ASGSR, *November 8-12, 2022*

CFD and Thermodynamic Model Predictions of No-Vent Tank Filling in Microgravity

Rebecca L. Winter[1], Vishwas Kulkarni[2], Michael Dreyer[2], Mohammad Kassemi[1]

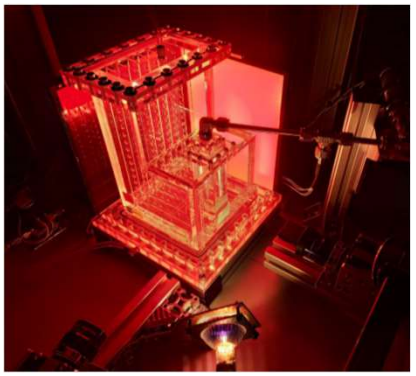
[1] Case Western Reserve University
National Center for Space Exploration Research
NASA Glenn Research Center
[2] Mechanical Engineering and Process Engineering Dept
University of Bremen

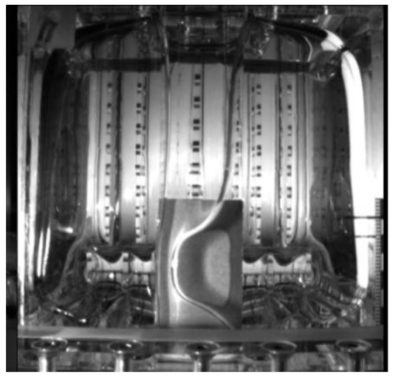
November 11, 2022

Background of ZBOT-FT

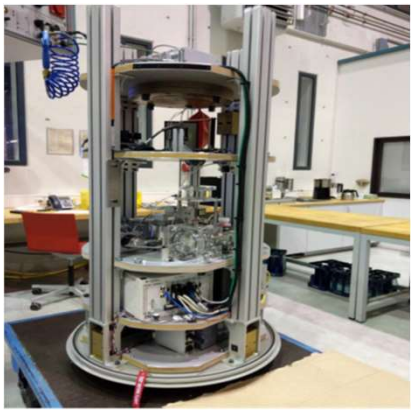
- ZBOT-FT (Filling & Transfer) is a collaboration between NASA/Case Western Reserve University and DLR (German Aerospace Center)/University of Bremen
- The project is designed to study how cryogenic or cryo-simulant fluid behaves when filling and emptying a tank in microgravity
- Application for in-flight storage and refueling of cryogenic liquid propellant

- Current CFD models in development
  - Donor tank
    - Study depressurization cavitation
  - Receiver tank
    - Study vented tank filling
    - Study unvented tank filling
  - Transfer line
    - Study flow and transition to nucleate boiling

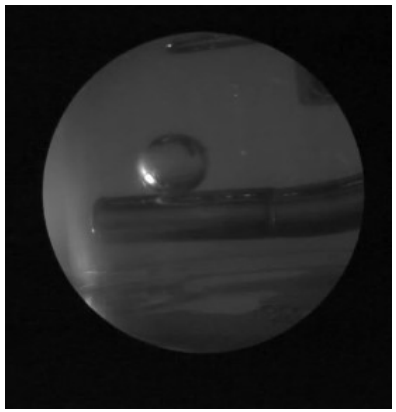




LAD performance tests in Donor Tank



Receiver Tank in Drop Tower Assembly for tank filling tests

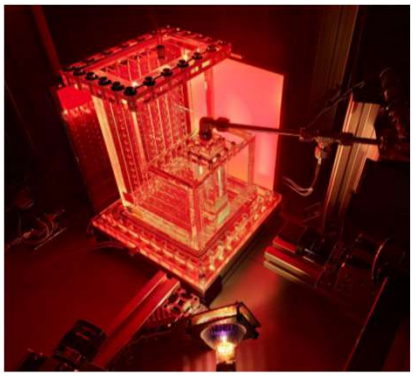


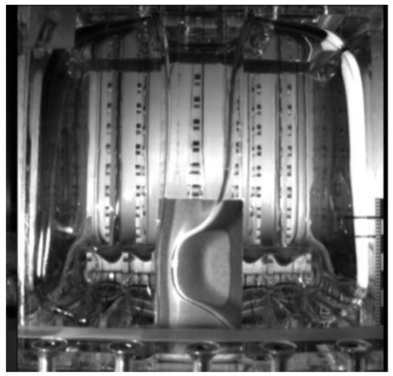
Drop Tower cavitation tests with LCH_4

Background of ZBOT-FT

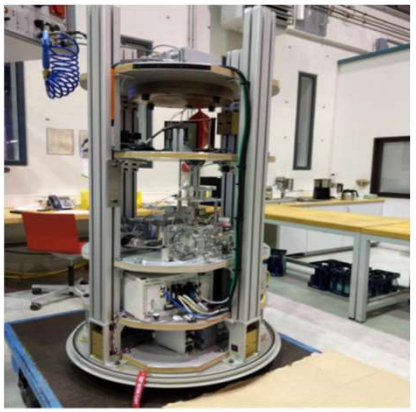
- ZBOT-FT (Filling & Transfer) is a collaboration between NASA/Case Western Reserve University and DLR (German Aerospace Center)/University of Bremen
- The project is designed to study how cryogenic or cryo-simulant fluid behaves when filling and emptying a tank in microgravity
- Application for in-flight storage and refueling of cryogenic liquid propellant

- Current CFD models in development
  - Donor tank
    - Study depressurization cavitation
  - Receiver tank
    - Study vented tank filling
    - Study unvented tank filling
  - Transfer line
    - Study flow and transition to nucleate boiling

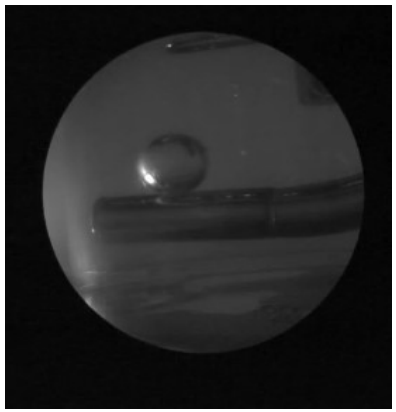




LAD performance tests in Donor Tank



Receiver Tank in Drop Tower Assembly for tank filling tests

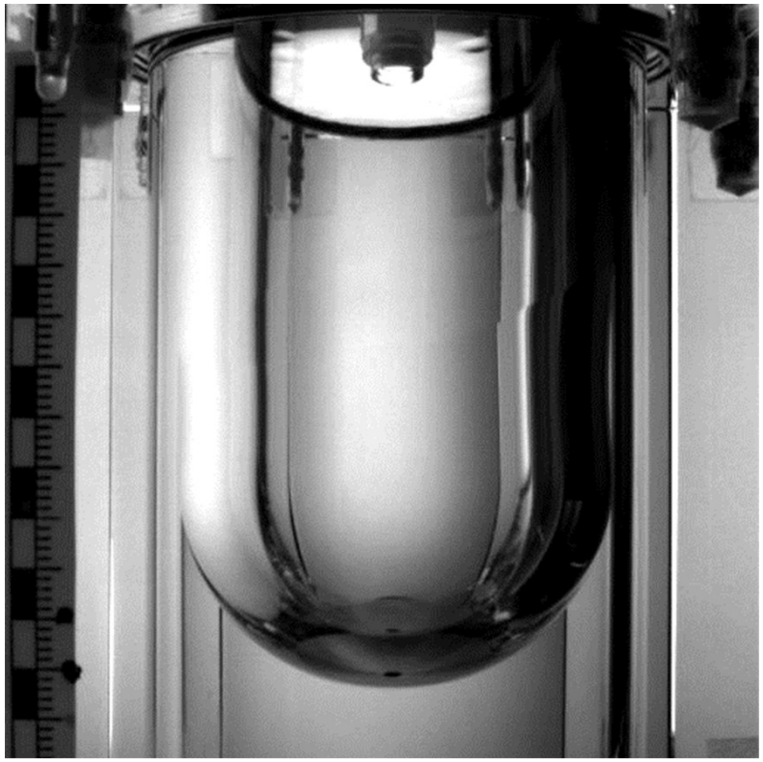


Drop Tower cavitation tests with LCH_4

Background of ZBOT-FT

- ZBOT-FT (Filling & Transfer) is a collaboration between NASA/Case Western Reserve University and DLR (German Aerospace Center)/University of Bremen
- The project is designed to study how cryogenic or cryo-simulant fluid behaves when filling and emptying a tank in microgravity
- Application for in-flight storage and refueling of cryogenic liquid propellant

- Current CFD models in development
  - Donor tank
    - Study depressurization cavitation
  - Receiver tank
    - Study vented tank filling
    - Study unvented tank filling
  - Transfer line
    - Study flow and transition to nucleate boiling



Vented tank filling drop tower experiment
1.4 ml/s inlet flow rate, initially filled 0.03 m

Motivation and Goals

- Motivation for studying unvented cryogen tanks
  - Cryogenic tanks are generally vented to prevent dangerous pressure rise
  - Fuel loss from venting may be up to 3% per month of storage [1]
  - Unvented tanks present a more efficient cryogenic storage option

- **The goal is to develop a model for studying the behavior of an unvented tank filling with cryogen simulant liquid in microgravity**

- Major questions
  - How does the unvented tank fill?
  - How well do thermodynamic and CFD models agree?
  - Does a 2D axisymmetric CFD model match the behavior of a 3D CFD model?
  - How does the geyser behave during a no-vent tank filling?
  - How do temperature and pressure behave in the tank?
  - How does inlet flow rate affect the behavior?

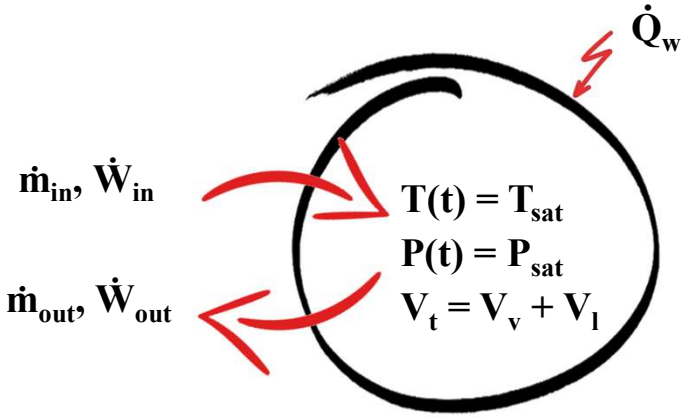
[1] Motil & Meyer. AIAA 2007-343, 2007.

Outline

- Thermodynamic Model
- 2D Axi-symmetric Computational Model
- 3D Computational Model
- Conclusions

One-Dimensional Thermodynamic Model – Setup

- The thermodynamic model gives a simplified theoretical understanding of the problem
  - Neglects geometry effects, no liquid-vapor interfaces
  - Spatially isothermal
  - Always at saturation conditions



- Based on mass and energy balance

$$\frac{dm_v}{dt} + \frac{dm_l}{dt} = \dot{m}_{in} - \dot{m}_{out}$$

$$\frac{d(m_v u_v)}{dt} + \frac{d(m_l u_l)}{dt} = \dot{Q}_w + \dot{m}_{in} u_{in_l} - \dot{m}_{out} u_{out_l} + \dot{W}_{in} - \dot{W}_{out}$$

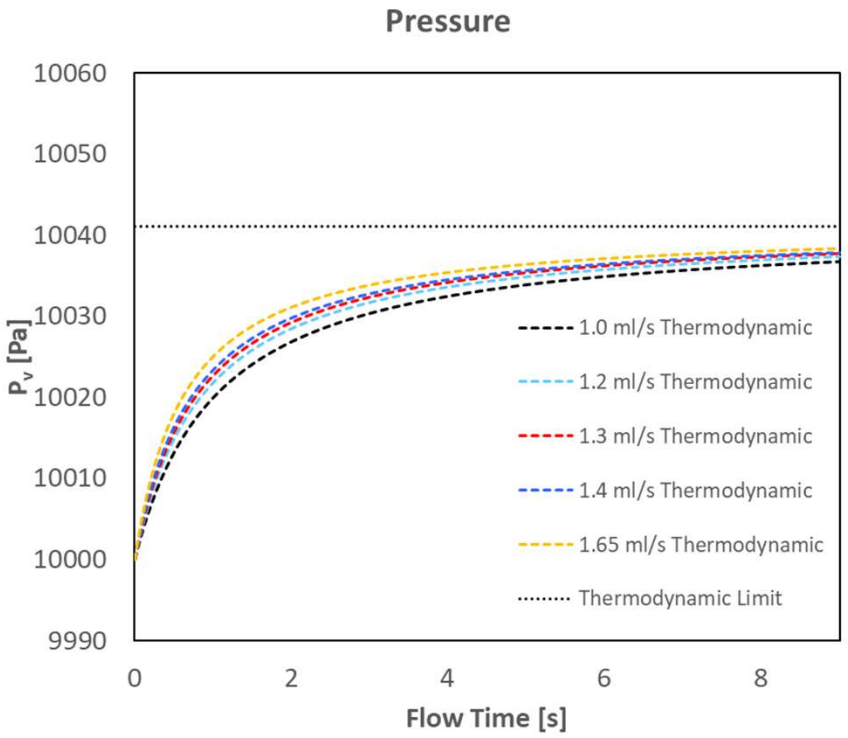
m is mass, u is velocity, $\dot{Q}_w$ is heat generation at the wall, W is work, $_l$ is liquid, $_v$ is vapor

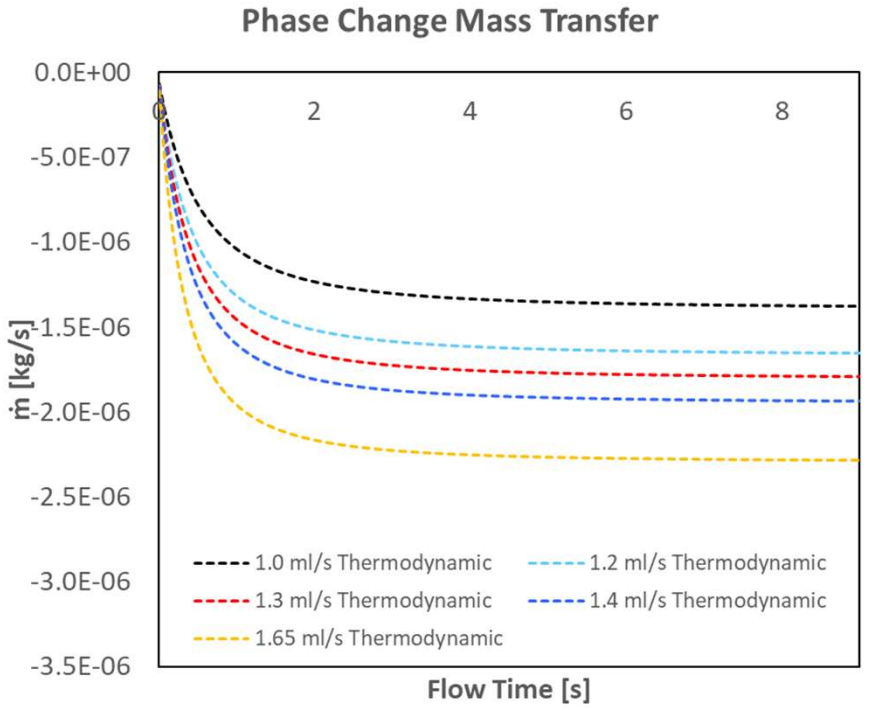
- These equations are solved and combined into the final evolution equation

$$\frac{dT}{dt} = \frac{\dot{Q}_w + h_{lv}(\dot{m}_{in_v} - \dot{m}_{out_v}) + \frac{h_{lv}\rho_v}{\rho_l - \rho_v}(\dot{m}_{in} - \dot{m}_{out}) + (c_{pv}\dot{m}_{in_v} + c_{pl}\dot{m}_{in_l})(T_{in} - T)}{c_{p_v}\rho_v V_v + c_{p_l}\rho_l V_t + \frac{h_{lv}}{\rho_l - \rho_v}\left(\rho_v \frac{d\rho_l}{dT} V_l - \rho_l \frac{d\rho_v}{dT} V_v\right) - V_t \frac{dP}{dT} + c_w m_w}$$

- Temperature is solved iteratively in Matlab

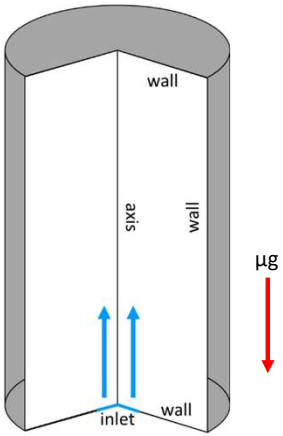
One-Dimensional Thermodynamic Model – Results

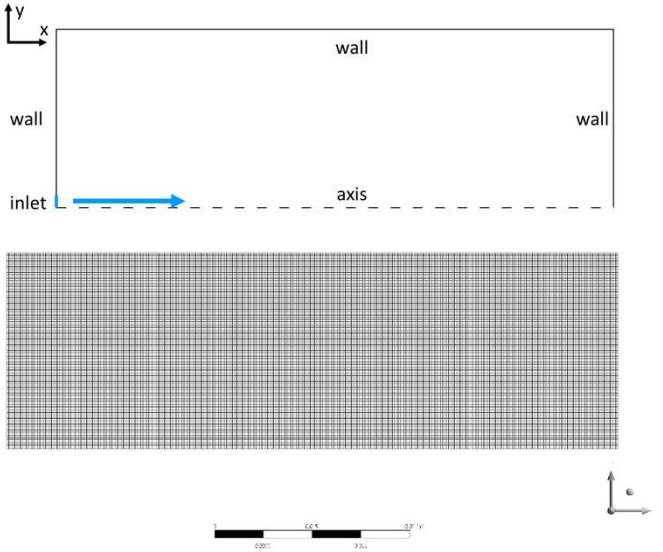




2D Axisymmetric CFD Model – Setup

- 2D axi-symmetric transient CFD model
- Problem Setup
  - 9 seconds of flow time
  - Microgravity -9.81E-6 m/s^2
- Solver Settings
  - Volume of Fluid (VOF) scheme with explicit formulation
  - Sharp interface modeling
  - Laminar fluid flow
  - Pressure-based solver with absolute velocity formulation
  - Phase change at interfaces based on the Shrage model
  - PISO solving method with least squares cell based gradient
  - PRESTO! pressure formulation
  - Compressive scheme for volume fraction
  - Second order upwind density, momentum, and energy
  - First order implicit transient formulation
  - 1E-4 s time steps
- Geometry and mesh
  - Tank height is 0.094 m, tank radius is 0.03 m, and inlet is at the center of the bottom boundary with a radius of 0.002 m
  - Adiabatic walls with wall adhesion
  - Structured uniform mesh with 3E-4 m elements, has 32,130 nodes and 31,714 elements
- Initial Conditions
  - PnP vapor
  - 283.18 K and 10,000 Pa.

