10th Thermal and Fluids Engineering Conference (TFEC)
March 9-12, 2025
Partially Online Virtual and at George Washington University, DC Conference

An Investigation into Microgravity Capillary Two-Phase Flows Regarding Passive Bubble Dynamics

TFEC-2025-57443 (Presentation Only)

Tyler R. Hatch
NASA Glenn Research Center, Cleveland, OH, USA
Cleveland State University, Cleveland, OH, USA

Dr. Yongxin Tao
Cleveland State University, Cleveland, OH, USA

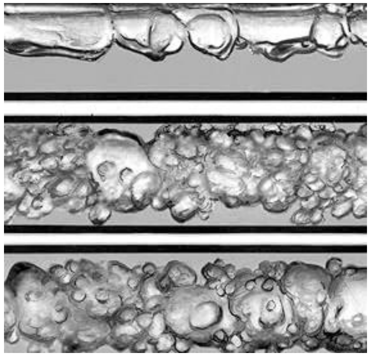
Dr. Mark Weislogel
IRPI, LLC, Wilsonville, OR, USA

Background

- Capillary Channel Flow (CCF) was an experiment performed on the International Space Station for studying a variety of inertial-capillary flows key to spacecraft fluid systems that cannot be studied on the ground.
- CCF sought to address several challenges in microgravity fluid physics related to the containment, storage, and handling of large liquid inventories (fuels, cryogens, and water) aboard spacecraft.
- To date, the results have assisted with the design, testing, and instrumentation of life support systems, phase separation devices, enhanced our current understanding of capillary flows in space.

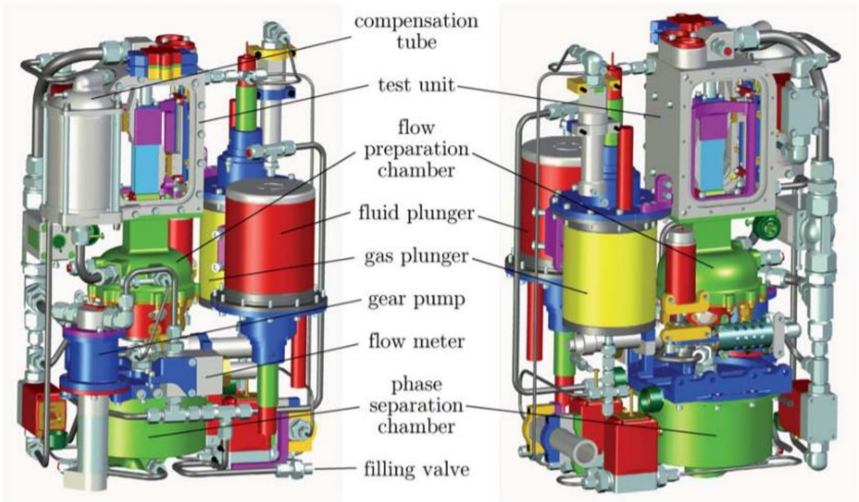
CCF Project Goals

- Provide performance limits for capillary dominated systems such as passive fluids management—capillary collection, pumping, and containment—and processes such as passive phase separation and transport. This is a pressing requirement for a wide range of spacecraft fluid systems.
- Use multiple test cell geometries and variable parameter ranges to investigate capillary systems' ability to passively change multiphase flow regimes. It will also study capillary dominated multiphase flow that may be exploited to assist other active or passive systems.
- Provide critical data for the uniquely low–gravity inertial-capillary flow regime important to liquid fuels and cryogen storage and management.



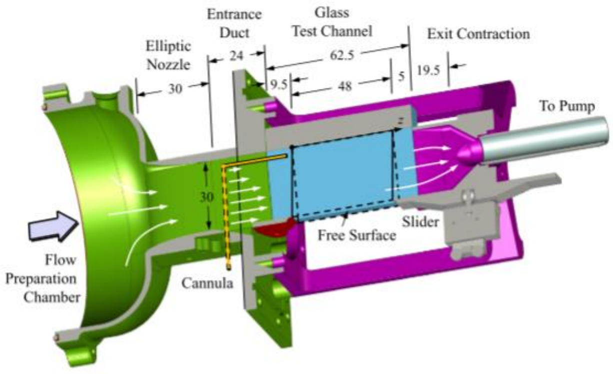
CCF Hardware

- The experiment was designed as a modular system consisting of the Fluid Management System (FMS), the Board Computer (BC), and two Experiment Units (EU), which included the Test Units (TU).
  - This versatility also enabled the use of the setup for other projects with similar technology-driven research objectives. TU2 included a gas bubble generator to test two-phase flow stability.
  - The FMS was equipped with components required to establish flow—pumps, plungers, and valves—while the EU contained the TU, a phase separation chamber, a compensation tube, cameras for the video observation as well as required illumination.
  - The BC performed experiment control, sampling of housekeeping data, and communication with both the MSG interfaces and the ground station.



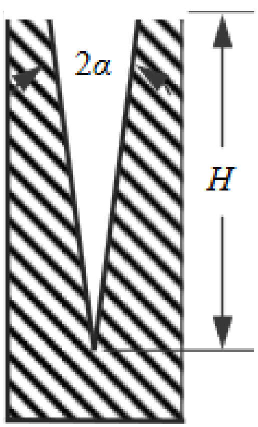
CCF Operations

- For investigation of the selected channel geometries—parallel plates channel, groove channel, and a wedge-shaped channel—and different channel dimensions, TUs were exchangeable.
- The experiments took place on the ISS in 2010, 2011, 2013, and 2014. ISS operations were controlled from ground stations in Portland, Oregon and Bremen, Germany.
  - Results collected show favorable agreement with predictions of critical flow rates and bubble separation rates. The results also indicate the nature of destabilizations and the myriad outcomes of gas liquid flows in the microgravity environment. Regarding critical flow rate limitations, steady uninterrupted flow is below the bubble-ingestion speed.
  - *For passive bubble separations in wedge sectioned conduits, the studies have suggested devices that can be applied directly to perform such task beyond such fundamental investigations*

