Thermal-Fluid Analysis of a Lithium Vaporizer for a High Power Magnetoplasmadynamic Thruster

Brian Eric St. Rock
Worcester Polytechnic Institute

Follow this and additional works at: https://digitalcommons.wpi.edu/etd-theses

Repository Citation
https://digitalcommons.wpi.edu/etd-theses/38

This thesis is brought to you for free and open access by Digital WPI. It has been accepted for inclusion in Masters Theses (All Theses, All Years) by an authorized administrator of Digital WPI. For more information, please contact wpi-etd@wpi.edu.
THERMAL-FLUID ANALYSIS OF A LITHIUM VAPORIZER FOR A HIGH POWER MAGNETOPLASMADYNAMIC THRUSTER

by

Brian St. Rock

A Thesis

Submitted to the Faculty

of the

WORCESTER POLYTECHNIC INSTITUTE

in partial fulfillment of the requirement for the

Degree of Master of Science

in

Mechanical Engineering

By

____________________________________

June 2006

APPROVED:

______________________________________________
Dr. John J. Blandino, Advisor
Professor, Mechanical Engineering Department

______________________________________________
Dr. Nikolaos A. Gatsonis, Committee Member
Assistant Professor, Mechanical Engineering Department

______________________________________________
Dr. Michael A. Demetriou, Committee Member
Associate Professor, Mechanical Engineering Department

______________________________________________
Dr. Mark W. Richman, Graduate Committee Representative
Associate Professor, Mechanical Engineering Department
ABSTRACT

A lithium vaporizer for a high-power magnetoplasmadynamic (MPD) thruster is modeled using a parallel approach. A one-dimensional, thermal-resistive network is developed and used to calculate the required vaporizer length and power as a function of various parameters. After comparing results predicted by this network model with preheat power data for a 200 kW MPD thruster, we investigate performance over a parameter space of interest for the Advanced Lithium-Fed, Applied-field, Lorentz Force Accelerator (ALFA\textsuperscript{2}) thruster. Heater power sensitivity to cathode tube emissivity, mass flow rate, and vapor superheat are presented. The cold-start heater power for 80 mg/s is found to range from 3.38 to 3.60 kW, corresponding to a vaporizer (axial) length of 18 to 26 cm. The strongest drivers of vaporizer performance are cathode tube emissivity and a conduction heat sink through the mounting flange. Also, for the baseline ALFA\textsuperscript{2} case, it is shown that increasing the vapor superheat from 100 K to 300 K has the effect of lowering the vaporizer thermal efficiency from 57\% to 49\%.

Also, a finite-volume computational fluid dynamic (CFD) is implemented in FLUENT 6.2 which includes conjugated heat transfer to the solid components of the cathode. This model uses a single-fluid mixture model to simulate the effects of the two-phase vaporizer flow with source terms that model the vaporization. This model provides a solution of higher fidelity by including three-dimensional fluid dynamics such as thermal and momentum boundary layers, as well as calculating a higher resolution temperature distribution throughout the cathode assembly. Results from this model are presented for three mass flow rates of interest (40 mg/s, 80 mg/s, and 120 mg/s). Using a fixed power and length taken from the conceptual ALFA\textsuperscript{2} design, the dryout point ranges from 12.3-17.6 cm from the base of the cathode assembly for 40 mg/s and 80 mg/s, respectively. For the 120 mg/s case, the two-phase flow never reaches dryout. Finally, results
two modeling approaches are compare favorably, with a maximum disagreement of 13.0 percent in prediction of the dryout point and 4.2 percent in predictions of the exit temperature.
ACKNOWLEDGMENTS

I wish to express my gratitude to everyone who has helped me in this endeavor. Most of all, I would like to thank my advisor Prof. John Blandino. His encouragement, advice, enthusiasm, and patience have made the time spent during my collegiate studies at WPI an extremely enjoyable and rewarding experience. His contributions to this thesis and to my education in general will always be remembered and appreciated.

I would like to thank Prof. Edgar Choueiri of Princeton University, Dr. Michael Lapointe of the NASA Marshall Space Flight Center, and the NASA Prometheus Advanced Systems and Technology Office for continued support of this work.

I would also like to thank the friendly, accommodating, and courteous staff of the Mechanical Engineering Department, specifically Barbara Edilberti, Pam St. Louis, and Barbara Furhman. Their guidance and friendship over the past couple years has always been appreciated, and their patience and willingness to help is without equal.

Sia Najafi and Randy Robinson also deserve special thanks for their assistance and technical expertise during all of my regularly occurring computer-related meltdowns. Their willingness to help never went unnoticed, and was always appreciated.

Thanks are due to all my fellow students in the graduate aerospace research group at WPI. Specifically to Anton, Ryan, and Jimmy; thanks for the good times. You have been more than just fellow students, you have been good friends. Further, I would like to thank the members of my graduate thesis committee for their time, attention, and advice.

Above all, I would like to express my sincere gratitude to my family, including my parents, and my fiancé and best friend, Sharon, who have always given me the greatest encouragement and support. Without them, none of this work would ever have been obtainable.
# TABLE OF CONTENTS

Abstract ................................................................................................................................ i
Acknowledgments .............................................................................................................. iii
Table of Contents ............................................................................................................... iv
List of Figures .................................................................................................................. viii
List of Tables ................................................................................................................... xiii
Nomenclature ................................................................................................................... xiv

1. Introduction ....................................................................................................................1  
   1.1 ALFA^2 Thruster .....................................................................................................2  
      1.1.1 Cathode Assembly ......................................................................................4  
   1.2 Motivation .............................................................................................................6  
   1.3 Objectives ............................................................................................................7  
      1.3.1 Goals .......................................................................................................7  
      1.3.2 Approach .................................................................................................8  
   1.4 Computer Programs and Resources .....................................................................9  
   1.5 Thesis Organization ............................................................................................10

2 Background ..................................................................................................................11  
   2.1 Flow Characterization .........................................................................................11  
   2.2 Previous Studies ..................................................................................................14  
   2.3 MAI Thruster .......................................................................................................15

3 Network Model ............................................................................................................17  
   3.1 Model Description ...............................................................................................18  
      3.1.1 Channel Region .........................................................................................21  
      3.1.2 Pre-Channel Region ..................................................................................25
3.2 Flow Regimes .................................................................................................................. 28
3.3 Convective Heat Transfer ................................................................................................. 32
  3.3.1 Liquid Region ............................................................................................................. 32
  3.3.2 Two-phase Region ..................................................................................................... 33
  3.3.3 Vapor Region ............................................................................................................. 36
3.4 Radiation View Factors ..................................................................................................... 36
3.5 Pressure Drop ................................................................................................................... 39
  3.5.1 Single-phase Flows .................................................................................................... 40
  3.5.2 Two-phase Flow ........................................................................................................ 41
3.6 Solution Methodology ...................................................................................................... 48
  3.6.1 Pre-Channel Region .................................................................................................. 48
  3.6.2 Channel Region .......................................................................................................... 50
3.7 Boundary Conditions and Simulation Parameters ............................................................. 51
3.8 Results and Discussion .................................................................................................... 55
  3.8.1 Model Validation ........................................................................................................ 56
  3.8.2 ALFA^2 Baseline ....................................................................................................... 57
  3.8.3 Performance Sensitivity ............................................................................................ 63
4 Finite-Volume Model ........................................................................................................... 72
  4.1 Model Description .......................................................................................................... 73
    4.1.1 Mass and Momentum Conservation ....................................................................... 74
    4.1.2 Energy Conservation ............................................................................................. 76
    4.1.3 Vapor Continuity ................................................................................................... 77
    4.1.4 Source Terms ......................................................................................................... 78
    4.1.5 Conjugate Heat Transfer ....................................................................................... 80
  4.2 Problem Simplifications ................................................................................................. 81
4.2.1 Cathode Partitioning ..........................................................................82
4.2.2 Geometry Modifications ....................................................................85
4.2.3 Channel Symmetry.............................................................................87
4.3 Mesh Description.....................................................................................90
4.4 Boundary Conditions.............................................................................96
  4.4.1 Thermal Boundary Conditions...........................................................96
  4.4.2 Hydrodynamic Boundary Conditions ..............................................103
4.5 Solution Methodology...........................................................................108
4.6 Results and Discussion.........................................................................115
  4.6.1 ALFA\(^2\) Baseline ........................................................................115
  4.6.2 Performance Sensitivity ..................................................................122
  4.6.3 Comparison with Network Model ...................................................128
5 Conclusions..............................................................................................131
  5.1 Network Model ..................................................................................131
  5.2 Finite-Volume Model ..........................................................133
  5.3 Observations .....................................................................................134
6 Future Work ............................................................................................138
  6.1 Transient Modeling ...........................................................................138
  6.2 Adaptive Grid Refinement ...............................................................139
  6.3 Pressure Dependence .........................................................................140
  6.4 Experimental Work ...........................................................................140
Appendix A: Material Properties ...............................................................141
Tungsten .................................................................................................141
Liquid Lithium .........................................................................................141
Appendix B: Raw Source Code .................................................................143
LIST OF FIGURES

Figure 1-1. Cutaway drawing of the conceptual, simplified ALFA$^2$ thruster showing major components. ........................................................................................................................................................................... 3

Figure 1-2. Axial cutaway drawing of the cathode assembly showing all main features and components of the cathode assembly................................................................. 5

Figure 2-1. Schematic of the MAI cathode showing the features similar to ALFA$^2$ [11]. ............... 16

Figure 3-1. Representative cross section of vaporizer tube showing pre-channel region which includes the lithium “pool”, heater flange, and mounting flange. ......................................................... 18

Figure 3-2. Schematic of the vaporizer section in region with channels................................. 21

Figure 3-3. Resistive network representation of vaporizer section in region with channels. ................................................................. 22

Figure 3-4. Segment of vaporizer tube showing element of vaporizer channel used in thermal network analysis......................................................................................................................................................... 22

Figure 3-5. Schematic of the vaporizer section in pre-channel region. ........................................ 25

Figure 3-6. Resistive network representation of the vaporizer pre-channel region........... 26

Figure 3-7. Flow map with experimental data from Cubaud & Ho, 2004 [17]......................... 29

Figure 3-8. Images of the various flow patterns and vapor structure in a microchannel flow [17] .................................................................................................................................................................................. 31

Figure 3-9. Illustration of the radiation views from the mounting flange. .......................................... 36

Figure 3-10. Illustration of the radiation views from the mounting flange to a) an internal cylinder and b) a truncated cone ......................................................................................................................... 37

Figure 3-11. Control volume illustrating the momentum flux across element........................................... 44

Figure 3-12. Cathode cross-section showing boundary conditions for the baseline simulation... 53
Figure 3-13. Comparison of network model results with data from the 200 kW MAI thruster. The eight mass flow rates used to generate the vaporizer performance curves represent the range of flow rates in Table 2-3.

Figure 3-14. Calculated vaporizer tube, fluid, and cathode tube temperatures as a function of axial position for the case of vapor superheat of 330 K (baseline ALFA\(^2\) length is 22.86 cm).

Figure 3-15. Heat flow through the network as a function of axial position. To highlight the system’s trends, the results are shown for a vapor superheat of 330 K. The baseline ALFA\(^2\) parameters are listed in Table 3-5.

Figure 3-16. Left: Additional vaporizer length required (beyond dryout) for various levels of vapor superheat. Right: Thermal efficiency as a function of vapor superheat. The plots asymptote due to a thermal equilibrium condition at a vapor superheat of approximately 330 K.

Figure 3-17. Evolution of the vapor quality and the void fraction during vaporization.

Figure 3-18. Cubaud and Ho flow map for the two-phase portion of the lithium flow.

Figure 3-19. Pressure versus axial position for ALFA\(^2\) baseline vaporizer channel.

Figure 3-20. ALFA\(^2\) vaporizer power calculated using network model. Sensitivity to emissivity of cathode tube surface for nominal mass flow rate of \(\dot{m} = 80\) mg/s.

Figure 3-21. Sensitivity of thermal efficiency to vapor superheat for different values of surface emissivity. Lowering the surface emissivity mimics the effect of including radiation shielding in the thruster design.

Figure 3-22. ALFA\(^2\) vaporizer power calculated using network model. Sensitivity to mass flow rate for a temperature-dependent emissivity.
Figure 3-23. Pressure drop sensitivity to total mass flow rate (top) and number of channels (bottom) for the baseline flow rate of 80 mg/s. ................................................................. 68

Figure 3-24. Sensitivity of the power/length requirements to vapor superheat (i.e. exit temperature) for the nominal mass flow rate of 80 mg/s. ................................................................. 69

Figure 3-25. Optimal vaporizer length for a range of flow rate, emissivity, and superheat........ 69

Figure 4-1. Illustration of differences between different velocity terms. .................................... 75

Figure 4-2. The void fraction plotted versus vapor quality for a range of density ratios. ........... 82

Figure 4-3. Top: The complete cathode assembly. Bottom: The cathode divided into subregions for computational purposes........................................................................................................... 84

Figure 4-4. Left: The circled areas indicate geometric features that are sources of poor elements. Right: The modified vaporizer geometry renders the domain more CFD-friendly. ............ 86

Figure 4-5. Left: Illustration of the helical symmetry of the cathode assembly. Right: Azimuthal cross-section of the cathode body across z showing relevant points. ............................... 88

Figure 4-6. Simplified vaporizer domain shown as modeled. ..................................................... 89

Figure 4-7. a) Partial listing of poor quality tetrahedral elements [38] and b) definition for element quality.............................................................................................................................. 92

Figure 4-8. Cross-section of a fluid channel showing the solid domain divided into 4 separate mappable regions. The fluid domain is the red rectangle. ................................................. 93

Figure 4-9. Portion of the helical channel defining the nomenclature of the faces in the solid domain.............................................................................................................................. 97

Figure 4-10. Illustration of the iterative solution of the internal boundary conditions.............. 99

Figure 4-11. Calculation of the view temperatures in the pre-channel region........................... 103
Figure 4-12. The operating pressure shown as a function of the Mach number at the dryout point. ........................................................................................................................................................................ 105

Figure 4-13. Flowchart of the solution process for the entire domain using the single channel approach........................................................................................................................................................................ 109

Figure 4-14. Solution algorithm of the segregated FLUENT solver. ............................................. 112

Figure 4-15. Contours of void fraction at the onset of boiling. ......................................................... 115

Figure 4-16. Evolution of void fraction across channel cross-sections at onset of boiling for the 80 mg/s flow rate. ........................................................................................................................................................................ 116

Figure 4-17. Cross-section of the temperature contours in the fluid and solid domain.............. 117

Figure 4-18. Temperature contours in the area of the actively cooled mounting flange......... 118

Figure 4-19. The calculated axial heat flux along the downstream wall ...................................... 119

Figure 4-20. Contours of velocity magnitude at exit of boiling onset region............................ 120

Figure 4-21. Contours of relative static pressure at exit of boiling onset region...................... 120

Figure 4-22. Temperature contours of cross-section in the prechannel region for the 80 mg/s flow rate. ........................................................................................................................................................................ 121

Figure 4-23. Pressure contours along the prechannel walls for each flow rate. ......................... 122

Figure 4-24. Temperature profiles as calculated by the FVM for the three flow rates tested. ... 124

Figure 4-25. Void fraction for the three flow rates tested.............................................................. 124

Figure 4-26. Pressure drop across vaporizer channels for three flow rates tested. Pressure is relative to the channel exit................................................................................................................................. 126

Figure 4-27. Assessment of the outlet temperatures and the distribution of total mass flow rate through pre-channel region................................................................................................................................. 127
Figure 4-28. Comparison between the network model and FVM predictions for the fluid temperature profile for the 80 mg/s case................................................................. 129

Figure 4-29. Comparison between the network model and FVM void fraction predictions for the 80 mg/s case............................................................................................................ 130
**LIST OF TABLES**

Table 2-1. Lithium properties used to evaluate microchannel criteria ........................................ 13

Table 2-2. Experimental data and numerical results of Emsellum et. al. [10]............................... 15

Table 2-3. MAI Flow rate range and heater power during preheat phase [11]............................ 16

Table 3-1. Definition of the resistances in the channel network equations ................................. 24

Table 3-2. Definition of the resistances in the pre-channel network equations........................... 27