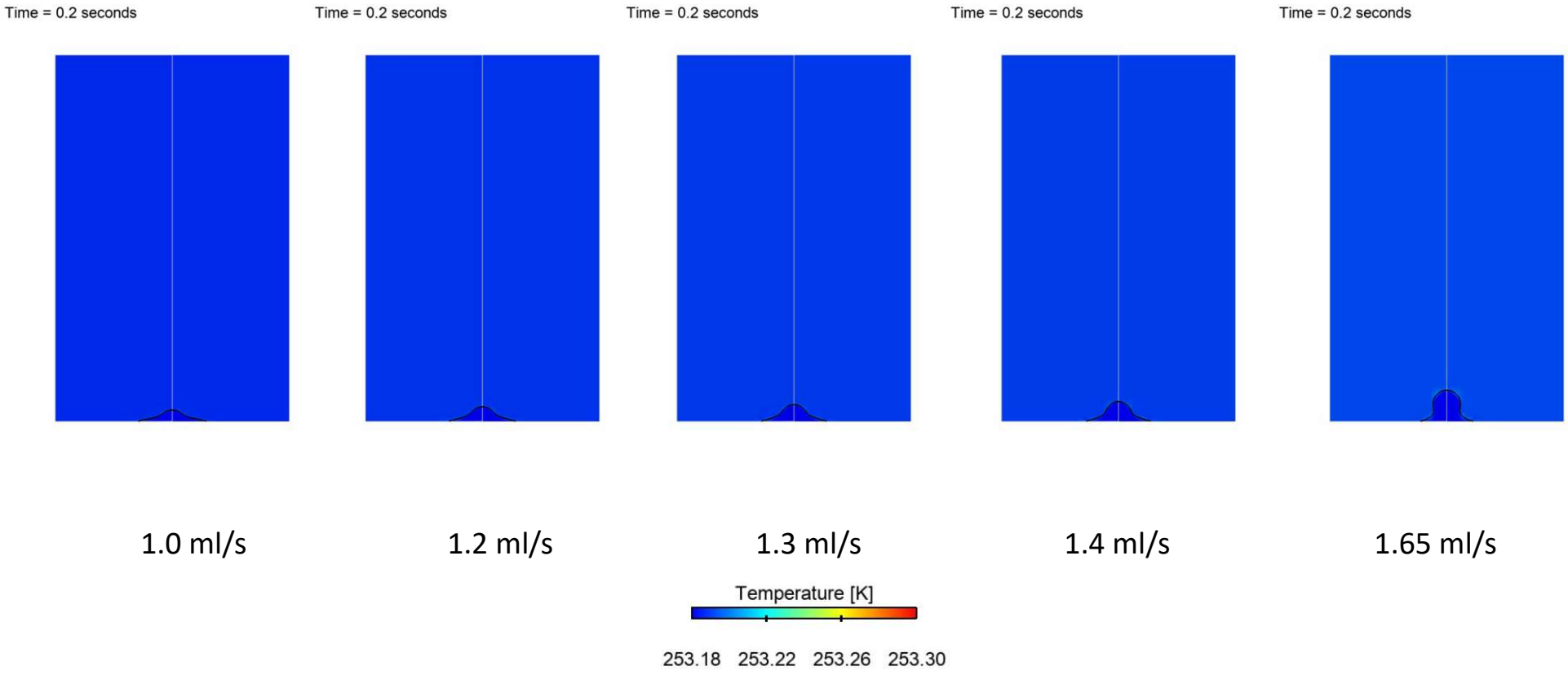




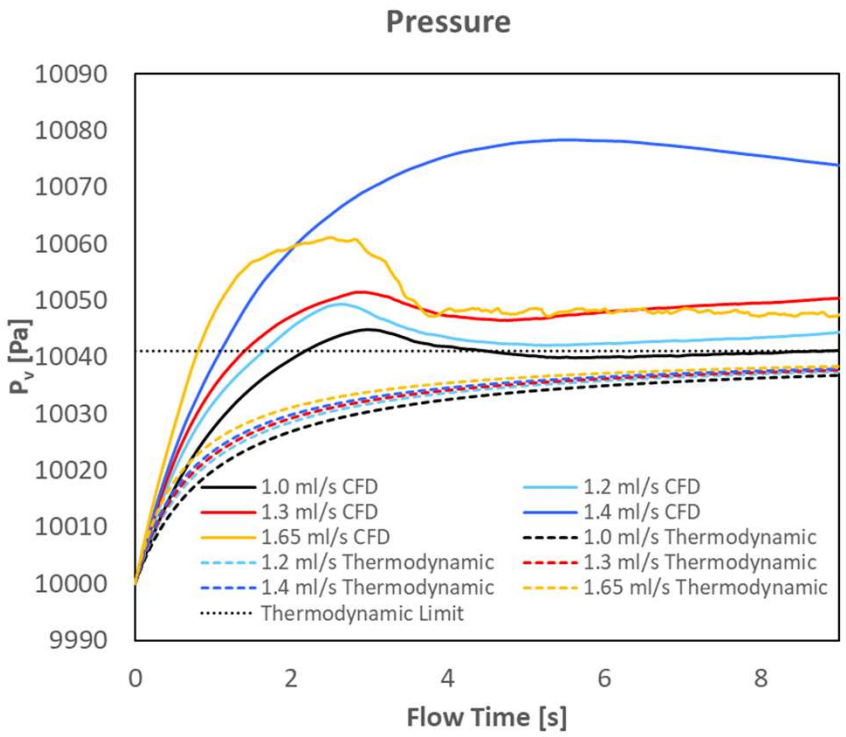
Properties of PnP (Perfluoro-normal-pentane C_5F_{12}) via DIPPR database at 253.18 K:

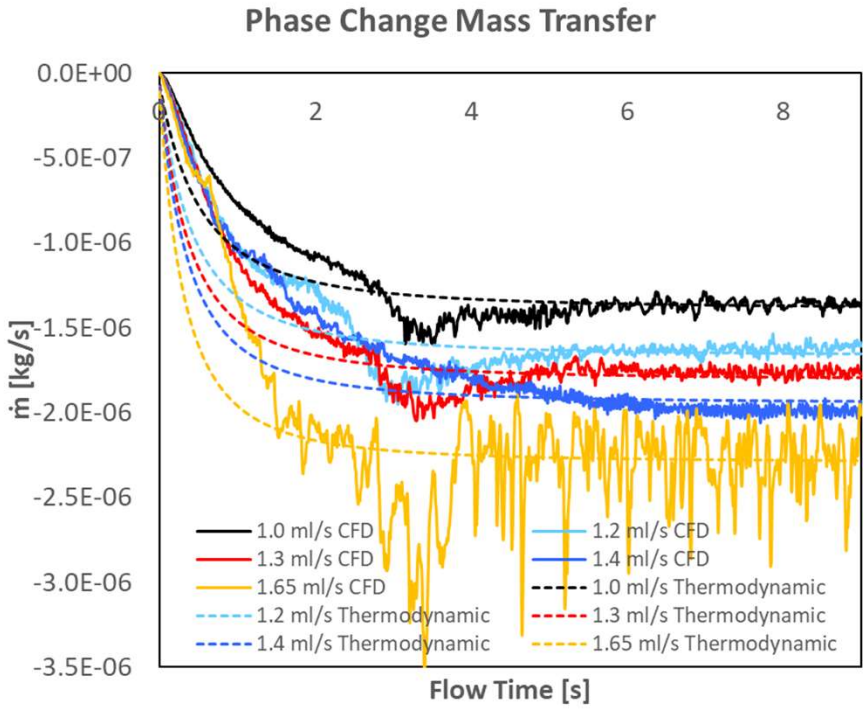
Property	Liquid	Vapor
Density	Boussinesq, 1743.919 kg/m^3	Ideal gas
Specific Heat	1131.825 J/kg K	692.049 J/kg K
Thermal Conductivity	0.0697077 W/m K	0.00729833 W/m K
Viscosity	9.0238E-4 kg/m s	9.5936E-6 kg/m s
Molecular Weight	288.034 kg/kmol	288.034 kg/kmol
Standard State Enthalpy	0 J/kmol	2.4375E7 J/kmol
Reference Temperature	298.15 K	298.15 K
Thermal Expansion Coefficient	0.00171415 K^{-1}	
Surface Tension Coefficient	0.0140107 N/m	
Wall Contact Angle	0°	

2D CFD Model – Results

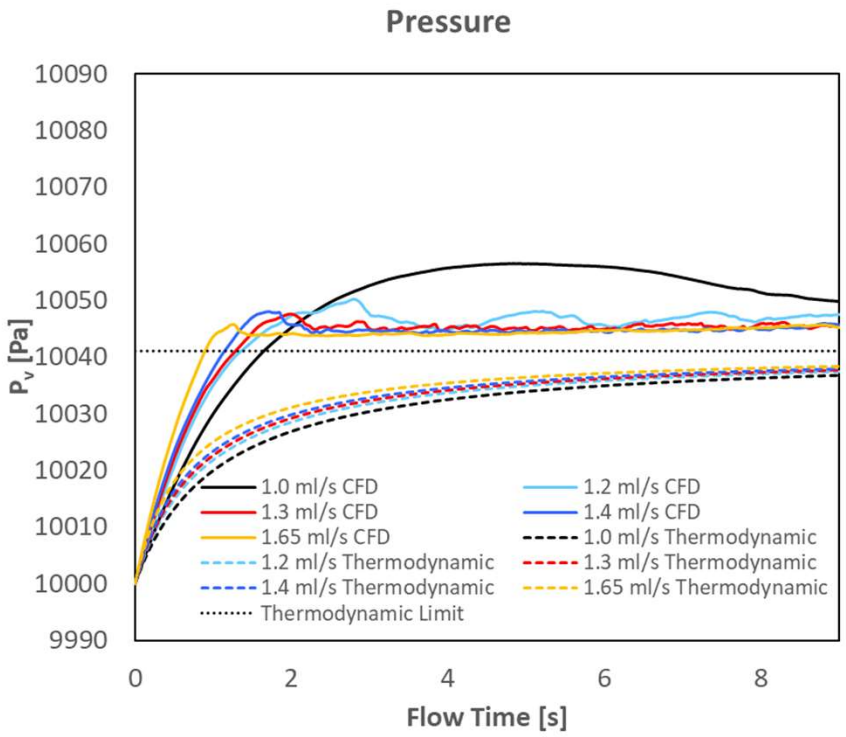


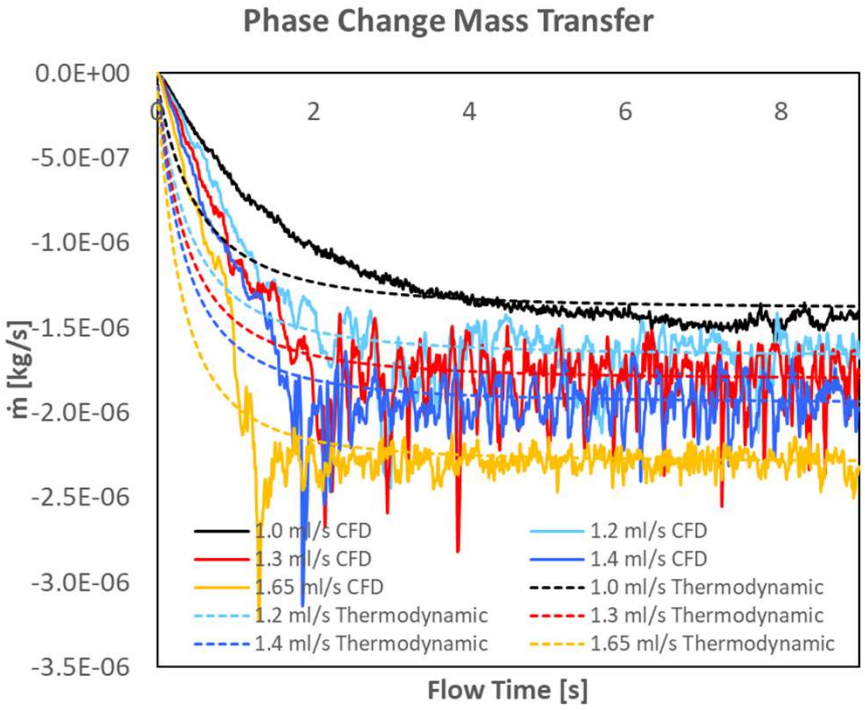
2D CFD Model – Results





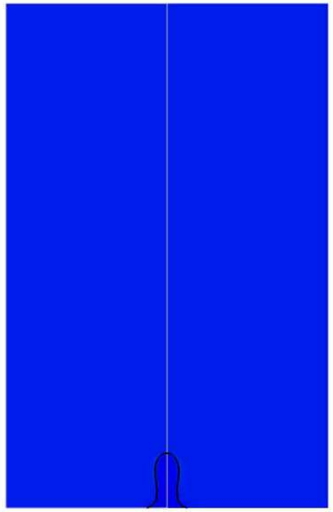
2D CFD Model – Poiseuille Inlet Profile





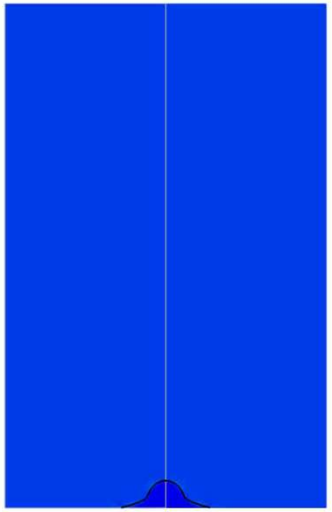
2D CFD Model – Inlet Velocity Profile

Time = 0.1 seconds

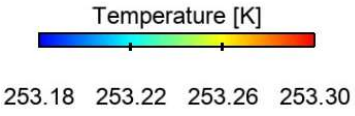


2D Poiseuille Inlet Profile
1.4 ml/s

Time = 0.2 seconds

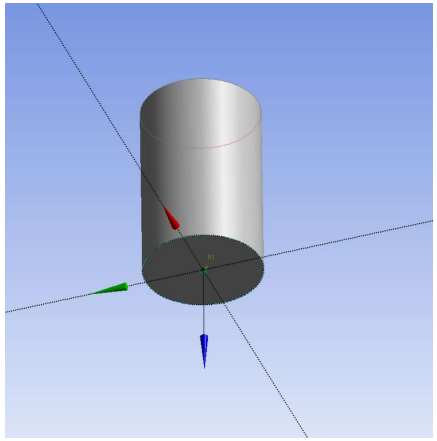


2D Flat Inlet Profile
1.4 ml/s



3D CFD Model – Setup

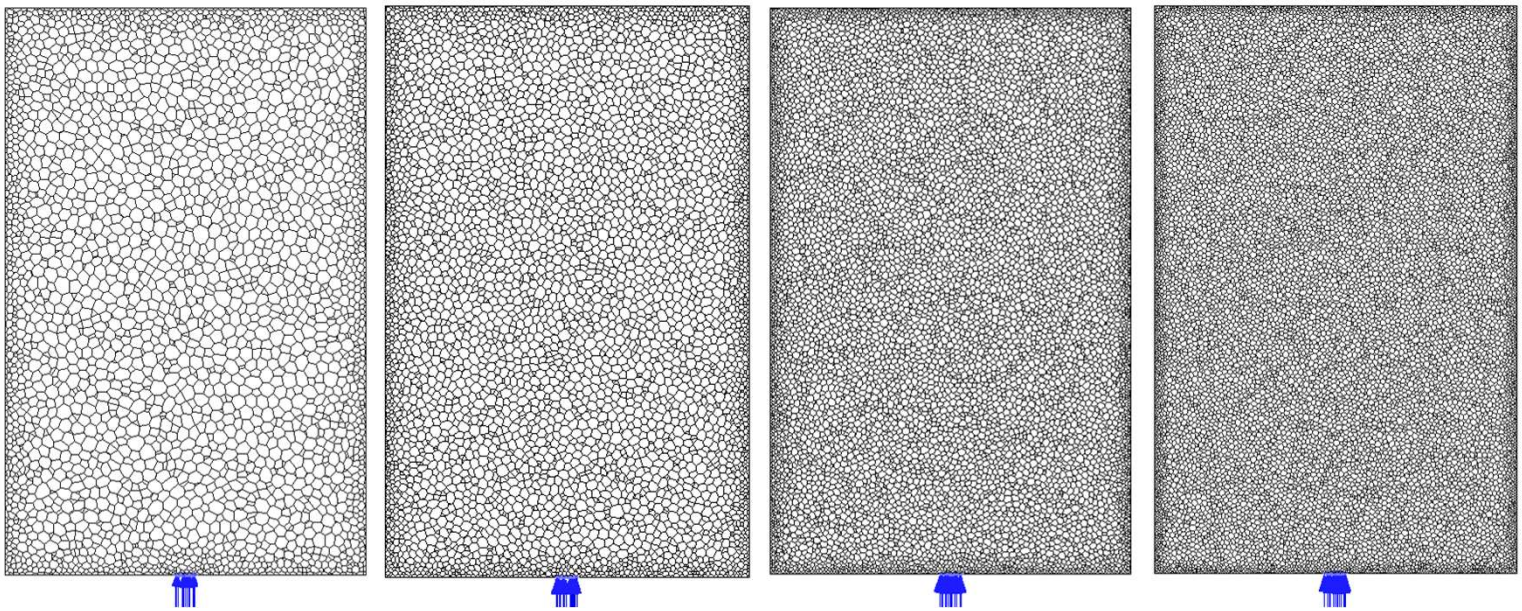
- 3D transient CFD model
- Problem Setup
  - 9 seconds of flow time
  - Microgravity -9.81E-6 m/s^2
- Solver Settings
  - Volume of Fluid (VOF) scheme with explicit formulation
  - Sharp interface modeling
  - Laminar fluid flow
  - Pressure-based solver with absolute velocity formulation
  - Phase change at interfaces based on the Shrage model
  - PISO solving method with least squares cell based gradient
  - PRESTO! pressure formulation
  - Compressive scheme for volume fraction
  - Second order upwind density, momentum, and energy
  - First order implicit transient formulation
- Geometry and mesh
  - Tank height is 0.094 m, tank radius is 0.03 m, and inlet is at the center of the bottom boundary with a radius of 0.002 m
  - Adiabatic walls with wall adhesion
  - Poly mesh, varied mesh size
- Initial Conditions
  - PnP vapor
  - 283.18 K and 10,000 Pa.



Properties of PnP (Perfluoro-normal-pentane C_5F_{12}) via DIPPR database at 253.18 K:

Property	Liquid	Vapor
Density	Boussinesq, 1743.919 kg/m^3	Ideal gas
Specific Heat	1131.825 J/kg K	692.049 J/kg K
Thermal Conductivity	0.0697077 W/m K	0.00729833 W/m K
Viscosity	9.0238E-4 kg/m s	9.5936E-6 kg/m s
Molecular Weight	288.034 kg/kmol	288.034 kg/kmol
Standard State Enthalpy	0 J/kmol	2.4375E7 J/kmol
Reference Temperature	298.15 K	298.15 K
Thermal Expansion Coefficient	0.00171415 K^{-1}	
Surface Tension Coefficient	0.0140107 N/m	
Wall Contact Angle	0°	

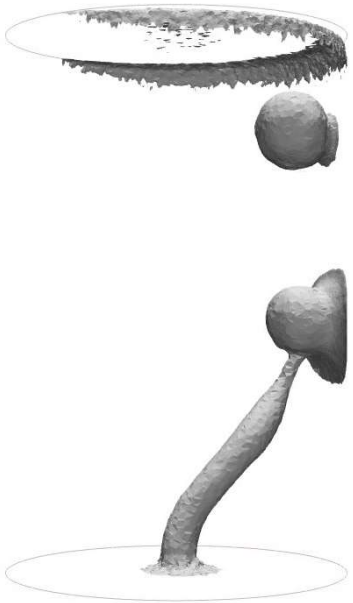
3D CFD Model – Uniform Mesh Study



1E-3 m elements 125k cells	7.5E-4 m elements 253k cells	5E-4 m elements 706k cells	4E-4 m elements 1.25m cells