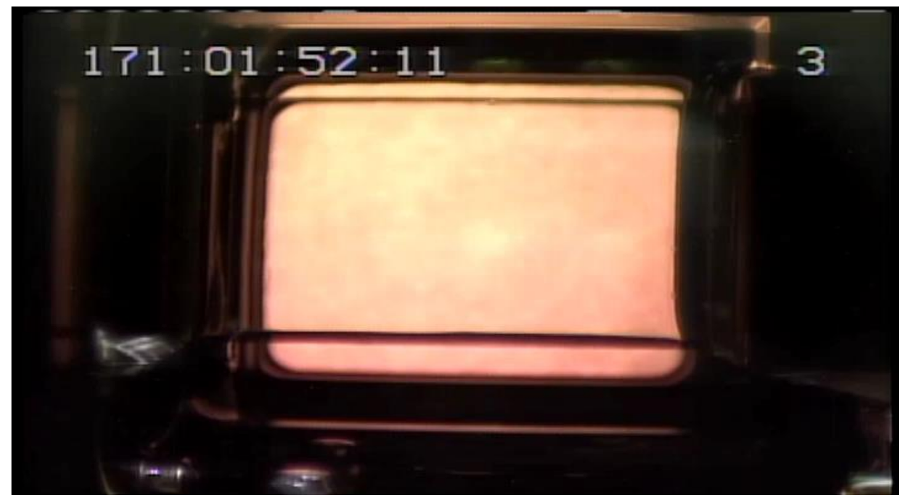


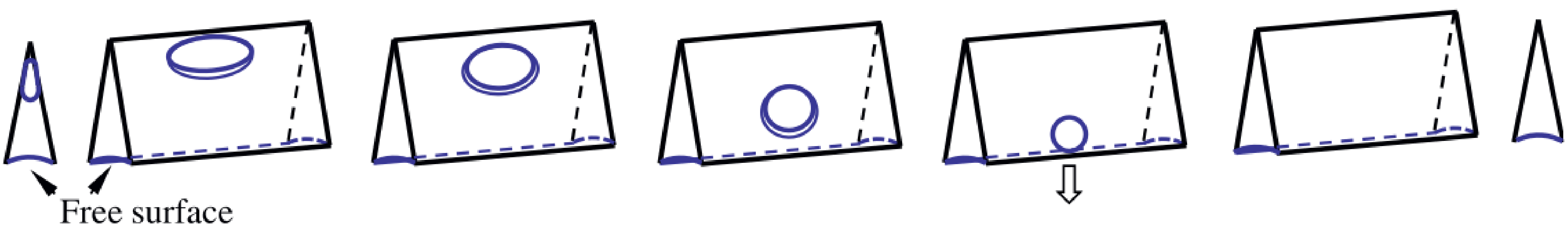
CCF Results

- EU-2 used a wedge channel that gave new insights into passive phase separation via geometry



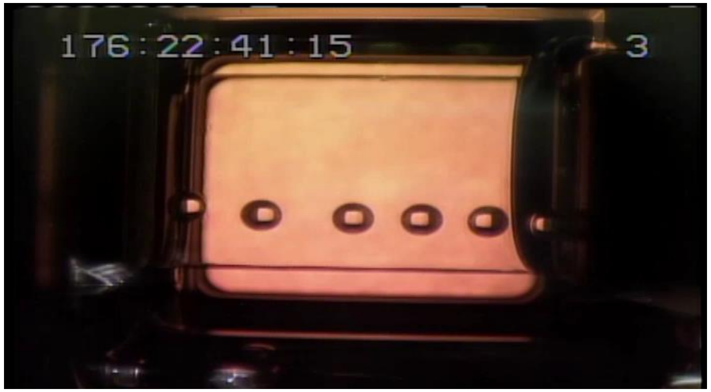
Cross section of wedge geometry interior flow with a corner at the vertex of the wedge, at total wedge angle 2α. The height of the wedge is shown as H and the radius of the circular section on the top is R.