Table 3-3. Constant and exponents required for calculating the parameter $C$ [25]............... 43

Table 3-4. MAI Parameter List.................................................................................................... 54

Table 3-5. Baseline ALFA $^2$ Parameter List ............................................................................. 54

Table 3-6. Sensitivity of the solution to boundary conditions..................................................... 70

Table 4-1. Meshing statistics and schemes for the FVM............................................................. 96

Table 4-2. Hydrodynamic conditions for the three flow rates investigated............................... 108

Table 4-3. Comparison between the network model and FVM................................................. 130
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a$</td>
<td>channel depth</td>
</tr>
<tr>
<td>$a_v$</td>
<td>vapor acceleration</td>
</tr>
<tr>
<td>$b$</td>
<td>cross-sectional channel width</td>
</tr>
<tr>
<td>$b'$</td>
<td>actual channel width</td>
</tr>
<tr>
<td>$B$</td>
<td>cross-section shape number</td>
</tr>
<tr>
<td>$d_H$</td>
<td>hydraulic diameter</td>
</tr>
<tr>
<td>$d_b$</td>
<td>characteristic bubble diameter</td>
</tr>
<tr>
<td>$E$</td>
<td>energy</td>
</tr>
<tr>
<td>$f_{drag}$</td>
<td>drag function for phase interaction</td>
</tr>
<tr>
<td>$F$</td>
<td>radiation view factor</td>
</tr>
<tr>
<td>$F_1$</td>
<td>empirical nucleate coefficient</td>
</tr>
<tr>
<td>$F_2$</td>
<td>empirical convective coefficient</td>
</tr>
<tr>
<td>$G$</td>
<td>mass flux</td>
</tr>
<tr>
<td>$g$</td>
<td>gravitational acceleration</td>
</tr>
<tr>
<td>$h$</td>
<td>local enthalpy</td>
</tr>
<tr>
<td>$h_F$</td>
<td>heat transfer coefficient</td>
</tr>
<tr>
<td>$h_{fg}$</td>
<td>latent heat of vaporization</td>
</tr>
<tr>
<td>$k$</td>
<td>thermal conductivity</td>
</tr>
<tr>
<td>$L$</td>
<td>component length</td>
</tr>
<tr>
<td>$L_c$</td>
<td>characteristic momentum flux</td>
</tr>
<tr>
<td>$L_{DRY}$</td>
<td>dryout point</td>
</tr>
<tr>
<td>$m$</td>
<td>mass flow rate</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$Nu$</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>$P$</td>
<td>static pressure</td>
</tr>
<tr>
<td>$P_h$</td>
<td>heater power</td>
</tr>
<tr>
<td>$Pe$</td>
<td>Peclet number</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$p$</td>
<td>thread pitch</td>
</tr>
<tr>
<td>$q''$</td>
<td>heat flux</td>
</tr>
<tr>
<td>$R$</td>
<td>phasic mass source term</td>
</tr>
<tr>
<td>$R''$</td>
<td>thermal resistance</td>
</tr>
<tr>
<td>$R_C''$</td>
<td>thermal contact resistance</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$r$</td>
<td>component radius</td>
</tr>
<tr>
<td>$s$</td>
<td>path coordinate along channel</td>
</tr>
<tr>
<td>$S$</td>
<td>solid heat source term</td>
</tr>
<tr>
<td>$S_E$</td>
<td>fluid energy source term</td>
</tr>
<tr>
<td>$S_m$</td>
<td>mixture mass source term</td>
</tr>
<tr>
<td>$St$</td>
<td>Stokes number</td>
</tr>
<tr>
<td>$t$</td>
<td>flange or wall thickness</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
</tr>
<tr>
<td>$\Delta T_{SH}$</td>
<td>vapor superheat</td>
</tr>
<tr>
<td>$U_c$</td>
<td>characteristic flow velocity</td>
</tr>
<tr>
<td>$V$</td>
<td>volume</td>
</tr>
<tr>
<td>$\nu$</td>
<td>flow velocity</td>
</tr>
<tr>
<td>$\nu_{dr}$</td>
<td>drift velocity</td>
</tr>
<tr>
<td>$\nu_{VL}$</td>
<td>slip velocity</td>
</tr>
<tr>
<td>$x$</td>
<td>vapor mass fraction or quality</td>
</tr>
<tr>
<td>$X_\alpha$</td>
<td>Martinelli parameter</td>
</tr>
<tr>
<td>$z$</td>
<td>axial coordinate</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>volume fraction</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>emissivity</td>
</tr>
<tr>
<td>$\mu$</td>
<td>dynamic viscosity</td>
</tr>
</tbody>
</table>
\( \rho \) density \( \lambda \) vapor source relaxation parameter
\( \sigma \) Stefan-Boltzmann constant \( \tau \) bubble relaxation time
\( \dot{\phi}_L \) Zieganick and Litvinov parameter \( \tau_c \) characteristic time scale

**Subscripts**

- \( AN \) pertaining to the anode
- \( b \) pertaining to a bubble
- \( C \) pertaining to the cathode assembly
- \( CT \) pertaining to the cathode tube
- \( F \) pertaining to the fluid
- \( HF \) pertaining to the heater flange
- \( I \) pertaining to the inner dimension
- \( i \) pertaining to the \( i \)-th phase
- \( MF \) pertaining to the mounting flange
- \( min \) pertaining to a minimum quantity
- \( m \) pertaining to a mixture averaged quantity
- \( m \) pertaining to molybdenum
- \( NC \) pertaining to the vaporizer area with no channels
- \( O \) pertaining to the outer dimension
- \( P \) pertaining to the inlet plenum
- \( R \) pertaining to radiation
- \( S \) pertaining to a heat sink
- \( sat \) pertaining to a saturation quantity
- \( t \) pertaining to tungsten
- \( VT \) pertaining to vaporizer tube
1. INTRODUCTION

A space-qualified solar or nuclear power source capable of producing several hundred kilowatts of electrical power could enable a number of ambitious robotic exploration missions to the outer planets and could also support exploration of the moon and Mars. In addition to purely robotic missions, a high-power solar or nuclear system would enable the use of high performance electric propulsion systems for the transport of cargo to Mars ahead of a human crew. Once the cargo vehicle reaches its destination, the reactor could be used to support science operations for an extended period of time.

Until early 2005, the Jupiter Icy Moons Orbiter (JIMO) was scheduled to be the flagship technology demonstration mission for the Prometheus program. This mission would use a combination of advanced gridded ion thrusters and high power Hall thrusters [1]. The combination of ion and Hall thrusters is needed to optimize the system performance which needs to balance high specific impulse (6000 – 8000 s) provided by the ion thrusters throughout most of the mission with the higher thrust needed for operations near Europa provided by the Hall thrusters. The Herakles thrusters, with a power per thruster capability of 30 kW represent the state of the art in ion thruster technology and would be operated in clusters to process the 200 kW available for a JIMO mission [2].

NASA is considering even more demanding missions than JIMO which require hundreds of kilowatts to several megawatts. These missions include outer solar system robotic missions such as a Saturn Orbiter (with moon tour) with a ΔV of 33 km/sec and an interstellar precursor mission with a ΔV of 28 – 53 km/s [3]. These studies have considered power levels ranging from 250 kW to 2.75 MW. Even with the advanced Herakles technology, the number of thrusters needed to process such high power leads to increasingly complex and massive clusters.
of thrusters, feed system components, and power processors. One option which has been considered is the use of lithium-fueled magnetoplasmadynamic (MPD) thrusters or Lorentz Force Accelerators (LFA) which have demonstrated the capability to process from 200 kW to 500 kW for a single thruster in tests performed at the Moscow Aviation Institute (MAI) [3].

The MPD thruster is an electromagnetic thruster in which the plasma is accelerated through a Lorentz body force resulting from the interaction between the current and magnetic fields (both self-generated and applied). This Lorentz force has two components: a pumping and a blowing component. The pumping component is a force directed inward that radially constricts the plasma flow. The blowing component is an axial force that produces the directed thrust. Because of this accelerating mechanism, MPD thrusters are referred to synonymously as Lorentz force accelerators (LFA). For a detailed discussion of these acceleration mechanisms and fundamental physics in these devices, the reader is directed to Jahn [4]. While MPD thrusters have been studied since the 1960s at numerous research laboratories, in the last twenty years, the largest contributors to the development of MPD thruster technology have been the Moscow Aviation Institute (MAI) and Princeton University.

While the focus of the present work is in devices using lithium propellant, gaseous propellants can also be used, including hydrogen and argon. In fact, the ability of MPD devices to accelerate a variety of gases has also led to its consideration for terrestrial materials processing applications as well [5].

1.1 ALFA$^2$ Thruster

Figure 1-1 shows a simplified cutaway view of a concept for the Advanced Lithium-Fed, Applied-field, Lorentz Force Accelerator (ALFA$^2$). The ALFA$^2$ thruster is a variant of an MPD
thruster that incorporates a multi-channel hollow cathode and lithium propellant in order to realize gains in efficiency and thruster lifetime [6]. The basic elements are the cathode assembly, anode, solenoids, and heat pipes. In the conceptual drawing of Figure 1-1, the vaporizer is concentric with a solenoid that applies a magnetic field. Depending on thruster design, the upstream solenoid from Figure 1-1 (shown in red) can be replaced by components such as a grounding screen or an isolator. In any arrangement, shielding can be used to reduce the radiation losses from the outer cathode surface.

![Figure 1-1. Cutaway drawing of the conceptual, simplified ALFA\textsuperscript{2} thruster showing major components.](image)

The thruster utilizes a multi-channel hollow cathode to sustain a steady-state discharge of several thousand amperes. In contrast to a single channel hollow cathode, multiple cathode channels will expose a larger surface area for thermionic emission to the neutral lithium propellant, thereby lowering cathode temperatures [7]. The ALFA\textsuperscript{2} thruster incorporates a number of additional innovative features designed to improve efficiency and extend lifetime.
The thruster is being designed to target a nominal performance of 60% to 63% thruster efficiency at a specific impulse ($I_{sp}$) value of 6200 s and a power level of 245 kW [3].

The lithium propellant, which has a melting point of approximately 180 °C (453 K), will be delivered by the propellant feed system to the thruster in a liquid state. The liquid must then be vaporized prior to delivery to the multichannel hollow cathode in order to sustain an electrical discharge of several thousand amperes. While the energy required by the vaporizer (including the latent heat of vaporization) will need to be provided only by electrical heating for cold-starts, a portion of this energy input will be provided during operation by heat from the discharge itself which is conducted and radiated back to the vaporizer channels.

1.1.1 Cathode Assembly

For an efficient, compact thruster, the vaporizer in the ALFA$^2$ concept is integrated into the cathode assembly. In Figure 1-2, an axial cutaway of the cathode assembly is shown highlighting all of the main components and features. The vaporizer and cathode tube are made from tungsten, and both flanges are made from molybdenum. Due to their suitability to high-temperature applications, these materials are ideal for the cathode assembly. The heater flange, along with isolator plates (not shown) are used to fix the graphite heater insert to the downstream end of the vaporizer.

The vaporizer tube has a helical groove cut into its outer surface, similar to that of a hollowed ACME-style threaded rod. The vaporizer tube can utilize multiple starts to provide multiple fluid channels (i.e. multiple helical threads/grooves) to better distribute the flow at the channel inlet and outlets. Following standard screw terminology, a start is a term that describes the number of independent grooves (or threads) on a screw. The vaporizer tube fits snugly into
the inner surface of the cathode tube, with the resulting gaps forming the channels for the fluid flow. The ALFA\(^2\) conceptual design utilizes a four-start vaporizer tube that creates fluid channels 0.75 mm deep and 2.5 mm wide. From a fluid mechanics standpoint, channel dimensions of this magnitude can result in microchannel flow characteristics, as will be discussed in later sections.

The heat load into the vaporizer (during a cold-start) is provided by a graphite heater insert. Heat is passed radiatively from the heater into the entire inside surface of the vaporizer tube. From the base of the heater flange to the location of the channel exit, the axial length of the ALFA\(^2\) vaporizer is 21.88 cm.

![Axial cutaway drawing of the cathode assembly showing all main features and components of the cathode assembly.](image)

When operating at the proposed performance point [3], the vaporizer will need to process approximately 80 mg/s of lithium. Liquid lithium enters the vaporizer through the feedtube, flows into an inlet plenum, and then into the helical channels. The inlet plenum aids in uniformly distributing the total mass flow rate between each individual channel. The liquid then flows through the helical channels, undergoing a phase change and eventual dryout. The neutral
lithium vapor then exits the vaporizer channel, flows into the outlet plenum, and is delivered to a multichannel emitter. Here, the neutral lithium vapor is ionized into a plasma by thermionic emission, and accelerated electromagnetically to produce directed thrust.

The mounting flange is used to attach the cathode assembly to a bus bar. A bus bar is a plate that conducts several thousand amperes into the cathode during operation. From a thermal standpoint, this bus bar attachment has undesirable consequences as it represents the largest conduction heat sink out of the cathode assembly. Additionally, all of the current required to sustain the arc must pass through this contact zone, severely restricting any effort to reduce parasitic heat losses by using insulating rings or by creating thermal chokes (i.e. necking). Any strategy employed to increase the thermal resistance to the bus bar is likely to have the unfavorable result of increasing the electrical resistance as well.

1.2 Motivation

In a lithium-fed MPD thruster, the vaporizer represents a critical component of the propellant supply system because the efficient operation of ALFA\(^2\) requires the reliable delivery of a controlled mass flow (tens of mg/s) of lithium vapor. Very few general studies have been completed from a thermal modeling standpoint on the cathode assemblies of MPD thrusters. Experimental data (such as temperature profiles) that could be used to guide an empirical design effort are also lacking. Enhancing the thermal performance of the cathode requires an understanding of the major performance drivers. Areas for improvement need to be identified, and a general understanding gained of the sensitivity of vaporizer performance to several key design parameters.

Power not used for lithium vaporization can be better utilized in other thruster and spacecraft subsystems. Minimizing the required heat load to sustain vaporization during
operation requires a careful cathode/vaporizer design in which the heat from the arc is matched with the mass flow rate. In addition, conducted and radiated heat losses from the thruster body and arc for an operating thruster must be considered in order to design the vaporizer for optimal thruster thermal efficiency.

The modeling efforts carried out in this work focus on addressing these fundamental thermal requirements. The insights gained from this analysis will eventually help guide the cathode design for use in a proof-of-concept version of the ALFA\textsuperscript{2} thruster that will operate in the 100-240 kW power range.

1.3 Objectives

1.3.1 Goals

To help meet the operational requirements for ALFA\textsuperscript{2}, modeling efforts are required to characterize the thermal performance of the vaporizer. Specifically, the objective of the model presented in the first part of this thesis is to provide a tool which can be used to quickly determine vaporizer performance sensitivity to mass flow rate, channel geometry, surface emissivity, number of channels (leads), and initial and final fluid thermodynamic states. A range of sensitivity studies can be used to decrease the parameter space for a more detailed model using a computational fluid dynamic (CFD) approach as presented in the second part of this work. The higher fidelity model can be implemented to elucidate flow details such as pressure drop, boundary layer development, and the structure and distribution of vapor in the channel.

In particular, emphasis is placed on the cold-start problem, during which an electric heater must provide all of the power required for full vaporization. The cold-start power requirement will represent a worst-case situation because no energy input will be available from
the plasma discharge. The primary performance metrics for a successful vaporizer design are divided into two main requirements:

- Maximize thermal efficiency
- Provide full vaporization with safety margin

Maximizing the thermal efficiency of the vaporizer requires minimizing the heat load required for full vaporization. From an operational standpoint, safety margin can be expressed in multiple ways. In this paper, however, margin in the vaporizer design is always expressed as vapor superheat. If full vaporization is not achieved at the vaporizer exit, liquid lithium will be expelled into the outlet plenum. This is undesirable because liquid expulsion can clog the multichannel hollow cathode or cause the thruster to run improperly. By superheating the lithium vapor above the saturation temperature, the vaporizer should be able to reliably deliver vaporized lithium even with some fluctuations in mass flow rate and heater power. The acceptable range of these fluctuations is a major goal of this work. In addition, vapor superheat will insure against any potential condensation in the outlet plenum during cold-starts.

