

Modeling Self-Pressurization and Spray Bar Pressure Control of a Cryogenic Storage Tank in Normal Gravity

Steven Soriano*

HX5 and NASA Glenn Research Center, Cleveland, OH 44135, USA

Olga Kartuzova[†] and Mohammad Kassemi[‡]

National Center for Space Exploration Research (NC SER), Case Western Reserve University and NASA Glenn Research Center, Cleveland, OH 44135, USA

Daniel Hauser[§]

NASA Glenn Research Center, Cleveland, OH 44135, USA

This paper presents Computational Fluid Dynamics (CFD) models for simulating self-pressurization and spray bar cooling processes in a large-scale liquid hydrogen storage tank under normal gravity. The self-pressurization model utilizes the kinetics-based Schrage equation with the Volume-Of-Fluid (VOF) model to account for the evaporative and condensing interfacial mass fluxes. It uses a laminar approach to model natural convection and compares the pressures and temperatures predicted by the CFD model with experimental data from the Multipurpose Hydrogen Test Bed (MHTB) self-pressurization experiment. Additionally, the predicted interfacial mass flux is presented. A CFD model for simulating pressure control using a spray bar is also presented. An Eulerian-Lagrangian approach models the interactions between the discrete droplets and the continuous ullage phase. The spray model is coupled with the VOF model by tracking particles in the ullage, removing particles from the ullage when they reach the interface, and then adding their contributions to the liquid. The T-sat model is used to calculate the droplet-ullage heat and mass transfer. In this model, droplets warm up to the saturation temperature corresponding to the ullage vapor pressure thereafter evaporating while remaining at the saturation temperature. The tank pressure, vapor, and liquid temperature evolutions are validated against data from the MHTB spray bar mixing experiment.

I. Nomenclature

A	=	Area density	p, P	=	Pressure
E	=	Energy	$\mathbf{q}$	=	Heat flux
$\mathbf{g}$	=	Gravity	$\dot{Q}$	=	Heat power
h	=	Surface curvature	R	=	Gas constant
k	=	Turbulent kinetic energy	T	=	Temperature
L	=	Latent heat	t	=	Time
M	=	Molar mass of fluid	$\mathbf{v}$	=	Velocity
$\mathbf{n}$	=	Normal vector	c_p	=	Heat capacity at constant pressure
			m	=	Mass

*Mechanical Engineer, Fluid and Cryogenic Systems Branch.

[†]Research Assistant Professor, Department of Mechanical & Aerospace Engineering, and AIAA Member.

[‡]Research Professor, Department of Mechanical & Aerospace Engineering, AIAA Associate Fellow.

[§]Fluids Engineer, Fluid and Cryogenic Systems Branch.

<u>Greek</u>			<u>Subscripts</u>		
α	=	Cell value of volume fraction	i	=	Interface or phase
β	=	Slope limiter	il	=	Liquid side of the interface
ϕ	=	Face value of volume fraction	iv	=	Vapor side of the interface
μ	=	Dynamic viscosity	sat	=	Saturation conditions
ρ	=	Density	l	=	Liquid
τ	=	Stress tensor	v	=	Vapor
ω	=	Specific turbulence dissipation rate	p	=	Particle

II. Introduction

This study aims to describe the development of a comprehensive two-phase CFD model for simulating the self-pressurization and pressure control of the MHTB tank at NASA Marshall Space Flight Center (MSFC) and its validation by MHTB ground-based testing results. The MHTB tank is 18-m³ (693 ft³) in volume and is representative of the size and shape of a liquid hydrogen (LH2) propellant tank in a fully integrated space transportation vehicle. The tank was designed to accommodate various components associated with different CFM concepts, including a spray-bar thermodynamic vent system (TVS) [1] for active pressure control and a Multilayer Insulation (MLI) system for passive thermal isolation [2].

In the two-phase CFD model of the MHTB test tank presented in this paper, the Volume-of-Fluid (VOF) model is used to capture the shape and evolution of the interface. A kinetic-based mass transfer submodel is employed to evaluate evaporative-condensing interfacial mass transfer. An Eulerian-Lagrangian coupled approach tracks the spray droplets and captures the interaction between droplets and the continuous vapor phase. Coupling the droplet and VOF models makes it possible to perform particle tracking in the vapor, removing particles from the vapor domain when they reach the interface. An in-house droplet-ullage heat and mass transfer model, the T-sat model, is used to account for changes in tank pressure during the spraying cycle. Kartuzova and Kassemi have previously used this model to simulate the MHTB spray bar pressure control and have shown good results [3]. The kinetic-based Schrage equation calculates the evaporative/condensing mass transfer to/from the droplet [4]. Droplet temperature and size changes are calculated and tracked during the simulations. Appropriate source terms are included in the governing equations to account for droplet-ullage mass, momentum, and energy transfer due to phase change.

Kartuzova and Kassemi previously conducted self-pressurization and spray bar simulations of the MHTB tank at 50% and 90% fill levels. Although a 25% fill level was tested experimentally, it was not validated against a CFD model. This work extends the validation of the CFD model by including the 25% fill level case for the MHTB tank.

This report compares the predicted ullage pressure and temperature evolutions against their experimental MHTB counterparts for model validation during self-pressurization. This work contributes results from an additional fill level from Kartuzova and Kassemi [3]. Simulation results for the spray-bar mixing process account for droplet and ullage heat and mass transfer to underscore the important contribution of droplet vaporization/condensation to the evolution of tank pressure during the pressure control interval. Additionally, pressure and temperature evolutions in the tank during the spray-bar mixing process are compared between CFD and experimental values.

A. Governing Equations

Fluid flow and heat transfer in the tank are described in terms of the continuity, Navier-Stokes and energy equations for both phases:

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

$$\frac{\partial}{\partial t}(\rho \mathbf{v}) + \nabla(\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \left[\mu_{eff} \left(\nabla \mathbf{v} + \nabla \mathbf{v}^T \right) \right] + \rho \mathbf{g} + \mathbf{F}_{vol}, \quad (2)$$

$$\frac{\partial}{\partial t}(\rho E) + \nabla(\mathbf{v}(\rho E + p)) = \nabla(k_{eff} \nabla T) + S_h. \quad (3)$$

The present study treats the liquid phase as incompressible with variable temperature-dependent properties, except for density. The liquid density is allowed to vary linearly with temperature in the body force term of the momentum

equation according to the Boussinesq approximation. The vapor is modeled as a compressible ideal gas. In this study, the movement of the interface is captured diffusely using the Volume of Fluid (VOF) method, as published by Hirt and Nichols [5]. Interfacial energy, momentum, and mass balances are applied using source terms in the diffuse interfacial region.