Motivation

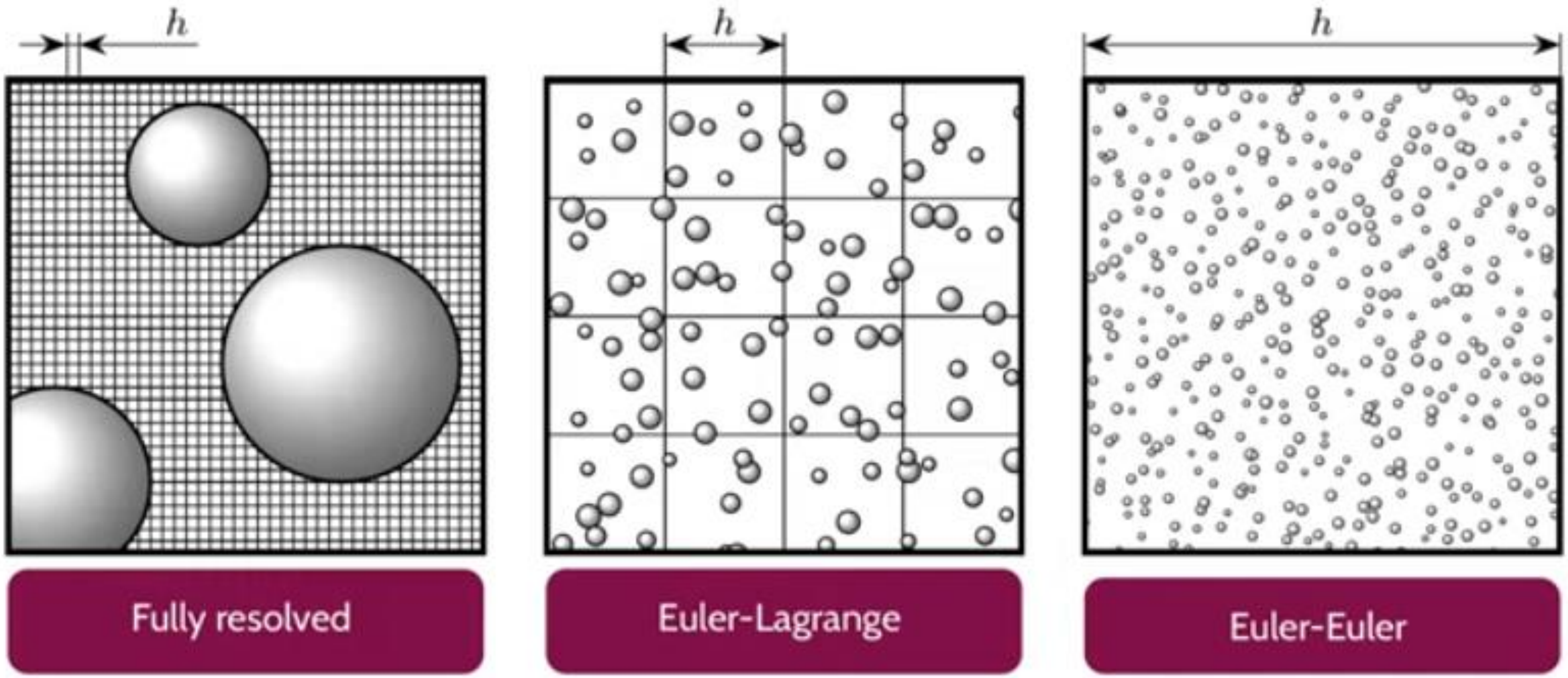
- However, there is a limiting point where the increasing curvature as the bubble grows leads to an imbalance of liquid pressure acting on the surface of the bubble.
- Higher pressure fluid flows around to the corner side of the bubble where the highest curvature exists and displaces the bubble away from the corner as a result.
  - These bubbles do not always have sufficient pressure differences to be pushed up to the free surface for ejection from the system.
  - There is a delicate balance of lift and drag forces acting on the gas bubble that can at times cause it to become “inscripted” at a specific elevation in the wedge channel, short of the free surface.



Can a computational model reveal the underlying physics leading to this phenomena?

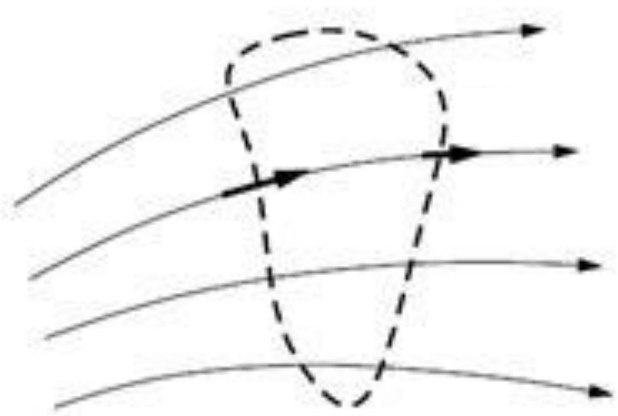
Modeling Approach

- Euler-Euler vs Euler-Lagrange



Eulerian Approach

- Volume of Fluid Model
  - Using a control volume, a single set of momentum equations is solved for flow through the domain
  - When 2 or more fluids are present, the volume fraction is tracked throughout the domain, and momentum + continuity are solved for each phase
  - Allows mass, momentum, and energy exchange between both phases
  - Generally used for transient flows but steady-state is also possible



Volume of Fluid Method

- Continuity based on volume fraction for q^{th} phase

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}\left(\alpha_q \rho_q\right) + \nabla * \left(\alpha_q \rho_q \vec{u}_q\right) = S_{\alpha_q} + \sum_{p=1}^{n}\left(\dot{m}_{pq} - \dot{m}_{qp}\right)\right]$$

- Momentum Equation for phase q

$$\frac{\partial}{\partial t}\left(\alpha_q \rho_q \vec{u}_q\right) + \nabla * \left(\alpha_q \rho_q \vec{u}_q \vec{u}_q\right) = -\alpha_q \nabla p + \nabla * \bar{\bar{\tau}}_q + \alpha_q \rho_q \vec{g} + \sum_{p=1}^{n}\left(\vec{R}_{pq} + \dot{m}_{pq}\vec{u}_{pq} - \dot{m}_{qp}\vec{u}_{qp}\right) + \vec{F}$$

- Where $\bar{\bar{\tau}}_q$ is the q^{th} phase stress strain tensor matrix

$$\bar{\bar{\tau}}_q = \alpha_q \mu_q\left(\nabla \vec{u}_q + \nabla \vec{u}_q^T\right) + \alpha_q\left(\lambda_q - \frac{2}{3}\mu_q\right)\nabla * \vec{u}_q \bar{\bar{I}}$$

- Additional lift forces to consider:
  - Saffman-Mei Lift Force

$$\vec{F}_L = -C_L \rho_q \alpha_p\left(\vec{u}_q - \vec{u}_p\right) \times \left(\nabla \times \vec{u}_q\right) \quad C_L = 6.46\frac{3}{2\pi\sqrt{Re_w}} \quad Re_w = \frac{\rho_q\left|\vec{u}_q - \vec{u}_p\right| d_p}{\mu_q}$$

  - Moraga, Legendre-Magnaudet, Tomiyama, Hessenkemper, models as well
  - No shear condition on bubbles

Volume of Fluid Method

- Surface tension force

  Continuum Surface Force (CSF) – Brackbill et al.