Of particular concern is identification of the power requirements, location of the dryout point in the lithium flow, and pressure drop required for feed system sizing. By studying the sensitivity of these performance quantities to various design parameters, the major performance drivers of the system can be identified. The end result of the modeling will be to provide a better understanding of two-phase flow field, thus elucidating potential for optimization.

1.3.2 Approach

For reasons that are discussed in detail in upcoming sections, the cathode model presents numerous computational difficulties. To meet the project goals identified above, a successful model should provide details about the physics occurring within the vaporizer while also
providing a tool that can be used quickly to observe the consequences of modifying design parameters. To achieve both of these objectives, a parallel modeling effort was required.

First, a low-order, analytical model was developed to provide a tool that could be used to quickly evaluate performance sensitivities. This approach is called a network model because it is based on a series (or network) of thermal resistances throughout the cathode. Secondly, a higher-order finite-volume CFD model with conjugate heat transfer was developed to provide a solution of higher-fidelity. This model is referred to as a mixture model because of the multiphase modeling approach employed in the CFD simulation. The results for both modeling techniques will be compared to determine areas of agreement/disagreement between the distinct solution approaches. Solution methodologies specific to each individual modeling approach are presented in each approach’s respective chapter.

1.4 Computer Programs and Resources

The network model was coded and solved using Matlab 7.0 R14 (Mathworks). For the CFD model, solid geometries of the volumes were created using Solidworks (Solidworks). The CAD geometries were stored with parasolid file extensions and imported into a commercially available meshing application, GAMBIT (Fluent). In GAMBIT, meshing algorithms were applied and subsequently imported into the commercial solver, Fluent 6.2.16 (Fluent). Post-processing was accomplished using a combination of built-in Fluent capabilities, TecPlot software (Amtec Engineering), as well as additional Matlab scripts. The raw source code for all computer simulations and post-processing algorithms is included in Appendix B.

Because of the size of the CFD model, parallel simulations were implemented in an effort to reduce computing time. The simulations were executed on a Sunfire V20z computer with dual
2.4 GHz AMD Opteron 64 processors with 8 GB of RAM on a LINUX Redhat Advanced Server 4 operating system. Even with high-performance computing resources, simulation time for the CFD model remained lengthy. To fully resolve and converge on a single CFD solution for the entire cathode assembly, approximately two months were required (on average).

1.5 Thesis Organization

This section outlines the basic structure of the thesis. Chapter 2 presents some background material relevant to both modeling efforts completed in this project, including the flow characterization of microchannels and previous studies on lithium MPD thrusters. Chapter 3 presents the network model, including the development and validation of the model using data obtained from MPD thruster testing at the Moscow Aviation Institute (MAI). Performance sensitivities are shown for several key design parameters, and detailed results are presented for the conceptual ALFA\(^2\) geometry. Chapter 4 presents a higher-fidelity finite-volume CFD model of the cathode assembly, including the governing equations and the solution approaches that were required to reduce the overall computational size of the model. Results of this modeling effort are presented for a range of flow rates, and are compared with predictions made by the network model. Conclusions specific to each modeling approach are discussed in Chapter 5, along with some general observations and design recommendations. Chapter 6 outlines future work that can be completed to improve on the models described in this thesis, as well as some recommendations for additional work that can support transient modeling. Expressions used to calculate lithium and tungsten properties throughout this thesis, as well as the raw source code for all simulations, are included in the Appendices.
2 BACKGROUND

Two-phase microchannel flows have been the subject of an increasing number of studies because of their broad application to a number of microfluidic heat transfer applications such as heat pipes, fuel cells, and combustors. Microchannels, in general, have potential for enhancing heat transfer in these applications. In fact, early-generation lithium vaporizers for MPD applications have been treated as an open-ended heat pipe [6].

In order to meet the goals for this thesis of providing a flexible modeling tool while characterizing details of the vaporizing flow field, a parallel modeling effort was required. These two modeling approaches are fundamentally very different. Therefore, all background and literature review specific to each individual modeling technique will be presented in each approach’s respective chapter.

2.1 Flow Characterization

In order to properly model and interpret the results of the two-phase lithium flow in the vaporizer, the behavior of the flow needs to be characterized in terms of whether or not the flow will exhibit macro or microchannel characteristics. The most general definition of a microchannel is the channel size at which observed flow characteristics begin to diverge from that of macroscale channels. The channel dimensions resulting in this crossover is strongly dependent on the properties of the fluid. In two-phase flows, for example, channel diameters being of the same order as a bubble diameter will behave different from that of macroscale bubbly flow.

As a practical matter, Kim et al. define a microchannel to be any channel with a hydraulic diameter between 0.1-5 mm [8]. This criterion is empirical in nature, and is based on the results
of a series of experiments using a variety of fluids. Because the flow dynamics are a strong function of the fluid properties, the actual hydraulic diameter corresponding to the macro-microflow crossover will vary for different fluids. Worth noting, however, is that the ALFA$^2$ vaporizer channels (see Section 1.1.1) fits within this loose definition, with hydraulic diameters of less than 1.15 mm.

To quantitatively classify the vaporizer channel as a microchannel, a more rigorous set of guidelines is required. Based on physical arguments, a set of criteria has been established that describe flows that exhibit microchannel behavior [8,9]. These three criteria are:

\[
Eo = \frac{(\rho_L - \rho_v)gd_H^2}{\sigma} << 1
\]  

\[
1 \leq We_{LS} = \frac{U_{LS}^2d_H\rho_L}{\sigma} \text{ or } We_{VS} = \frac{U_{VS}^2d_H\rho_V}{\sigma} \leq 100
\]  

\[
Re_{LS} = \frac{U_{LS}d_H\rho_L}{\mu_L} \gg 1
\]

where $\sigma$ is the surface tension of lithium, and $\rho_L$ and $\rho_v$ are the liquid and vapor densities respectively. The liquid Reynolds number $Re_{LS}$ is based on the superficial liquid velocity, which is the velocity of the liquid in the channel if it was flowing as a single-phase flow. That is,

\[
U_{LS} = \frac{G_L}{\rho_L},
\]  

where $G_L = (1-x)G_T$ is the liquid mass flux for a quality $x$. The superficial vapor velocity $U_{VS}$ is defined in a similar manner, with $G_v = xG_T$.

Examination of these non-dimensional parameters gives better physical insight into the characteristics of a microchannel flow. An Eotvos number less than unity implies that surface tension dominates buoyancy effects. This is an important insight, as the diminished importance of buoyancy effects indicates that the flow dynamics will exhibit a general insensitivity to
channel orientation. This allows for the usage of correlations based on horizontal channels for the pressure drop and convection coefficients in the helical channel. Also, a liquid Reynolds number much greater than unity means that inertia forces are also significant. The Weber number is an additional non-dimensional number used to determine the relative magnitudes of the two dominant forces in the microchannel system, inertia and surface tension. From Eq. (2-2), it follows that surface tension will dominate the inertial forces of the liquid or be dominated by the inertial forces of the vapor by no greater than two orders of magnitude in microchannel flow dynamics.

Table 2-1. Lithium properties used to evaluate microchannel criteria

<table>
<thead>
<tr>
<th>Parameter/Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total mass flux, $G_f$</td>
<td>10.87 kg/m$^2$s</td>
</tr>
<tr>
<td>Liquid density, $\rho_L$</td>
<td>399 kg/m$^3$</td>
</tr>
<tr>
<td>Vapor density, $\rho_v$</td>
<td>0.0522 kg/m$^3$</td>
</tr>
<tr>
<td>Hydraulic diameter, $d_H$</td>
<td>1.15x-3 m</td>
</tr>
<tr>
<td>Surface Tension, $\sigma$</td>
<td>0.214 N/m</td>
</tr>
<tr>
<td>Liquid Viscosity, $\mu_L$</td>
<td>1.21x-4 kg/ms</td>
</tr>
</tbody>
</table>

The relevant set of lithium properties is listed in Table 2-1 for a saturation temperature of 1620 K (at atmospheric pressure). Using these properties and evaluating Eq. (2-1), we see that the Eotvos number is 0.024. In the limit as $x \to 0$, the liquid Reynolds number is 103. As quality increases ($x \to 1$), the liquid Reynolds number will steadily decrease to zero, eventually dipping below unity. As a practical matter, however, the criterion stated in Eq. (2-4) is satisfied
until \( x = 0.9903 \), which accounts for the majority of the two-phase flow. Similarly, the Weber number for the liquid phase as \( x \to 0 \) is 2.81, satisfying the requirement until \( x = 0.404 \). The vapor Weber number goes to zero as \( x \to 0 \). In the limit as \( x \to 1 \), \( We_v = 12.2 \), therefore satisfying Eq. (2-2) for all qualities. Based on the above criteria, the channel for the baseline ALFA\(^2\) vaporizer can be categorized as a microchannel for the two-phase lithium flow. This classification will drive the modeling approaches and aid in interpreting the modeling results.

### 2.2 Previous Studies

The primary body of analytical and experimental work focused on MPD thrusters comes from Princeton University. While the ALFA\(^2\) vaporizer has not been modeled specifically, the results and insights of these earlier studies help support the present modeling efforts by providing a basis for choosing and applying boundary conditions.

A thermal analysis of an early-generation lithium vaporizer (i.e. open-ended heat pipe) was conducted by Emsellem, Kodys, and Choueiri [10]. This study was inspired, in part, by the observation of condensed lithium in the cathode emitter during their experimental work. Therefore, the objective of the study was to identify and evaluate any major heat sinks in the steady-state system, as well as to develop innovative strategies for thermal management within the thruster assembly. From an operational standpoint, condensation of lithium vapor can have drastic consequences, as it can restrict or clog the lithium flow through the multichannel emitter.

Their analysis was performed using ANSYS software to calculate temperature distributions in the cathode and thruster assemblies. Thermocouples had been used to experimentally record the temperatures at various locations in the cathode assembly during operation. This data was subsequently used to compare with the numerical results. These results are summarized in Table 2-2 for the locations sampled in the cathode assembly.
The model did not include the effect of the lithium flow in the temperature distribution throughout the cathode. While the geometry and the heater placement are slightly different for this early generation cathode, several key results/trends can be extended to the present study, including:

- Abrupt temperature decrease below the heat sunk mounting flange.
- Actively cooled power leads result in asymmetries in the isotherms of the system.
- Large fall-off in cathode temperature in the direction of the cathode tip for cold-starts.

The temperature of the cathode root was calculated to be 1003 K (730 °C), in excellent agreement with their experimental results. In the present study, this location corresponds to one of the boundaries of the computational domain. Therefore, this result provides a good estimate for the boundary condition to be used along this surface.

<table>
<thead>
<tr>
<th>Thermocouple Location</th>
<th>Numerical</th>
<th>Experimental</th>
<th>Disagreement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Under Cathode Heater</td>
<td>1720 °C *</td>
<td>1720 °C</td>
<td>N/A</td>
</tr>
<tr>
<td>Center of Cathode Tip</td>
<td>430 °C</td>
<td>470 °C</td>
<td>4.5 %</td>
</tr>
<tr>
<td>Cathode Root at Flange</td>
<td>730 °C</td>
<td>725 °C</td>
<td>0.7 %</td>
</tr>
<tr>
<td>Backplate/Insulator Ring</td>
<td>580 °C</td>
<td>610 °C</td>
<td>5.1 %</td>
</tr>
<tr>
<td>Anode Lip</td>
<td>280 °C</td>
<td>305 °C</td>
<td>8.5 %</td>
</tr>
</tbody>
</table>

* Note: For this value, they use an experimentally measured value as a boundary condition in the numerical model.

2.3 MAI Thruster

During the 1990s, a lithium-fed MPD thruster was designed, built, and tested at the Moscow Aviation Institute (MAI) [11]. The main objective of their work was to demonstrate the technical feasibility of MPD thruster operation. Due to the similarities with the ALFA²
cathode, the limited test data available for the MAI cathode is used to validate the modeling approaches presented in this work. This test data is listed in Table 2-3.

The 200 kW MAI thruster provides a good baseline for comparison because it shares many of the same features of the ALFA$^2$ cathode design. Figure 2-1 highlights these similarities in the cathode and vaporizer design. While the basic structure of the assembly remains the same, a couple of noteworthy differences are the fixturing method of the graphite heater and the single piece construction of the mounting flange with a portion of the cathode tube.

<table>
<thead>
<tr>
<th>Test</th>
<th>Minimum Flow (mg/s)</th>
<th>Maximum Flow (mg/s)</th>
<th>Heater Power (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>55</td>
<td>107</td>
<td>3600</td>
</tr>
<tr>
<td>2</td>
<td>96</td>
<td>106</td>
<td>3600</td>
</tr>
<tr>
<td>3</td>
<td>92</td>
<td>104</td>
<td>3600</td>
</tr>
<tr>
<td>4</td>
<td>104</td>
<td>125</td>
<td>3600</td>
</tr>
<tr>
<td>5</td>
<td>128</td>
<td>128</td>
<td>3600</td>
</tr>
</tbody>
</table>

Note: MAI vaporizer tube length = 21.0 cm

Figure 2-1. Schematic of the MAI cathode showing the features similar to ALFA$^2$ [11].
3 NETWORK MODEL

In this chapter, a first-order thermal model of the cathode/vaporizer assembly is introduced and explained in detail. This thermal model is called the network model because the approach adopted was to use a one-dimensional (radial) network of thermal resistances to determine the temperatures at multiple radial locations between the inner vaporizer tube wall and the outer cathode tube wall. This radial network in turn is solved at different axial positions along the vaporizer length. The primary benefit of such a model is to quickly test the thermal performance sensitivity of the vaporizer to several key design parameters such as surface emissivity, vaporizer geometry, flow rate, boundary conditions, and thermodynamic end state. In this way, the network model can stand alone or be used in conjunction with higher fidelity CFD models to decrease the parameter space required for investigation. To validate the solution approach, results are compared with limited data collected for the Russian-built MAI thruster discussed in Section 2.3 [11].

This chapter is organized as follows. Section 3.1 describes how the network model is developed, as well as defines the thermal resistances used in the model. A flow map is presented in Section 3.2 that is used to assess likely flow patterns in the channel. Sections 3.3, 3.4, and 3.5 describe the auxiliary relations used in the network model for convection coefficients, radiation view factors, and pressure drop, respectively. The solution methodology is detailed in Section 3.6, and the boundary conditions and simulation parameters are reported in Section 3.7. Finally, results are presented in Section 3.8 for the conceptual ALFA\textsuperscript{2} vaporizer including performance sensitivity to various design parameters.
3.1 Model Description

The model is divided into two segments; one for the portion of the vaporizer tube that includes the helical flow channels and one for the pre-channel portion extending from the base of the heater flange to the inlet to the helical channels. Figure 3-1 represents an axial cutaway of a portion of the vaporizer tube closest to the inlet. The inlet tube is not modeled explicitly, but the initial lithium temperature is taken to be a value representative of what it would be entering the tube. The vaporizer tube, cathode tube, and mounting flange radii are measured from the centerline indicated by the horizontal dotted line. The geometric variables shown in Figure 3-1, along with the channel dimensions described later were selected to represent either the MAI or ALFA\textsuperscript{2} thruster as closely as possible. The two components of the model are described in the following sections.

![Figure 3-1. Representative cross section of vaporizer tube showing pre-channel region which includes the lithium “pool”, heater flange, and mounting flange.](image)
The network model is categorized as a first-order model based on the assumption that the heat flow is mostly in the radial direction. This implies that the net axial heat flow is small relative to the radial heat flux. This assumption allows for an analytical solution of the radial temperature distribution throughout the cathode assembly.