B. VOF Model

In the VOF method, a volume fraction is defined in each cell such that the volume fractions of all the phases sum to unity. In the cell, the change in the interface can be tracked by solving a continuity equation for the volume fraction of the q^{th} phase:

$$\frac{1}{\rho_q} \left[\frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \mathbf{v}_q) \right] = S_{\alpha_q}, \quad (4)$$

where the volume fraction for the primary phase is determined from:

$$\sum_{q=1}^n \alpha_q = 1. \quad (5)$$

In the VOF method, the field variables and properties are defined in terms of the volume fraction, which for a general system can be written as:

$$\rho = \sum_{q=1}^n \alpha_q \rho_q, \quad \mu_{eff} = \sum_{q=1}^n \alpha_q \mu_{eff_q}, \quad k_{eff} = \sum_{q=1}^n \alpha_q k_{eff_q}. \quad (6)$$

In this fashion, the continuity, momentum, and energy equations, as described by Eq. (1)-(3), can be solved throughout the domain for the temperatures and velocities in the two phases. In the VOF model, energy (E) and temperature (T) are treated as mass-averaged variables:

$$E = \frac{\sum_{q=1}^n \alpha_q \rho_q E_q}{\sum_{q=1}^n \alpha_q \rho_q}, \quad (7)$$

where E_q is based on the specific heat of the q^{th} phase and the shared temperature.

Evaporation and condensation at the interface are modeled as a source term in the continuity equation for the volume fraction (Eq. 4), i.e.:

$$S_{\alpha_q} = \dot{\mathbf{m}}_i \cdot \mathbf{A}_i, \quad (8)$$

where $\mathbf{A}_i$ is an interfacial area density vector, and $\dot{\mathbf{m}}_i$ is a mass flux vector, which for near equilibrium conditions can be determined based on the Schrage equation [4]:

$$|\dot{\mathbf{m}}| = \left(\frac{2\sigma}{2 - \sigma} \right) \left(\frac{M}{2\pi R} \right)^{1/2} \left(\frac{P_i}{T_i^{1/2}} - \frac{P_v}{T_v^{1/2}} \right). \quad (9)$$

Here σ is the accommodation coefficient (a value of 0.001 was used); M is the molar mass of the fluid; R is the universal gas constant; P_i and P_v are, respectively, the interfacial and vapor pressures (it was assumed that $P_i \cong P_{sat}$); T_i and T_v are, respectively, the interfacial and vapor temperatures (it was assumed that $T_i = T_v \cong T_{sat}$ at the interface). Finally, $\mathbf{A}_i$ is defined as:

$$\mathbf{A}_i = |\nabla \alpha|, \quad (10)$$

where α is the volume fraction of the primary phase.

In the present implementation, the surface tension forces at the interface are modeled via the Continuum Surface Force (CSF) model of Brackbill et al. [6]. In this model, the surface tension forces at the interface are transformed into a volume force ($\mathbf{F}_{vol}$), which is added as a source to the momentum equation:

$$\mathbf{F}_{vol} = \sum_{\text{pairs } i,j, i < j} \sigma_{ij} \frac{\alpha_i \rho_i h_i \nabla \alpha_j + \alpha_j \rho_j h_j \nabla \alpha_i}{\frac{1}{2} (\rho_i + \rho_j)}, \quad (11)$$

where h_i is the surface curvature calculated from the local gradients in the surface normal at the interface:

$$h_i = \nabla \cdot \hat{\mathbf{n}}. \quad (12)$$

C. Lagrangian Spray Model

A customized Lagrangian Spray model of the ANSYS Fluent CFD code [7] was utilized to simulate the MHTB tank's cooling by spraying cold liquid in a vapor region. This model uses the Eulerian-Lagrangian approach, where the fluid phase (ullage) is treated as a continuum by solving the Navier-Stokes equations. The dispersed phase is solved by tracking a large number of particles (spray droplets) through the calculated flow field. The droplets exchange mass, momentum, and energy with the fluid phase. In the Lagrangian spray model, the droplet trajectory is calculated by integrating the force balance on the droplet. This force balance equates the droplet inertia with the forces acting on the droplet as:

$$\frac{d\vec{u}_p}{dt} = F_D (\vec{u} - \vec{u}_p) + \frac{\vec{g} (\rho_p - \rho)}{\rho_p} + \vec{F}, \quad (13)$$

where $\vec{u}$ is the fluid phase velocity, $\vec{u}_p$ is the particle velocity, ρ is the fluid density, ρ_p is the density of the droplet, F_D is the drag force per unit droplet mass, and $\vec{F}$ is an additional acceleration. Integration of time in Eq. 13 yields the velocity of the droplet at each point along the droplet trajectory. The spherical drag law proposed by Morsi and Alexander [8] is applied for the droplet drag force calculation. The droplet energy equation is solved in the user subroutine as:

$$m_p c_{p,p} \frac{dT_p}{dt} = h \cdot A_p (T_\infty - T_p) - L \dot{m}_p, \quad (14)$$

where m_p is the droplet mass, $c_{p,p}$ is the heat capacity of the droplet, A_p is the surface area of the droplet, T_∞ is the local temperature of the continuous phase, h is the convective heat transfer coefficient, L is the latent heat, and $\dot{m}_p$ is the rate of evaporation/condensation on the droplet, obtained from the Schrage relation (Eq. 9) in kg/s.

The droplet mass transfer is calculated using the developed T-sat model, where the mass transfer rate is calculated from the droplet energy balance (Eq. 14). In the T-sat model, the droplet is allowed to warm up to the saturation temperature corresponding to the vapor pressure, after which it uses all of the energy supplied by the ullage for evaporation while it stays at the saturation temperature, such that:

$$\begin{aligned} \text{if } T_p < T_{\text{sat}}(P_v) &\Rightarrow \dot{m}_p = 0; T_p \text{ is calculated from Eq. 14,} \\ \text{if } T_p \geq T_{\text{sat}}(P_v) &\Rightarrow T_p = T_{\text{sat}}(P_v); \dot{m}_p \text{ is calculated from Eq. 14.} \end{aligned} \quad (15)$$

In this case, the evaporation rate can be evaluated directly from the droplet energy balance (Eq. 14).