  – Surface tension is a continuous, 3D effect across an interface instead of a boundary condition
  – Surface curvature is dependent on local gradients at surface
  – Source term is added in momentum equation for 2 phase flow studied here:

$$F_{vol} = \sigma_{ij} \frac{\rho \kappa_i \nabla \alpha_i}{\frac{1}{2}(\rho_i + \rho_j)} \qquad F_{CSF} = \sigma \kappa \nabla \alpha$$

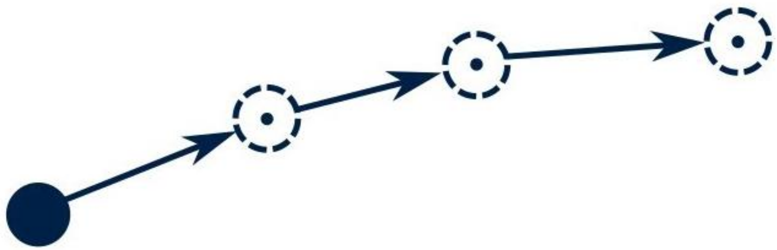
  Continuum Surface Stress (CSS)

  – Conservative approach, avoids curvature calculation – anisotropic
  – Source term is dependent on volume fraction directly
  – Advantageous for variable surface tension
  – Performs better in sharp corners which could be hard to resolve in solver

$$F_{CSS} = \nabla \left[\sigma \left(|\nabla \alpha| I - \frac{\nabla \alpha \times \nabla \alpha}{|\nabla \alpha|} \right) \right]$$

Lagrangian Approach

- Discrete Phase Model
  - Fluid phase is solved via differential forms of Navier-Stokes equations and Newton's laws of motion
  - Dispersed phase (bubbles) are solved by tracking individual particles
  - Allows mass, momentum, and energy exchange between both phases
  - Neglecting particle-particle interactions or breakup
  - Must have a low volume fraction of dispersed phase (<10%)
    - Mesh spacing larger than particle diameter
    - However large mass is allowed (x100%)



Discrete Phase Method

- Force balance for the particle (bubble)

$$m_p \frac{d\vec{u}_p}{dt} = m_p \frac{\vec{u}-\vec{u}_p}{\tau_r} m_p \frac{\vec{g}(\rho_p-\rho)}{\rho_p} + \vec{F}$$

where the droplet relaxation time in the drag force term is

$$\tau_r = \frac{\rho_p d_p^2}{18\mu} \frac{24}{C_d Re}$$

Using a relative Reynolds number $Re = \frac{\rho d_p |\vec{u}-\vec{u}_p|}{\mu}$

- Combine the force balance with particle (bubble) velocity

$$\frac{d\vec{x}}{dt} = \vec{u}_p$$

Coupled ODE equations

- Particle (bubble) velocity can be expressed as

$$\frac{d\vec{u}_p}{dt} = \frac{1}{\tau_p}\left(\vec{u} - \vec{u}_p\right) + \vec{a}$$ with additional acceleration forces besides drag force

Discrete Phase Method

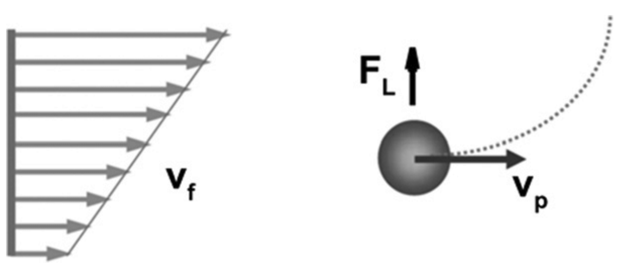
- Additional forces to consider:
  - Because the particle (bubble) may be rotating, we must look at the Magnus force caused by a pressure differential along the surface. Rotational Lift force is:

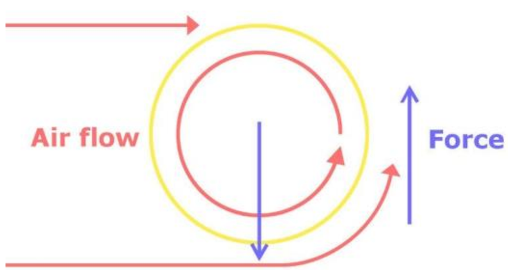
$$\vec{F}_{RL} = \frac{1}{2} A_p C_{RL} \rho_f \frac{|\vec{V}|}{|\vec{\Omega}|} (\vec{V} \times \vec{\Omega}) \text{ using } C_{RL} \approx 0.4$$

  - Shear causing lift (Saffman Lift) may also contribute here:

$$\vec{F}_S = m_p \frac{2K\nu^{1/2}\rho d_{ij}}{\rho_p d_p (d_{lk} d_{kl})^{1/4}} (\vec{u} - \vec{u}_p) \text{ where } K = 2.594$$

  - ANSYS Fluent has a “Hybrid” mode that allows DP +VOF coupled together with Macro-particles
  - However: No internal particle (bubble) forces, and it is treated as incompressible
  - May not allow a moving boundary on the surface – solves no slip condition





Case Description

- Simulations performed using ANSYS Fluent version 2024 R2
- 3D wedge geometry was modeled
- 2 phase VOF simulation, microgravity ($10^{-6}g$)
- Compressible ideal gas (Nitrogen gas + Novec HFE 7500 fluid)
- Contact angle 0° (high wettability engineering fluid)
- Surface tension effects via Continuum Surface Stress (CSS)
- Tetrahedral mesh elements, polyhedral conversion limited by sharp corner angle in geometry
- Adiabatic, no heat transfer currently (may add later)
- Convergence Criteria:
  - Residuals: 10^{-3} convergence criteria, except continuity: 10^{-6} to ensure "watertightness"