In radial coordinates, the general form of energy conservation through a solid is [12]:

$$k \nabla^2 T + \dot{q} = \rho c \frac{\partial T}{\partial t}$$  \hspace{2cm} (3-1)

where $\dot{q}$ is volumetric heat generation. For a steady-state system with no heat generation, the Laplacian operator can be expanded,

$$\frac{1}{r} \frac{\partial}{\partial r} \left( kr \frac{\partial T}{\partial r} \right) + \frac{1}{r^2} \frac{\partial}{\partial \theta} \left( k \frac{\partial T}{\partial \theta} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right) = 0.$$  \hspace{2cm} (3-2)

For a uniform conductivity through each radial shell, and based on the radial heat flow assumption (from the inner vaporizer tube wall to the outer cathode tube wall), the conservation of energy in the vaporizer becomes,

$$\frac{1}{r} \frac{d}{dr} \left( r \frac{dT}{dr} \right) = 0.$$  \hspace{2cm} (3-3)

Applying the temperature boundary conditions at the outer and inner radial locations (subscripts “$o$” and “$i$”, respectively),

$$T = T_i \quad \text{at} \quad r = r_i$$

$$T = T_o \quad \text{at} \quad r = r_o$$  \hspace{2cm} (3-4)

the solution to Eq. (3-3) is given by,

$$T(r) = T_i - (T_i - T_o) \frac{\ln \left( \frac{r}{r_i} \right)}{\ln \left( \frac{r}{r_o} \right)}.$$  \hspace{2cm} (3-5)
This temperature is seen to be a function of radial position. Also, the heat load into an element of the inner vaporizer tube wall can be calculated from

\[ q_i = -kA \left( \frac{dT}{dr} \right)_{r=r_i} \]  

(3-6)

Substituting the temperature expression in Eq.(3-5), the heat into the inner surface of the vaporizer tube becomes,

\[ q_i = \frac{kA}{\ln\left(\frac{r_i}{r_o}\right)} (T_i - T_o). \]  

(3-7)

Equation (3-7) can be expressed in the form of an Ohm’s Law,

\[ q_i = \frac{(T_i - T_o)}{R_i} \]  

(3-8)

where the temperature gradient, \((T_i - T_o)\), is the driving potential, the heat flow, \(q_i\), is analogous to electrical current, and \(R_i\) is the thermal resistance to the heat flow which, for conduction, is given by,

\[ R_i = \frac{\ln\left(\frac{r_i}{r_o}\right)}{kA} \]  

(3-9)

By applying Eq. (3-8) at specified intervals through the composite vaporizer assembly and solving the system of equations simultaneously, the temperatures of any set of radial locations can be determined. Although the heat conduction equation was used in the development of Eq. (3-8), the result is more general. In areas where convection or radiation replace conduction, the thermal resistance from Eq. (3-9) is modified to reflect these changes in the mode of heat transfer.
3.1.1 Channel Region

Figure 3-2 and Figure 3-3 represent a section of the vaporizer tube and cathode tube which has been cut axially, along with its corresponding resistive network. The channel height, channel width, and groove width are given by $a$, $b$, and $c$ respectively. The shaded regions represent cross sections of a helical channel through which lithium flows into (or out of) the page. Figure 3-4 shows a portion of the vaporizer tube with a single, helical vaporizer channel. At a given axial position ($z$-coordinate), conservation of energy is applied through a volume with a perimeter indicated by the rectangular dotted line and a depth $\Delta s$ along the channel (flow path) coordinate ($s$ coordinate). The axial and channel-wise coordinates, $z$ and $s$, are shown in Figure 3-4. This figure also shows the outline of a representative control volume element. The channel and groove widths ($b'$ and $c'$) which define the control volume are slightly smaller than the values ($b$ and $c$) measured in an axial cross section (Figure 3-2) because of the angle introduced by the groove pitch, $p$. The pitch shown in Figure 3-4 corresponds to a single lead or channel. For cases with multiple channels, the pitch for that specific channel is used to correctly relate the fluid path length to cathode length along the $z$-coordinate (i.e., $z = z(s, p)$). The step length $\Delta s$ (along the channel) is taken to be a small fraction of the channel width ($\Delta s \ll b$) so that a planar geometry for the control volume can be assumed.

![Figure 3-2. Schematic of the vaporizer section in region with channels.](image)
Figure 3-3. Resistive network representation of vaporizer section in region with channels.

Figure 3-4. Segment of vaporizer tube showing element of vaporizer channel used in thermal network analysis.

The temperatures represented in the network and the corresponding heat flow vectors are also shown in Figure 3-2. The direction of the heat flow vectors in the figure correspond to what has been taken as the positive sense. In generating the vaporizer performance curves, heat flux to the inner wall of the vaporizer tube is the independent variable and assumed uniform along the length of the vaporizer tube. Heat flows radially outward from the inner vaporizer tube wall,
through the solid groove and fluid into the cathode tube. The equivalent resistive network for this system is shown in the schematic in Figure 3-3. The node representing the outer wall temperature is connected to one of the two circuit elements as shown depending on whether the point of integration is under the flange.

The region of the flow under the mounting flange presents a unique challenge. The heat flow in this region is inherently multi-dimensional since elements of the network will see different thermal resistances to the radiating surfaces at each of the flange faces. To approximate this variation, an approach was adopted in which the flange is represented by four separate nodes (Network B in Figure 3-3). These four nodes are located at the inner and outer radii ($MF,B$ and $MF,O$), as well as at two equally-spaced radial locations within the flange ($MF,1$ and $MF,2$).

The radiation view factors from these surfaces to the surroundings, including the upstream cathode tube (view $A$), heater flange (view $B$), the vacuum chamber (view $C$), the downstream cathode tube (view $D$), and the anode (view $E$), were calculated from closed form analytical expressions for the approximate geometry as described in Section 3.4. The three radiating surfaces of the mounting flange are the upstream and downstream facing surfaces ($\Delta A_{US}$ and $\Delta A_{DS}$, respectively) and the outer facing rim surface ($\Delta A_{RIM}$). These surfaces are divided into a number of smaller area elements equal to the number of network elements under the flange. So for example, if each fluid channel has 270 elements or integration steps under the flange, and there are four channels, then the fraction of the area on each face used in the network system solution for one of the integration steps will be equal to $1/(4 \times 270) = 1/1080^{th}$ of the total surface area for that face. The surface of the mounting flange in contact with the mounting bus bar ($\Delta A_{COND}$) is also handled in this manner. These surface areas are incorporated into the thermal resistances listed in Table 3-1.
Table 3-1. Definition of the resistances in the channel network equations

<table>
<thead>
<tr>
<th>Res.</th>
<th>Equation</th>
<th>Res.</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_1$</td>
<td>$\frac{t_{CT} - a}{k_{VT} \left( b' + c' \right) \Delta s}$</td>
<td>$R_2$</td>
<td>$\frac{a}{2k_{VT} c' \Delta s}$</td>
</tr>
<tr>
<td>$R_{2C}$</td>
<td>$\frac{1}{c' \Delta s} \left( \frac{a}{2k_{VT}} + R_c' \right)$</td>
<td>$R_3$</td>
<td>$\frac{t_{CT}}{k_{CT} \left( b' + c' \right) \Delta s}$</td>
</tr>
<tr>
<td>$R_{F,B}$</td>
<td>$\frac{1}{h_y b' \Delta s}$</td>
<td>$R_{F,T}$</td>
<td>$\frac{1}{h_y b' \Delta s}$</td>
</tr>
<tr>
<td>$R_{F,Y}$</td>
<td>$\frac{1}{2h_y a \Delta s}$</td>
<td>$R_{R}'$</td>
<td>$\left( \frac{1}{\epsilon_{CT}} + \frac{r_{C,O}}{r_{AN}} \left( \frac{1}{\epsilon_{AN}} - 1 \right) \right) \frac{T_{W,O}^{\alpha s}}{\Delta s (b' + c')}$</td>
</tr>
<tr>
<td>$R_4$</td>
<td>$\frac{r_{MF} - r_{C,O}}{3k_{MF} \left( b' + c' \right) \Delta s}$</td>
<td>$R_5$</td>
<td>$\frac{r_{MF} - r_{C,O}}{3k_{MF} \left( b' + c' \right) \Delta s}$</td>
</tr>
<tr>
<td>$R_6$</td>
<td>$\frac{r_{MF} - r_{C,O}}{3k_{MF} \left( b' + c' \right) \Delta s}$</td>
<td>$R_{R,O}'$</td>
<td>$\frac{T_{MF,O}^{\alpha s}}{\sigma \epsilon F_t \Delta A_{RM}}$</td>
</tr>
<tr>
<td>$R_{R,A}'$</td>
<td>$\frac{T_{MF,2}^{\alpha s}}{\sigma \epsilon F_t \Delta A_{US}}$</td>
<td>$R_{R,B}'$</td>
<td>$\frac{T_{MF,2}^{\alpha s}}{\sigma \epsilon F_t \Delta A_{US}}$</td>
</tr>
<tr>
<td>$R_{R,C}'$</td>
<td>$\frac{T_{MF,2}^{\alpha s}}{\sigma \epsilon F_t \Delta A_{US}}$</td>
<td>$R_{R,D}'$</td>
<td>$\frac{T_{MF,2}^{\alpha s}}{\sigma \epsilon F_t \Delta A_{US}}$</td>
</tr>
<tr>
<td>$R_{R,E}'$</td>
<td>$\frac{T_{MF,3}^{\alpha s}}{\sigma \epsilon F_t \Delta A_{OS}}$</td>
<td>$R_{C,BB}$</td>
<td>$\frac{R^*}{\Delta A_{COND}}$</td>
</tr>
</tbody>
</table>

This is only an approximate approach for a couple of reasons. First, the view factors used correspond to those calculated for the entire face area, and secondly, this approach assumes each element under the flange will have the same thermal resistance to the radiating surface, regardless of whether it is right under an edge or within the middle of the flange. To the accuracy of the network model as a whole, this approximation is reasonable. In particular, this approach allows both conduction and radiation heat loss from each fluid element under the
flange to be captured. In addition, the approach captures the fact that different elements of the flange radiating surfaces will be at different temperatures as a result of the fluid below it.

3.1.2 Pre-Channel Region

In the channel network just described, the network equations are solved at different locations along the channel to determine the axial temperature profile. In the pre-channel region, there is no axial temperature profile calculated. Rather, the set of conservation equations is solved once in order to estimate the heat loss through this portion of the vaporizer tube, and also to calculate the temperature of the fluid at the point where it enters the vaporizer channels. The liquid “pool” acts as an inlet plenum. The low mass flux through this pool (due to a larger cross-sectional flow area) can result in significant heating of the fluid. The possibility of premature boiling in this pool region is something to be avoided as it will result in unsteady flow.

Figure 3-5. Schematic of the vaporizer section in pre-channel region.

The pre-channel portion of the vaporizer tube and its corresponding resistive network are shown in Figure 3-5 and Figure 3-6. The multidimensional geometry is captured in an
approximate way in the thermal resistances used in the network equations. For example, the flange surface area (facing downstream) increases the surface area available for radiative emission, which lowers the impedance to radiation. In addition, the upstream facing flange area is clamped down to a base temperature, $T_{S,HF}$, through a contact resistance. Another feature of the pre-channel region is the location of the liquid pool, which is not centered in the control volume. The pool actually is bounded by a channel wall on the downstream side (on the right in Figure 3-5). This is accounted for by using the area of both channel walls in the thermal resistance $R_{F,V}$.

Figure 3-6. Resistive network representation of the vaporizer pre-channel region
Note that in Figure 3-5, the feedtube geometry is not included. The feedtube would have effect making the prechannel non-axisymmetric, thus lowering the overall resistance in a single azimuthal direction. As a practical matter, any heat that bleeds out of the pre-channel region via the feedtube will mostly serve to pre-heat the inlet fluid, and thus have little or no impact on the overall thermal efficiency of the vaporizer. In addition, the feedtube cross-sectional area is very small (less than one percent of the total outer surface area of the pre-channel region). For these reasons, the effect of the feedtube is neglected in the pre-channel resistive network, thus preserving axial symmetry in calculating the thermal resistances.

Table 3-2. Definition of the resistances in the pre-channel network equations

<table>
<thead>
<tr>
<th>Res.</th>
<th>Equation</th>
<th>Res.</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_4$</td>
<td>$\frac{L_{NC} - t_{HF}}{\pi \left[ \left( r_{C,j} + t_{VT} - a \right)^2 - r_{C,l}^2 \right] k_{VT}}$</td>
<td>$R_5$</td>
<td>$\frac{1}{2\pi t_{HF} k_{VT}} \ln \left( \frac{r_{C,j} + t_{VT}}{r_{C,l}} \right)$</td>
</tr>
<tr>
<td>$R_6$</td>
<td>$\frac{1}{2\pi (L_{NC} + L_p - t_{HF}) k_{VT}} \ln \left( \frac{r_{C,j} + t_{VT} - a}{r_{C,l}} \right)$</td>
<td>$R_7$</td>
<td>$\frac{1}{2\pi (L_{NC} - t_{HF}) k_{VT}} \ln \left( \frac{r_{C,j} + t_{VT} - 0.5a}{r_{C,l} + t_{VT} - a} \right)$</td>
</tr>
<tr>
<td>$R_8$</td>
<td>$\frac{1}{2\pi (L_{NC} - t_{HF}) k_{VT}} \ln \left( \frac{r_{C,j} + t_{VT}}{r_{C,l} + t_{VT} - 0.5a} \right)$</td>
<td>$R_9$</td>
<td>$\frac{(L_{NC} - t_{HF})}{\pi \left( \left( r_{C,j} + t_{VT} \right)^2 - \left( r_{C,j} + t_{VT} - a \right)^2 \right) k_{VT}}$</td>
</tr>
<tr>
<td>$R_{10}$</td>
<td>$\frac{1}{2\pi t_{HF} k_{CT}} \ln \left( \frac{r_{C,O}}{r_{C,j} + t_{VT}} \right)$</td>
<td>$R_{11}$</td>
<td>$\frac{1}{2\pi (L_{NC} + L_p - t_{HF}) k_{CT}} \ln \left( \frac{r_{C,O}}{r_{C,j} + t_{VT}} \right)$</td>
</tr>
<tr>
<td>$R_{12}$</td>
<td>$\frac{1}{2\pi (L_{NC} - t_{HF})} \ln \left( \frac{\left( \frac{r_{C,O}}{r_{C,j} + t_{VT}} \right)^2}{r_{C,i}^2 - \left( r_{C,j} + t_{VT} \right)^2} \right) k_{CT}$</td>
<td>$R_{13}$</td>
<td>$\frac{1}{2\pi t_{HF} k_{HF}^3} \ln \left( \frac{r_{C,O} + 0.5 \left( r_{HF} - r_{C,O} \right)}{r_{C,O}} \right)$</td>
</tr>
<tr>
<td>$R_{15}$</td>
<td>$\frac{1}{2\pi t_{HF} k_{HF}} \ln \left( \frac{r_{HF}}{r_{C,O} + 0.5 \left( r_{HF} - r_{C,O} \right)} \right)$</td>
<td>$R_{16}$</td>
<td>$\frac{(L_{NC} - t_{HF})}{\pi \left( \frac{r_{C,O}}{r_{C,i} + t_{VT}} \right)^2 - \left( \frac{r_{C,O}}{r_{C,i} + t_{VT}} \right)^2} k_{CT}$</td>
</tr>
<tr>
<td>$R_{R,OB}$</td>
<td>$\frac{T_{W,OB}^3}{\sigma \varepsilon \pi r_{C,O} \left( L_{NC} + L_p - t_{HF} \right)}$</td>
<td>$R_{R,HF}$</td>
<td>$\frac{T_{HF}^3}{\sigma \varepsilon \pi t_{HF} t_{HF}^3}$</td>
</tr>
<tr>
<td>$R_{F,T}$</td>
<td>$\frac{1}{2\pi h_F \left( r_{C,j} + t_{VT} \right) L_p}$</td>
<td>$R_{F,V}$</td>
<td>$\frac{1}{4\pi h_F \left( r_{C,j} + t_{VT} - \frac{a}{2} \right)^2 \alpha}$</td>
</tr>
</tbody>
</table>

$R_{F,B} = \frac{1}{2\pi h_F \left( r_{C,j} + t_{VT} - a \right) L_p}$
To better account for the influence of the conduction heat sink at the base of the heater flange, axial heat flow paths are included in the network system. In Figure 3-6, the network is divided into two segments. Segment $A$ is the portion of the cathode tube and vaporizer tube directly under the heater flange. Segment $B$ includes all of the remaining cathode assembly in the pre-channel region, including the inlet plenum. To better estimate the heat lost through the contact surface, $T_{S, HF}$, these two regions are coupled by including axial heat flow resistances in the network system.