The correlation of Ranz and Marshall [9][10] is used to evaluate the droplet heat transfer coefficient:

$$Nu = \frac{h d_p}{k_\infty} = 2.0 + 0.6 Re_d^{1/2} Pr^{1/3}, \quad (16)$$

where d_p is the droplet diameter, k_∞ is the thermal conductivity of the continuous phase, Re_d is the Reynolds number based on the particle diameter and the relative velocity, and Pr is the Prandtl number of the continuous phase.

The relative Reynolds number is calculated as:

$$Re_d \equiv \frac{\rho d_p |\vec{u}_p - \vec{u}|}{\mu}, \quad (17)$$

where μ is the molecular viscosity of the fluid.

After the droplet energy equation is solved in the user subroutine, each droplet's mass, diameter, and temperature are updated. The mass, momentum, and energy transfer between the droplets and the continuous phase are modeled via source terms added to Eq. 1-3.

The Lagrangian Spray model is coupled with the VOF model via a user subroutine, which performs particle tracking in the ullage, removes particles from the ullage when they reach the interface, and then adds their contributions to the liquid mass, momentum, and energy equations through source terms. These source terms are added to those liquid cells near the location where the spray drops crossed the liquid-vapor interface. Because the VOF method produces a diffuse

interface, the criteria for determining when the spray drops have crossed into the bulk liquid is defined as the liquid drop entering a fluid cell with a liquid volume fraction > 0.1 .

III. Numerical Implementation

The study presented is based on the MHTB tank self-pressurization and spray bar mixing experiments conducted in normal gravity [2]. The MHTB tank consists of a cylindrical-mid section with 3.05 m diameter and 3.05 m height and two 2:1 elliptical end caps.

The interfacial mass transfer formulation used with the VOF scheme, droplet/ullage heat, and mass transfer and the coupling between the VOF and the Lagrangian Spray models has been developed, coded, and implemented into a customized in-house version of the ANSYS Fluent version 24.1 CFD code. In the VOF model, the interfacial mass transfer is calculated in the user subroutine. Sources are applied to the momentum and relevant scalar equations based on the assumption that mass "created" or "destroyed" will have the same momentum and energy of the phase from which it was "created" or "destroyed." All two-phase calculations are performed in the user subroutine, which is called once per outer iteration before any of the field equations are solved.

The computational domain for the 25% fill ratio self-pressurization case was discretized using an unstructured mesh of 12,609 cells, as depicted in Fig. 1. The computation domain used for the spray-bar mixing simulation was discretized using a tetrahedral mesh of 256,729 cells. In both the ullage and liquid regions, the conservation equations are evolved in time using a bounded second-order time-stepping routine with a time step size on the order of 5×10^{-2} seconds for self-pressurization cases and 1×10^{-3} seconds for spray cases. The second-order monotone upwinding scheme discretizes the convective fluxes in the momentum, energy, and turbulence equations. The PISO method is used for the pressure-velocity coupling.

For the volume fraction equation, the Compressive discretization scheme is used. The Compressive scheme is a second-order reconstruction scheme for the VOF equation based on the slope limiter [7]. In this scheme:

$$\phi_f = \phi_d + \beta \nabla \alpha_d, \quad (18)$$

where ϕ_f is the face value of volume fraction, ϕ_d is the donor cell value of volume fraction, β is the slope limiter, and $\nabla \alpha_d$ is the donor cell volume fraction gradient.

In the Lagrangian spray cases, cold liquid enters the liquid phase as a "tiny jet," modeled as a source term defined in a user subroutine and added to Fluent's mass, energy, and momentum equations. In the 25% fill ratio case, there are 9 liquid jets in the liquid region and 33 spray injections in the vapor region. The spray-bar hole diameter is 1.7e-03 m with a flow-through length of 0.7e-03m. The spray injection type is a plain orifice atomizer with four-particle streams per injection.

Two-way coupling between the droplets and continuous phase is enabled. To model heat and mass transfer between the droplets and ullage, first, a value for the mass transfer rate is calculated in the subroutine. Then, the droplet energy equation is solved, and the droplet temperature is updated. The mass and diameter of the droplet are updated to account for the amount of evaporated or condensed mass. Then, corresponding mass and energy sources are added to the ullage.

Convergence criteria are set to 1×10^{-5} for all equations except the energy equation, which is set to 1×10^{-8} .

A. Self-Pressurization

The self-pressurization study was conducted using a 2D axisymmetric model. The gravitational vector is aligned with the tank's central axis in the negative axial direction. A uniform heat flux of 0.5266 W/m^2 was applied to the tank wall to match the experiment's average wall heat flux. These values are based on previous thermal analysis and multinode simulations. Conduction through the tank wall was not considered. The spray bar assembly approximated as lying along the tank centerline, is treated as an adiabatic surface. A zero velocity field is used as an initial condition for the self-pressurization simulation. The initial tank temperature for self-pressurization is set to the experimental tank temperature profile. The details of the self-pressurization case are presented in Table 1.

Table 1 Experimental conditions for tank self-pressurization test.

Case #	Initial Liquid Fill, %	Initial Pressure, kPa	Final Pressure, kPa	Test Duration, hours
1	25	122.1	135.1	18.5

B. Spray-Bar Mixing

The temperature, velocity, and phase fields at the end of self-pressurization are transferred from 2D to 3D geometry and used as initial conditions for the spray-bar mixing case. The 3D spray-bar mixing case is a 90° sector of the complete tank with two symmetry planes. Nine submerged spray-bar holes are modeled as point sources in the liquid region, and 33 spray injections are modeled using the discrete phase model (DPM) in the vapor region. Both liquid jets and spray injections have the same uniform temperature of 21.11 K and a liquid flow rate of 5.7×10^{-4} kg/s. This temperature is the average liquid temperature at the end of self-pressurization. The spray injection type is a plain orifice atomizer with four particle streams per injection. Table 2 highlights the details of the spray-bar mixing experiment.

Table 2 Experimental conditions for spray-bar mixing test.

