Modeling and simulation of jet mixing for cryogenic propellant tank

Eymon Lan1, Michael Harris2, Wei Ji1, Mamoru Ishii3, Shanbin Shi1

1Department of Mechanical, Aerospace, and Nuclear Engineering, Rensselaer Polytechnic Institute

110 8th Street, Troy, NY 12180, USA

2National Aeronautics and Space Administration Kennedy Space Center, KSC, FL 32899, USA

3School of Nuclear Engineering, Purdue University, West Lafayette, IN 47907, USA

lane@rpi.edu, michael.f.harris@nasa.gov, jiw2@rpi.edu, ishii@purdue.edu, shis3@rpi.edu

# ABSTRACT

The efficient and accurate simulation of long-term cryogenic fluid behavior in microgravity is critical to space mission planning. To prevent boiloff of precious propellant, axial jets are utilized in thermodynamic vent system to induce flow mixing and eliminate thermal stratification in cryogenic propellant tanks. For fast simulation of cryogen systems, a nodal model with validated correlations can be developed to accurately model complex phenomena.

In this work, a tank nodal model based on a 2D nodalization scheme was developed with SINDA/FLUINT, provided by CRTech. In the new tank model, fluid lumps were discretized to account for the growth of jet radius after injection and boundary layer flow along the wall. To enhance the modeling accuracy of internal flow driven by jets, the analytical solutions of Schlichting were implemented for predicting the jet radius and volumetric flow rates. To validate the correlations, the analytical solutions were compared to data from high fidelity particle image velocimetry (PIV) experiments in literature, which studied the jet mixing in thermally stratified pools using jets with small (~3%) density difference compared to ambient fluid. From the comparison, Schlichting’s analytical solutions overestimate volumetric flow rates for all the experimental cases examined. A modification factor was introduced to reduce uncertainties from 27% to 7%.

To additionally assess computational fluid dynamics (CFD) capabilities for prediction of cryogenic propellant behavior, a numerical study on the jet flow mixing was performed using ANSYS Fluent by implementing existing Reynolds-averaged Navier-Stokes (RANS) based turbulence models such as Realizable k-ε model and shear stress transport k-Ω model. First, a mesh independence study was performed to assess sensitivity of the simulated results such as velocity profiles at quasi steady state through mesh refinements. Then, CFD simulations were conducted using Eulerian-Eulerian fluid model with the turbulence models to simulate the experimental cases. Uncertainties in simulated velocity profiles were identified and quantified by comparing numerical results to the experimental data. The evaluated CFD modeling approach will be used to simulate the jet mixing in microgravity.

This work was supported by an Early-Stage Innovations grant from NASA’s Space Technology Research Grants Program.

**Keywords**: Cryogenic propellant tank; Jet mixing; Nodal model; CFD simulation