The resistances used in the different sections are significantly different as seen in Table 3-1 and Table 3-2. The entire pre-channel portion of the vaporizer tube is solved with one resistive network. Therefore, a planar geometry cannot be assumed as is done in the channel network in which the conservation of energy is applied to what are essentially differential rectangular elements along the fluid path. The thermal resistances developed for the pre-channel therefore reflect this radial geometry.

3.2 Flow Regimes

In order to properly classify the flow pattern for a given set of conditions, a flow map is needed. A flow map attempts to correlate the structure of vapor packets to specific flow parameters. Insight into the flow patterns likely to be observed throughout the boiling region can be helpful in not only interpreting the results, but also in driving the selection of the auxiliary models such as convection coefficients and pressure correlations.

Many different flow maps based on different sets of criteria have been proposed in the literature for macro-scale flows [13,14], and more recently for micro-scale flows [15,16]. For the present study, the flow map used was empirically developed and comes from recent work
done by Cubaud and Ho [17]. As with most experimental two-phase research, the fluid combination used was deionized (DI) water and air. The type of channel used to develop the Cubaud and Ho flow map, however, is very similar to the channel used in the ALFA$^2$ (and MAI) configuration. The Cubaud and Ho map uses a horizontal, rectangular channel with hydraulic diameters of 0.200 mm and 0.525 mm. The ALFA$^2$ channel is rectangular and 1.15 mm (nominally) in hydraulic diameter.

Figure 3-7. Flow map with experimental data from Cubaud & Ho, 2004 [17]

Figure 3-7 includes the experimental data set and flow pattern transitions obtained in the study. The flow mapping is based only on the superficial phase velocities (defined by $J_L$ and $J_G$ in Figure 3-7), which define what the velocity of a phase would be if it was flowing through the channel as a single phase [23]. Because the phasic superficial velocities depend on the liquid and vapor flow rates, fluid properties, and channel geometry, the effects of these parameters are
captured by the flow mapping of Figure 3-7. When plotted on a logarithmic axis, the flow map distinguishes between five distinct flow structures (Figure 3-8):

- Bubbly flow
- Wedging flow
- Slug flow
- Annular flow
- Dry Flow

In bubbly flow regime, the vapor phase is distributed as discrete spherical bubbles in a continuous liquid phase. The bubble diameter is smaller than the channel height, and the bubbles are free to collide and coalesce, thus forming larger bubbles. This flow type is observed when the void fraction is very small ($\alpha_v < 0.3$). In the bubbly flow regime, gravitational forces (buoyancy) can be significant, as the individual bubbles are free to translate under an off-axis buoyancy force [17].

The wedging flow consists of larger, elongated bubbles. The vapor wedge is confined by the channel walls, where the bubble diameter would otherwise be larger than the characteristic channel gap. Plugs of liquid separate individual vapor slugs. A typical range of void fractions for this flow pattern is $0.3 < \alpha_v < 0.8$. In the wedge flow regime, buoyancy becomes dominated by surface tension and inertial effects, which makes the gravitational effects less important.

As the void fraction increases to larger values, $\alpha_v > 0.8$, vapor wedges begin to coalesce and form larger vapor slugs. In this vapor slug regime, bubble size is much greater than the channel gap. A film of liquid separates the vapor slug from direct contact with the wall. Liquid plugs are thinner and more dispersed in the flow. As the void fraction increases further,
the liquid film between the slug and the wall starts becomes smaller and the liquid plugs begin to evaporate. When the liquid plug evaporates, the vapor slugs coalesce into a continuous flow of vapor within the channel core. This type of flow pattern is annular flow, so named because of the liquid annulus surrounding the central vapor flow. This flow regime is complex and inherently unsteady because of transient effects such as ripples, flooding, and the appearance of wave structures in the liquid film [17].

In regions of extremely high void fractions, $\alpha_v > 0.995$, a dry flow regime is observed. The liquid film along the wall evaporates into the central vapor core. In rectangular channels,
the remaining liquid tends to gather in the channel corners. When the last of the liquid film evaporates, the dryout point has been reached, and a single-phase vapor flow results.

In general, these flow patterns are consistent with the types seen in other two-phase research [15,16], although differences can exist in the transition regions. The flow patterns are different from those observed in macrochannels due to the suppression of buoyancy effects, completely suppressing separated flows. In the flow patterns observed in microchannel flow of Figure 3-8, the liquid phase is typically trapped along the wall. The exception is the bubbly flow regime where bubbles can stick to the channel walls if adhesion forces are greater than drag forces. This is ideal in a sense, because the trapped liquid film prevents the low thermal conductivity vapor from insulating the flow.

3.3 Convective Heat Transfer

Correct evaluation of the heat transfer coefficients is critical if one is to gain useful insight into the performance characteristics of the vaporizer. Because they define the resistance of the heat flow into the fluid, the heat transfer coefficients will determine the locations of boiling onset and dryout points, and will ultimately govern the overall length and thermal efficiency of the vaporizer.

3.3.1 Liquid Region

In the pre-channel portion of the vaporizer, as well as in the region of the channels where the flow is entirely liquid (from fluid inlet to the onset of boiling), the correlation employed comes from Sleicher and Rouse [18], formulated specifically for low Prandtl number fluids ($Pr << 1$), and is given by,

$$Nu = 6.3 + 0.167 Re_D^{0.85} Pr^{0.93}$$

(3-10)
Here, $Re_d$ is evaluated at the local fluid temperature, and $Pr$ is evaluated at the wall surface temperature, which is taken to be a local, weighted average over all four channel walls at a given position along the channel. In this liquid region, the mean Prandtl number of the liquid lithium is $Pr = 0.0071$. To calculate the Reynolds number in the pre-channel region, the area used for calculating the hydraulic diameter is the annular cross-section of the inlet plenum. Because the flow model is one-dimensional, the bulk temperature of the element is taken to be the local fluid temperature at that given position. The significant increase in fluid temperature from the feedtube inlet to the onset of boiling (~1100 K) makes capturing the effects of temperature-dependent fluid properties important.

### 3.3.2 Two-phase Region

In the multiphase region, two heat transfer mechanisms interplay to account for the total heat transfer into the fluid [19]. Initially, nucleate boiling will dominate the boiling regime. At higher vapor fractions, convective boiling becomes more dominant as the remaining liquid is evaporated trapped along the channel walls in evaporated into the vapor core. No and Kazimi’s correlation accounts for these effects and was developed specifically for calculating the boiling heat transfer of liquid metals [20]. The correlation takes the form:

$$Nu = 0.152 \frac{F_1}{F_2} Pr_L Re_{LO}^{0.90} \phi_L$$

(3-11)

where, the subscript “$L$” refers to the conditions of the saturated liquid. The Reynolds number is based on the liquid mass flux,

$$Re_{LO} = \frac{G(1-x)d_H}{\mu_L}$$

(3-12)
where $x$ is the quality (or vapor mass fraction), representing the fraction of the total mass flow that has been converted to vapor. The Peclet Number used in Eq. (3-19), $\text{Pe}_{LO}$, is also corrected in this manner.

In the correlation given by Eq. (3-11), the parameter $\phi_L$ represents an empirical correction factor developed by Zeigarnick and Litvinov [Eq. (3-14)] in terms of a simplified Martinelli parameter [Eq.(3-15)] [20]. This parameter is defined as,

$$\phi_L = (\phi_L)_{tt}^{0.88}$$

(3-13)

where

$$(\phi_L)_{tt} = \left[1 + \frac{20}{X_H} + (X_H)^{-2}\right]^{1/2}$$

(3-14)

and,

$$X_H = \left(\frac{1-x}{x}\right)^{0.9} \left(\frac{\rho_V}{\rho_L}\right)^{0.5} \left(\frac{\mu_L}{\mu_V}\right)^{0.1}$$

(3-15)

The values for $F_1$ and $F_2$ are dependent on $\text{Re}_{LO}$ and are obtained from the following expressions:

For $\text{Re}_{LO} < 50$

$$F_1 = 1.5$$

$$F_2 = 0.7071(\text{Re}_{LO})^{0.5} \text{Pr}_L$$

(3-16)

For $50 \leq \text{Re}_{LO} \leq 1125$

$$F_1 = 1.563$$

$$F_2 = 5 \text{Pr}_L + \left(\frac{5}{E}\right) \ln \left[1 + E \text{Pr}_L \left(\frac{5}{5} - 1\right)\right]$$

(3-17)
For $Re_{LO} > 1125$

$$F_1 = 1.818$$

$$F_2 = 5\Pr_L + \left(\frac{5}{E}\right)(1 + 5E\Pr_L) + \left(\frac{6}{E\gamma}\right)\ln\left[\frac{2M + \gamma - 1}{1 + \gamma - 2M}\left(\frac{1 + \gamma - \beta}{\beta + \gamma - 1}\right)\right]$$

(3-18)

The additional parameters used in these relations are defined as follows:

$$E = 0.00375P_{eLO}\left[1 - \exp\left(-0.00375P_{eLO}\right)\right]$$

(3-19)

$$\delta^+_C = 0.4818\left(Re_{LO}\right)^{0.585}$$

(3-20)

$$\gamma = \left[1 + \left(\frac{10M}{E\delta^+_N\Pr_L}\right)^{0.5}\right]$$

(3-21)

$$M = 1 - (\alpha)^{0.5}$$

(3-22)

$$\alpha = \left[1 + \left(\frac{1-x}{x}\right)\left(\frac{\rho_v}{\rho_l}\right)\right]^{-1}$$

(3-23)

$$\delta^+_N = 0.133\left(Re_{LO}\right)^{0.7614}$$

(3-24)

$$\beta = 60\left(\frac{M}{\delta^+_N}\right)$$

(3-25)

From Eqs. (3-12)-(3-15), we see that the Nusselt number calculated using this correlation has a strong dependence on the vapor mass fraction. As the vapor fraction approaches one (full vaporization), both the liquid phase Reynolds number and the Martinelli parameter approach zero. Although somewhat offset by the Reynolds number approaching zero, the appearance of the Martinelli parameter in the denominator of the Zeigarnick and Litvinov factor causes this correlation to predict unreasonably high heat transfer coefficients for high quality values ($x > 0.95$). Therefore, in the implementation of this algorithm, the quality is limited to a maximum value of 0.95 when evaluating Eq. (3-15).
3.3.3 Vapor Region

In the vapor region, the flow remains laminar and has a Prandtl number of order 1. The Nusselt number for this fully-developed flow will tend to be a constant value determined only by the channel’s cross-sectional aspect ratio \( (a/b') \) from Figure 3-2 and Figure 3-4). The Nusselt number used for the simulation was obtained by interpolation of channel geometries listed in Holman [21]. For the baseline channel geometry of the ALFA\(^2\) vaporizer, the value of the heat transfer coefficient is calculated from \( \text{Nu} = 4.95 \).

3.4 Radiation View Factors

To calculate the thermal radiation resistances from Table 3-1, the view factors from the mounting flange to various components of the cathode assembly must be calculated. As illustrated in Figure 3-9, the upstream facing side mounting flange has view to the upstream outer cathode surface (\( F_A \)), the heater flange (\( F_B \)), and the vacuum chamber (\( F_C \)). On the downstream facing side, the inner portion of the mounting flange views the downstream cathode tube surface (\( F_D \)) and the anode or grounding screen (\( F_E \)). These view factors are determined using closed-form expressions taken from Naraghi & Chung for radiation views from annular rings (i.e. flanges) to various classes of axisymmetric bodies [22].

![Figure 3-9. Illustration of the radiation views from the mounting flange.](image-url)
Figure 3-10. Illustration of the radiation views from the mounting flange to various components of the cathode assembly.

On the upstream side of the flange, the view factor to the outer cathode surface (surface \( A \)) can be directly calculated. Referring to geometry of Figure 3-10, the expression for view \( A \) is given by [22],

\[
F_A = \frac{1}{\pi (1 - R^2)} \left\{ (1 - R^2) \tan^{-1} \left( \frac{1 + R^2}{1 - R^2} \right)^{\frac{1}{2}} - \left[ (1 + R^2 + L^2 - 4R^2)^{\frac{1}{2}} \right. \right. \\
\left. \tan^{-1} \left[ \frac{(1 + R^2 + L^2 + 2R^2)(1 - R^2)}{(1 + R^2 + L^2 - 2R^2)(1 + R^2)} \right]^{\frac{1}{2}} \right. \\
\left. + \frac{1}{2} L^2 \cos^{-1} R + 2RL \tan^{-1} \left[ \frac{(1 - R^2)^{\frac{1}{2}}}{L} \right] \right\}
\]

where

\[
R = \frac{R_{CO}}{R_{MF}}, \text{ and } L = \frac{L_F}{R_{MF}} \tag{3-27}
\]

and \( L_F \) is the axial distance between the mounting flange and the heater flange.

To visualize the view factor from the mounting flange to the heater flange, an auxiliary cone is traced in Figure 3-10. This cone is constrained by the outer cathode tube radius (or the
mounting flange inner diameter) and the outer radius of the heater flange. The view factor from the mounting flange surface to this truncated cone is given by the expression,

\[
F_{B,tot} = \frac{1}{\pi(1-R^2)} \left\{ -A'B'tan^{-1}\left(\frac{A'C'}{B'D'}\right) + (C'D')^2 tan^{-1}\left(\frac{D'}{C'}\right) + \right.
\]

\[
\left. \sin \psi \left[ \left( \frac{H^2 + 2HR}{\tan \alpha} \right) tan^{-1} \sqrt{\frac{E'}{H}} + E'tan^{-1}\left( \frac{H}{\sqrt{E'}} \right) \right] + \right.
\]

\[
\left( \frac{H^2}{2\cos^2 \psi} + HR \tan \psi \right) \cos^{-1}(R) \right\} \right.
\]

where

\[
A' = \sqrt{H^2 + (1 + H\tan \psi + R)^2}
\]

\[
B' = \sqrt{H^2 + (1 - H\tan \psi - R)^2}
\]

\[
C' = \sqrt{1-R}
\]

\[
D' = \sqrt{1+R}
\]

\[
E' = (1 - R^2)\cos^2 \psi
\]

and \(R\) has been previously defined in Eq. (3-27).

The view factor from the mounting flange to the cone (\(F_{B,tot}\)) obtained from Eq. (3-28) includes the radiation view to the outer cathode tube, \(F_A\). Therefore, the view to the only the heater flange is obtained using,

\[
F_B = F_{B,tot} - F_A.
\]

The remaining view factor on the upstream side of the mounting flange is to the far field ambient temperature, \(T_{INF}\). Because the three view factors capture all possible views and are mutually exclusive, the ambient view is equal to,
\[ F_C = 1 - F_B - F_A. \] (3-30)

The downstream side of the flange is handled in a similar manner, with the exception that there are only two views; the downstream outer cathode surface and the concentric anode. Replacing \( L_f \) with the distance between the mounting flange and the cathode tip, \( F_D \) can calculated using Eq. (3-26). The remaining view to the anode is then calculated using \( F_E = 1 - F_D \).

### 3.5 Pressure Drop

In general, the pressure losses through the channel length will have three components. These three components are frictional, gravitational, and accelerational (or inertial). By expressing the momentum equation in terms of pressures, the total pressure gradient (provided by the feed system for example) across an element can be expressed by the relationship [23],

\[
\left( \frac{dP}{ds} \right)_{\text{tot}} = \left( \frac{dP}{ds} \right)_{\text{fric}} + \left( \frac{dP}{ds} \right)_{\text{grav}} + \left( \frac{dP}{ds} \right)_{\text{acc}} \] (3-31)

This equation is just a restatement of the momentum equation with the “applied” pressure gradient on the left hand side and all other terms collected on the right hand side. In all regions of the vaporizer channel, the head loss due to gravitational effects is neglected because over the length of the helical channel, there is no significant net elevation change. Furthermore, the magnitude of the Eotvos number (\( \text{Eo} \ll 1 \), as discussed in Section 2.1) suggests that gravitational effects (i.e. buoyancy) will be negligible compared to the effects of surface tension and inertia. In the single phase regions (liquid or vapor), the change in momentum flux across an element is also negligible, assuming a fully-developed flow. Therefore, in these single-phase regions, the “accelerational” pressure losses (following the terminology of Whalley [23]) are ignored. In the
two phase region, however, increases in the void fraction across an element can make inertia forces significant, resulting in potentially large increases in the flux of momentum through an element, $\Delta s$. Therefore, accelerational pressure losses need to be included in the pressure drop model for the two-phase flow regime.