User Defined Function (UDF):

- Defines bubble position and velocity injection into flow via patching
- Bubble Volume: 0.18 – 0.138 mL (4-6mm diameter on average)
  - Defines coarsest meshing limit to fully resolve individual bubbles

Initial and Boundary Conditions

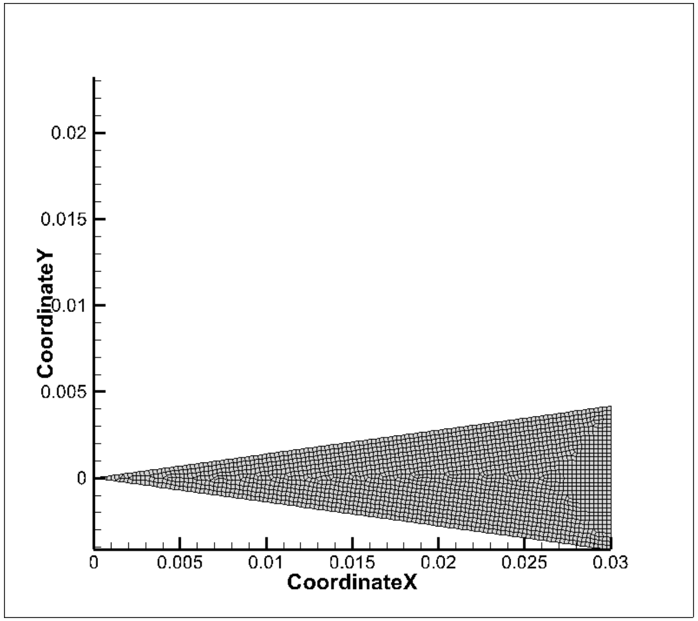
- Gauge pressure: 14.7 *PSI*
  - Experiment channel open to ISS cabin air
  - Same for pressure outlet
- Temperature: 298 *K* - Adiabatic
- Velocity Inlet
  - Z velocity 0.3766 *cm/s*
  - converted from total flow rate of $Q = 0.47\ {}^{cm^3}/_{s}$ using cross sectional area 1.248 cm^2
- 6cm long, 2cm tall,15° wedge angle
- Turbulent kinetic energy 0.375
- HFE 7500 Properties