Case #	Initial Liquid Fill, %	Initial Pressure, kPa	Final Pressure, kPa	Test Duration, s
1	25	135.0	131.0	35

C. Computational Mesh

An initial mesh of 15,690 cells was generated for modeling self-pressurization. As shown in Figure 1, the mesh is refined near the wall and in the interfacial area. A mesh convergence study, including a finer and coarser mesh, will be conducted to compare results and will be included in the final paper.

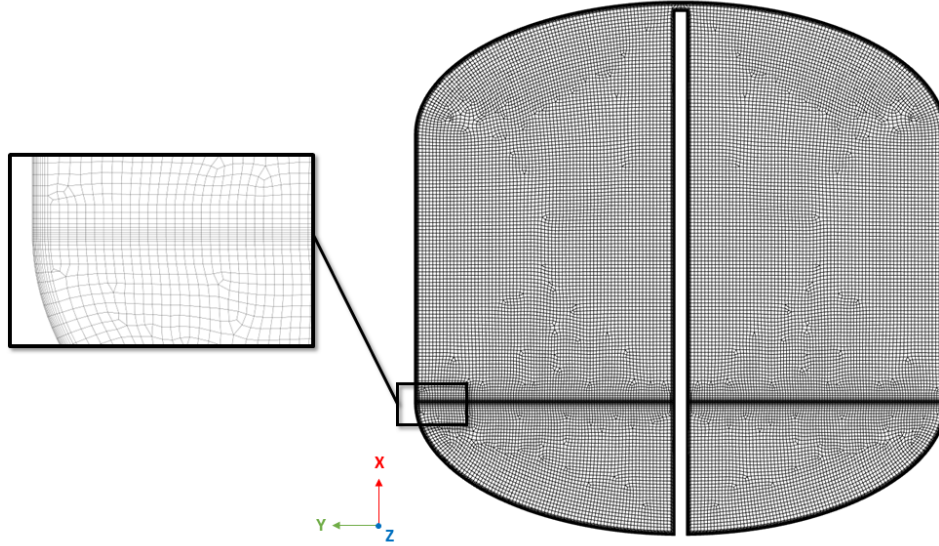


Fig. 1 Computational mesh for the 25% fill ratio self-pressurization case (15,690 cells).

The computational mesh used for the spray-bar mixing simulation is shown in Figure 2. Tetrahedral cells are used to discretize the domain, with 256,729 cells. There is mesh refinement near the tank's walls, spray-bar wall, and interfacial area.



Fig. 2 Computational mesh for the 25% fill ratio spray-bar mixing case (256,729 cells).

IV. Results & Discussion

A. Self-Pressurization

The simulations presented in this paper is based on the MHTB tank self-pressurization and spray bar mixing experiments conducted in normal gravity [1]. The experiment was conducted using three different tank liquid fill ratios (25%, 50%, and 90%). The CFD results for the 50% and 90% tank fill ratios were presented in previous publications [11] [12].

A laminar model is initially used to model the self-pressurization of the MHTB tank with 25% fill ratio based on favorable previous self-pressurization results for 50% and 90% fill ratios [11]. The tank pressure and interfacial mass transfer rate evolutions are presented in Figure 3 and Figure 4, respectively. The experimental tank pressure is plotted with the calculated CFD pressure in Figure 3. However, there is no experimental interfacial mass transfer available for comparison. The tank pressure estimated by the laminar VOF model closely matches the experimental data. The laminar VOF model initially predicts evaporation at the interface for the first 1,500 seconds and again between 40,000 and 50,000 seconds during the self-pressurization period, with condensation occurring at other times.

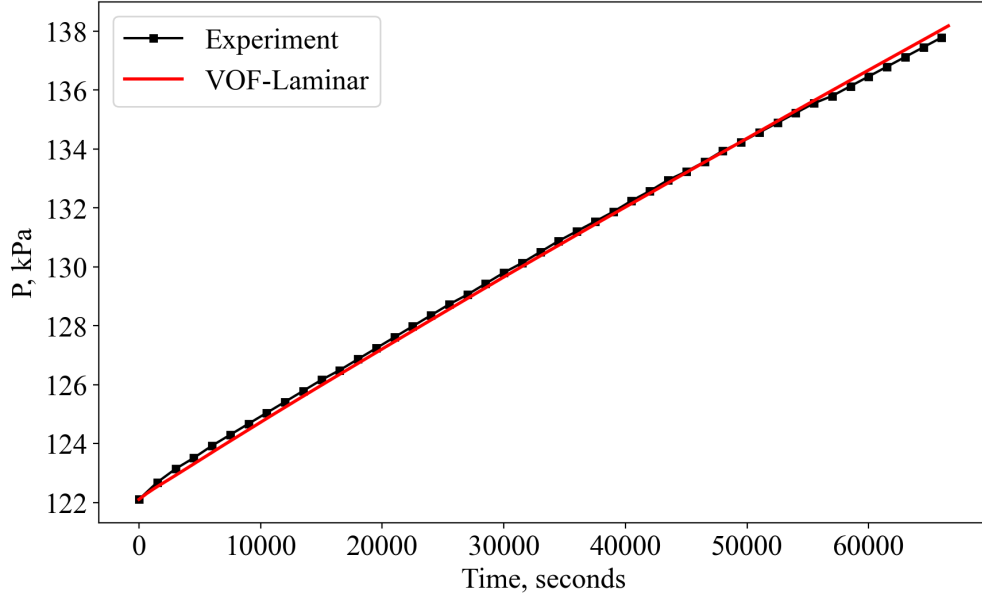


Fig. 3 Tank pressure evolutions during self-pressurization of 25% fill ratio MHTB Tank.