3D CFD Model – Uniform Mesh Study

1E-3 m elements
125k cells

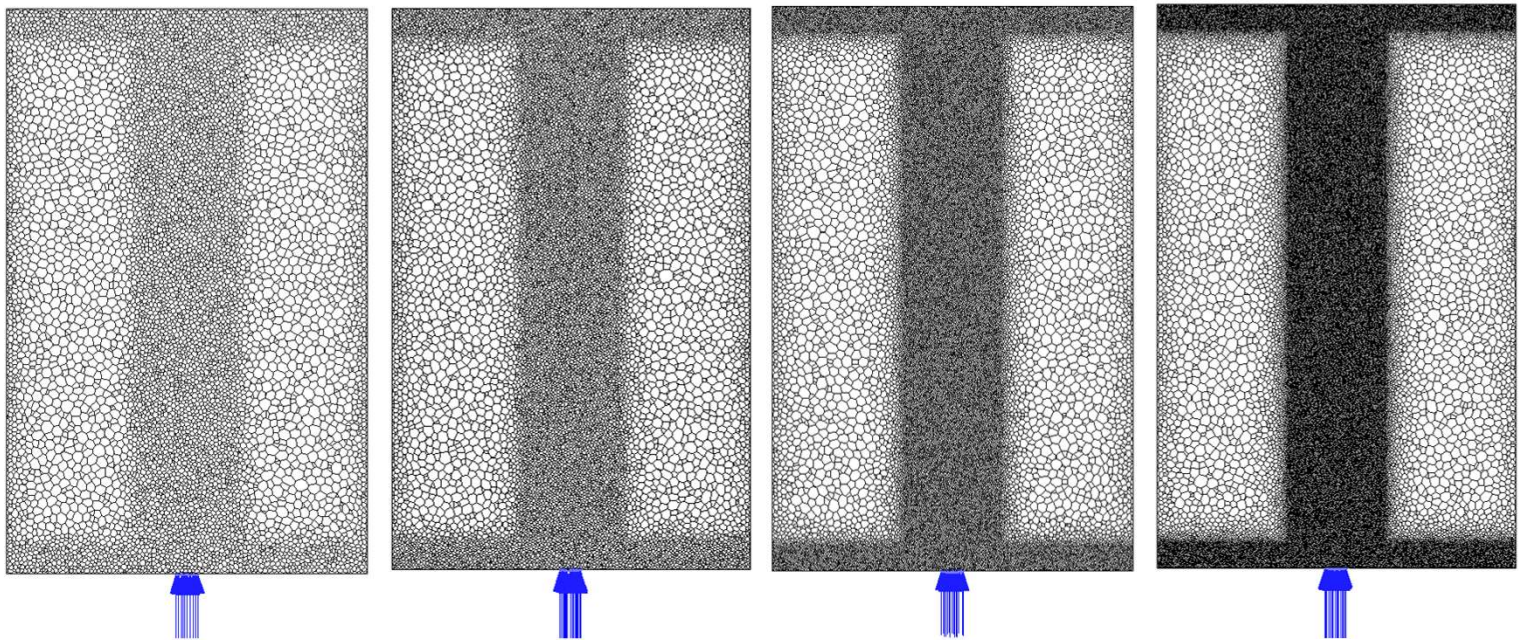


7.5E-4 m elements
253k cells

5E-4 m elements
706k cells

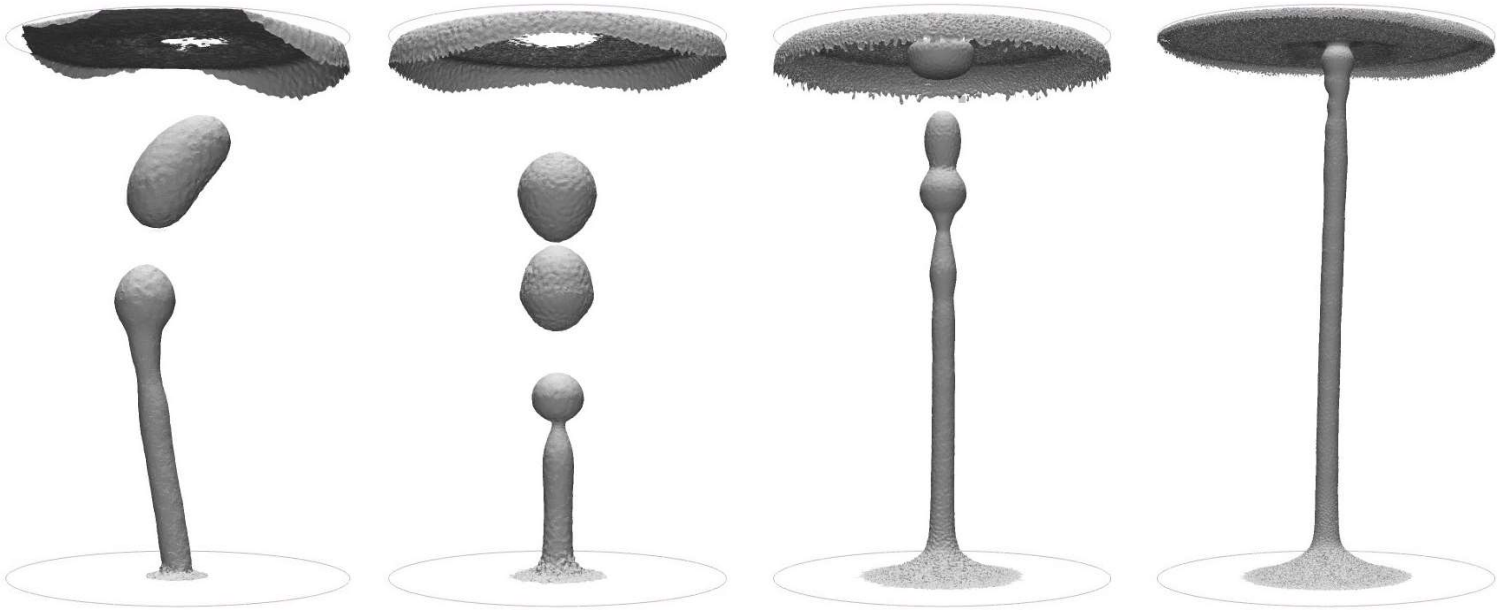
4E-4 m elements
1.25m cells

3D CFD Model – Region of Interest Refined Mesh Study



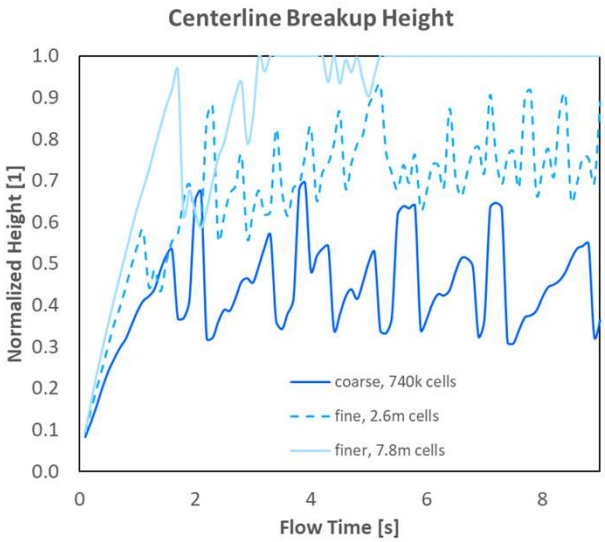
Coarser	Coarse	Fine	Finer
7E-4 m ctr elements	5E-4 m ctr elements	3E-4 m ctr elements	2E-4 m ctr elements
380k cells	740k cells	2.6m cells	7.8m cells

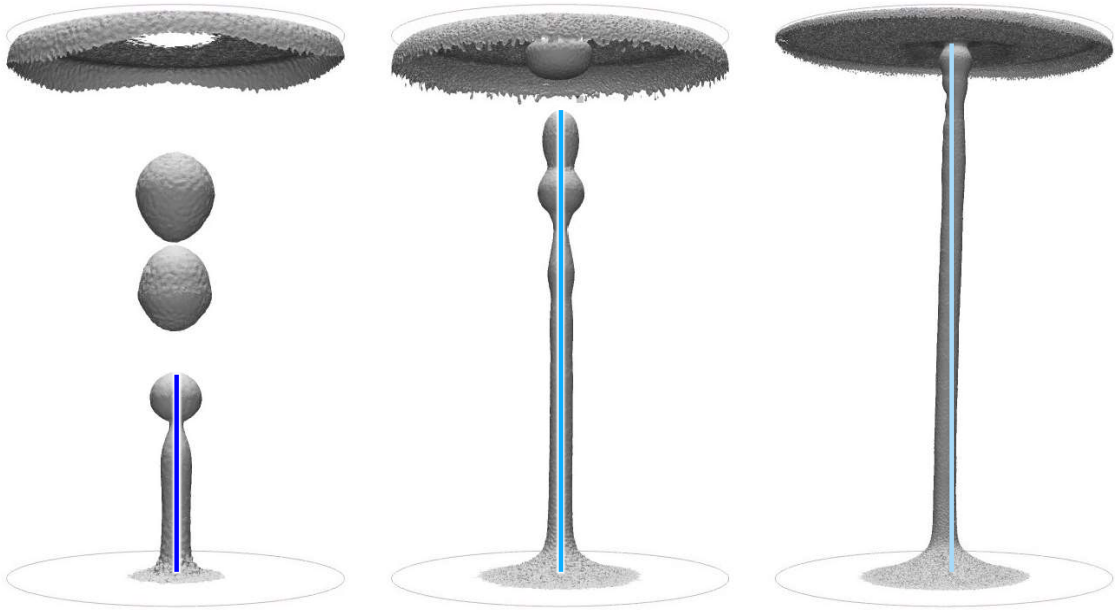
3D CFD Model – Region of Interest Refined Mesh Study



Coarser	Coarse	Fine	Finer
7E-4 m ctr elements	5E-4 m ctr elements	3E-4 m ctr elements	2E-4 m ctr elements
380k cells	740k cells	2.6m cells	7.8m cells

3D CFD Model – Region of Interest Refined Mesh Study



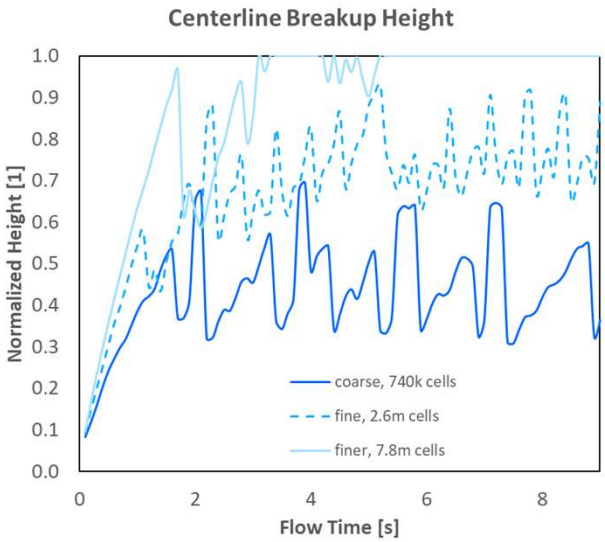


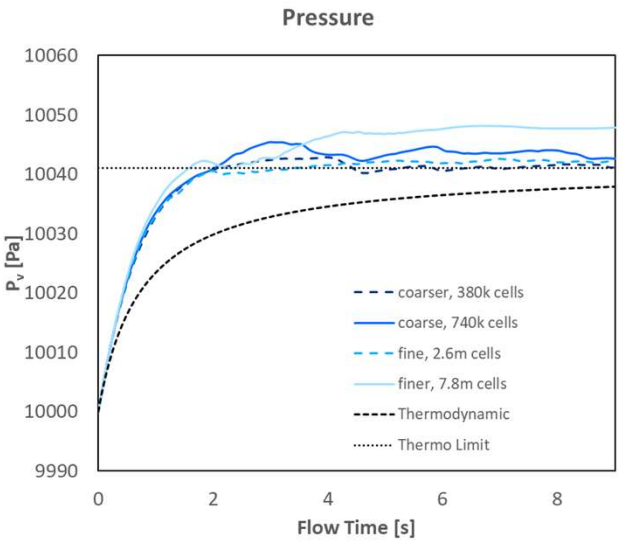
Coarse
5E-4 m ctr elements
740k cells

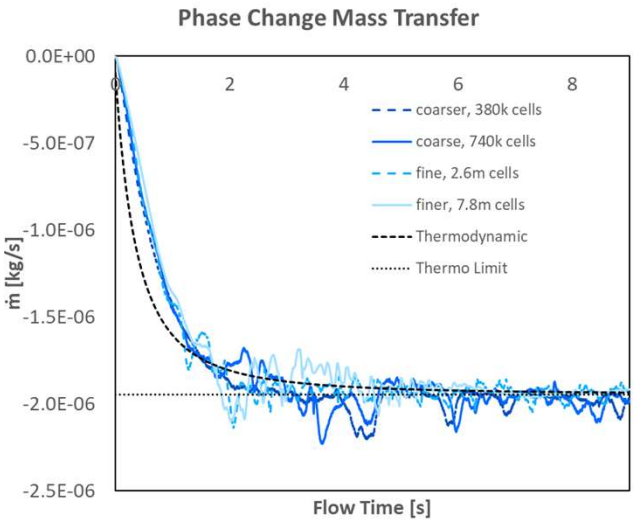
Fine
3E-4 m ctr elements
2.6m cells

Finer
2E-4 m ctr elements
7.8m cells

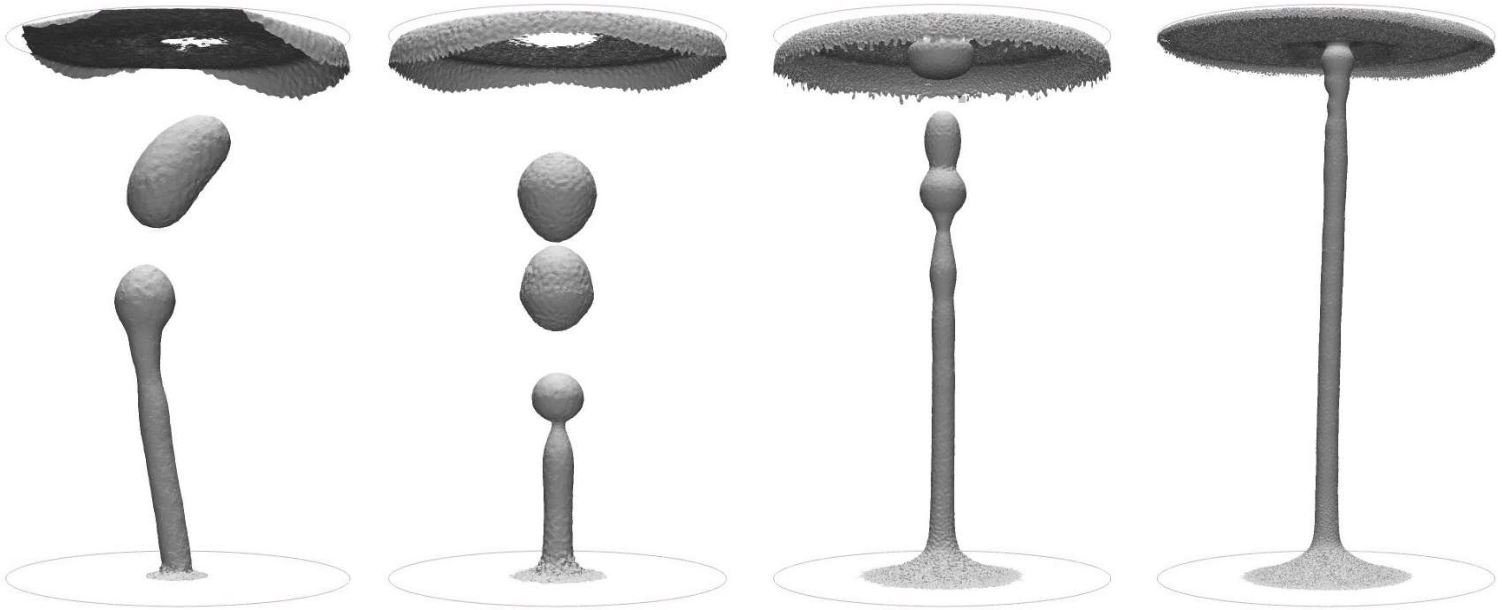
3D CFD Model – Region of Interest Refined Mesh Study







3D CFD Model – Region of Interest Refined Mesh Study



Coarser	Coarse	Fine	Finer
7E-4 m ctr elements	5E-4 m ctr elements	3E-4 m ctr elements	2E-4 m ctr elements
380k cells	740k cells	2.6m cells	7.8m cells

