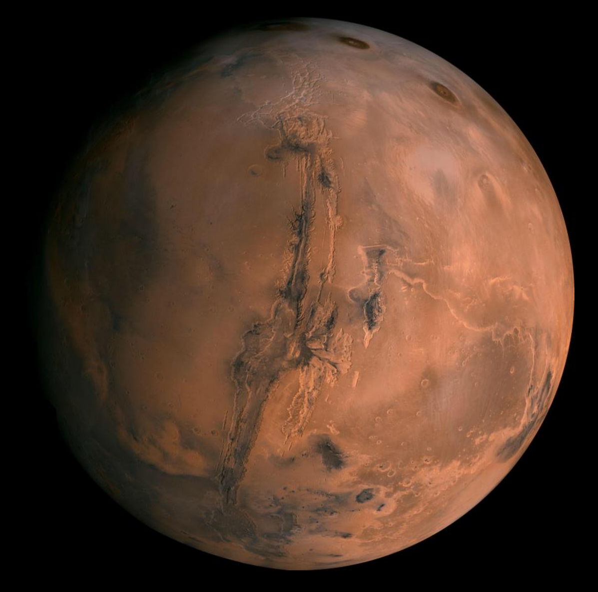
Validation of Cryogenic Propellant Tank Filling using Computational Fluid Dynamics Simulation

H. Q. Yang, CFD Research Corp (ASEG - MSFC ER42)
Jacob Brodnick, NASA (MSFC ER42)



The Fluid Dynamics Branch at MSFC has positioned itself to support a wide range of customers in need of Cryogenic Fluid Management (CFM) analysis. A computational fluid dynamics (CFD) tool used for all manner of internal and external propulsion applications has been extended and refined to better model cryogenic propellant storage and tanking operations. Through the CFM Portfolio project, several validation activities were initiated. Validation of propellant tank self-pressurization, autogenous pressurization, slosh-induced ullage collapse, and jet-induced mixing all aid in defining model accuracy. The on-going validation effort has enabled confident application of the tool to in-line design and evaluation of CFM hardware and operations. Recent project support included defining the impact of in-space slosh dynamics on reaction control system mass for Space Launch System (SLS) upper stages. Propellant mixing strategies were defined to improve performance of a thermal vent system for a Commercial Lunar Payload Services (CLPS) partner. Design support of in-space maneuvers, tank hardware, and autogenous pressurization operations was also provided through Human Landing System (HLS) collaboration work. The branch has engaged the CFM community to share recent findings and capabilities through several forums including conferences, technical interchange meetings, and workshops. Development and demonstration of CFM modeling capabilities continues in this work on the no-vent fill of propellant tank in micro-gravity to meet the needs of NASA and its industry partners in the endeavor to sustainably reach the Moon and beyond.

CFD Model Tool Development

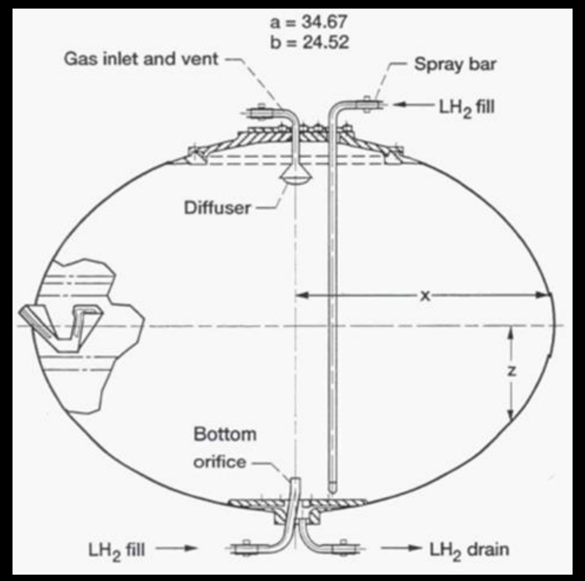
- Volume of Fluid (VOF) based solution methodology for multiphase flow.
- Developed in Loci framework.
- The formulation includes initialization, interface reconstruction and VOF advection. Can keep the sharp interface without diffusion through geometrical interpolation.
- Coupled to pressure-based flow solver, Loci/STREAM with the modifications to face density, accounting for body forces due to gravity and surface tension and modification of the energy equation.
- millions of cells on thousands of processors with good scaling properties

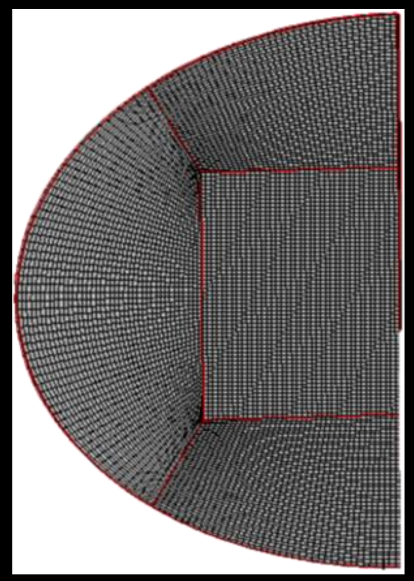
Validation of No-Vent Fill (NVF)

- The No-Vent Fill (NVF) technique can prevent liquid fuel from being expelled overboard for in space propellant transfer, and requires minimal hardware.
- Modeling the NVF process highlights the importance of ullage condensation, which is enhanced by agitation or mixing of the tank contents.
- During the no-vent fill operation, this mixing is induced by an axial jet of incoming liquid from the bottom orifice.
- Validated CFD tool enables prediction for low gravity environment.