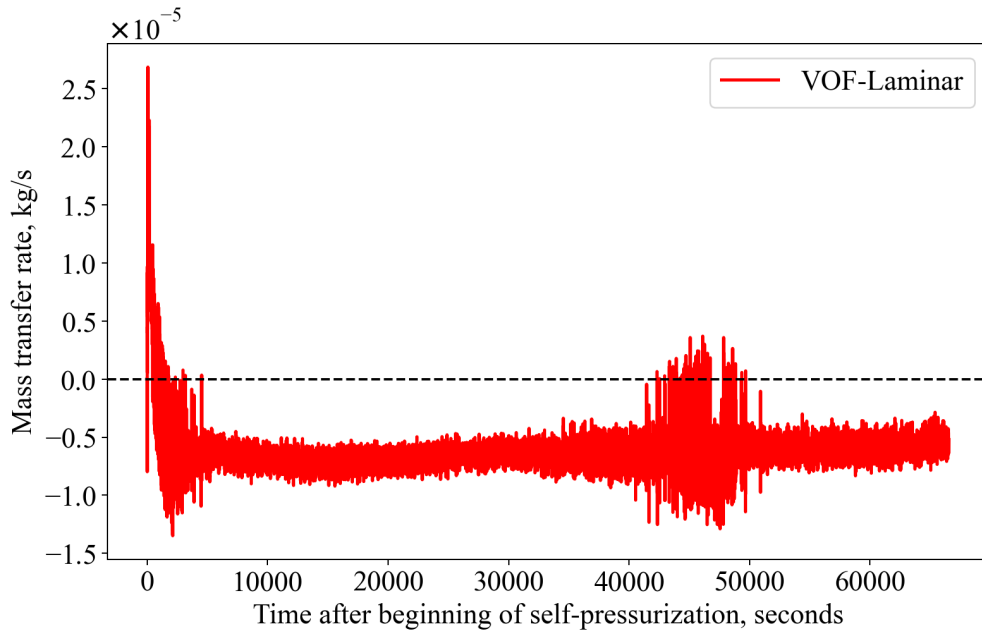


Fig. 4 Interfacial mass transfer rate during self-pressurization of 25% fill ratio MHTB Tank.

Temperatures along the vertical rake at 0.74168 m from the tank centerline are compared at the end of self-pressurization between CFD results and experimental data in Figure 5. Although the laminar VOF model over-predicts the vapor temperatures compared to the experimental values, it agrees with the slope of the temperature distribution from the experiment and the experimental temperatures near the interface.

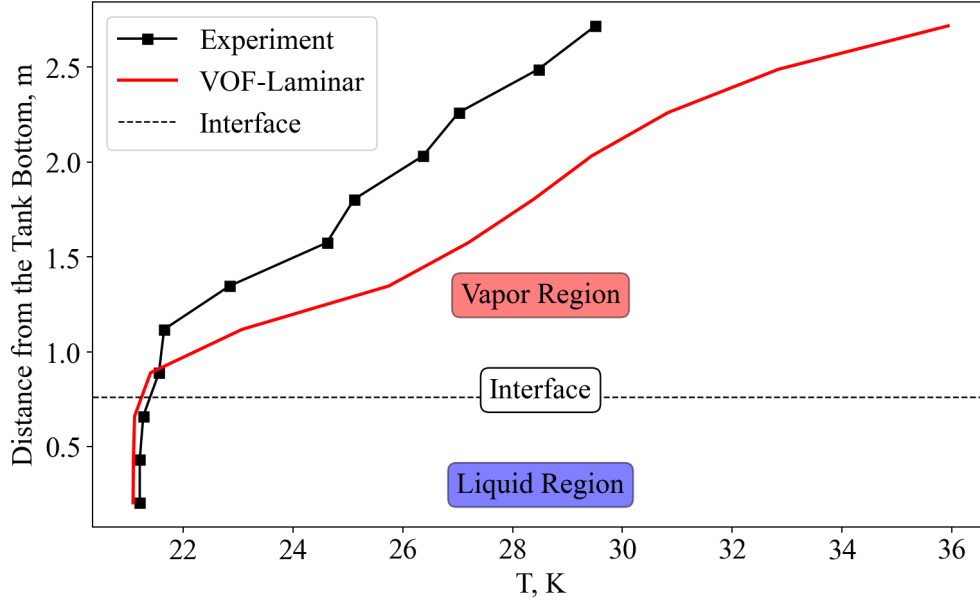


Fig. 5 Temperatures along the vertical rake (0.74 m off the centerline) at the end of tank self-pressurization.

The final paper will also include results with a turbulent VOF model for self-pressurization to compare with the laminar VOF.

B. Spray-Bar Mixing

The results from the end of self-pressurization were revolved around the tank's central axis into 100 azimuthal slices to create a 3D 90° sector and interpolated onto a 3D 90° sector mesh. The mesh is refined near the interface, tank, and spray-bar walls. Previous simulations have investigated the comparison between calculating the droplet mass transfer using a kinetics-based Schrage equation or a T-sat model that assumes that the droplet remains at saturation temperature and computes the mass transfer from the droplet using an energy balance [3]. The T-sat model is used for this simulation as this model predicted the pressure evolutions better than the Schrage equation based on previous results and does not need an accommodation coefficient [3]. Only the first spray on/off cycle, after self-pressurization in the test segment, is simulated and reported. In this cycle, liquid entering the Joule Thomson device is bypassed. Consequently, the liquid temperature exiting the spray bar is the same as the liquid temperature entering the pump near the bottom of the tank.

Figure 6 illustrates the tank pressure evolution during the spray injection, alongside the corresponding experimental values, using a laminar VOF model akin to previous self-pressurization simulations. The figure demonstrates that the CFD model accurately predicts the changes in tank pressure relative to the experimental data. Heat and mass exchange with the droplet influence the ullage pressure during the spray injection.

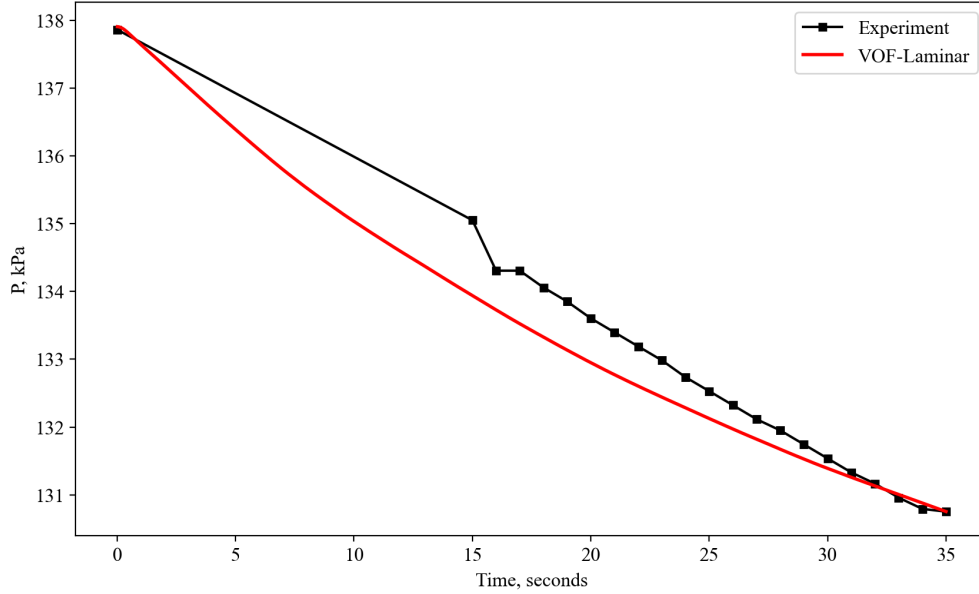


Fig. 6 Tank pressure evolution during spray cooling of 25% fill ratio MHTB Tank.