3D CFD Model – Time Step Study

1E-3 s time step
Largest step size

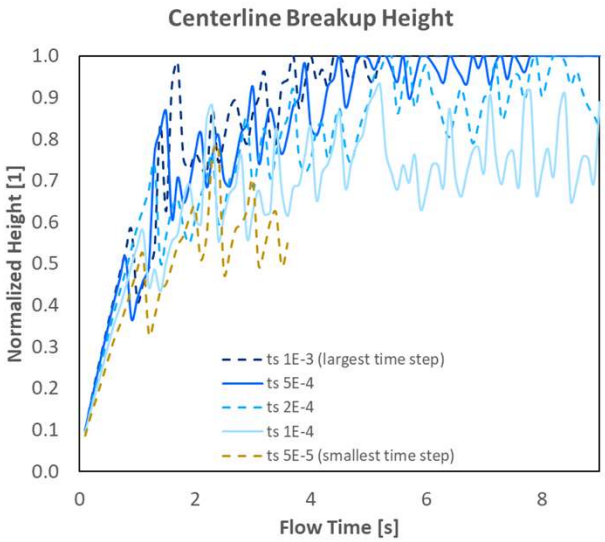
5E-4 s time step

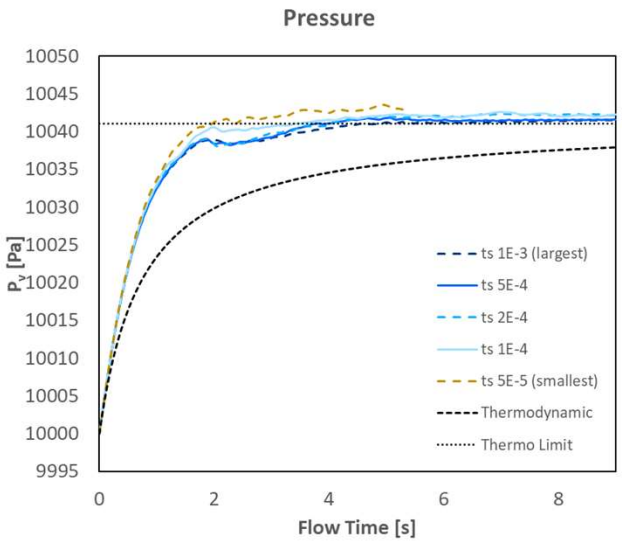
2E-4 s time step

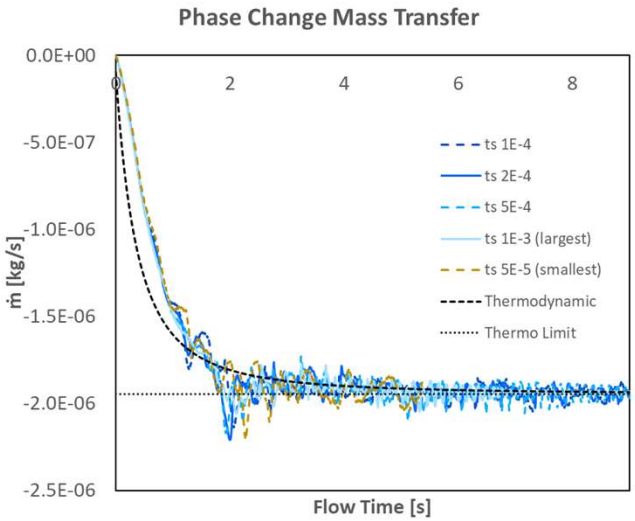
1E-4 s time step

5E-5 s time step
Smallest step size

3D CFD Model – Time Step Study







3D CFD Model – Time Step Study

1E-3 s time step
Largest step size

5E-4 s time step

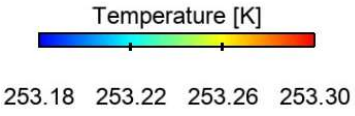
2E-4 s time step

1E-4 s time step

5E-5 s time step
Smallest step size

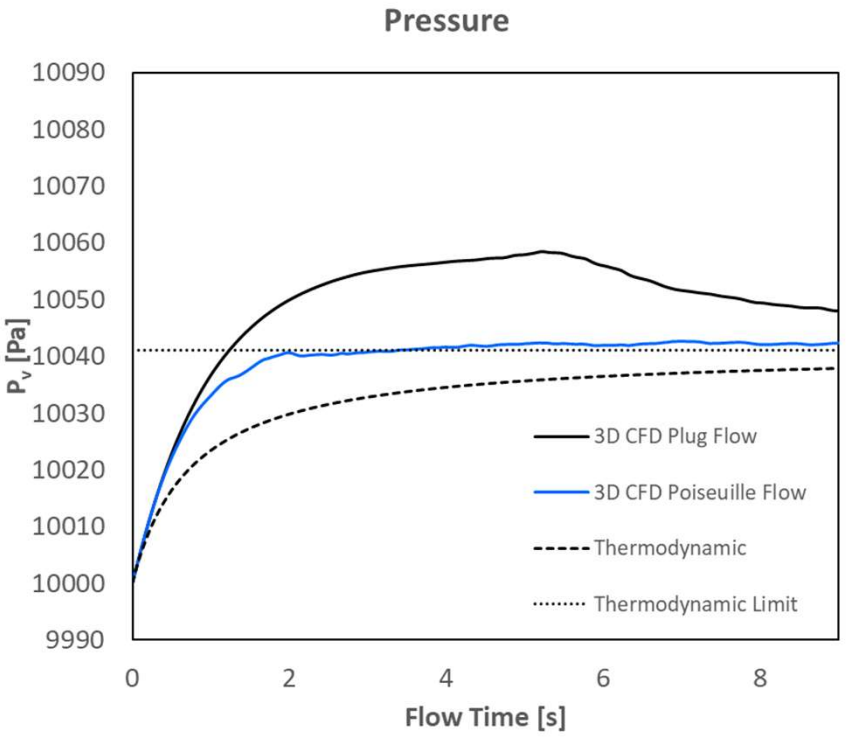
3D CFD Model – Inlet Velocity Profile

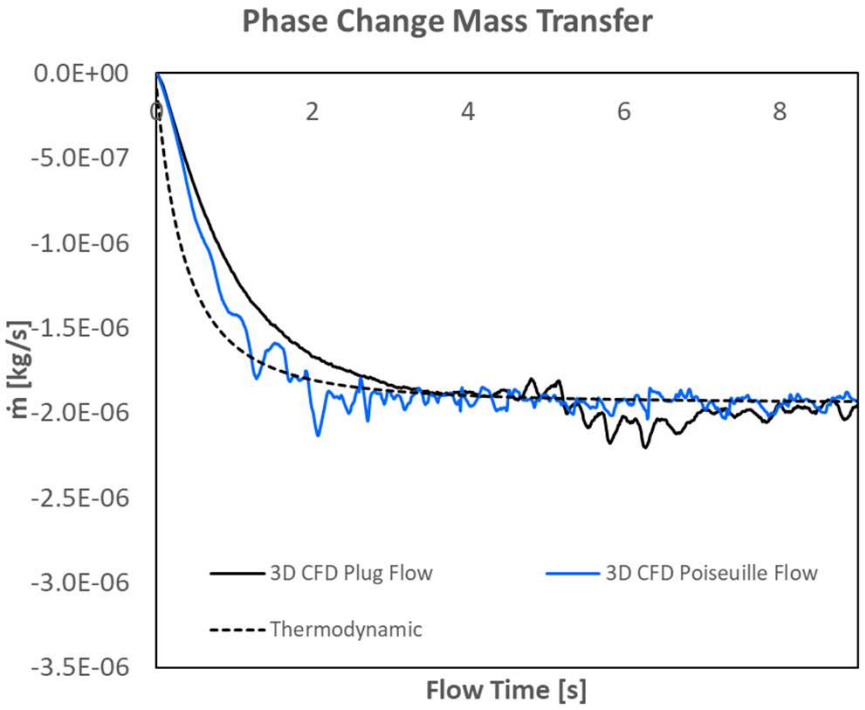
3D Plug Flow Inlet Profile
1.4 ml/s



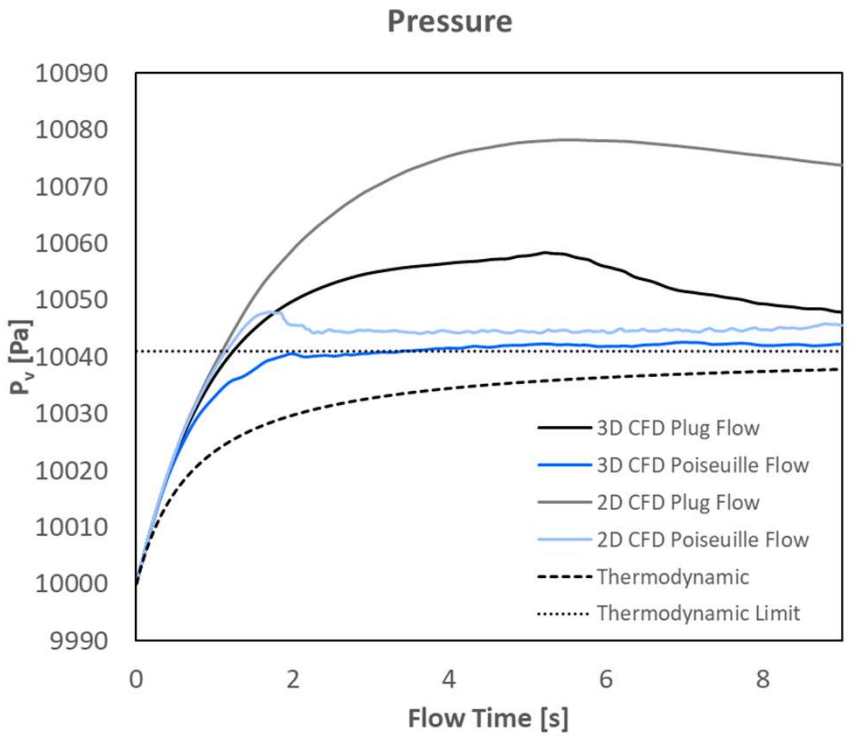
3D Poiseuille Inlet Profile
1.4 ml/s

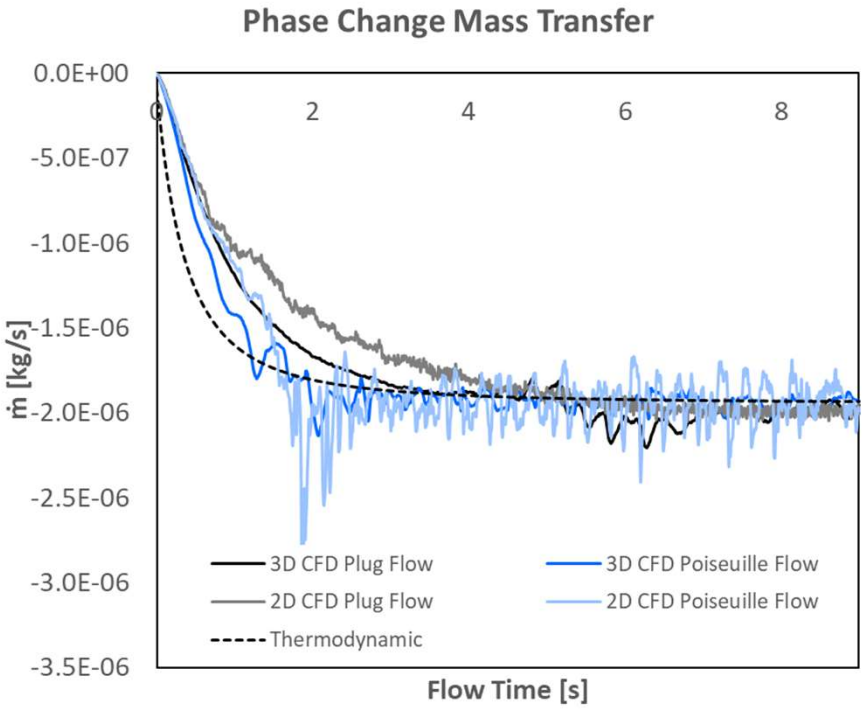
3D CFD Model – Inlet Velocity Profile



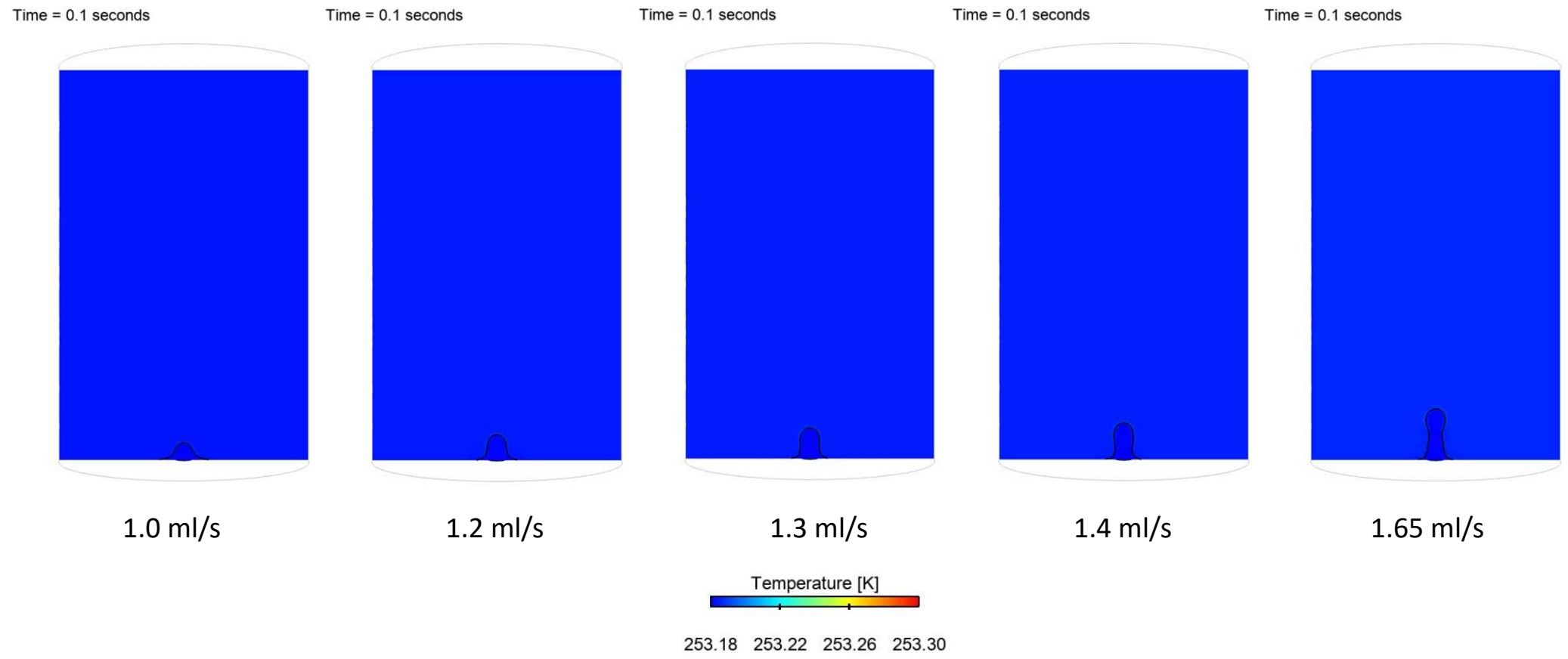


3D CFD Model – Inlet Velocity Profile

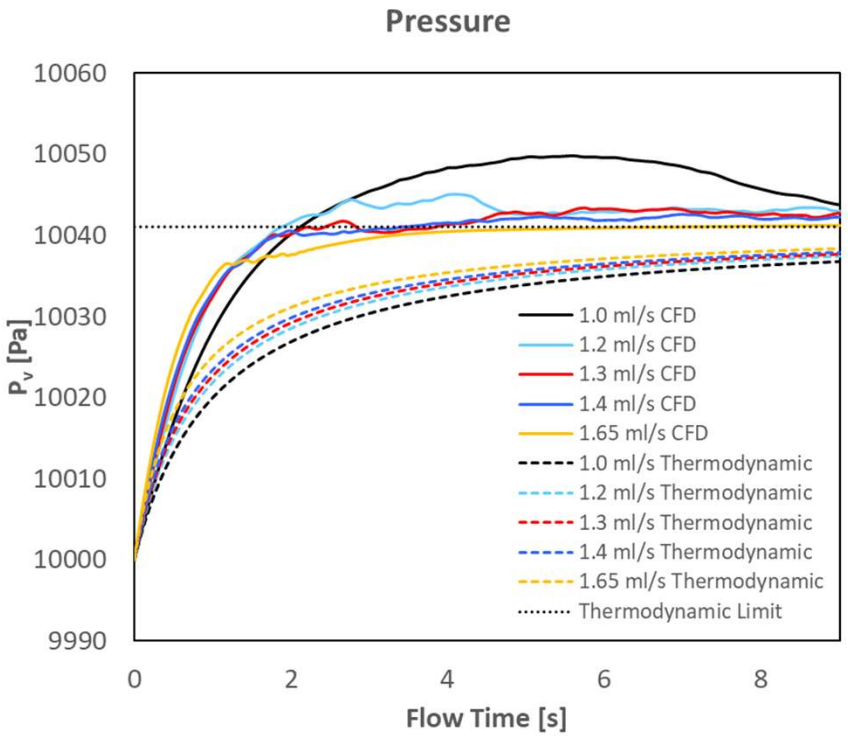


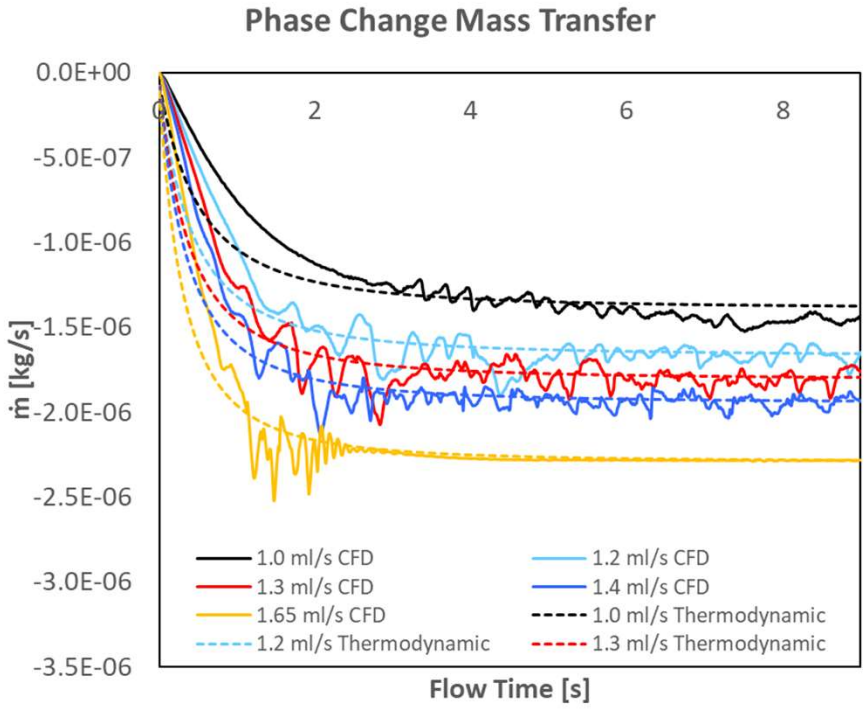


3D CFD Model – Results

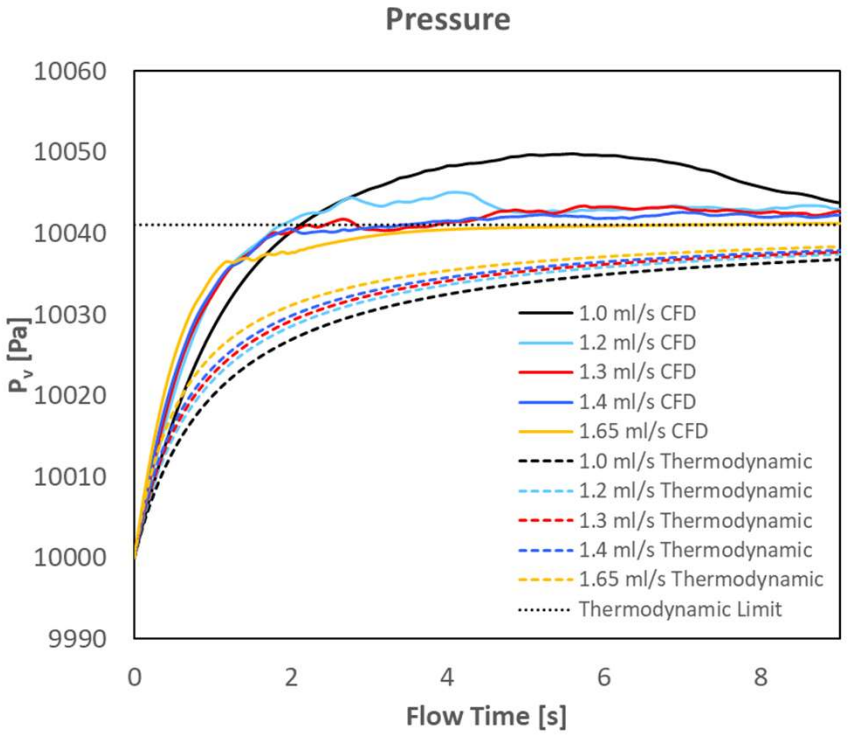


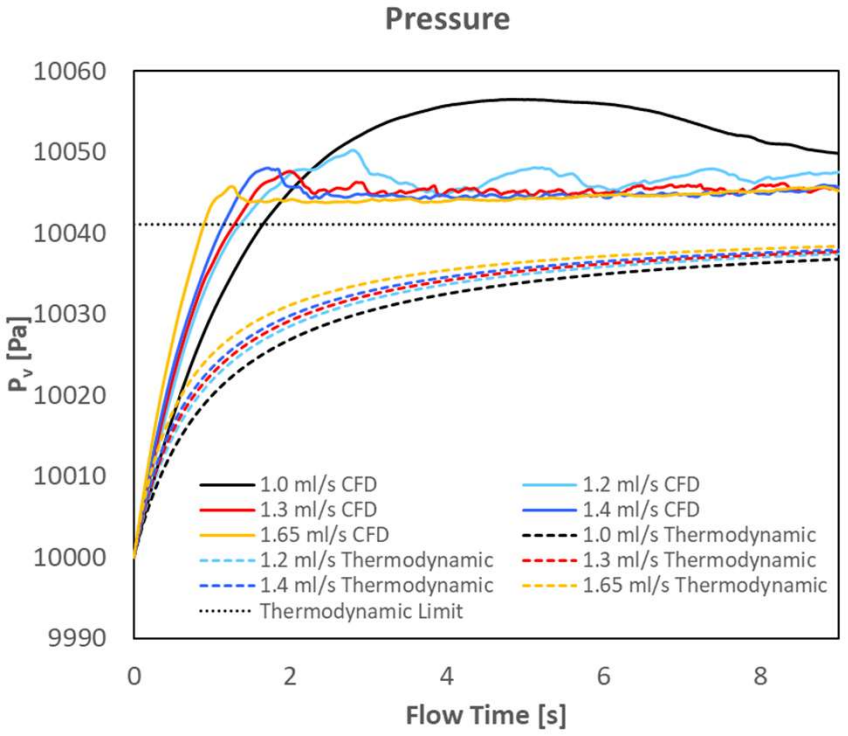
3D CFD Model – Results



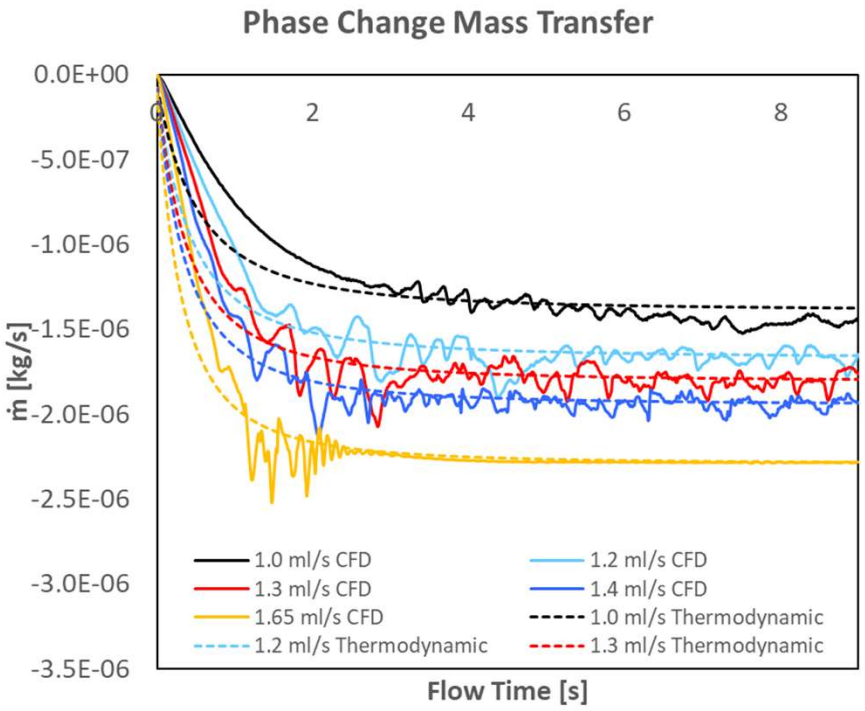


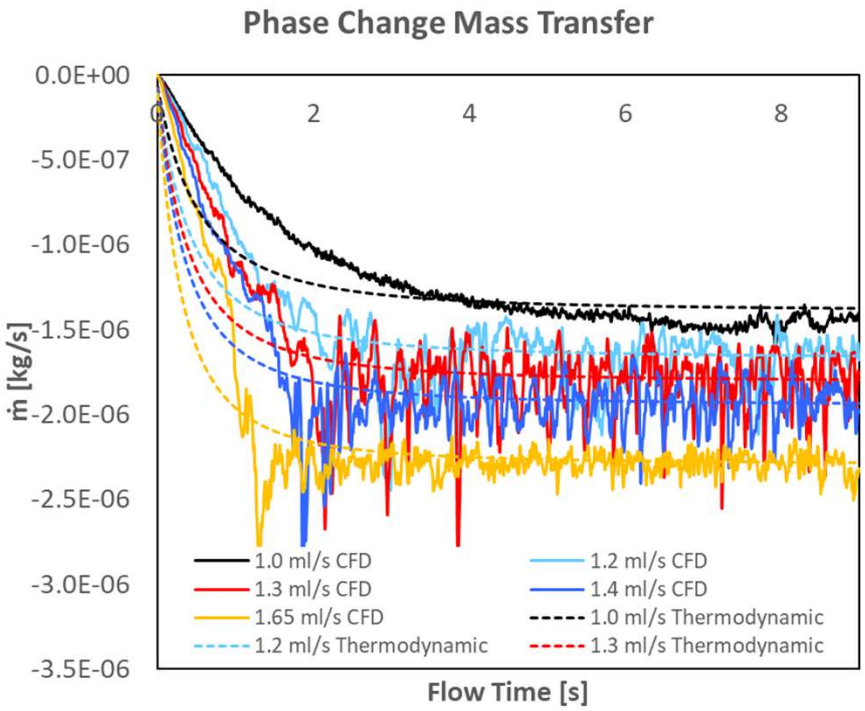
3D CFD Model – Results





3D CFD Model – Results





Conclusions

- Summary
  - Developed a thermodynamic model, a 2D axisymmetric CFD model, and a 3D CFD model
  - Performed parametric studies investigating effects of inlet flow rate and inlet velocity profile
  - Demonstrated agreement between CFD and thermodynamic models of both the final pressure and mass transfer rates
- Findings
  - The 2D axisymmetric model is limited in its ability to capture the full geyser behavior, despite the symmetric geometry and boundary conditions of the tank, potentially because of:
    - Computational errors at the axis
    - Inherently three-dimensional nature of the geyser flow pattern
  - The CFD results predict a steeper pressure rise in the early transitory phase compared to thermodynamic
  - Geyser behavior is highly affected by the inlet velocity profile
- Future
  - When corresponding experiments are completed, will validate the model against the results of the experiments in microgravity for different inlet flow rates

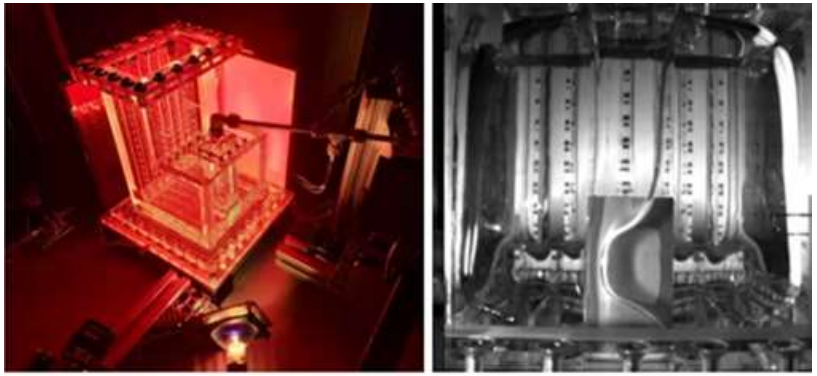
Acknowledgements

- This work is supported by NASA Biological and Physical Sciences (BPS) Division.
- I would like to thank principal investigator Mohammad Kassemi, fellow researchers Olga Kartuzova and Sonya Hilton, and collaborators Michael Dreyer, Vishwas Kulkarni, and the rest of the Dreyer Lab at the University of Bremen.