3.5.1 **Single-phase Flows**

Recent studies in the field of single phase microchannel flow have shown that pressure drop correlations developed for macrochannel flows perform reasonably well at predicting the pressure drop in microchannels [8,24]. Therefore, in the fully developed, laminar regimes, the frictional pressure losses are calculated using the Darcy-Weisbach equation [21], which includes dependence on the flow characteristics ($\text{Re}_D$), channel geometry ($A_{sa}, d_H$), and local mean volumetric kinetic energy ($\rho v^2/2$). Along the path coordinate $s$, the Darcy-Weisbach equation takes the form,

$$
\left( \frac{dP}{ds} \right)_{fri} = \frac{4f}{d_H} \frac{\rho v^2}{2}
$$

(3-32)

where $f$ is the Fanning friction factor. For laminar flows, the friction factor is a function only of the local Reynolds number and channel geometry [12], and is defined as,

$$
f = \frac{C}{\text{Re}_D}
$$

(3-33)

where,

$$
C = 16\exp\left(0.294B^2 + 0.068B - 0.318\right)
$$

(3-34)

where $C$ is a function of a dimensionless parameter $B$; the cross-section shape number,
For the baseline geometry of \( \text{ALFA}^2 \), the cross-section shape number is \( B = 0.563 \) and the constant \( C = 13.3 \). This single phase pressure drop model agrees well with experimental data from experimental studies using similar sized rectangular channels \[8\]. This equation is solved a single time in the pre-channel region to estimate the pressure loss in the feedtube and inlet plenum, and then across each single phase element along the helical channel.

### 3.5.2 Two-phase Flow

**Frictional Pressure Losses**

The subject of two-phase frictional pressure loss in microchannels has been the focus of very few experimental studies \[36\]. Unfortunately, unlike single-phase flows, macrochannel correlations perform poorly at predicting two-phase pressure losses in microchannel flow \[8\]. Therefore, correlations formulated specifically for two-phase, non-adiabatic flows are used to form the two-phase pressure drop model.

To calculate the pressure drop in the boiling regime, the concept of a two-phase frictional multiplier \( \phi_L \) is used. In general form, the frictional pressure drop in the two-phase region is given by \[23\],

\[
\left( \frac{dP}{ds} \right)_{2p,f} = \phi_L \left( \frac{dP}{ds} \right)_{l,f}
\]

where the two-phase multiplier, \( \phi_L \), is defined using the Lockhart-Martinelli correlation,

\[
\phi_L = 1 + \frac{C}{X} + \frac{1}{X^2}
\]
where the Martinelli parameter $X$ is a function of the single phase pressure drops of each individual phase,

$$X = \left[ \frac{(dP/ds)_{L,\text{frie}}}{(dP/ds)_{V,\text{frie}}} \right]^{1/2}.$$  \hspace{1cm} (3-38)

The parameter $C$ is dependent on the characteristics of the lithium flow. The correlation used for $C$ in the present study assumes a wedging/slug flow structure (see Section 3.2) that takes into account the effects of surface tension, channel gap size, and flow rate on the two-phase pressure gradient. It was developed specifically for rectangular microchannel flow [25] and is defined as,

$$C = A\lambda^\alpha \psi^\beta \text{Re}_{LO}^\gamma$$  \hspace{1cm} (3-39)

with,

$$\lambda = \frac{\mu^2_L}{\rho_L \sigma d_H}$$  \hspace{1cm} (3-40)

and,

$$\psi = \frac{\mu_L U_{LS}}{\sigma}.$$  \hspace{1cm} (3-41)

The Darcy-Weisbach expression [Eq. (3-32)] is used to calculate the single phase pressure drops needed for evaluation of the Martinelli parameter in Eq. (3-38). The constants $A$, $q$, $R$, and $S$ in Eq. (3-39) are obtained from Table 3-3 for different combinations of liquid-vapor flow regimes. The liquid Reynolds number $\text{Re}_{LO}$ is defined by Eq. (3-12) and the superficial velocity of the liquid $U_{LS}$ is given by Eq. (2-4).
Table 3-3. Constant and exponents required for calculating the parameter $C$ [25].

<table>
<thead>
<tr>
<th>Flow Regime Liquid</th>
<th>Flow Regime Gas</th>
<th>$A$</th>
<th>$q$</th>
<th>$R$</th>
<th>$S$</th>
<th>Range of $X$</th>
<th>Range of $Re_{LO}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laminar Liquid</td>
<td>Laminar Gas</td>
<td>$6.833 \times 10^{-8}$</td>
<td>-1.317</td>
<td>0.719</td>
<td>0.557</td>
<td>0.776-14.176</td>
<td>175-1480</td>
</tr>
<tr>
<td>Laminar Liquid</td>
<td>Turbulent Gas</td>
<td>$6.185 \times 10^{-2}$</td>
<td>0</td>
<td>0</td>
<td>0.726</td>
<td>0.303-1.426</td>
<td>293-1506</td>
</tr>
<tr>
<td>Turbulent Liquid</td>
<td>Laminar Gas</td>
<td>3.627</td>
<td>0</td>
<td>0</td>
<td>0.174</td>
<td>3.276-79.415</td>
<td>2606-17642</td>
</tr>
<tr>
<td>Turbulent Liquid</td>
<td>Turbulent Gas</td>
<td>0.408</td>
<td>0</td>
<td>0</td>
<td>0.451</td>
<td>1.309-14.781</td>
<td>2675-17757</td>
</tr>
</tbody>
</table>

Accelerational Pressure Losses

In the following, a momentum flux model is developed to calculate the inertial (or accelerational) pressure drop across a fluid element. Figure 3-11 illustrates a segment of the channel region that is used as the control volume. The momentum balance for this control volume can be written simply as:

Rate of momentum in $= \dot{m}v_{in} = \rho_{m,in}A v_{in}^2$

Rate of momentum out $= \dot{m}v_{out} = \rho_{m,out}A v_{out}^2$

where $A$ is the constant cross-sectional area of the channel. Because liquid is being converted to low-density vapor across the element, the mixture (or overall) density will tend to decrease from the inlet to outlet. Thus, to conserve mass, the mixture velocity $v$ will increase across the (constant area) element, resulting in a net flux of momentum. This momentum flux is equivalent to the accelerational force exerted on the flow in this segment, and is defined as,

\[ \text{Rate of change of momentum} = F_{acc} = A\left(\rho_{m,out}v_{out}^2 - \rho_{m,in}v_{in}^2\right) \]  \hspace{1cm} (3-42)

where the inlet and outlet velocities at the control volume boundaries are defined as,

\[ v_{in} = \frac{G}{\rho_{m,in}}, \quad \text{and} \quad v_{out} = \frac{G}{\rho_{m,out}}. \]  \hspace{1cm} (3-43)
Continuity ensures that the mass flux through the vaporizer channel, \( G \) [kg/m\(^2\)-s], will be constant regardless of the quality or mixture density. Substituting Eq. (3-43) into Eq. (3-42) and rearranging, the inertial component of the pressure drop becomes:

\[
\Delta P_{\text{acc}} = \frac{F_{\text{acc}}}{A} = G^2 \left( \frac{1}{\rho_{m,\text{out}}} - \frac{1}{\rho_{m,\text{in}}} \right) \quad (3-44)
\]

Defining the mixture density as a volume-weighted average of the phase densities, \( \rho_m = \alpha \rho_v + (1 - \alpha) \rho_L \), the final form of the accelerational pressure drop is obtained:

\[
\Delta P_{\text{acc}} = G^2 \left( \frac{1}{\alpha_{\text{out}} \rho_v + (1 - \alpha_{\text{out}}) \rho_L} - \frac{1}{\alpha_{\text{in}} \rho_v + (1 - \alpha_{\text{in}}) \rho_L} \right) \quad (3-45)
\]

Because the network model assumes a constant vapor density and the two-phase mixture is at a constant temperature (\( T = T_{\text{sat}} \)), the inertial pressure drop \( \Delta P_{\text{acc}} \) becomes a function only of the increase in the vapor volume fraction across the element. Correct evaluation of the vapor volume fraction is therefore required for proper characterization of the accelerational component of the pressure losses in the two-phase boiling regime.

![Figure 3-11. Control volume illustrating the momentum flux across element.](image)

The vapor volume fraction, or synonymously the void fraction, is a function of the fluid quality, the density ratio between phases, and the velocities of the individual phases [23],
where the velocity ratio $v_v / v_L$ is frequently referred to as the slip ratio, $S$. In the limiting case of $S = 1$, the system reduces to a homogeneous flow with phases interacting so fully that they travel at the same velocity. In general, however, the vapor phase can be moving at a much greater velocity than the liquid phase, resulting in slip ratios significantly greater than unity. Physically, the less dense vapor phase tends to peel past the higher density liquid phase.

In the network model, as well as in experimental work, the quality and phase densities from Eq. (3-46) are known quantities at each control volume. The only remaining unknown quantities are the individual phasic velocities. For this reason, correlations for the void fraction are actually a correlation for the slip ratio determined by the relative (or slip) velocities of the two phases [23]. In calculating the slip ratio, the two most important parameters are,

- density ratio, $\rho_v / \rho_L$
- vapor quality, $x$

The slip ratio also shows a lesser dependence on the mass flux $G$, the flow patterns (as discussed in Section 3.2), and hydraulic diameter $d_H$ [23]. The correlation used in the network model comes from Chisholm [26], and includes dependence on the two most dominant drivers of the slip ratio; the phase density ratio and vapor quality,

$$ S = \left( 1 + x \left( 1 - \frac{\rho_L}{\rho_v} \right) \right)^{0.5} \quad (3-47) $$

From Eq. (3-47), it is seen that $S \to 1$ as $\rho_L / \rho_v \to 1$. This is consistent with the characterization of a homogeneous flow. In the limit as the vapor quality goes to zero or to unity (at the onset of boiling and at the dryout point, respectively) we have also,
as $x \to 0, \quad S \to 1$ \hspace{1cm} (Limit 1)

as $x \to 1, \quad S \to (\rho / \rho_v)^{0.5}$ \hspace{1cm} (Limit 2)

The lower bound (Limit 1) is the intuitive statement that there is no relative velocity between phases at the onset of boiling. The statement made in the upper bound (Limit 2) is less obvious.

The following analysis is an effort to illustrate the physical meaning of the upper bound of Eq. (3-47). Consider the translational momentum flux of each individual phase, $L_i = G v_i$ [N/m$^2$], at some arbitrary quality $x$,

$$L_L = G v_L (1 - x) \quad (3-48)$$

$$L_V = G v_V x \quad (3-49)$$

where the phasic velocities are defined generally as [23],

$$v_L = \frac{(1 - x)G}{(1 - \alpha) \rho_L} \quad (3-50)$$

$$v_V = \frac{xG}{\alpha \rho_v} \quad (3-51)$$

Note that these velocities cannot be calculated explicitly because of a dependence on both the quality (known) and the void fraction (unknown). Substituting the phase velocities into the momentum expressions,

$$L_L = \frac{G^2 (1 - x)^2}{\rho_L (1 - \alpha)} \quad (3-52)$$

$$L_V = \frac{G^2 x^2}{\rho_v \alpha} \quad (3-53)$$
By summing each phasic momentum flux, the total momentum flux through the control volume can be expressed as,

\[
L_{\text{tot}} = G^2 \left( \frac{x^2}{\alpha \rho_v} + \frac{(1-x)^2}{(1-\alpha) \rho_l} \right).
\] (3-54)

Differentiating Eq. (3-54) with respect to the void fraction and setting the result equal to zero will define a critical point that corresponds to either a maximum or minimum momentum flux,

\[
\frac{\partial L_{\text{tot}}}{\partial \alpha} = G^2 \left( \frac{-x^2}{\alpha^2 \rho_v} + \frac{(1-x)^2}{(1-\alpha)^2 \rho_l} \right) = 0.
\] (3-55)

It can be shown that Eq. (3-54) has a positive second derivative for all possible values of quality, so this extremum corresponds to a minimum momentum flux. Simplification of Eq. (3-55) provides a relationship between the void fraction and vapor quality for this minimum momentum flux,

\[
\frac{\alpha}{(1-\alpha)} = \frac{x}{(1-x)} \left( \frac{\rho_l}{\rho_v} \right)^{0.5}.
\] (3-56)

Finally, by direct comparison with Eq. (3-46), we arrive at the statement that,

\[
S = \frac{v_v}{v_l} = \left( \frac{\rho_l}{\rho_v} \right)^{0.5}.
\] (3-57)

Therefore, Eq. (3-57) defines the value of the slip ratio corresponding to a minimum momentum flux. This expression for the slip ratio is equivalent to the upper bound of Eq. (3-47). This exercise highlights the overall trend of the Chisholm slip correlation, which varies from \( S = 1 \) to \( S = \left( \frac{\rho_l}{\rho_v} \right)^{0.5} \). That is, the two-phase flow will be homogeneous at the onset of boiling, and will produce a minimum momentum flux at the dryout point.
This is consistent with the flow pattern discussion of Section 3.2 and the inertial pressure drop expression of Eq. (3-45). At the onset of boiling, individual vapor slugs are separated by plugs of liquid. The liquid and vapor have a strong interaction, and therefore move at nearly the same velocities (i.e. the flow is homogeneous). As the flow moves into the “dry” flow regime, the flow is mostly vapor from a volume fraction standpoint. Therefore, smaller changes in the void fraction should be expected at these already high values of void fraction. Consequently, from Eq. (3-45), less of an increase in void fraction across a control volume will result in increasingly lower inertial pressure losses. In other words, the contribution to the required pressure gradient resulting from the phase change will be decreasing.

3.6 Solution Methodology

3.6.1 Pre-Channel Region

The pre-channel region is divided into two segments; a portion which includes a slice under the heater flange and a slice including the inlet plenum (Figure 3-5 and Figure 3-6). The fraction of the total heater power into these two segments (\( q_{IN,A} \) and \( q_{IN,B} \)) and the fluid inlet temperature (\( T_{IN} \)) are used as the independent variables in the network equation. The heat into this region is calculated from a given heater flux and the feedtube inlet temperature is taken to be only slightly greater than the melting point of lithium (\( T_{IN} = 500 \) K). While conservative, this value is consistent with previous work focused on the lithium feed system, where liquid lithium temperatures were measured and found to range from 523 K – 623 K [27].

On the outer cathode surface, a radiation boundary condition to the vacuum chamber is applied with view to a 300 K ambient condition. The pre-channel cathode area emits to a black-body enclosure, assuming the surface area of the vacuum chamber is much larger than the
cathode surface area. To the left of the heater flange, the system is clamped to a 1000 K sink temperature, chosen to match the modeling results and experimental data summarized in Section 2.2 [10]. This heat sink is included to account for parasitic heat loss through conduction to the mounting surface.

To obtain the solution for the pre-channel portion of the vaporizer, the system of equations based on the network from Figure 3-6 and resistances from Table 3-2 is solved using a Gaussian elimination (with partial pivoting) algorithm in coded in Matlab. The radiation resistances, $R_{r,OB}'$ and $R_{r,RF}'$ are introduced in the network equation to linearize the radiation boundary conditions along the outer cathode surface. By grouping terms that include the wall temperature into the radiation resistances as shown in Table 3-2, the equation network becomes a “linear” system at the cost of making the coefficient matrix a function of the unknown surface temperature. This requires that the network equation be solved iteratively. By providing an initial estimate for the outer wall temperatures, $T_{W,OB}'$ and $T_{HF,T}'$, an estimated value for the radiation resistance can be calculated in the coefficient matrix. After solving the system of equations based on this “guess”, the calculated wall temperatures are compared with the initial guess to test convergence. If this comparison satisfies the convergence criterion, the solution for the pre-channel is stored and becomes the inlet condition for the channel region. If the convergence criterion is not satisfied, the outer wall temperatures are updated using a bisection technique using the calculated values for $T_{W,OB}'$ and $T_{HF,T}'$. These updated “guesses” are used for the next iteration, the resistances are recalculated, and the process is repeated until the convergence criterion is satisfied. Because there is a relatively small radial temperature gradient across the cathode cross-section (from the inner wall of the vaporizer tube to the outer wall of the
cathode tube), the convergence criterion is chosen to be very small. For all simulations, the convergence criterion for the outer wall temperatures was set to $1 \times 10^{-9}$ K.