Temperature sensors measured the temperature at various locations in the vapor and liquid regions during the experiment. The temperature evolutions at three different measurement locations in the vapor region and one in the liquid region are compared against the experimental data in Figures 7 - 10. Since the CFD temperatures in the vapor region at the end of self-pressurization over-predicted the experimental temperatures (Figure 5), a normalized temperature plot was presented in Figures 7 - 10 where both the experiment and CFD temperatures were normalized against their respective initial temperatures ($T - T_0$). The result is a plot where the CFD and experimental $T - T_0$ values start at 0 and decrease through the spray period. This highlights a comparison of the temperature behavior between the CFD and experimental results.

Figure 7 shows the temperature evolutions at the top of the vapor region (TD1). At this location, the CFD model captures the slope of the temperature decline in the experiment with close fidelity. More oscillations in the vapor temperatures are clear in the CFD simulations than in the experiment. The sampling of temperatures during the experiment is sparse and may not capture more frequent temperature oscillations during the spray; however, the final temperature difference between CFD and the experiment is almost equal.

Figure 8 shows the temperature evolutions in the middle of the vapor region (TD5). The CFD model predicts a lower temperature difference than experimental values at this location toward the end of the spray cycle; however, it maintains an adequate temperature decay during the first 22 seconds of the spray cycle.

A comparison of temperature evolutions at the bottom of the vapor region near the interface (TD8) and measured during the experiment is shown in Figure 9. Despite the initial sparse sampling of the experimental values, it is evident that the CFD model underpredicted the temperature change compared to the experiment in this region.

Lastly, the temperature evolutions in the liquid region (TD11) are compared with the experiment in Figure 10. The CFD and experimental results show little change in liquid temperatures during the spray cycle.

Further discussion on this topic will be included in the final paper.

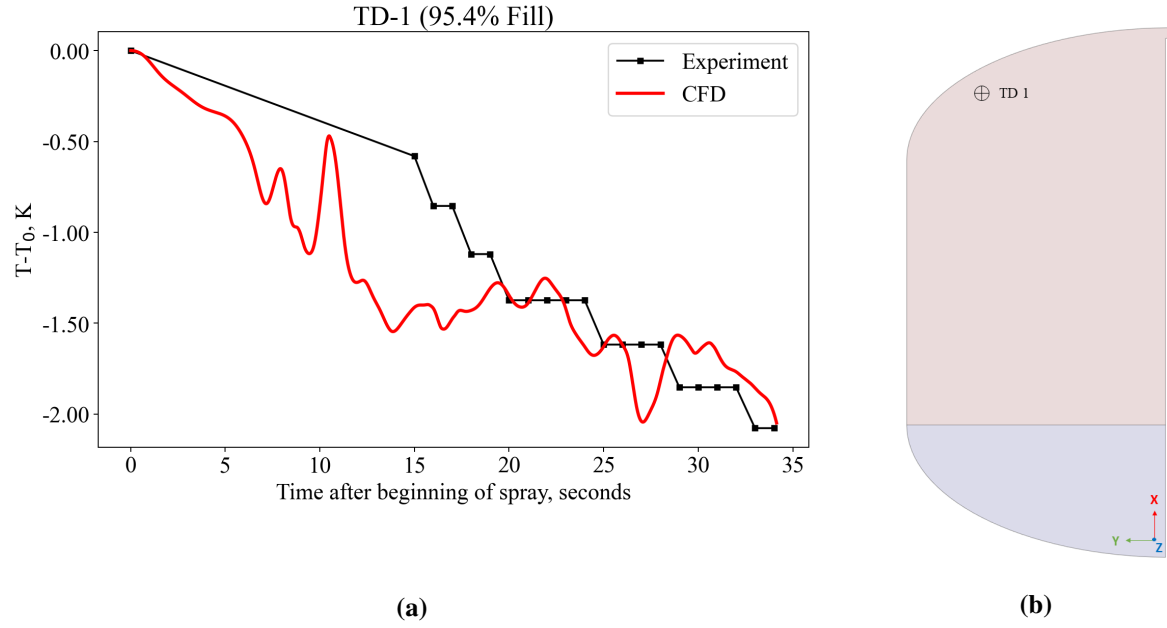


Fig. 7 (a) Normalized temperature evolutions for TD1 during the first spray cycle; (b) Schematic of the tank with TD1 location shown.