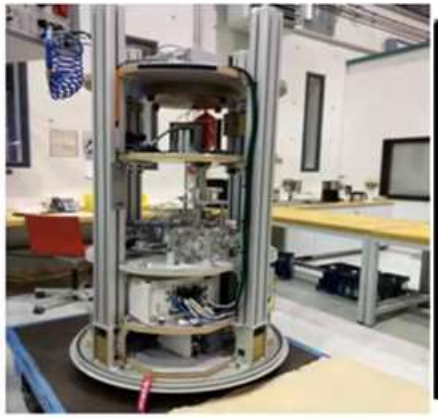
Backup Slides

Background of ZBOT-FT

- ZBOT-FT (Filling & Transfer) Experiment was initiated as a collaboration between DLR, NASA, UoB, and CWRU at the 2018 AIC conference in Bremen
- The following roles were established:
  - Ground-based Phase (2021)
    - DLR: Funds the Drop Tower Hardware build-up and Experiment
    - UoB: Performs ground-based experiments (drop tower and parabolic flight)
    - NASA: Funds the CFD model development
    - CWRU/NASA Team: Performs CFD model development and validation
  - Flight Phase (2026)
    - DLR: Funds the Flight Hardware build-up
    - NASA: Funds all aspects of integration of the experiment into ISS and directs the review process
    - UoB & CWRU: Prepares the SRD and all the documents for the various review processes
    - UoB: Works with the German hardware builder
    - CWRU/NASA Team: Supports with CFD modeling and ISS operations



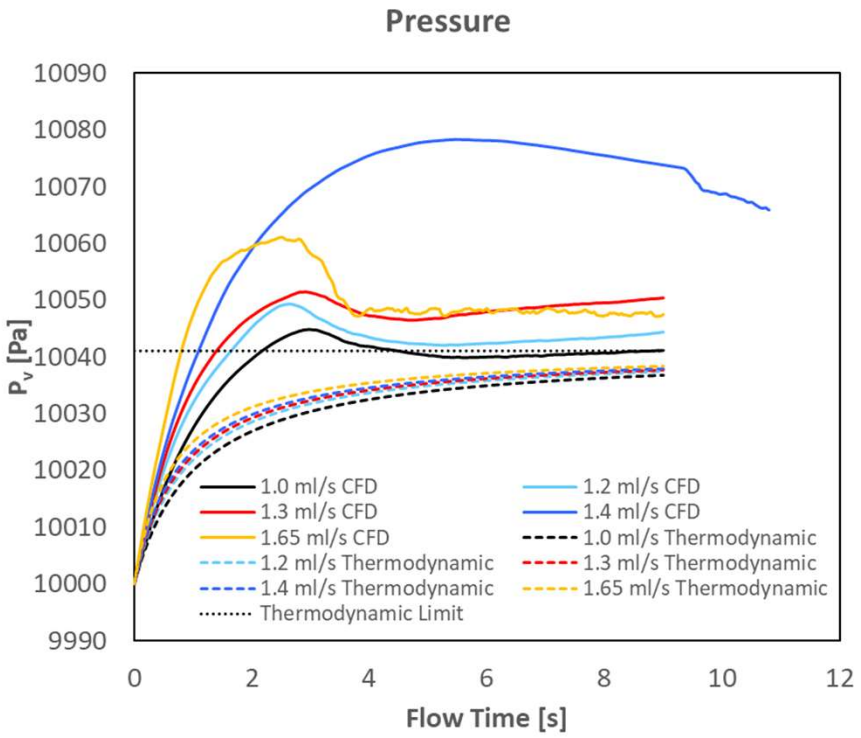
LAD performance tests in Donor Tank

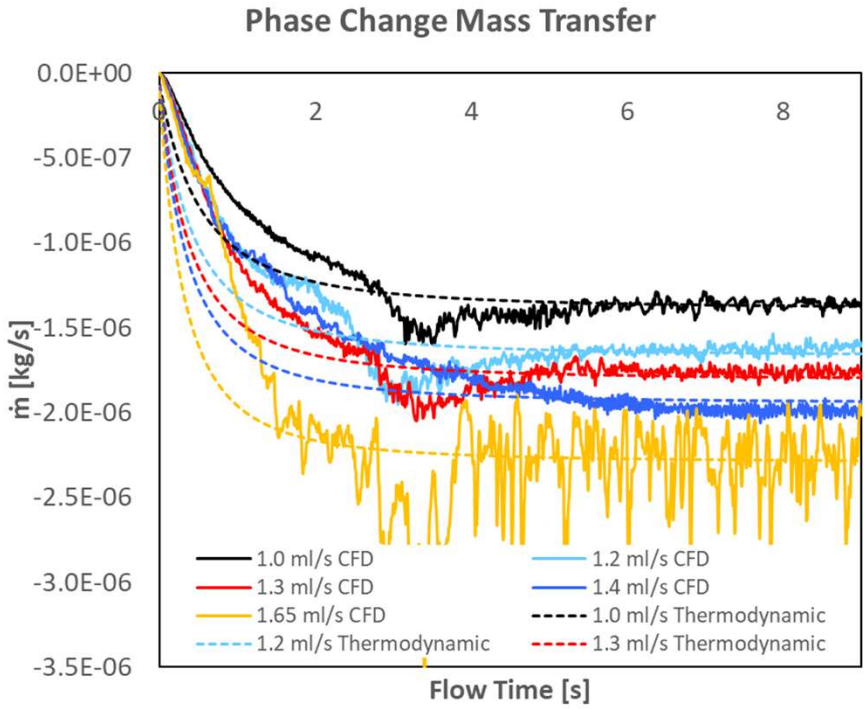


Receiver Tank in the Drop Tower Assembly – tank filling tests

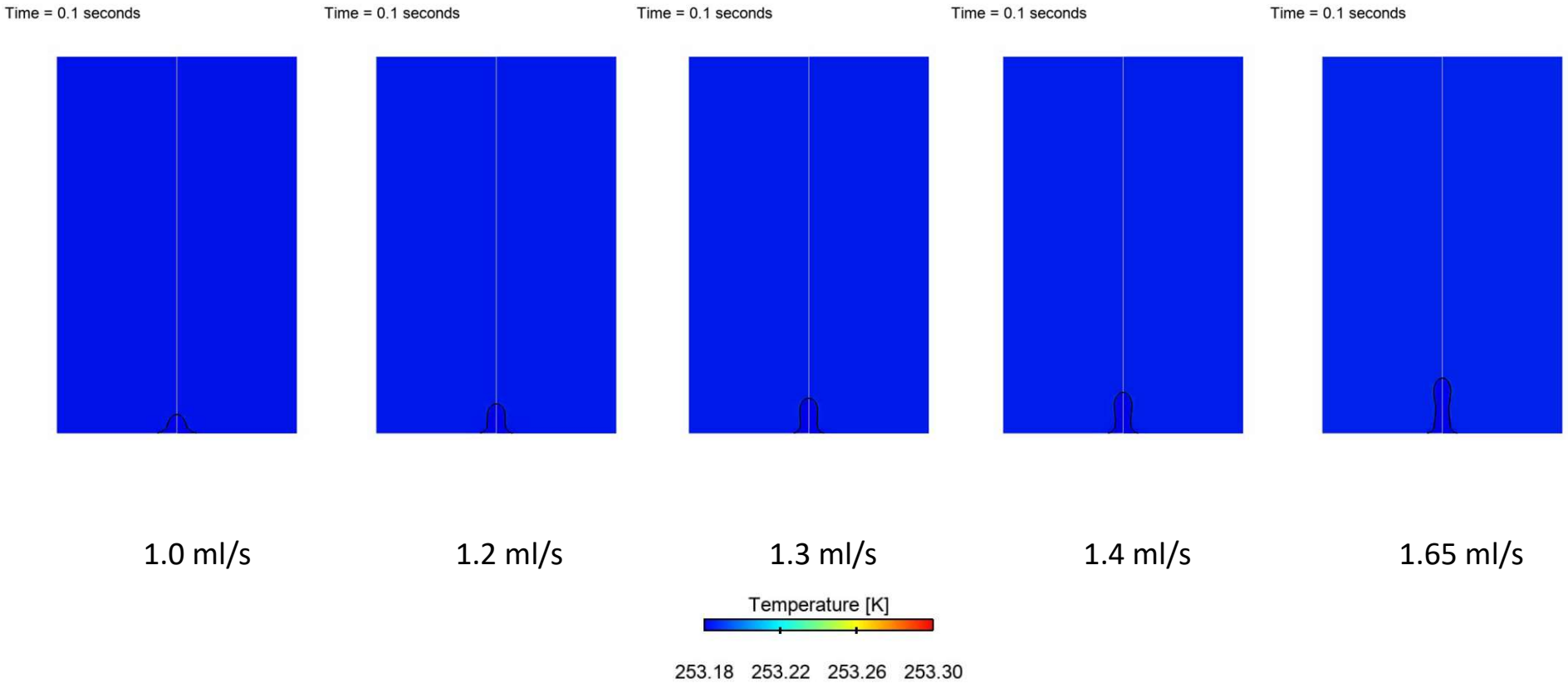
Drop Tower cavitation tests with LCH4

2D CFD Model – Results



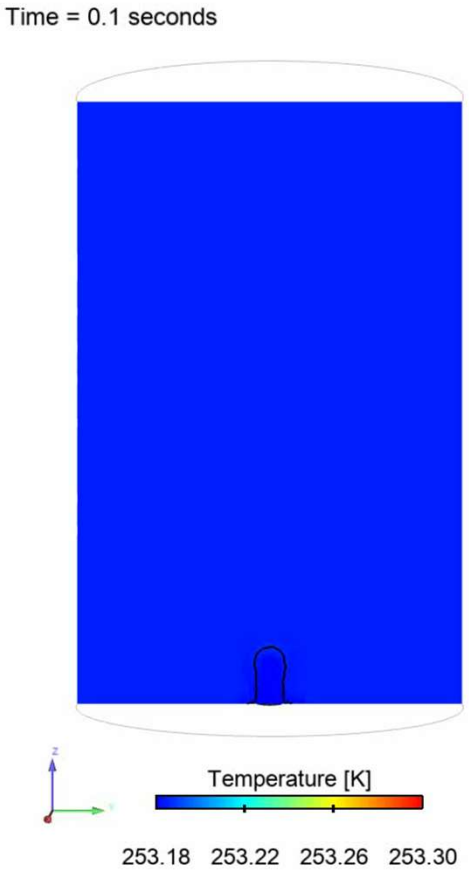


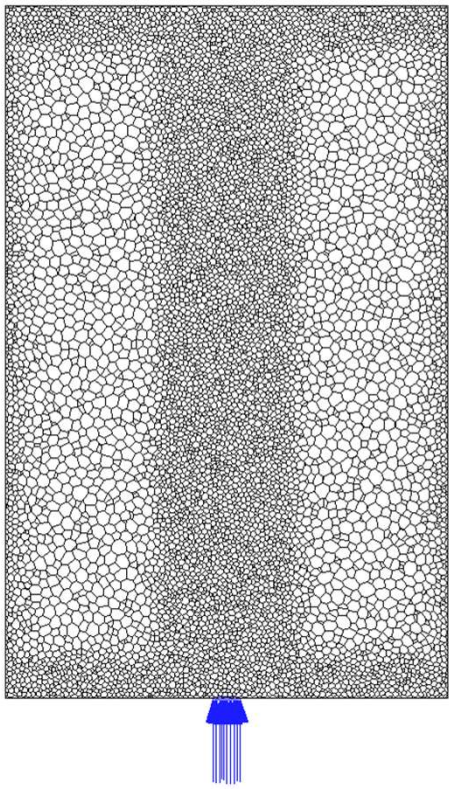
2D CFD Model – Poiseuille Inlet Profile



3D CFD Model – Mesh Study

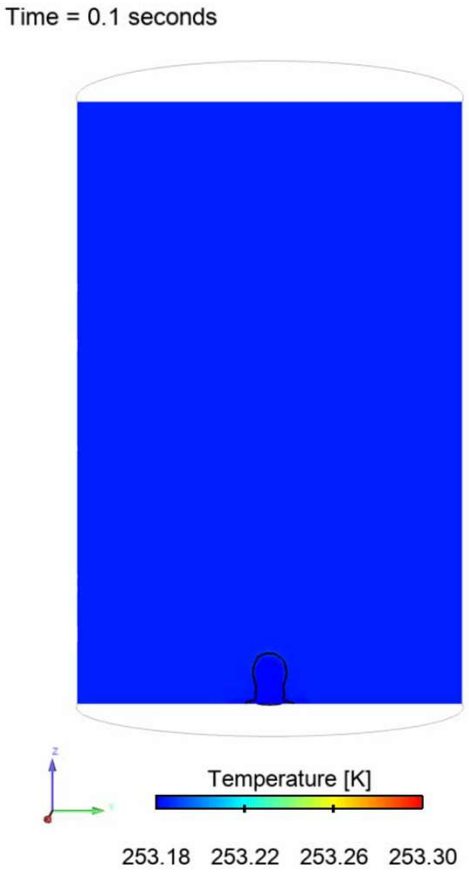
- Coarser mesh has 390,000 cells, and a central region element size of 7E-4 m

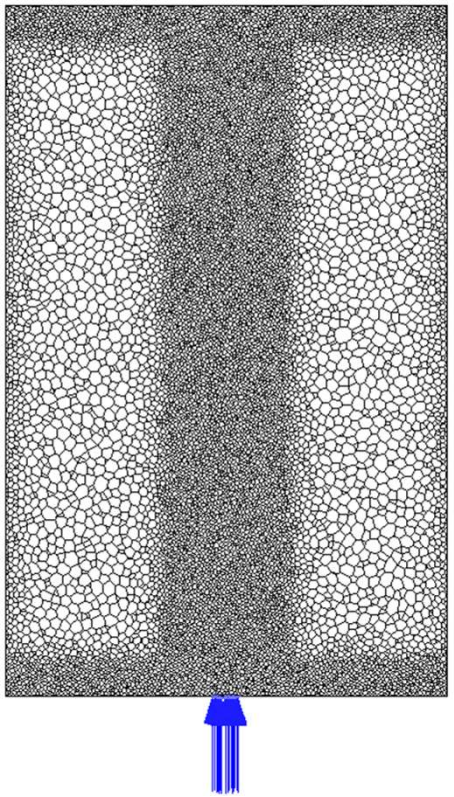




3D CFD Model – Mesh Study

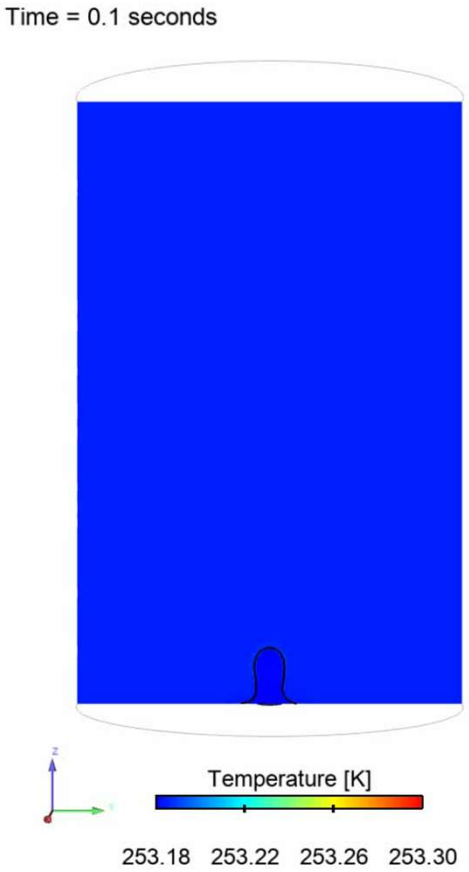
- Coarse mesh has 740,000 cells, and a central region element size of 5E-4 m

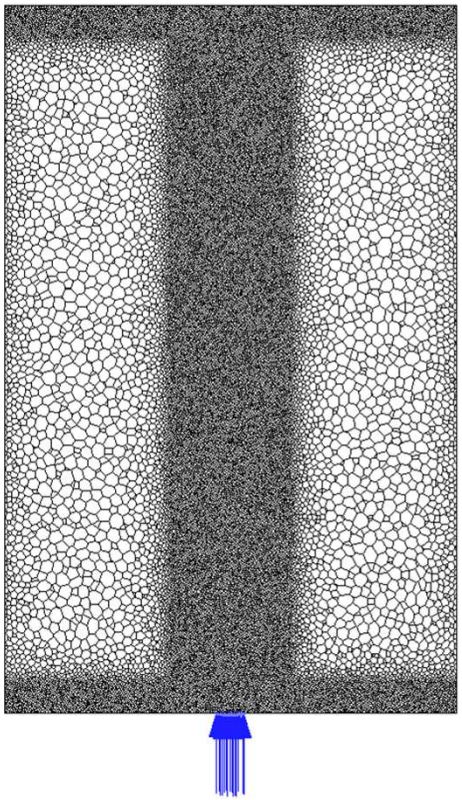




3D CFD Model – Mesh Study

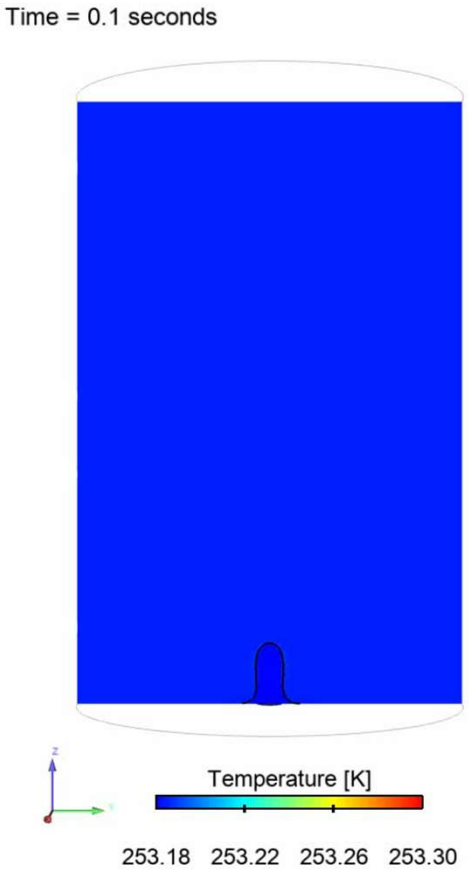
- Fine mesh has 2.6 million cells, and a central region element size of 3E-4 m