### 3.6.2 Channel Region

The solution for the resistive network in the channel region is obtained in a different manner from the pre-channel region. In the channel region, the network equation is still solved iteratively for the outer wall temperature, as described above, but for each differential element along the path coordinate $s$ from Figure 3-4. Consistent with the planar geometry assumption, small integration steps are used ($\Delta s \sim 0.1$ mm) in the simulations. An additional unknown $q_{1u}$ is included in the network equation in order to explicitly calculate the magnitude of the heat passing into the fluid. Also, while the mass flow rate through the pre-channel section is the total overall mass flow rate through the vaporizer (for the baseline ALFA$^2$ design, $\dot{m} = 80$ mg/s), the portion of this mass flow through each of the individual channels is the total divided by the number of channels. This makes use of the assumption that the mass flux is divided uniformly over each of the channels.

At each element in the channel, the fluid temperature for the next step is updated based on a calculated temperature gradient, $\Delta T_f$. This updated value for the fluid temperature is then used to calculate the relevant set of material properties based on relations from Appendix A and used as the independent variable (along with $q_{1u}$) in the next solution step. Tungsten properties used in the simulation are also temperature-dependent. The average temperature in each solid domain (i.e. cathode tube, vaporizer tube, and flanges) is used to calculate its thermal conductivity, and the outer surface temperature is used to calculate the surface emissivity.
When the fluid temperature reaches the saturation temperature ($T_{SAT} = 1620$ K at 1 atm), the simulation moves into the multiphase region, and new correlations are employed. In this multiphase region, the network equation is modified slightly to account for the fact that heat into the fluid results in an increase in latent heat rather than an increase in the temperature. Therefore, the coefficient matrix is modified to enforce the constraint $\Delta T_v = 0$.

Within this phase change region, the value of the vapor mass fraction is calculated at each step by the evaluating the ratio of the latent heat absorbed by the two-phase mixture to the total amount of latent heat required to complete the phase change (i.e. to achieve dryout). This ratio is the vapor quality (or vapor mass fraction):

$$x = \frac{\dot{h}_{ABS}}{\dot{m} h_{fg}}$$

(3-58)

where $h_{fg}$ is the latent enthalpy of vaporization [J/kg], and $\dot{h}_{ABS}$ is an integration of $q_H$ [W] over the fluid path. The value for the quality is used to determine the completion of the phase change (dryout point) and to monitor any potential condensation under the flange. When $x = 1$, the boiling process is complete, and the simulation is triggered to begin using the vapor phase correlations.

### 3.7 Boundary Conditions and Simulation Parameters

As the integration point moves axially through the channel region, the radiation view from the outer surface of the cathode switches from the laboratory environment to the anode assembly. This transition can be seen in the simplified thruster layout in Figure 1-1. To the left of (and including) the mounting flange, the radiation heat transfer is treated as black body emission to a large surrounding enclosure at 300 K. To the right of the mounting flange, the
cathode assembly is surrounded by either an insulator (ALFA\textsuperscript{2} design) or a tungsten grounding screen (MAI design). In either case, the radiation view changes to 600 K, and the heat flux boundary condition is modeled as two emitting, infinite, concentric cylinders (i.e. the view factor from cathode to anode is assumed equal to unity),

\[
q_{C-A} = \frac{\sigma A_{C,0} \left( T_{C,0}^4 - T_{AN}^4 \right)}{\frac{1}{\varepsilon_{C,0}} + \frac{1 - \varepsilon_{AN}}{\varepsilon_{AN}} \left( \frac{r_{C,0}}{r_{AN}} \right)}
\]

(3-59)

This simplification is valid for the large aspect ratio of the cathode assembly ($L/r_{C,0} > 10$) and the relative proximity of the anode to the cathode.

The mounting flange is also located within the channel region. In the segment under this flange, the network equation changes due to the conduction heat path to the bus bar, as well as the additional equations governing the multiple radiation views. As is seen in Figure 3-12, over its entire upstream facing surface area, the mounting flange has view to the outer cathode tube, heater flange, as well as the vacuum chamber (views $A$, $B$, and $C$, respectively). The temperatures of these surfaces are known, either from the solution to the pre-channel region (cathode tube, $T_{W,OB}$ and heater flange, $T_{HF,T}$) or from simulation inputs (chamber temperature, $T_{INF}$). On the downstream facing side, the mounting flange surface has different boundary conditions on its upper and lower halves. On the lower half, the flange has view to an insulator (or grounding screen depending on the design) and the hot end of the cathode tube (views $D$ and $E$). The insulator temperature is a simulation input, but the temperature for the downstream end of the cathode tube $T_{CATH}$ must be supplied a priori. Because the multiphase regime dominates the portion of the vaporizer downstream of the mounting flange, a reasonable estimate for this
radiation view is assumed to be twenty degrees lower than the saturation temperature. Subsequent checks of this assumption validate the usage of this temperature.

![Figure 3-12. Cathode cross-section showing boundary conditions for the baseline simulation.](image)

The remaining upper half of the mounting flange (facing downstream) is in direct contact with an actively cooled cathode mounting plate (bus bar). Over this surface, a 500 K conduction boundary condition is used to simulate parasitic heat losses to the mounting plate. Because this plate-flange interface will need to conduct several thousand amperes of current from the bus bar to the cathode, it is not practical to increase the thermal resistance to this flange by using an insulating plate. The outer radius/rim of the flange is assumed to emit only to the environment/chamber. To calculate the various view factors, closed form expressions for the representative flange areas are taken from Naraghi and Chung [22] for a class of annular disks emitting to axisymmetric bodies (i.e. cylinders and cones), from which the individual view factor to each component is calculated.

The simulation parameters (and additional assumed values) for the MAI and ALFA² vaporizers are shown in Table 3-4 and Table 3-5. These values were selected to best represent
the actual parameters for the MAI and ALFA\textsuperscript{2} thruster. For the MAI vaporizer, the vaporizer tube is made from molybdenum and the cathode tube is constructed from tungsten. Except for molybdenum flanges, the conceptual ALFA\textsuperscript{2} design specifications include all tungsten construction. Properties for tungsten, including emissivity and thermal conductivity, are temperature-dependent and are listed in Appendix A.

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Value</th>
<th>Conditions/Properties</th>
<th>Value/Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a)</td>
<td>2.0 mm</td>
<td>(\varepsilon_{CT} = \varepsilon_i(T))</td>
<td>0.08-0.24</td>
</tr>
<tr>
<td>(b)</td>
<td>4.0 mm</td>
<td>(\varepsilon_{AN})</td>
<td>0.7</td>
</tr>
<tr>
<td>(c)</td>
<td>2.0 mm</td>
<td>(R_{c})</td>
<td>7\times10^{-6} \text{ m}^{2} \text{ K}/\text{ W}</td>
</tr>
<tr>
<td>(t_{VT})</td>
<td>4.0 mm</td>
<td>(T_{IN})</td>
<td>500 K</td>
</tr>
<tr>
<td>(t_{CT})</td>
<td>3.5 mm</td>
<td>(T_{AN})</td>
<td>600 K</td>
</tr>
<tr>
<td>(r_{c,l})</td>
<td>15 mm</td>
<td>(\Delta T_{SH})</td>
<td>100 K</td>
</tr>
<tr>
<td>(r_{AN})</td>
<td>48 mm</td>
<td>(k_m)</td>
<td>98 \text{ W/mK}</td>
</tr>
<tr>
<td>(L_{NC})</td>
<td>18 mm</td>
<td>(k_i = k_i(T))</td>
<td>103-137 \text{ W/mK}</td>
</tr>
<tr>
<td>(L_p)</td>
<td>30 mm</td>
<td>Channels</td>
<td>3</td>
</tr>
</tbody>
</table>

Note: For 3 channels, \(p = 18 \text{ mm}, b' = 3.95 \text{ mm}, c' = 1.975 \text{ mm}\)

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Value</th>
<th>Conditions/Properties</th>
<th>Value/Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a)</td>
<td>0.75 mm</td>
<td>(\varepsilon_{CT} = \varepsilon_i(T))</td>
<td>0.08-0.24</td>
</tr>
<tr>
<td>(b)</td>
<td>2.5 mm</td>
<td>(\varepsilon_{AN})</td>
<td>0.7</td>
</tr>
<tr>
<td>(c)</td>
<td>2.5 mm</td>
<td>(R_{c})</td>
<td>7\times10^{-6} \text{ m}^{2} \text{ K}/\text{ W}</td>
</tr>
<tr>
<td>(t_{VT})</td>
<td>3.0 mm</td>
<td>(T_{IN})</td>
<td>500 K</td>
</tr>
<tr>
<td>(t_{CT})</td>
<td>9.0 mm</td>
<td>(T_{AN})</td>
<td>600 K</td>
</tr>
<tr>
<td>(r_{c,l})</td>
<td>13.5 mm</td>
<td>(\Delta T_{SH})</td>
<td>100 K</td>
</tr>
<tr>
<td>(r_{AN})</td>
<td>58 mm</td>
<td>(k_m)</td>
<td>98 \text{ W/mK}</td>
</tr>
<tr>
<td>(L_{NC})</td>
<td>14.2 mm</td>
<td>(k_i = k_i(T))</td>
<td>103-137 \text{ W/mK}</td>
</tr>
<tr>
<td>(L_p)</td>
<td>20.2 mm</td>
<td>Channels</td>
<td>4</td>
</tr>
</tbody>
</table>

Note: For 4 channels, \(p = 20 \text{ mm}, b' = c' = 2.453 \text{ mm}\)
For the temperatures calculated throughout the cathode assembly, the range of surface emissivity and thermal conductivity are 0.08-0.24 and 103-137 Wm\(^{-1}\)K\(^{-1}\), respectively. The value for the thermal contact resistance was estimated based on an order of magnitude comparison with similar metal-to-metal contact surfaces from Holman [21]. The exit condition for vapor superheat was taken to be 100 K for all cases.

The component arrangement and dimensions for the analysis were selected so as to approximate the preliminary design to be used in initial laboratory testing, not the simplified, conceptual design shown in Figure 1-1. One benefit of the network model is that any of these parameters can be varied to quickly test the effect on the vaporizer performance.

### 3.8 Results and Discussion

A vaporizer performance curve represents the locus of length-power solutions for a range of heat flux values. The knee in these curves represents the optimal solution from a thermal efficiency standpoint. Above the knee, the upper right portion of the curve represents the low-flux, long-length solution branch. Below the knee, the lower right portion of the curve represents the high-flux, short-length solution branch. Either of these branches represents a departure from the optimal thermal efficiency, either because the increased outer wall temperatures (lower branch) or increased surface area (upper branch) have the effect of increasing the radiation losses.

In the following discussion, the thermal efficiency of the vaporizer is defined by comparing the calculated power to the minimum achievable thermodynamic power. This minimum power requirement is determined only by the inlet temperature, the desired outlet temperature, and fluid properties (specific heat, latent heat of vaporization, and saturation
temperature). Therefore, a thermal efficiency equal to 100 percent implies that all of the supplied energy is absorbed into the lithium.

### 3.8.1 Model Validation

Due to similarities in the functional design (see Section 2.3), the network model is validated using test data for the MAI 200 kW thruster. There are a number of unknown parameters from the MAI tests which are inputs to the model and could affect this comparison. Among others, these include the inlet and outlet thermodynamic states and the boundary conditions on the flanges. In these instances, assumed values were used and taken to be equivalent to the ALFA\(^2\) baseline vaporizer (shown in Figure 3-12).

![Figure 3-13. Comparison of network model results with data from the 200 kW MAI thruster. The eight mass flow rates used to generate the vaporizer performance curves represent the range of flow rates in Table 2-3.](image)

Figure 3-13 shows the results of the network model compared with experimental data for five tests of the MAI 200 kW thruster listed in Section 2.3. The eight flow rates represent the
minimum-maximum bounds for the five tests shown in Table 2-3. All five tests were conducted using the same power and length, so the eight flow rates shown in Table 2-3 correspond to a single data point in Figure 3-13.

While showing an overall agreement in the sense that it brackets the experimental data, the model underpredicts the power requirement for low mass flow rates and overpredicts the high flow rates. A likely cause for this trend is the constant thermodynamic end state that was used for all cases (i.e. 100 K superheated vapor). In reality, for a constant length and power, higher flow rates will tend to have lower exit temperatures, while lower flow rates will dryout sooner and thus have higher exit vapor temperatures. The effect of choosing too high of an end condition would be to bias the curves to the right, while choosing too low of an end condition would bias the curves to the left. Ideally, if the values for the exit temperature for each flow rate tested were known, it would be expected that all the curves would all converge into a single length-power vaporizer curve.

3.8.2 ALFA\(^2\) Baseline

In Figure 3-14, the temperatures calculated by the network model are shown as a function of axial position for the ALFA\(^2\) baseline case listed in Table 3-5. The position coordinate begins at 3.44 cm which is the axial position corresponding to the transition between the pre-channel and channel regions of the vaporizer. The inlet temperature to the channel is the calculated exit temperature from the pre-channel section. In Figure 3-14 and Figure 3-15, we have calculated the profiles out to approximately \(z = 33\) cm (vapor superheat of 330 K) to capture the leveling off of the fluid temperature in the vapor region. For the ALFA\(^2\) baseline parameters, the lithium is seen to be fully vaporized at a point 0.98 cm from the end of the vaporizer tube which has a baseline design length of \(z = 22.86\) cm. In terms of the path length \(s\) along the channel, the
The dryout point is 4.9 cm from the channel exit. The discontinuities in the wall temperatures as the fluid moves under the flange and into the superheated region are a direct result of neglecting any axial heat transfer in the channel region.

Also seen in Figure 3-14 is the effect of the 500 K heat sunk mounting flange on the behavior of the temperature profile. The overall effect is to extract heat out of the fluid, initially causing fluid condensation (quality reduction). After all vapor has been converted to liquid, the heat sink results in significant reductions in temperature of all components, as well as the fluid. Worth noting is that the simulation shows only limited sensitivity to the value for this boundary condition. If the temperature of this heat sink is doubled (1000 K), there is only a modest decrease in the power requirement of 3.9%. This result suggests that the conduction heat flow path through the mounting flange is a major driver of vaporizer performance regardless of its contact temperature.

The leveling of the temperature profile (as \( x \to 33 \text{ cm} \)) is a consequence of decreasing the heat flux to the fluid as seen in Figure 3-15. The discontinuities evident at the points where vaporization begins (~7.5 cm) and ends (~22 cm) are numerical artifacts that result from transitioning to different convection correlations at these points. Under the flange, however, the spikes in the heat flux are consistent with the modeling approach. In this region, the temperature of the cathode drops rapidly due to the 500 K contact surface of the flange. The fluid temperature, however, remains constant until the two-phase flow has fully condensed (i.e. reached a quality of zero). The large temperature gradients that result between the saturated fluid and the relatively cold walls causes the large spikes in the heat flux (at \( z \sim 5 \text{ cm} \)), and are a direct result of neglecting axial heat flow.
Figure 3-14. Calculated vaporizer tube, fluid, and cathode tube temperatures as a function of axial position for the case of vapor superheat of 330 K (baseline ALFA$^2$ length is 22.86 cm).

Figure 3-15. Heat flow through the network as a function of axial position. To highlight the system’s trends, the results are shown for a vapor superheat of 330 K. The baseline ALFA$^2$ parameters are listed in Table 3-5.
From Figure 3-15, it is seen that the largest source of thermal inefficiency is radiation losses from the cathode outer tube; a fact that will be further discussed in the following section. In the superheated vapor region \((L > 22 \text{ cm})\) we see that the heat flux into the vapor from the channel top and bottom \(\left(\dot{q}_{FH,T}^{v} \text{ and } \dot{q}_{FH,B}^{v}\right)\) and sides \(\left(\dot{q}_{FV}^{v}\right)\) approach zero. This is a consequence of the temperature difference between the channel walls and vapor approaching zero. Because the heat flux into a control volume is constant, the heat flow into the fluid cannot increase without limit and some heat is always radiated away from the outer surface of the cathode tube. As the length increases, the fluid temperature will approach equilibrium with the channel walls and the outer surface temperature will steadily increase and approach a value such that all the additional supplied power is radiated away.