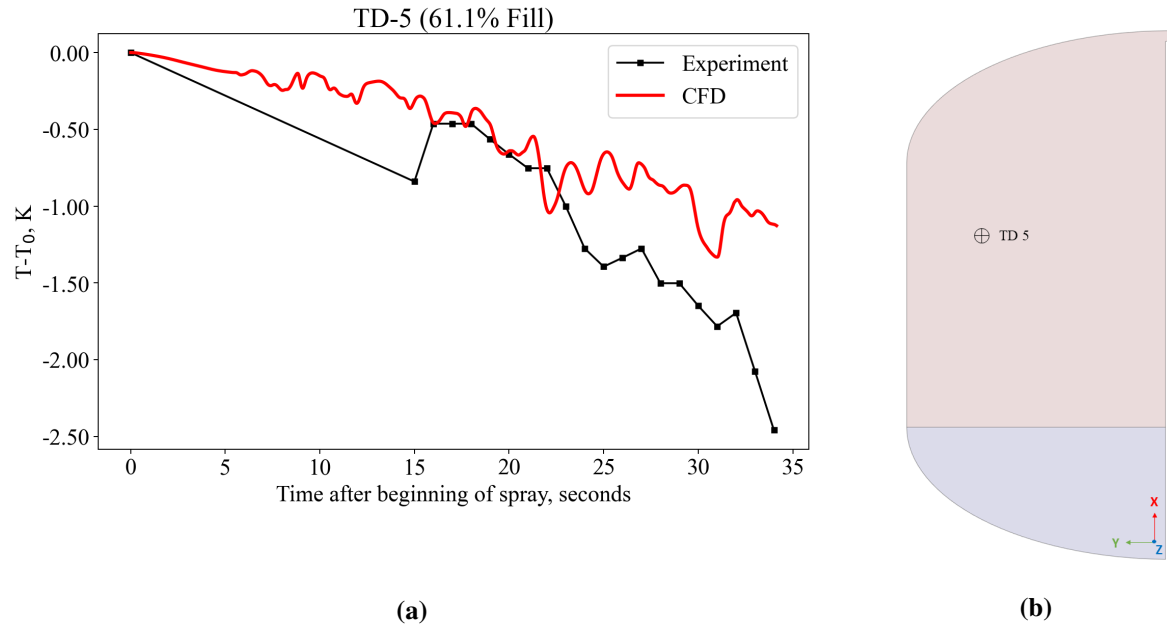


Fig. 8 (a) Normalized temperature evolutions for TD5 during the first spray cycle; (b) Schematic of the tank with TD5 location shown.

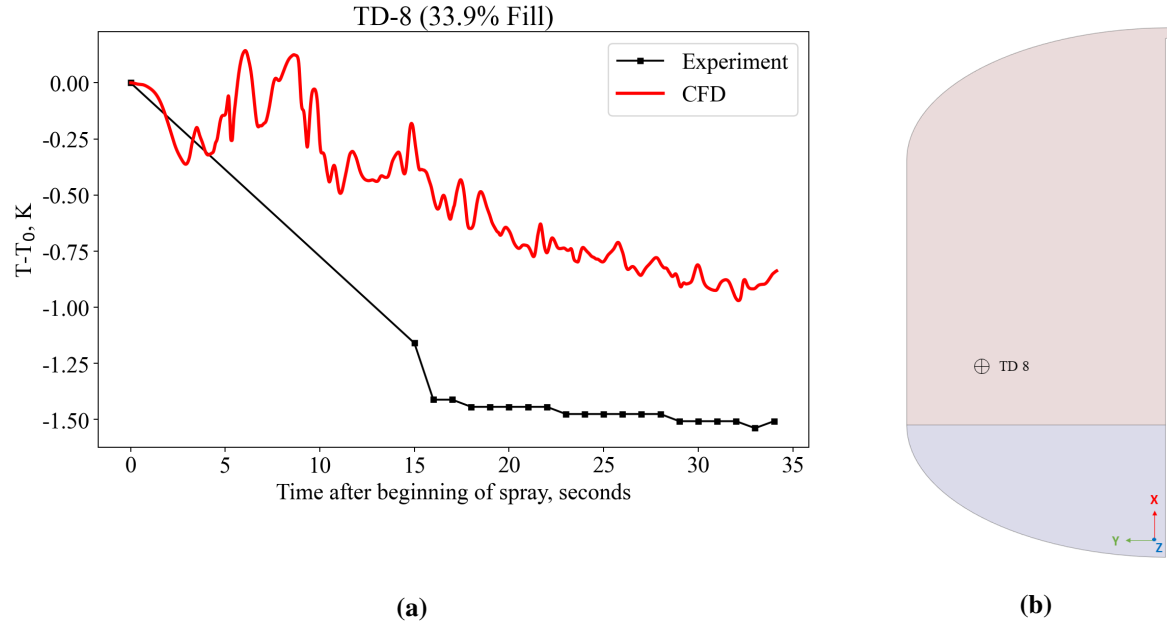


Fig. 9 (a) Normalized temperature evolutions for TD8 during the first spray cycle; (b) Schematic of the tank with TD8 location shown.

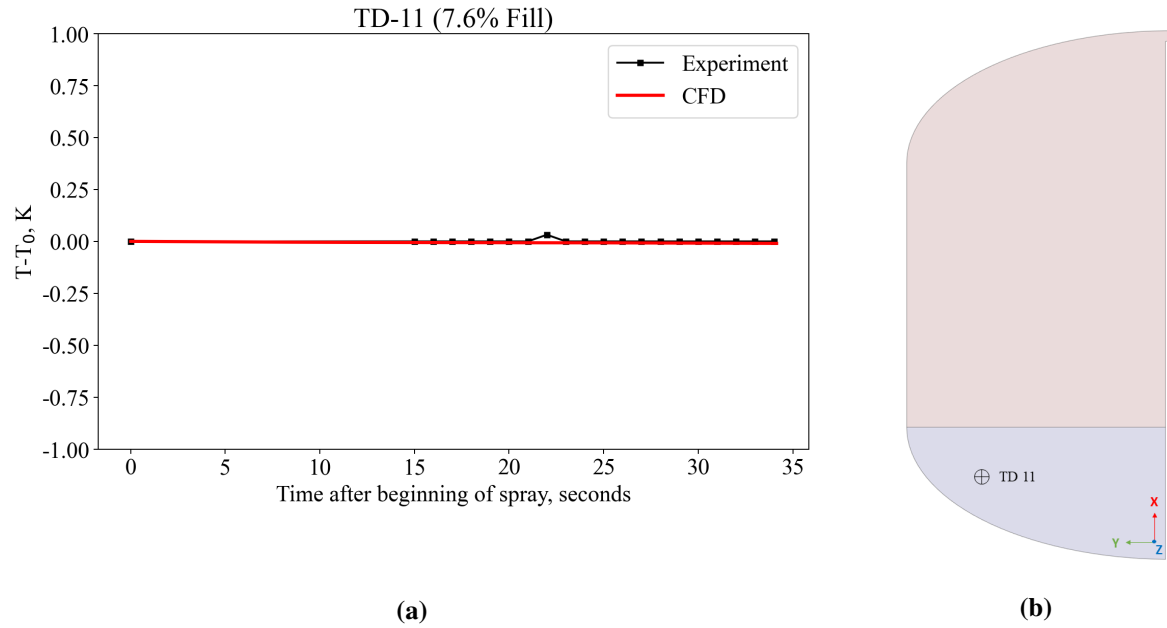


Fig. 10 (a) Normalized temperature evolutions for TD11 during the first spray cycle; (b) Schematic of the tank with TD11 location shown.