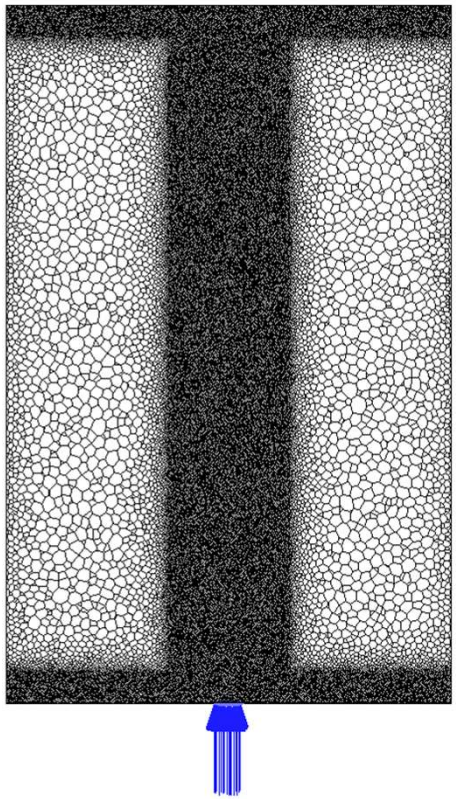




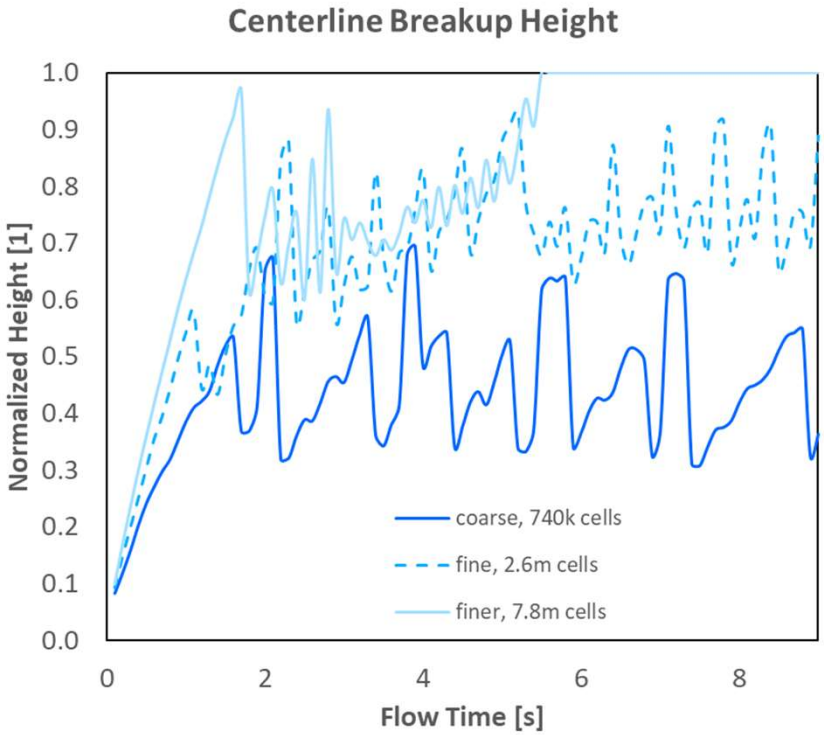
3D CFD Model – Mesh Study

- Finer mesh has 7.8 million cells, and a central region element size of 2E-4 m

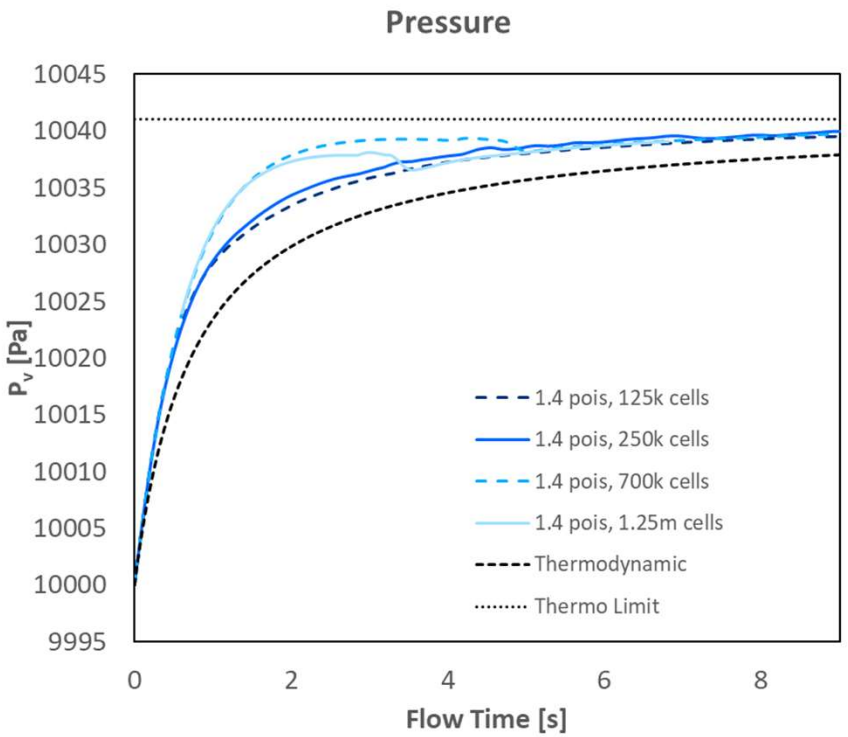


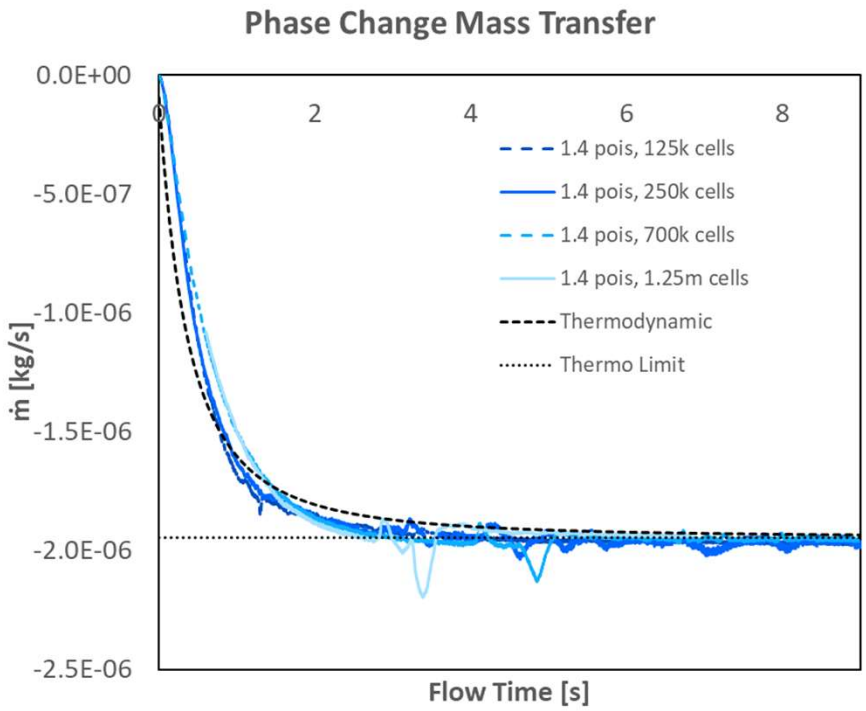


3D CFD Model – Region of Interest Refined Mesh Study

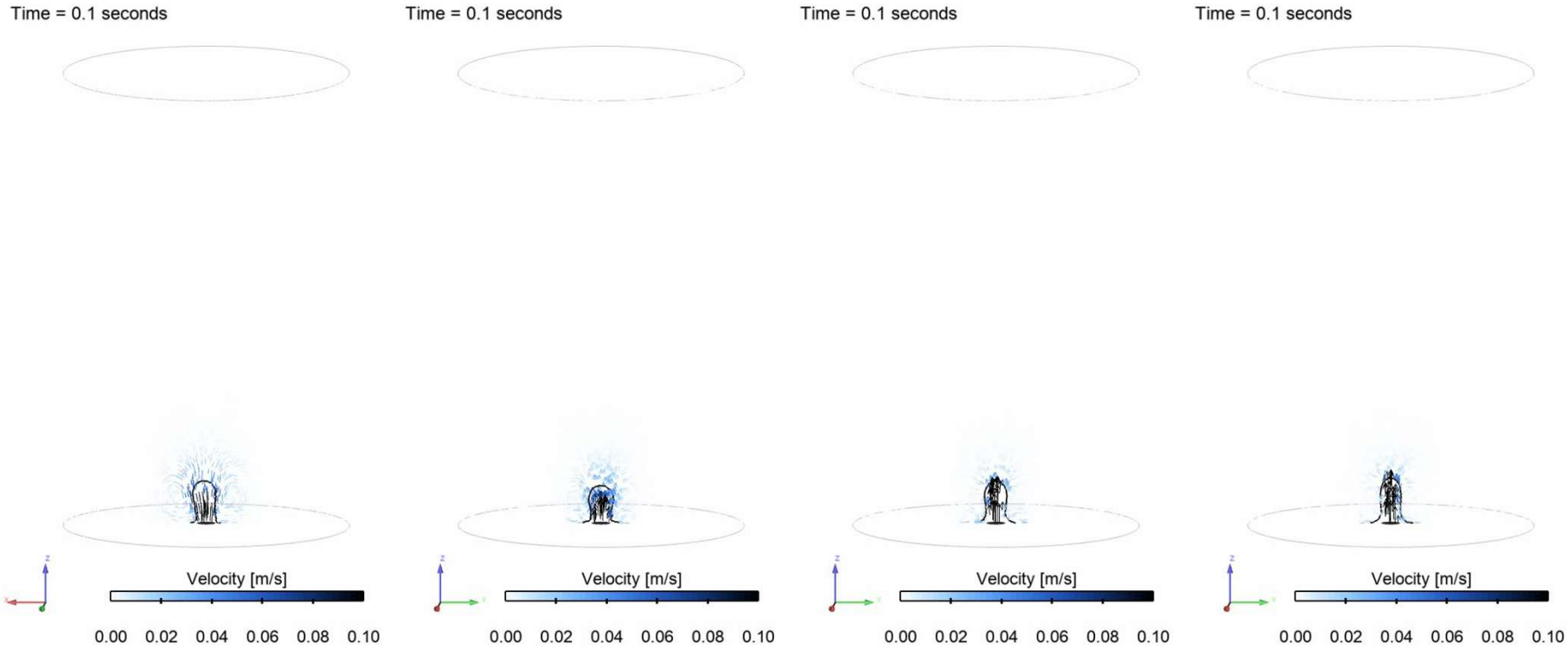


3D CFD Model – Uniform Mesh Study

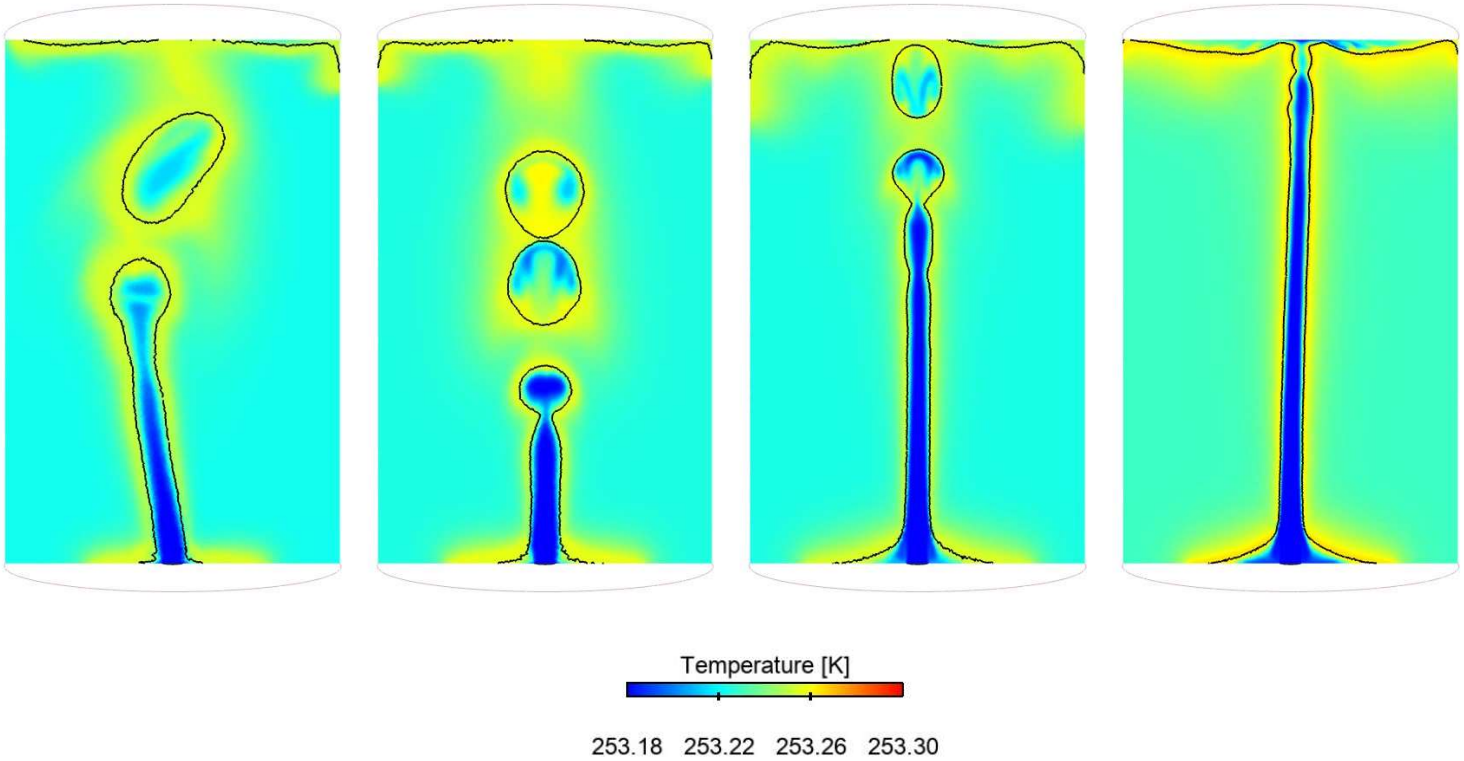




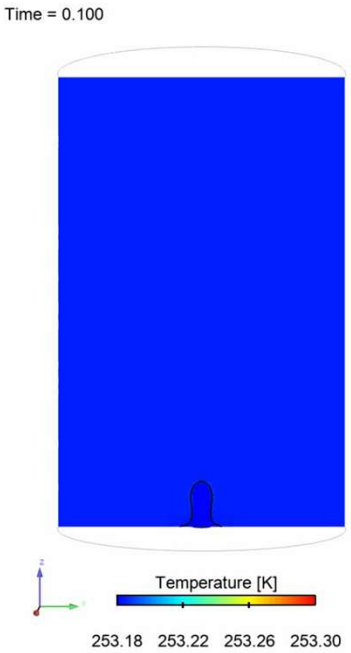
Region of Interest Refined Meshes



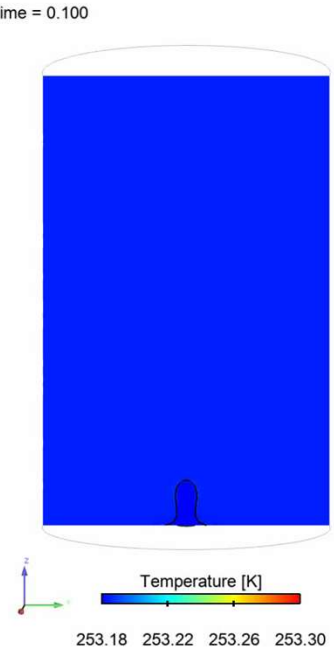
3D CFD Model – Region of Interest Refined Mesh Study