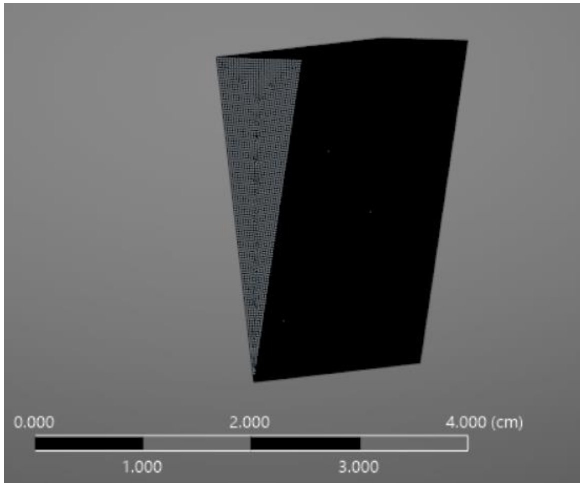
$\rho = 1614\ {}^{kg}/_{m^3}$

$\mu = 0.00125\ {}^{kg}/_{m\ s}$

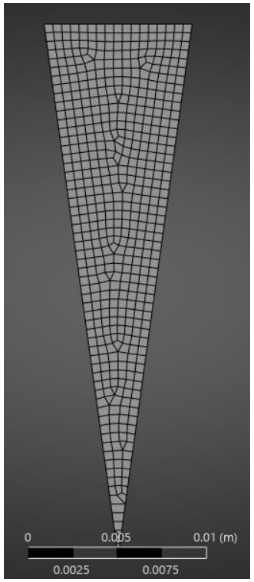
$\sigma = 0.0162\ {}^{N}/_{m}$

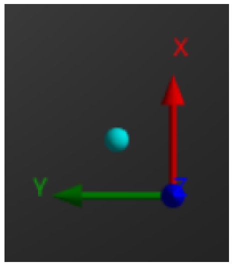
- Quartz glass exterior walls



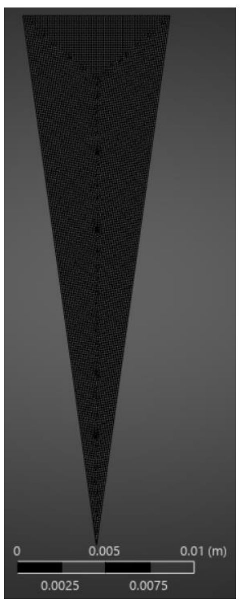


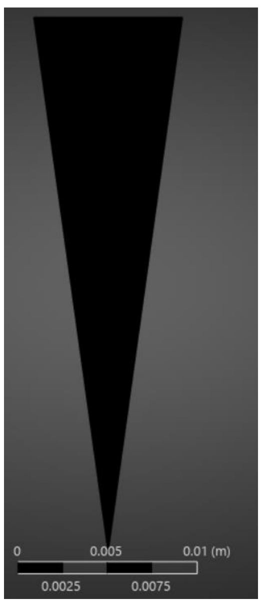
Mesh Grid Independence Study

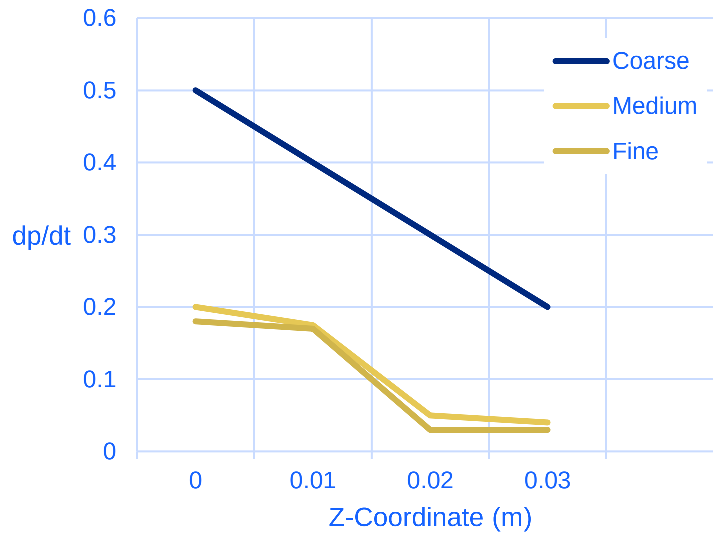




	Sizing Basis	Element size (mm)	Total number of elements	Total number of nodes
Coarse mesh	Aspect Ratio	0.5	58,394	51,168
Medium mesh	Aspect Ratio	0.1	6,253,481	6,078,240
Fine mesh	Skew	0.05	48,892,797	48,206,400

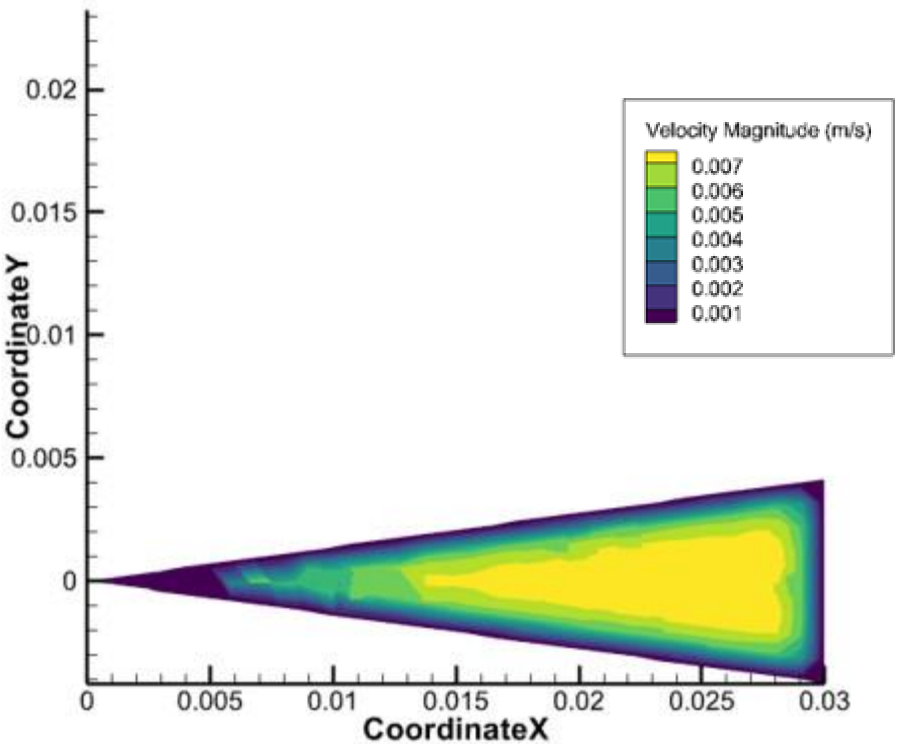




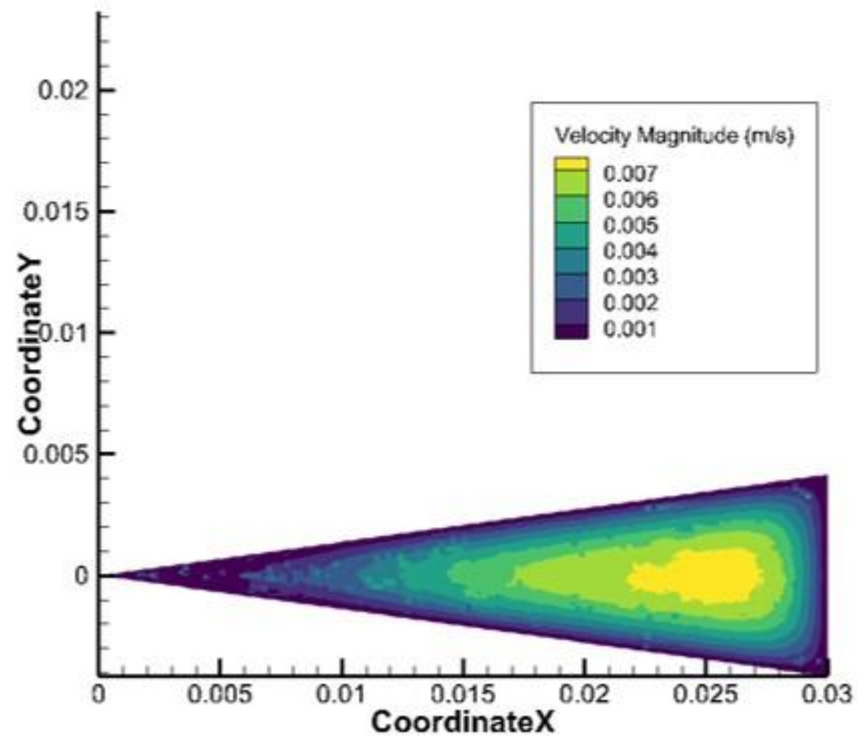


Current Models

- VOF vs DP, with gravity term still included (-9.81 m / s^2)