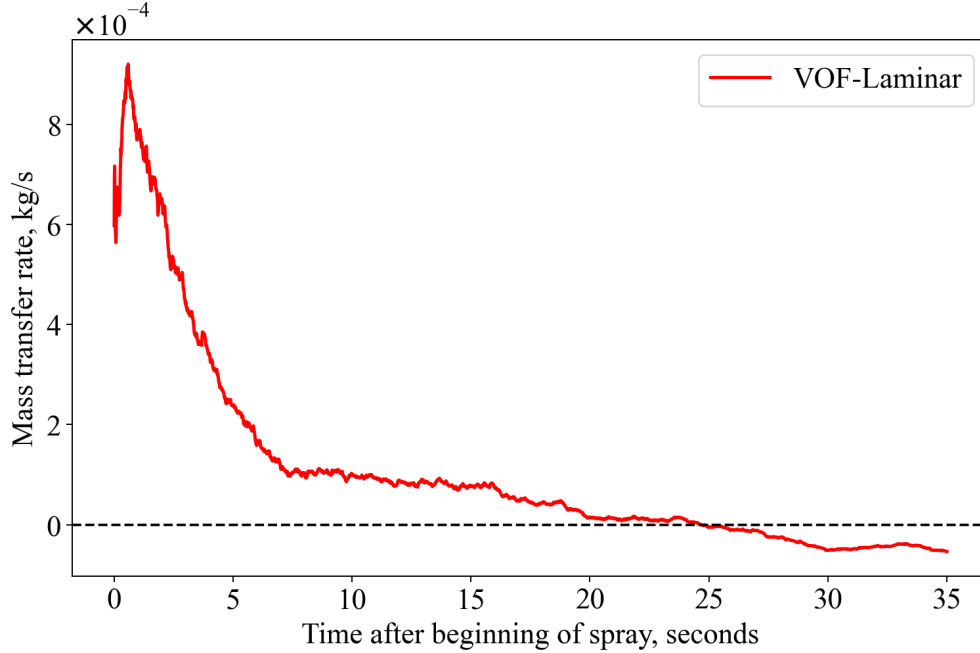


Fig. 11 Interfacial mass transfer rate, during the first spray cycle.

Figure 11 depicts the evolution of the mass transfer rate during the spray process. Although there is no experimental data for comparison, the model illustrates significant evaporation at the interface in the early stages of the spray. This is a response of the interface to the vapor cooling and pressure changes in the vapor. Towards the end of the spray cycle, the model shows increased condensation, suggesting that the vapor temperatures have reached saturation levels and are condensing back into liquid.

V. Conclusion

Preliminary CFD results for self-pressurization and spray-bar mixing of a cryogenic storage tank in normal gravity are presented. To evaluate the fidelity of the CFD models, predicted tank pressure and fluid temperatures are compared with data provided by the MHTB experiments during self-pressurization and spray-bar mixing.

The model predicts tank pressure very well during self-pressurization. However, using the laminar model, the predicted vapor temperatures at the end of self-pressurization are slightly overestimated. The use of a turbulence model during self-pressurization is currently being investigated, and the results will be presented in the final paper.

Compared with the experiment, the model predicts the temperature and pressure behavior exceptionally well during the spray-bar mixing process. Improving the temperature results at the end of self-pressurization will only improve this model's results. Further analysis on both topics, self-pressurization, and spray-bar mixing, will be expanded in the final paper.

Acknowledgments

The authors thank the Modeling Portfolio Project within the STMD CFM Project Office for funding this work.

References

- [1] Hastings, L., Flachbart, R., Martin, J., Hedayat, A., Fazah, M., Lak, T., Nguyen, H., and Bailey, J., "Spray bar zero-gravity vent system for on-orbit liquid hydrogen storage," Tech. rep., 2003.
- [2] Martin, J., and Hastings, L., "Large-scale liquid hydrogen testing of variable density multilayer insulation with a foam substrate," Tech. rep., 2001.
- [3] Kartuzova, O. V., and Kassemi, M., "Modeling droplet heat and mass transfer during spray bar pressure control of the

multipurpose hydrogen test bed (MHTB) tank in normal gravity,” *52nd AIAA/SAE/ASEE Joint Propulsion Conference*, 2016, p. 4673.

- [4] Schrage, R. W., *A theoretical study of interphase mass transfer*, Columbia University Press, 1953.
- [5] Hirt, C. W., and Nichols, B. D., “Volume of fluid (VOF) method for the dynamics of free boundaries,” *Journal of computational physics*, Vol. 39, No. 1, 1981, pp. 201–225.
- [6] Brackbill, J. U., Kothe, D. B., and Zemach, C., “A continuum method for modeling surface tension,” *Journal of computational physics*, Vol. 100, No. 2, 1992, pp. 335–354.
- [7] *ANSYS Fluent User’s Guide*, ANSYS, Inc., release 24.1 ed., 2023.
- [8] Morsi, S., and Alexander, A., “An investigation of particle trajectories in two-phase flow systems,” *Journal of Fluid mechanics*, Vol. 55, No. 2, 1972, pp. 193–208.
- [9] Ranz, W. E., and Marshall, J., W. R., “Evaporation from Drops, Part I,” *Chemical Engineering Progress*, Vol. 48, No. 3, 1952, pp. 141–146.
- [10] Ranz, W. E., and Marshall, J., W. R., “Evaporation from Drops, Part II,” *Chemical Engineering Progress*, Vol. 48, No. 4, 1952, pp. 173–180.
- [11] Kartuzova, O. V., Kassemi, M., Moder, J. P., and Agui, J. H., “Self-pressurization and spray cooling simulations of the multipurpose hydrogen test bed (MHTB) ground-based experiment,” *50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference*, 2014, p. 3578.
- [12] Kartuzova, O. V., and Kassemi, M., “CFD modeling of the multipurpose hydrogen test bed (MHTB) self-pressurization and spray bar mixing experiments in normal gravity: effect of the accommodation coefficient on the tank pressure,” *51st AIAA/SAE/ASEE joint propulsion conference*, 2015, p. 3769.