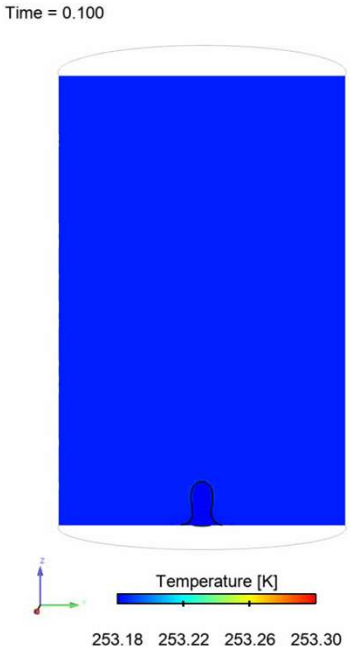
3D CFD Model – Time Step Study



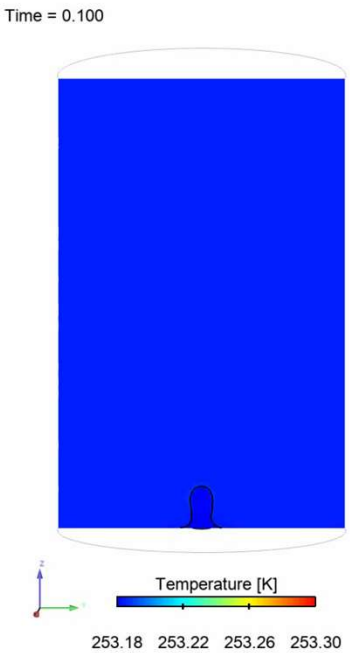
1E-3 s time step
Largest step size



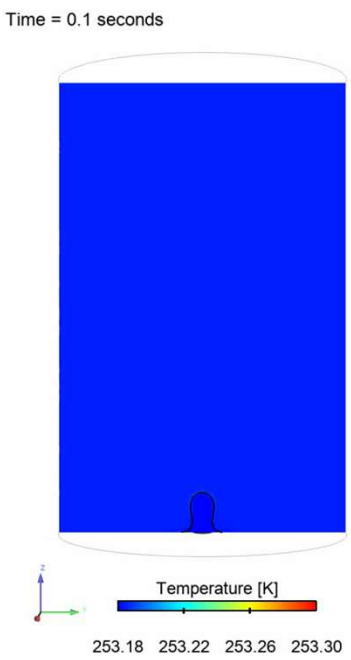
5E-4 s time step



2E-4 s time step



1E-4 s time step



5E-5 s time step
Smallest step size

3D CFD Model – Time Step Study

1E-3 s time step
Largest step size

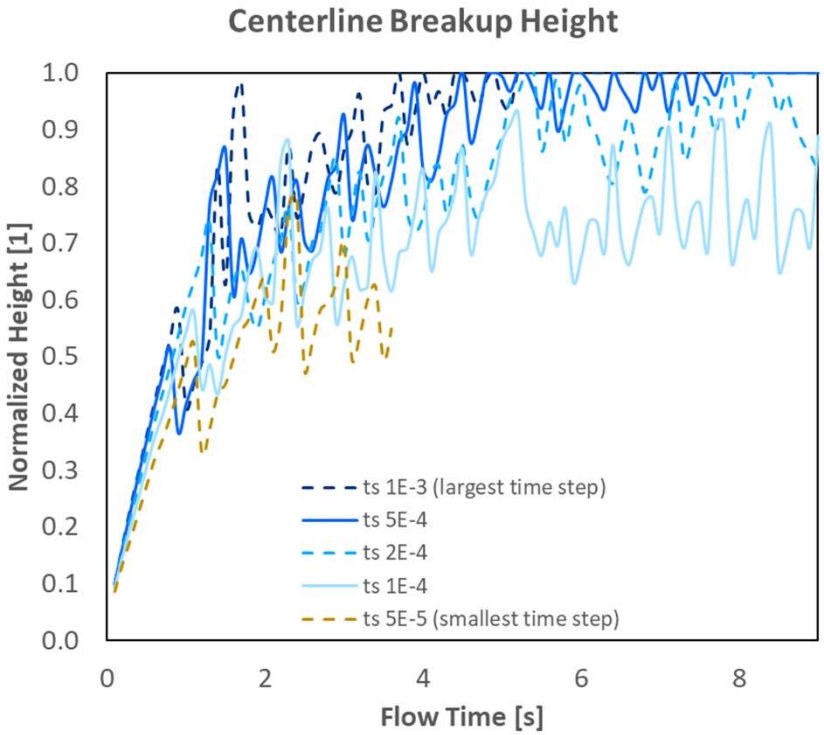
5E-4 s time step

2E-4 s time step

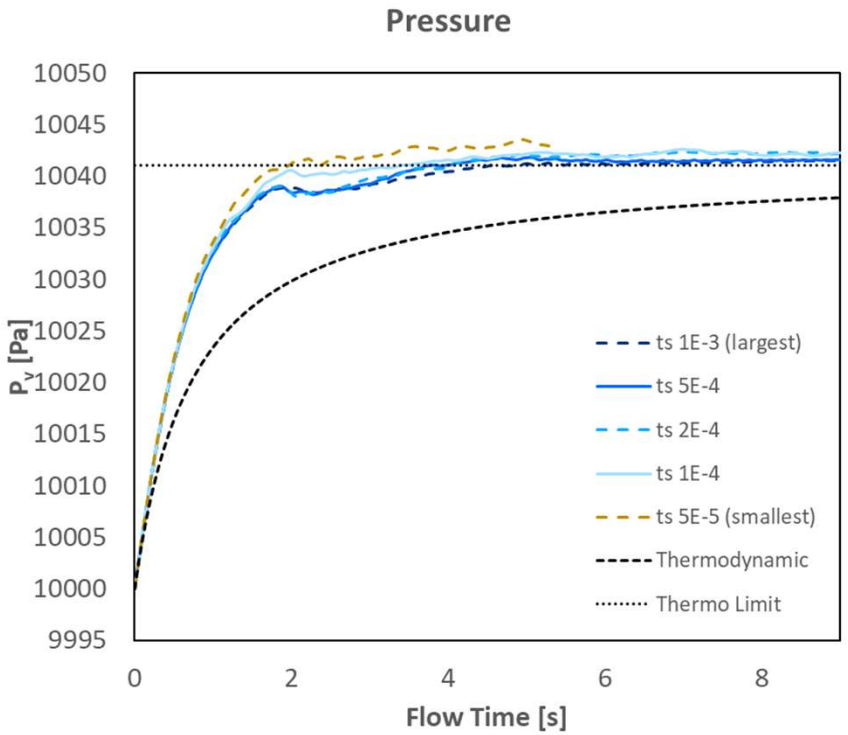
1E-4 s time step

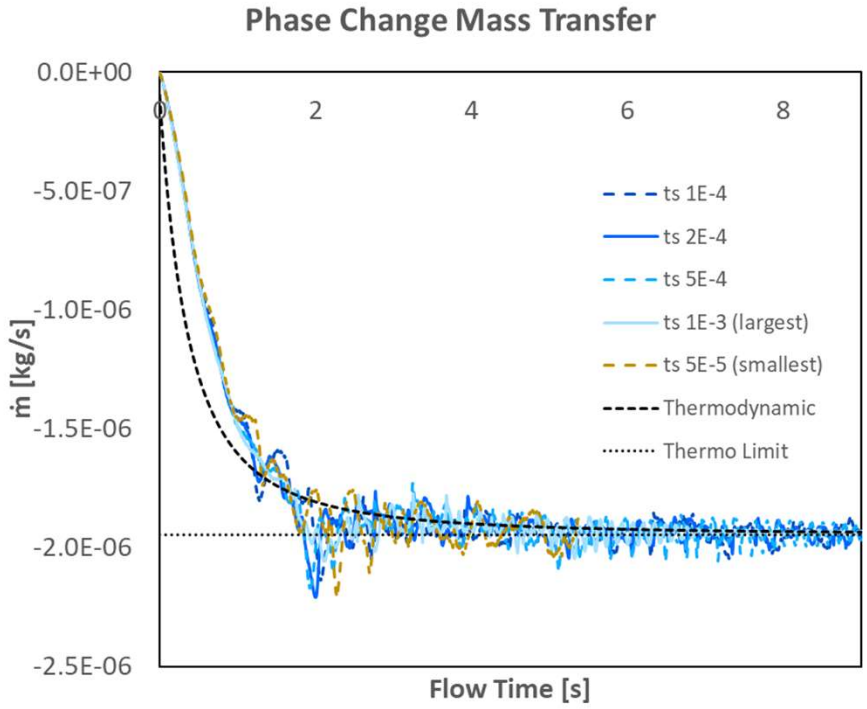
5E-5 s time step
Smallest step size

3D CFD Model – Time Step Study

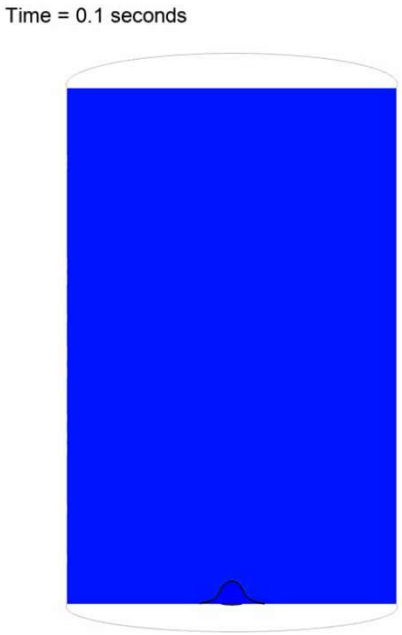


3D CFD Model – Time Step Study

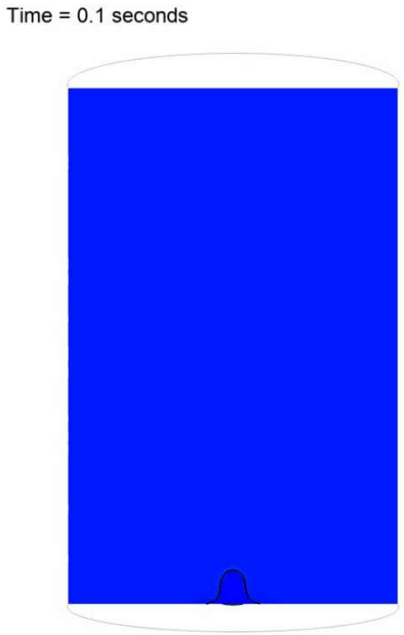




3D CFD Model – Results for Different Flow Rates

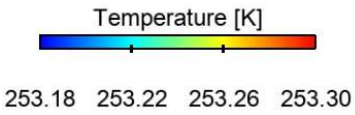


1.0 ml/s

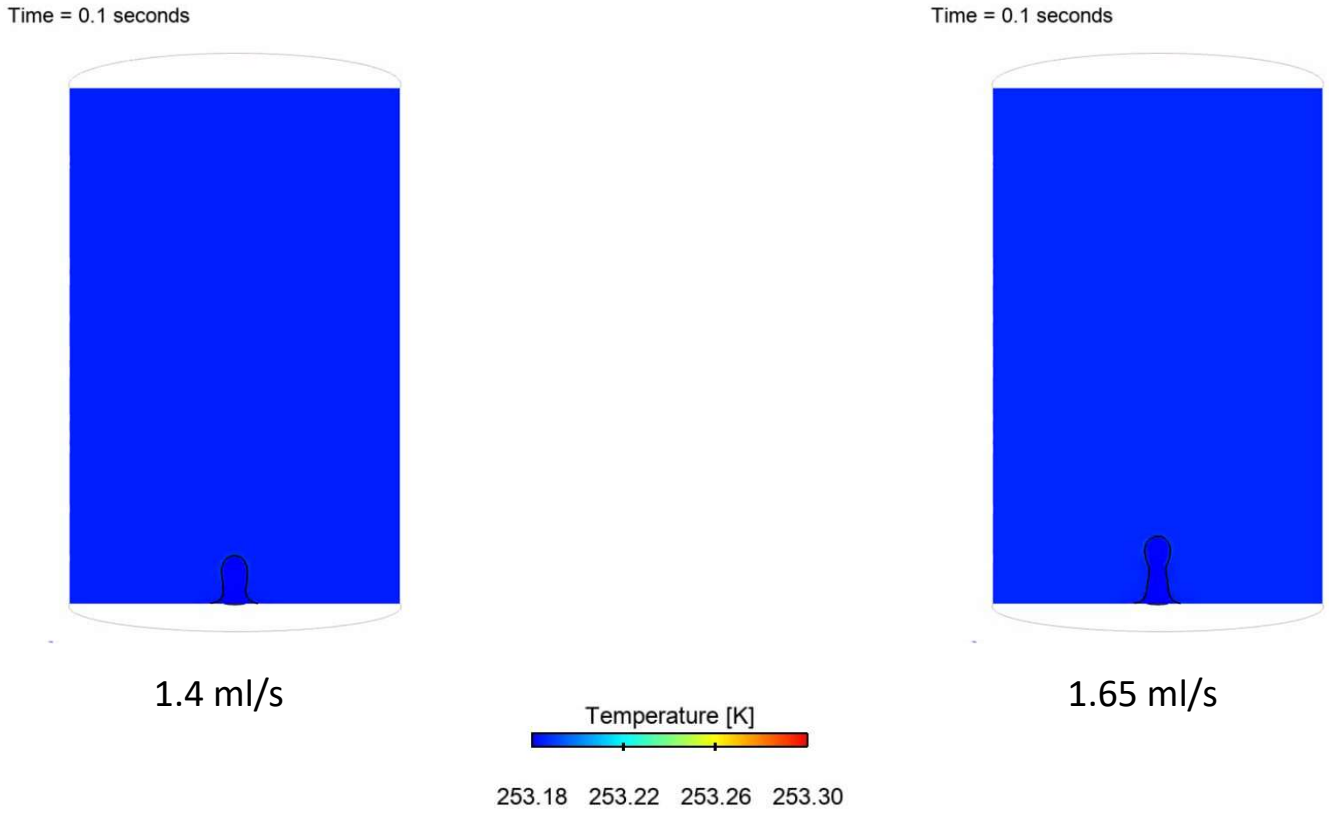


1.2 ml/s

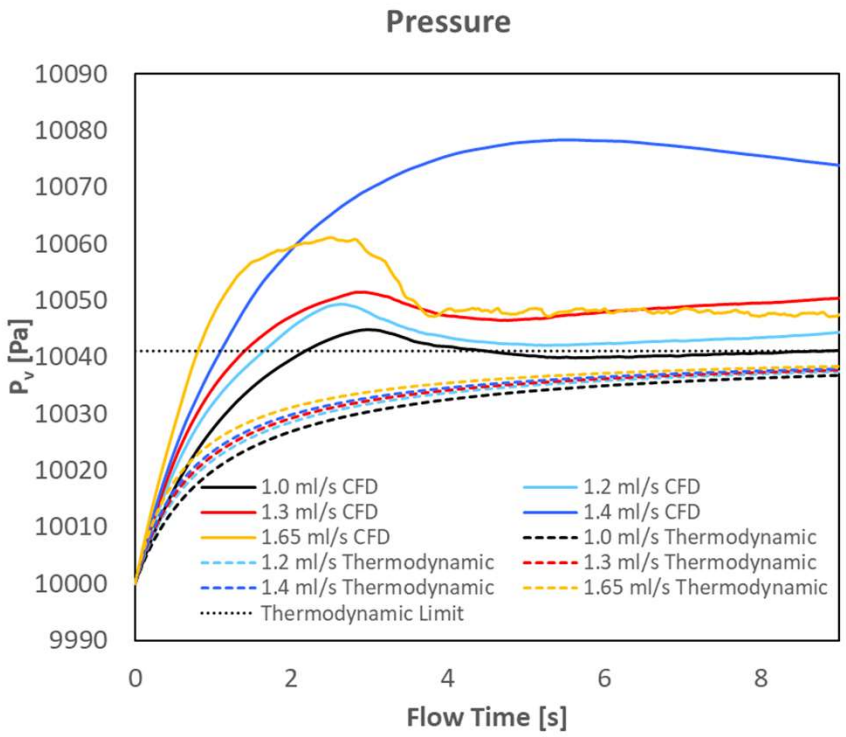
1.3 ml/s

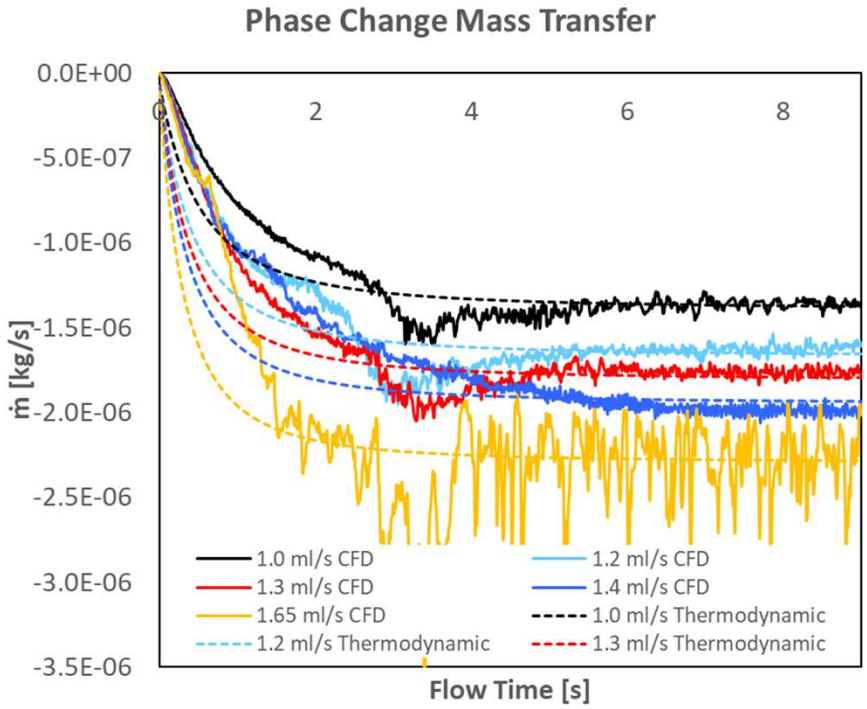


3D CFD Model – Results for Different Flow Rates

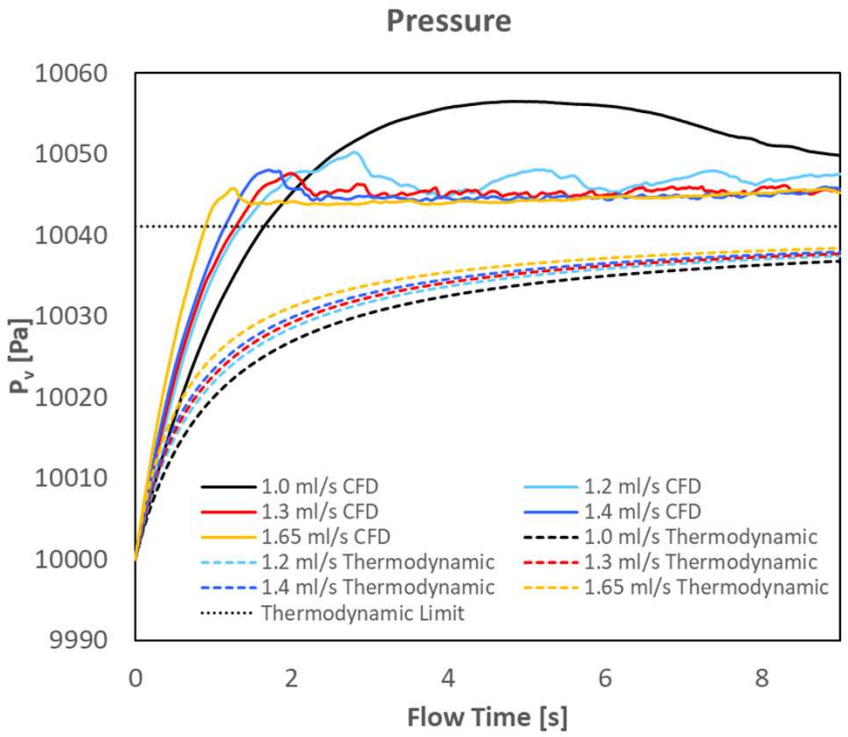


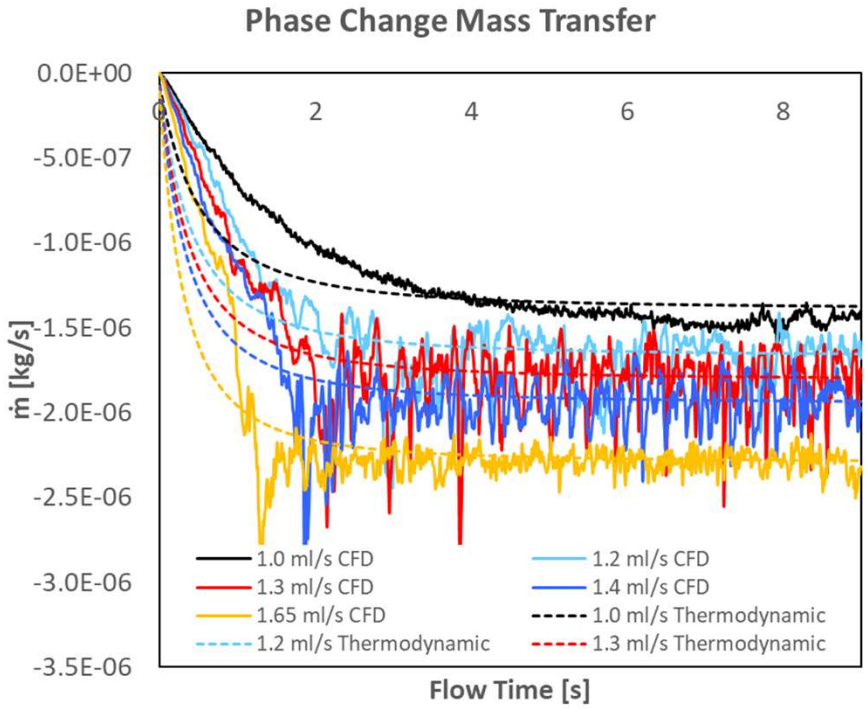
2D CFD Model – Flat Inlet Profile



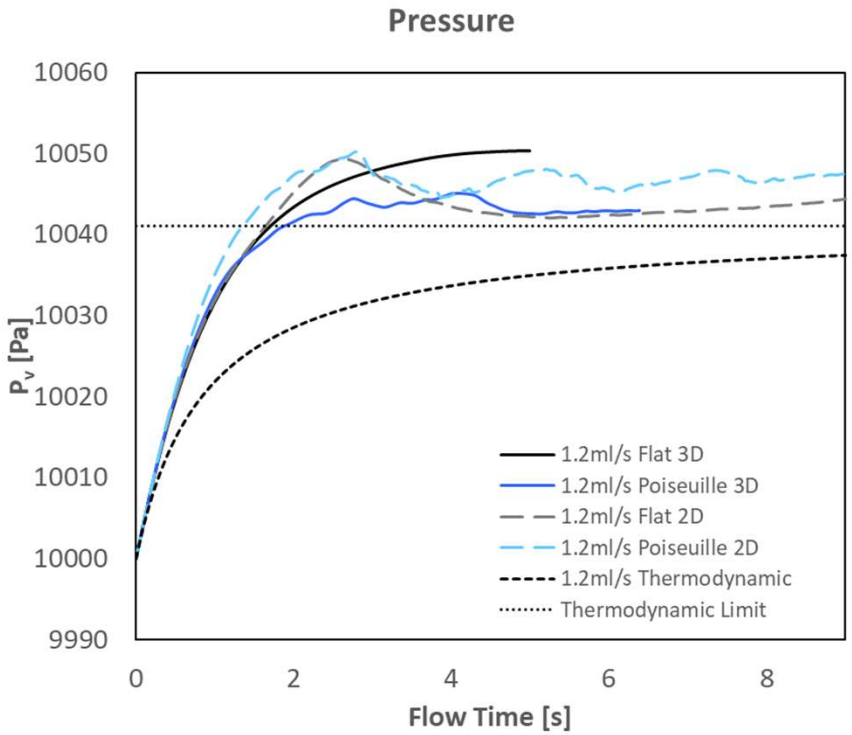


