK, 1997.
- [23] Hirt C.W. and Nichols B.D., Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. *Journal of Computational Physics*, 39 DOI: 10.1016/0021-9991(81)90145-5, pp. 201–225, 1981.
- [24] Jasak, H., Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows. *PhD Thesis*. Imperial College, London, UK, 1996.
- [25] Issa R. I., Solution of the implicitly discretized fluid flow equations by operator splitting *J. Comput. Phys.* 62 40–65, 1986.
- [26] Pozrikidis C., *Introduction to Theoretical and Computational Fluid Dynamics* (Oxford: Oxford University Press). 1997.
- [27] Brackbill J.U., Kothe D.B. and Zemach C., A continuum method for modeling surface tension, *J. Comput. Phys.*, 100 335–54, 1992.
- [28] Stephens D. G., Leonard H. W., and Perry Jr T. W., Investigation of the Damping of Liquids in Right-Circular Cylindrical Tanks, Including the Effects of a Time-Variant Liquid Depth, 1962.
- [29] Miles J. W., On the sloshing of liquid in a cylindrical tank, Tech. Report, Thompson Ramo Wooldridge Inc. Los Angeles CA, 1956.
- [30] Case K. and Parkinson W., Damping of surface waves in an incompressible liquid, *Journal of Fluid Mechanics*, Vol. 2, No. 2, 1957, pp. 172–184.
- [31] Mikishev G., An experimental investigation of free oscillations of a liquid in containers, *News of the Academy of Sciences of USSR, The Branch of Technical Sciences, Mechanics and Machinery* (Izvestiya Akademii Nauk SSSR, Otdelenie Tekhnicheskikh Nauk Mekhanika: Mashinostroenie), Vol. 4, 1961, pp. 48–53.
- [32] W.W. Kim and S. Menon. A new dynamic one-equation subgrid-scale model for large eddy simulations. In 33rd Aerospace Sciences Meeting and Exhibit, Reno, NV, USA, 1995.

[33] F.R. Menter, M. Kuntz, and R. Langtry. Ten years of industrial experience with the SST turbulence model. In Proceedings of the fourth international symposium on turbulence, heat and mass transfer, pages 625–632, Antalya, Turkey, 2003.

[34] Kırılancı O.U. and Celebi M.S., A comparative study of sub-grid scale (SGS) models in the large eddy simulation (LES) of liquid sloshing, Journal of Physics, Vol. 318(4), December 2011.