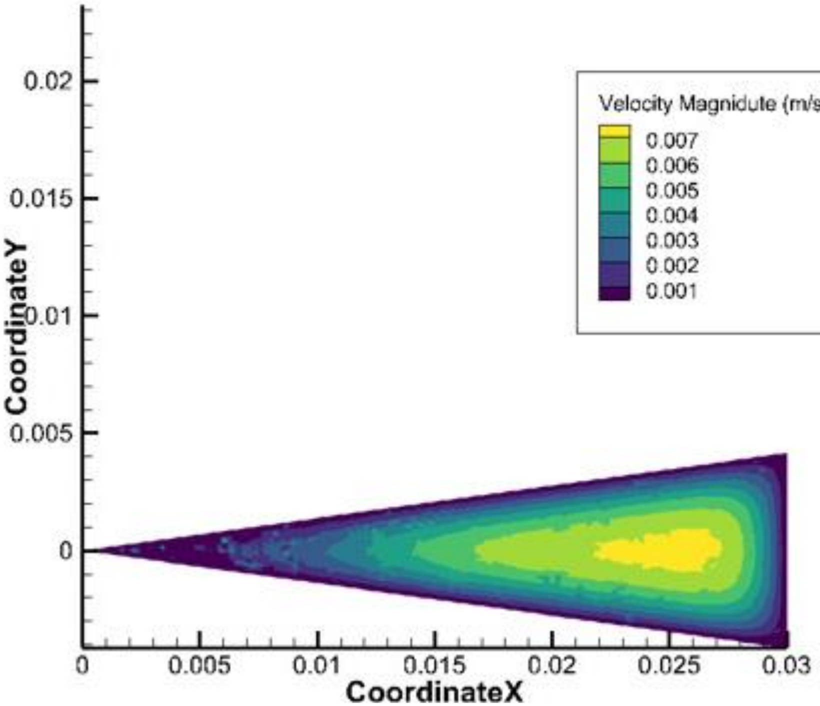
Volume of Fluid



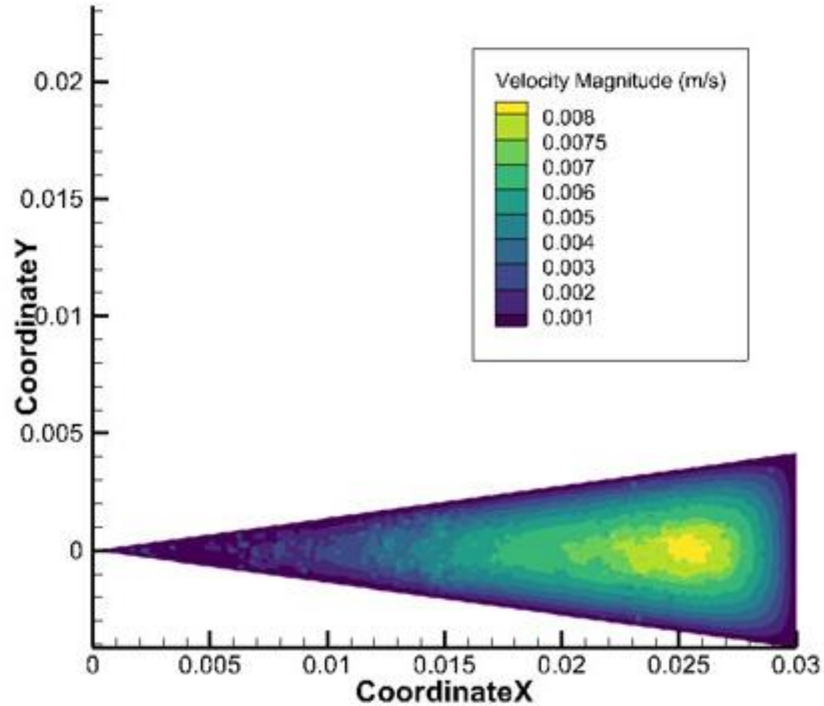
Discrete Phase

Current Models

- VOF vs DP, microgravity ($1x10^{-6}$ * g)



Volume of Fluid

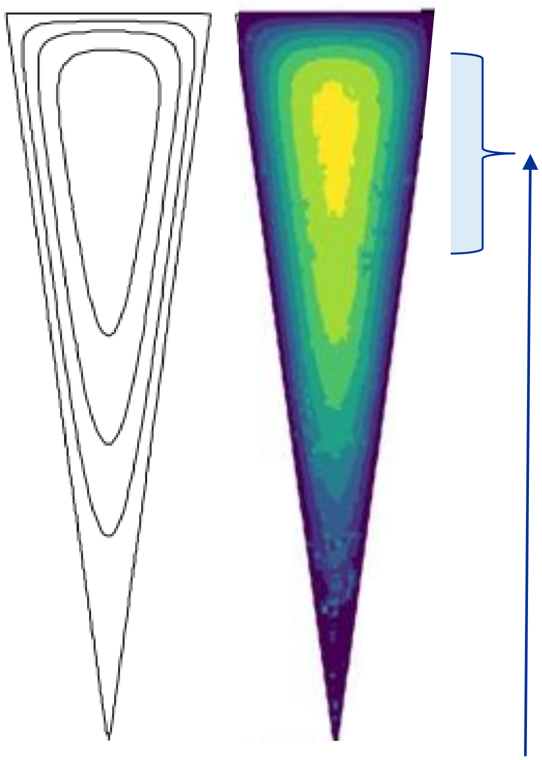


Discrete Phase

Validation

- Discrete phase seems to match literature more exactly than VOF (Klatte, 2012)
- However, VOF seems better suited for bubble investigation
- Drop tower experiments verified by ISS data
- Surface Evolver modeling of interface
  - but no active model representing bubble motion
- Flow velocity increases from 0.38 *cm/s*

  to 0.7 *cm/s* in center of wedge channel
- Inscription range: 0.03 – 0.13 *mm* from bottom