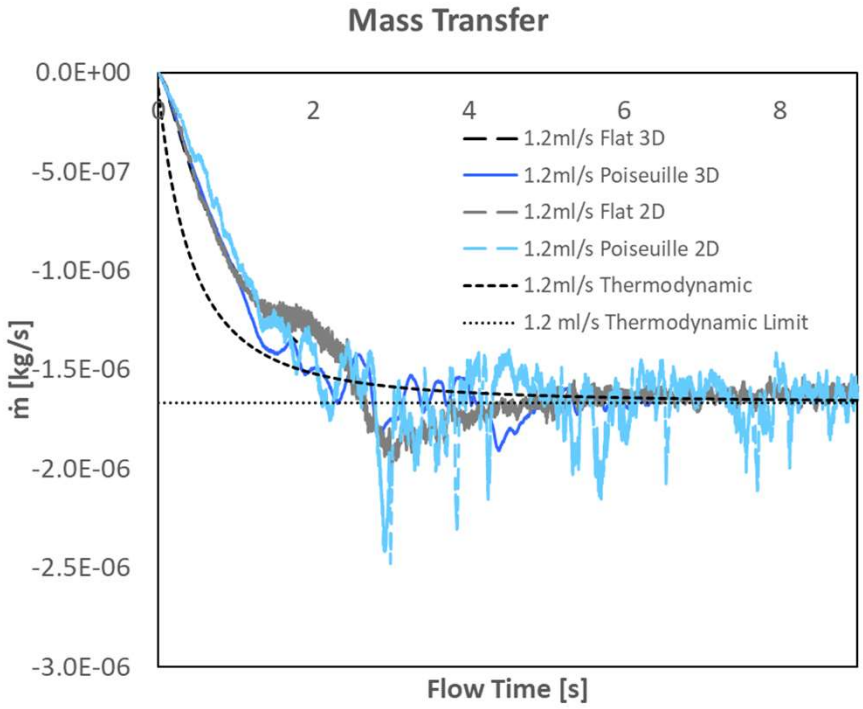
2D Model Poiseuille Inlet



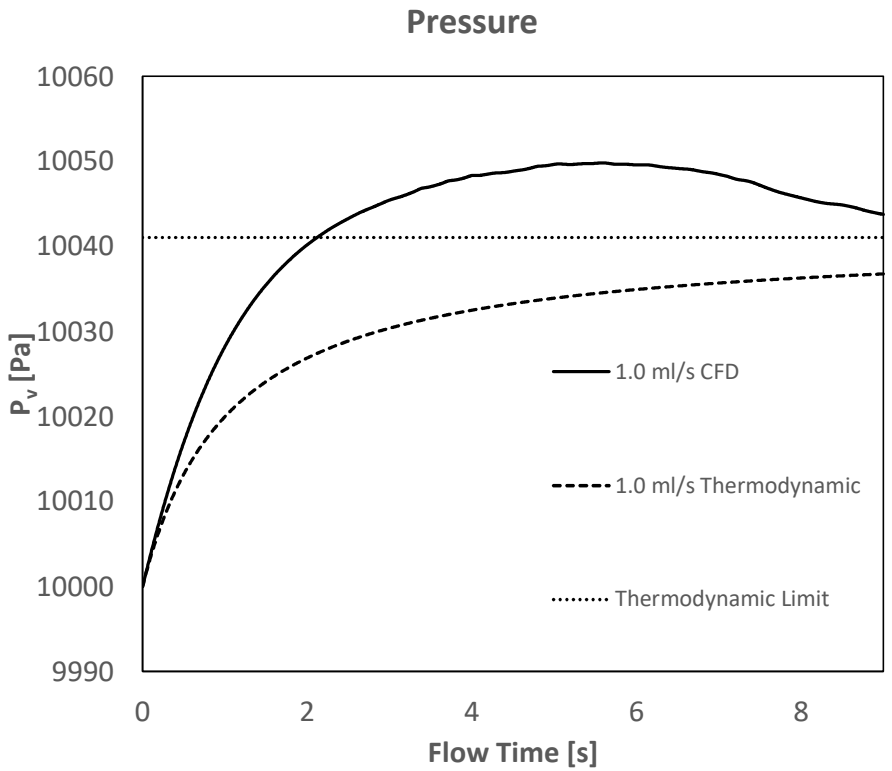


1.2 ml/s

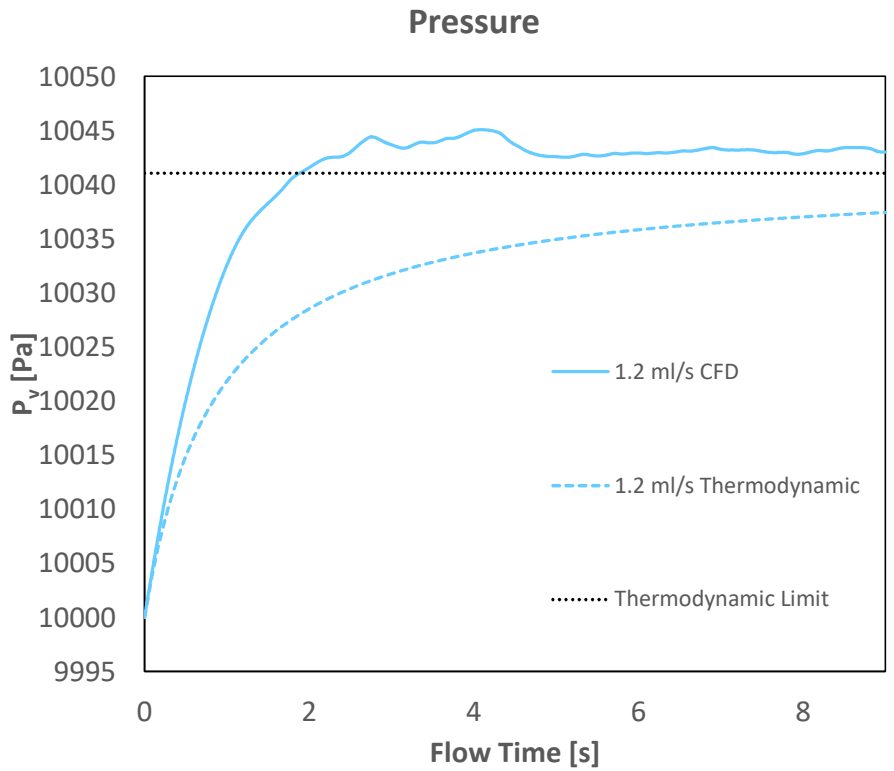

Pressure
10060
10050
10040
10030
10020
10010
10000
9990
P_v [Pa]
0
2
4
6
8
Flow Time [s]
1.2ml/s Flat 3D
1.2ml/s Poiseuille 3D
1.2ml/s Flat 2D
1.2ml/s Poiseuille 2D
1.2ml/s Thermodynamic
Thermodynamic Limit


Mass Transfer
0.0E+00
-5.0E-07
-1.0E-06
-1.5E-06
-2.0E-06
-2.5E-06
-3.0E-06
$\dot{m}$ [kg/s]
2
4
6
8
Flow Time [s]
1.2ml/s Flat 3D
1.2ml/s Poiseuille 3D
1.2ml/s Flat 2D
1.2ml/s Poiseuille 2D
1.2ml/s Thermodynamic
1.2 ml/s Thermodynamic Limit

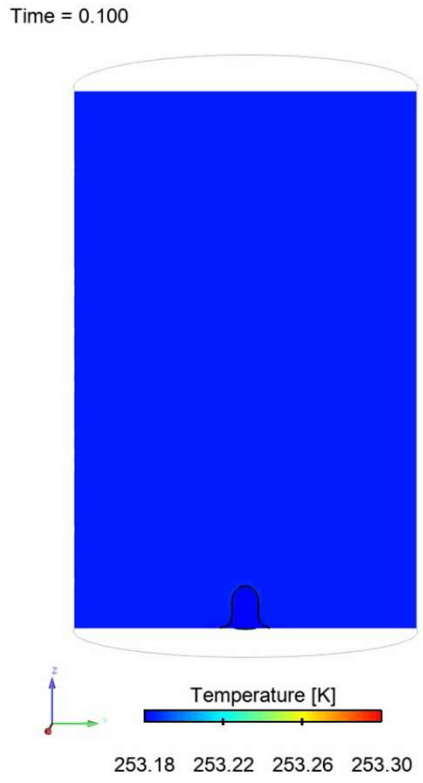
3D CFD Model – Results for 1.0 ml/s

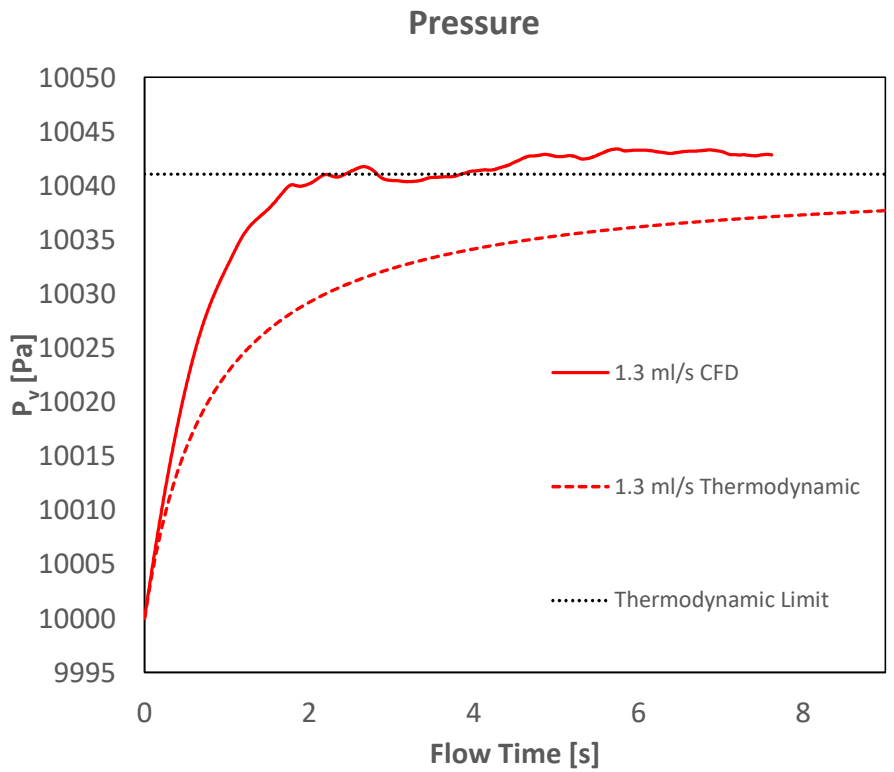


3D CFD Model – Results for 1.2 ml/s

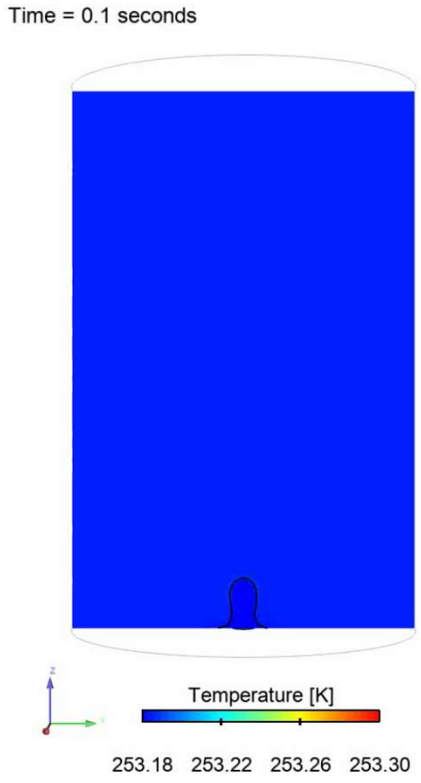


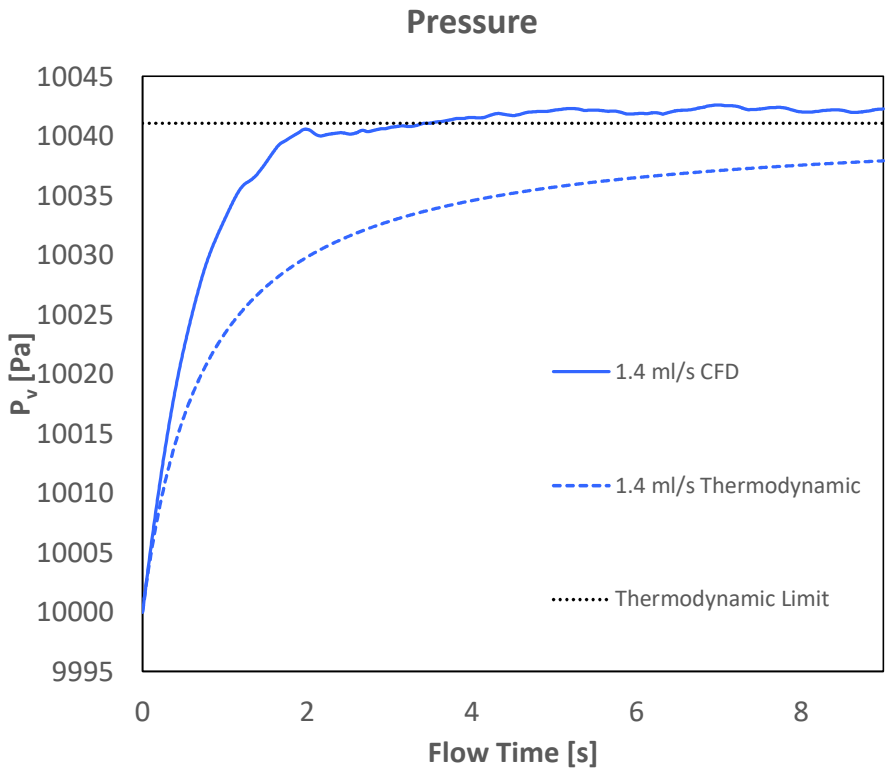
3D CFD Model – Results for 1.3 ml/s



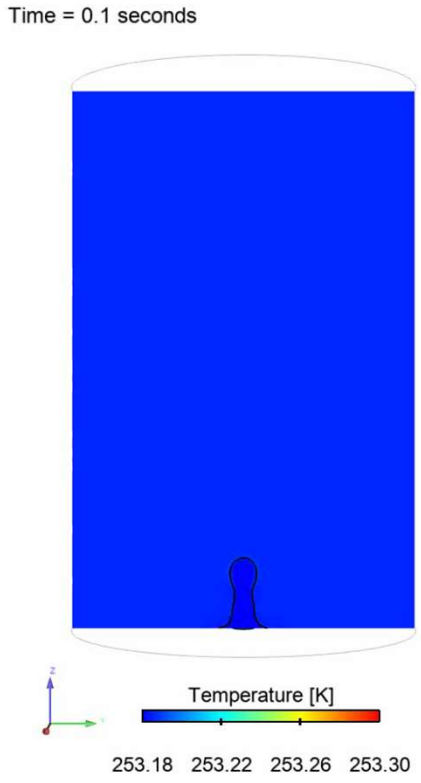


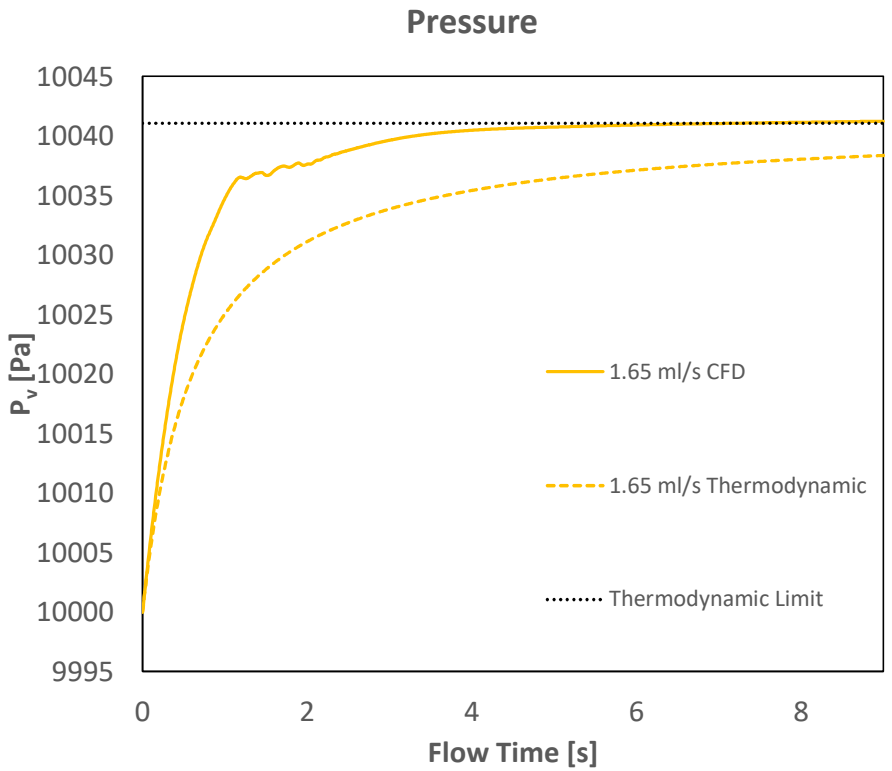
3D CFD Model – Results for 1.4 ml/s



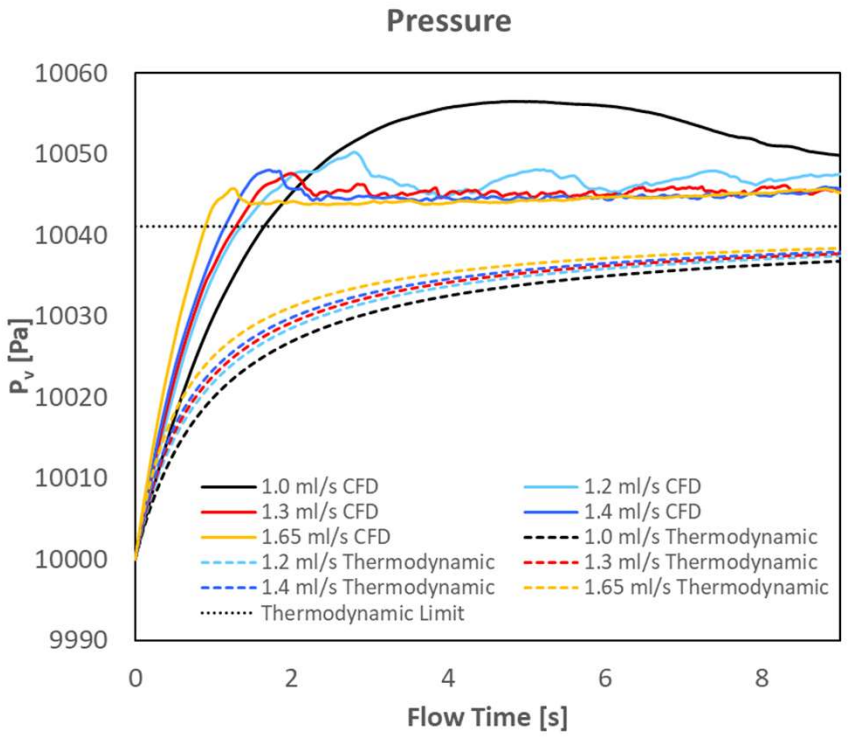


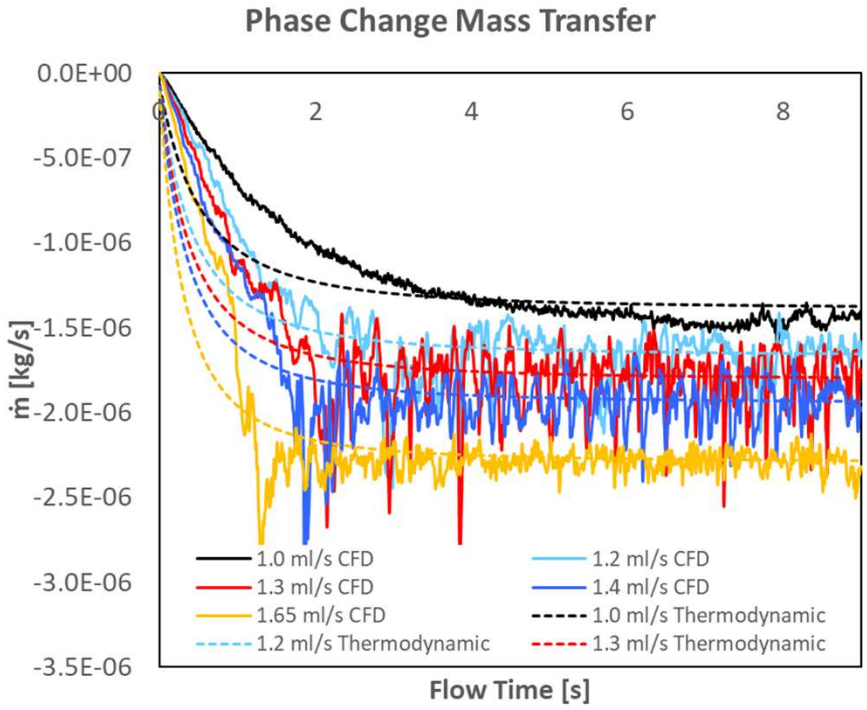
3D CFD Model – Results for 1.65 ml/s





2D Model Poiseuille Inlet





2D Model Poiseuille Inlet