CFD Model Development

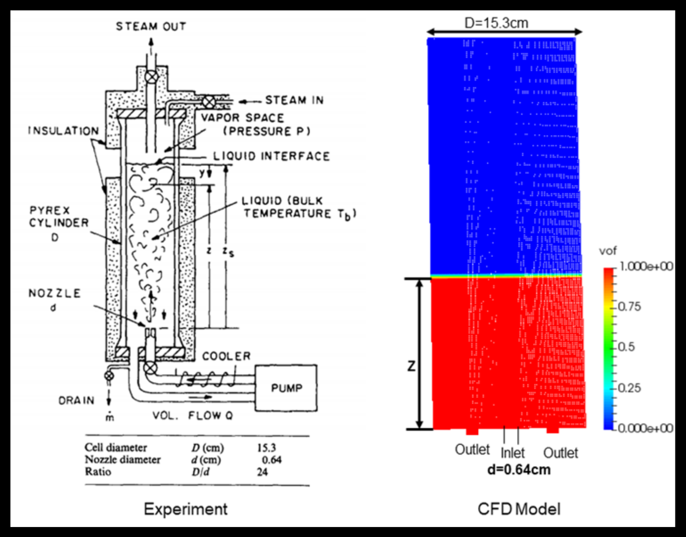
- k-ω-SST-DDES.
- 2D-aximmetric.
- Implicit Phase Change
- Hertz-Knudsen Model coefficient
- Enthalpy-based Temperature Form
- Antoine Saturation Equation
- CFL number: 0.1
- NIST Properties