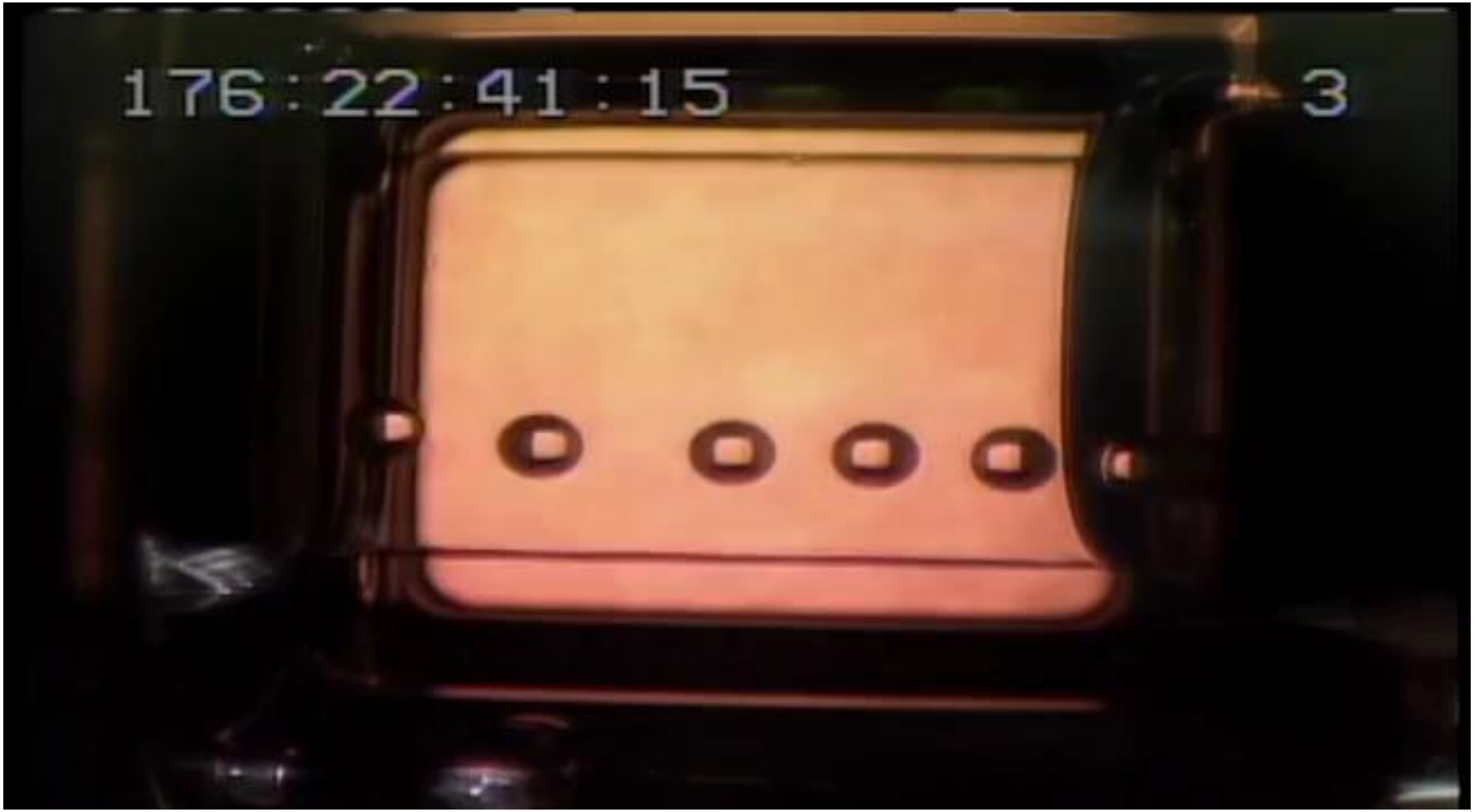
- Next steps:
  - Open channel model with free surface interactions
  - Integrate bubble motion code into model

Conclusions

- CFD model was developed and validated against theoretical velocity contours.
- Discrete phase methods offer advantages in force calculations on a particle but may not capture the entire picture here as VOF can.
- Sharp wedge corner limits the ability for polyhedral meshing and surface tension force representation.
- Bubble diameter is a limiting factor in mesh sizing to fully resolve the driving forces acting upon it.
- Once open channel model has been validated also, computational bubble motion data will be compared against 0g ISS experimental data.

- Acknowledgements
  - Dr. Mark Weislogel, IRPI
  - Dr. Yongxin Tao, Cleveland State University
  - NASA GRC Academic Education Program (AEP)

Thank you!



Abstract

Spacecraft fluid systems require unique considerations to function effectively in microgravity. An open wedge channel can passively separate bubbles from two-phase flows, mimicking the role of buoyancy on Earth. For large bubbles, or those that merge within the channel, capillary forces drive them upwards and outwards, allowing them to coalesce and escape through the free surface. In low-gravity environments, smaller bubbles tend to follow paths along their inscribed elevations rather than leaving the liquid. These 'unseparated' bubbles can lead to downstream pump failure, cavitation, flow instabilities, and other adverse effects in spacecraft fluid systems. Understanding these mechanisms is crucial for reliable passive capillary fluid management in space applications, such as liquid fuel and propellant transport, thermal fluid circulation, water recycling, and plant watering systems.

The DLR and NASA Capillary Channel Flow (CCF) experiment on the International Space Station has provided a valuable database on these phenomena. Utilizing this resource, we aim to benchmark our numerical study of zero-gravity bubbly two-phase flow in open wedge channels, focusing on the interplay of Saffman lift and the Magnus effect on bubble motion. Our findings are expected to guide the development of more robust passive phase separation methods for spacecraft plumbing systems.