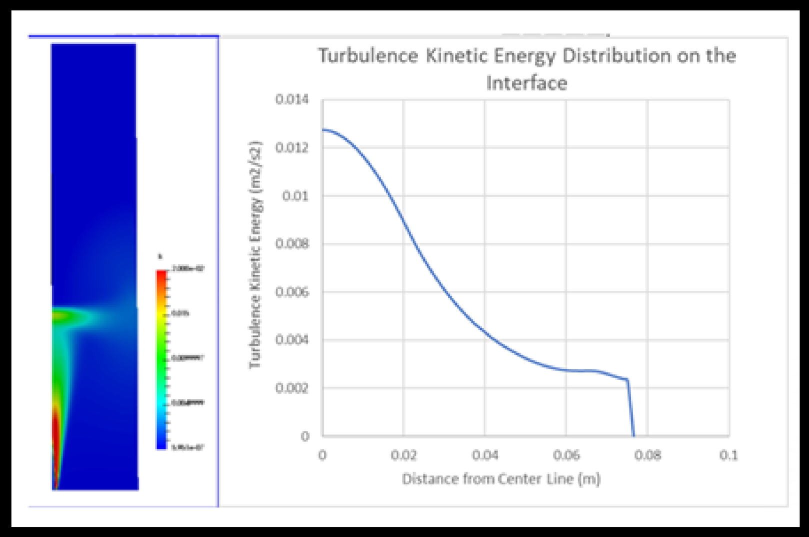


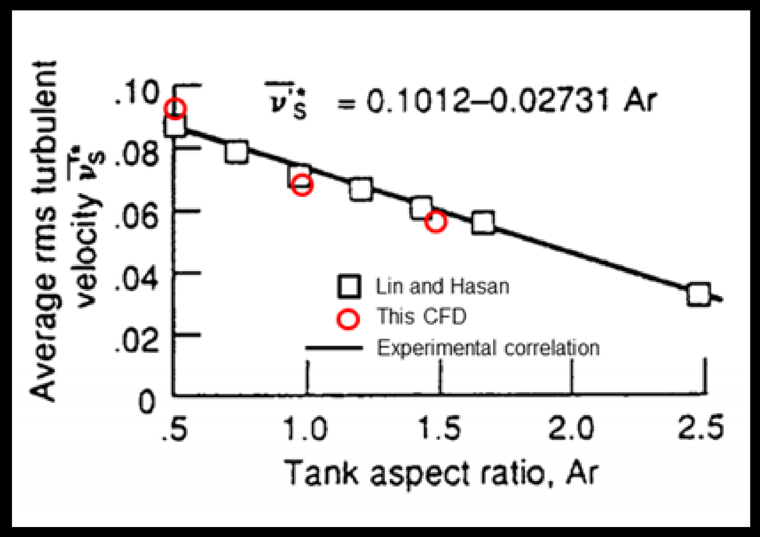


Interface Turbulence Velocity due to Jet Mixing

- Good agreement with experiment

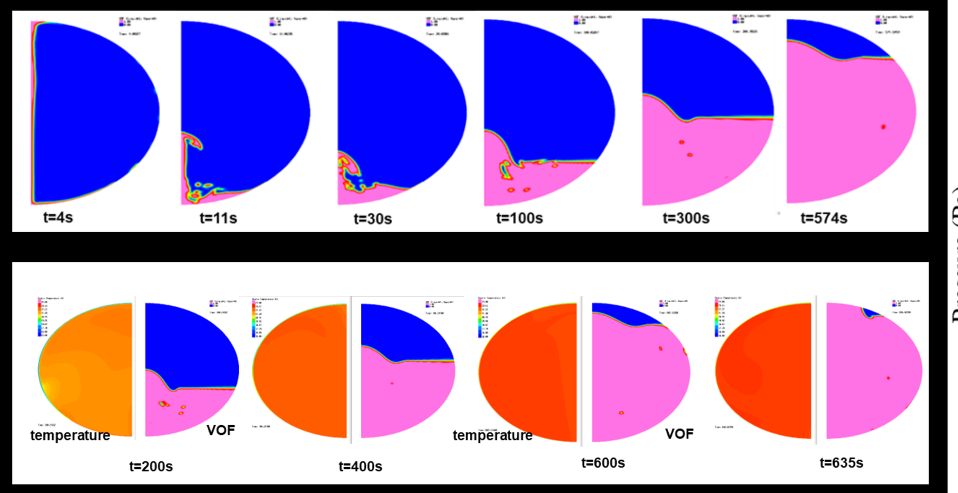


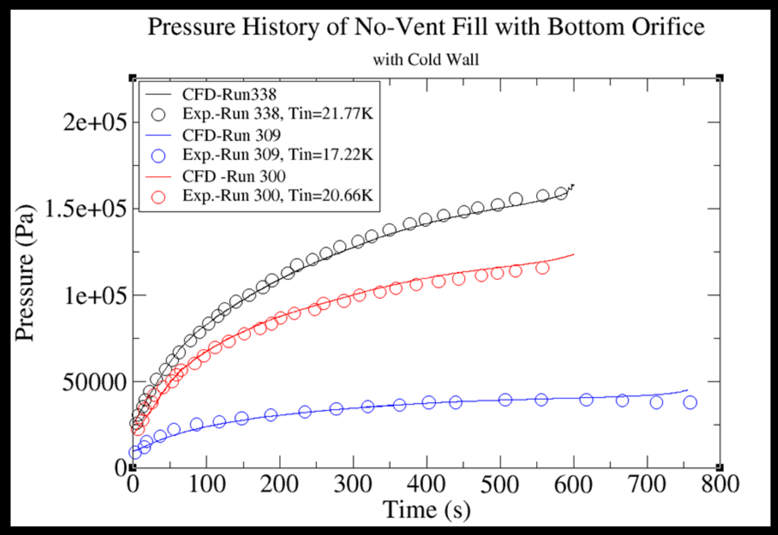


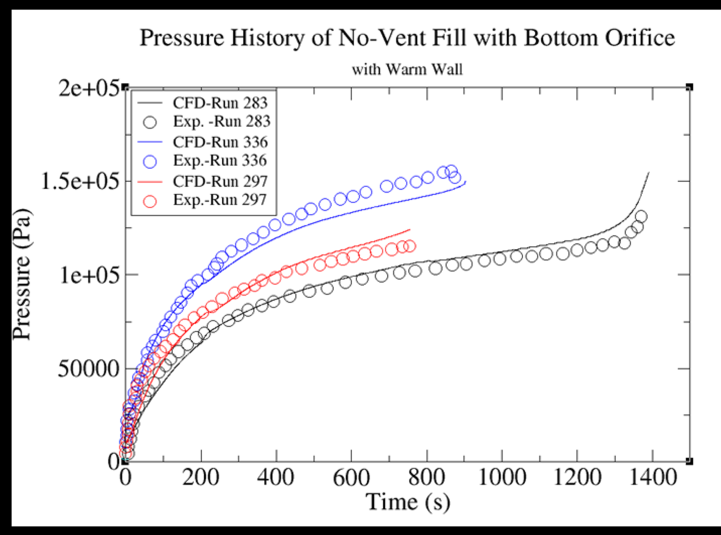


Validation for Cold and Hot Walls

- Good agreement with exp. for different inlet fluid temperatures, flow rates and wall temperatures







Reduced-Order 1D CFD Model

- Same input file, fast running.
- Essential physics well captured.