![Figure 3-16. Left: Additional vaporizer length required (beyond dryout) for various levels of vapor superheat. Right: Thermal efficiency as a function of vapor superheat. The plots asymptote due to a thermal equilibrium condition at a vapor superheat of approximately 330 K.](image)

This behavior is further evident in Figure 3-16 which presents the cost of the “safety margin” in terms of increased length and decreased thermal efficiency for increasing values of superheat. The percentage length increase is measured relative to the length needed to reach the
dryout point, $L_{\text{DRY}}$. From Figure 3-14 for the baseline case, $L_{\text{DRY}} = 21.88$ cm (along the $z$-coordinate). Any increase beyond this point will add safety margin (with respect to liquid expulsion which should be avoided) but will also increase the surface area available for heat loss and will lower the thermal efficiency as shown in Figure 3-16. Beyond 200 K of vapor superheat, the loss of efficiency becomes significant.

In Figure 3-17, we show the evolution of the vapor quality and the corresponding void fraction (i.e. vapor volume fraction). The two-phase region is seen to account for the majority of the channel due to the high enthalpy of vaporization for lithium (19.56 MJ/kg). For the baseline case, the vapor quality is seen to increase fairly linearly, while the void fraction exhibits a sharp increase at the onset of boiling. This is caused by the large density difference between the liquid phase (399 kg/m$^3$) and vapor phase (0.05 kg/m$^3$). Because of this large density difference, a small amount of mass that is converted to vapor will require a much larger volume. For example, 1 mg of liquid lithium will occupy a volume of 2.5 mm$^3$. The same amount of mass of vaporized lithium will fill a volume of 20,000 mm$^3$. Thus, the lithium vapor quickly dominates the volume of the channel while only accounting for a very small fraction of the mass.

![Figure 3-17. Evolution of the vapor quality and the void fraction during vaporization.](image-url)
Figure 3-18 shows the flow map for the baseline ALFA\textsuperscript{2} case. For clarity, the markers are reported at every 100 integration steps. Consistent with the steep increase in void fraction from Figure 3-17 and the subsequent dependence of the flow pattern on the void fraction (Section 3.2), Figure 3-18 shows that it is likely that the flow quickly transitions from a wedging flow to an annular and dry type flow pattern. The majority of the two-phase flow is a dry flow, where liquid is trapped only in the corners of the channel and a thin liquid film surrounds the vapor core. Of particular interest is that the isolated bubbly flow regime is completely suppressed, as the characteristic bubble diameter exceeds the channel diameter (due to the relatively large surface tension effects of lithium). The physical picture is that the liquid film surrounding the vapor core has much less volume, but still accounts for the majority of the mass flow.

![Flow Map](image)

Figure 3-18. Cubaud and Ho flow map for the two-phase portion of the lithium flow.

In Figure 3-19, the pressure profile across the vaporizer is shown, with an overall pressure drop of 5.4 kPa (41 torr). The pressure drop is seen to be most significant in the vapor
region of the flow, which is caused, in part, by an increase in flow Reynolds number due to a lower vapor viscosity. Also, the low density vapor flows at a much greater velocity, causing the single-phase pressure drop given by the Darcy-Weisbach equation [in Eq. (3-32)] to be large. Consequently, along with increasing the required heater power, adding safety margin (i.e. increasing exit temperature) can have the undesired effect of increasing the pressure drop by increasing the length of the channel containing this high velocity vapor flow. Therefore, Figure 3-19 represents an additional cost (for feed system sizing) as a result of introducing margin into the system. As a point of comparison, adding only 1.47 cm of vaporizer length will increase the vapor superheat at the exit from 100 K to 200 K, but will increase the pressure drop by 5.1 kPa. This represent a 94.5 percent increase in the pressure drop across the vaporizer channels.

![Figure 3-19. Pressure versus axial position for ALFA² baseline vaporizer channel.](image)

3.8.3 Performance Sensitivity

The next six figures present the results of the network model for the baseline ALFA² design using a temperature-dependent emissivity for the outer cathode tube surface. In Figure 3-20, the sensitivity of vaporizer thermal performance to cathode tube emissivity is shown for
the nominal flow rate of $\dot{m} = 80$ mg/s. For each individual curve, the emissivity is held constant (not temperature-dependant) to observe the effects on radiation losses. We can see in Figure 3-20 that the emissivity of the outer cathode tube is a strong driver of vaporizer thermal performance. The steepening of the curves, as well as the lower, optimal power point as the emissivity is decreased, is a result of an increasing efficiency as more of the heat is captured by the fluid.

![Figure 3-20. ALFA² vaporizer power calculated using network model. Sensitivity to emissivity of cathode tube surface for nominal mass flow rate of $\dot{m} = 80$ mg/s.](image)

Also, the knee in the curve is not as pronounced for lower emissivities, indicating that the departure from the optimum operating point is not as severe as radiation losses are decreased. Also, seen in Figure 3-20 is that lowering the emissivity tends to increase the optimal length. This is due to the fact that as the radiation heat loss is reduced, the longer length results in lower heat fluxes. This then results in lower pre-channel temperatures, reducing the conduction heat
loss out through the base of the heater flange. For the current ALFA$^2$ design, the vaporizer tube length is 22.86 cm which corresponds to a calculated heater power of approximately 3.9 kW (for a constant emissivity of 0.3).

The root cause for the rapid drop-off in thermal efficiency observed in Figure 3-16 is the increase of radiative heat losses for large values of exit superheat due to increases in the outer surface area (due to increases in vaporizer length). To combat this issue, radiation shielding can be implemented in the design. When positioned around the cathode, shielding (such as tantalum foil [37]) can be used to remove this asymptotic efficiency decrease. In fact, increases in thermal efficiency can actually result from radiation shielding as shown in Figure 3-21. This figure was obtained by regenerating Figure 3-16 using lower values of (constant) surface emissivity that simulates the effect of the shielding.

![Figure 3-21](image-url)

**Figure 3-21.** Sensitivity of thermal efficiency to vapor superheat for different values of surface emissivity. Lowering the surface emissivity mimics the effect of including radiation shielding in the thruster design.
The leveling (and eventual increase) of the thermal efficiency is a consequence of reducing the radiation losses on the outer cathode surface. A longer vaporizer means that the heater power is distributed along a larger cathode surface, resulting in lower surface heat fluxes. By reducing the magnitude of the heat flux, less energy is available to be lost to the conduction heat sinks at the upstream end of the vaporizer.

The implication of using radiation shielding is therefore to shift the dominant heat loss mechanism from radiation to conduction, thereby favoring a longer cathode to transfer energy further from the upstream flanges. This behavior is also evident in Figure 3-20, as the vaporizer performance curves steepen as emissivity is lowered. While an optimal performance point exists at $\varepsilon = 0.2$, the vaporizer efficiency does not suffer as significant of a drop off (compared to the higher surface emissivities). As emissivity approaches zero, the upper branch of the performance curve will steepen to the point such that no optimal point exists. That is, the curve will monotonically decrease from short-length, high-power to long-length, low-power solutions.

Figure 3-22. ALFA$^2$ vaporizer power calculated using network model. Sensitivity to mass flow rate for a temperature-dependent emissivity.
Figure 3-22 shows the thermal performance sensitivity of the vaporizer to mass flow rate. For given initial and final thermodynamic states, the minimum power (vaporizer thermal efficiency 100%) scales linearly with the mass flow rate. This is evident in Fig. 3-22 from the nearly equal spacing of the curves. As a point of reference, at $\dot{m} = 80 \text{ mg/s}$, the minimum power to vaporize the lithium is $P_{h,\text{min}} = 1.992 \text{ kW}$, so the maximum vaporizer efficiency for assumed geometry and inlet/outlet states is approximately 60%.

The slight variation in the shape of the curves is a result of second-order effects such as changes in convective heat transfer coefficient as the Reynolds number in the channels increase. This enhances the heat transfer, resulting in a slight increase in the optimal length as the mass flow rate increases. Because more heat is absorbed into the fluid, less heat is available to be lost through radiation. A longer vaporizer tube lowers the heat flux, which in turn lowers the surface temperature of the dead region in the pre-channel region (i.e. the section which has no fluid flow, $L_{NC}$ in Figure 3-1). This aids in reducing the heat loss through radiation and through the conduction heat sink at the base of the flange.

Figure 3-23 shows the sensitivity of the pressure drop to mass flow rate for a constant vaporizer length. The constant length used was the ALFA$^2$ baseline length of 22.8 cm. The pressure drop scales non-linearly with the mass flow rate because of the nonlinear dependence on the Reynolds number and mass flux variations. For the ALFA$^2$ baseline, the pressure drop is 5.4 kPa (∼ 41 torr). Note that by holding the length constant while changing the mass flow rate, a corresponding change in heater power is applied. For each mass flow rate reported in Figure 3-23, the heater power required to provide the baseline cathode length can be obtained by using Figure 3-22. This non-linear pressure drop scaling is more severe when fewer channels are used. Decreasing the number of channels directly increases the mass flux through each channel, as
well as increases the Reynolds number of the flow (potentially causing a turbulent vapor region). In high mass flux cases, this has the effect of significantly increasing the pressure drop through the two-phase region of the flow.

![Graph](image_url)

Figure 3-23. Pressure drop sensitivity to total mass flow rate (top) and number of channels (bottom) for the baseline flow rate of 80 mg/s.

In Figure 3-24 we show vaporizer length-power curves similar to those presented for the emissivity and mass flow rate sensitivity analysis. This curve provides an estimate of the increase in heater power needed to increase the level of vapor superheat for a given length. An estimate of what constitutes an acceptable safety margin will require some analysis of the flow instabilities which could lead to oscillation of the dryout point. If this fluctuation is large enough,
liquid could be ejected into the plenum. If this information is available, the required margin can be expressed in terms of superheat and Figure 3-24 consulted to estimate the heater power needed to provide that margin.

Figure 3-24. Sensitivity of the power/length requirements to vapor superheat (i.e. exit temperature) for the nominal mass flow rate of 80 mg/s.

Figure 3-25. Optimal vaporizer length for a range of flow rate, emissivity, and superheat.
Figure 3-25 shows the optimal vaporizer lengths with respect to variations in the three parameters investigated in foregoing discussion. The optimal lengths are taken from Figure 3-20 for surface emissivity, Figure 3-22 for mass flow rate, and Figure 3-24 for changes in vapor superheat. Additional sensitivity studies were completed to provide additional data points to smooth out these curves. The trend seen in Figure 3-25 reinforces the point that as the vaporizer operates more efficiently, either by decreasing radiation losses, enhancing heat transfer into the fluid, or limiting the safety margin, the optimum operating length tends to become longer.

<table>
<thead>
<tr>
<th>Boundary Location</th>
<th>Boundary Condition</th>
<th>Power, kW</th>
<th>Length, cm</th>
<th>Comments/Sensitivity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mounting Flange</td>
<td>500 K</td>
<td>3.52</td>
<td>22.8</td>
<td>Baseline Case</td>
</tr>
<tr>
<td></td>
<td>1500 K</td>
<td>3.25</td>
<td>21.4</td>
<td>7.7% power decrease</td>
</tr>
<tr>
<td>Heater Flange</td>
<td>500 K</td>
<td>3.52</td>
<td>22.8</td>
<td>No Sensitivity</td>
</tr>
<tr>
<td></td>
<td>1500 K</td>
<td>3.52</td>
<td>22.8</td>
<td>No Sensitivity</td>
</tr>
<tr>
<td>Ambient</td>
<td>0 K</td>
<td>3.52</td>
<td>22.8</td>
<td>No Sensitivity</td>
</tr>
<tr>
<td></td>
<td>500 K</td>
<td>3.52</td>
<td>22.8</td>
<td>No Sensitivity</td>
</tr>
<tr>
<td>Insulator/Anode</td>
<td>300 K</td>
<td>3.55</td>
<td>23.4</td>
<td>0.8% power increase</td>
</tr>
<tr>
<td></td>
<td>1000 K</td>
<td>3.38</td>
<td>22.3</td>
<td>4.0% power decrease</td>
</tr>
<tr>
<td>Fluid Inlet</td>
<td>453 K*</td>
<td>3.52</td>
<td>22.8</td>
<td>No Sensitivity</td>
</tr>
<tr>
<td></td>
<td>600 K</td>
<td>3.52</td>
<td>22.8</td>
<td>No Sensitivity</td>
</tr>
</tbody>
</table>

* The melting point of lithium is 453 K

Finally, we show in Table 3-6 the sensitivity of the calculated solution (in terms of power and length) to variations in the applied thermal boundary conditions. The last column of the table comments on the effects of varying the boundary conditions with respect to the solution to the 80 mg/s flow rate case. To facilitate a comparison, the baseline result is shown in the first row of the table. We see that varying each boundary condition over a fairly large (but
reasonable) temperature range results in only modest changes in the power requirements. The largest boundary sensitivities are the temperatures of mounting flange heat sink and the anode. This makes sense physically because these boundaries represent the dominant heat loss mechanisms from the cathode. In both cases, conservative values for these boundary conditions were used.

Also seen in Table 3-6 is a general invariance of the calculated solution to the prechannel boundary conditions, including temperatures of the vacuum chamber (ambient), heater flange heat sink, and fluid inlet. This result indicates that the prechannel region, as modeled by the thermal resistive network, is not a strong driver of vaporizer performance. Also, this result suggests that the energy required to heat the lithium to the saturation temperature is much less than the latent heat required to fully vaporize the lithium. While all boundary conditions have some influence on the solution, in most cases the difference in the solution is not detectable within the accuracy of the network model as a whole.
4 FINITE-VOLUME MODEL

In this chapter, a three-dimensional finite volume model (FVM) is implemented to analyze the thermal-fluid behavior of the ALFA$^2$ vaporizer. Compared to the network model, the FVM calculates a higher fidelity solution by coupling the energy transport in the fluid domain to the heat conduction through the solid channel walls. Because the thermal boundary conditions at these fluid-solid interfaces are not known in advance, the heat transfer between these surfaces is said to be conjugated.

By coupling the energy transport as well as including phase change mechanisms and three-dimensional fluid dynamics, details about the momentum, thermal, and vapor boundary layers are captured. This model has a much finer spatial resolution than the network model, and is used to provide details for the ALFA$^2$ baseline vaporizer design. The heater power used to investigate these details is taken from the results of the network model for the ALFA$^2$ vaporizer length and 100 K of vapor superheat. Three flow rates were evaluated to bracket the potential operating envelope of flow rates for the point ALFA$^2$ design: 40 mg/s, 80 mg/s, and 120 mg/s, with 80 mg/s being the nominal operating point.

This chapter is organized as follows. Section 4.1 describes the modeling approach and governing equations that comprise the FVM. In Section 4.2, several approaches taken to make the computational problem more tractable are discussed, including a method of cathode partitioning and exploitation of geometrical symmetry. The description of the meshing schemes employed to discretize the solid and fluid domains are discussed in Section 4.3, and treatment of the boundary conditions and initial conditions used in the FVM are presented in Section 4.4. Section 4.5 discusses the solution methodology and simulation parameters. Finally, Section 4.6 presents the results of the FVM, including comparisons with network model predictions.
Profiles for the temperature, pressure drop, and volume fraction throughout the vaporizer for each of the three flow rates, as well as contour plots of the flow velocity and pressure are presented.

4.1 Model Description

To model the two-phase flow system, a single-fluid mixture model was chosen. This type of model is a scaling technique that is used to approximately capture the detailed physics of multiphase systems in large computational domains. The approach is to use a phase-weighted average for the fluid properties so that predictions of the overall behavior of two-phase flows can be calculated.

The fundamental assumption in formulating the mixture model governing equations is that the two phases are well-mixed, such that the overall properties of the amalgam can be obtained by using a weighted average of the fluid properties for the individual phases. These mixture properties are weighted based on the local volume fraction of the \( i \)-th phase, \( \alpha_i = V_i / V_{\text{total}} \).

Unlike volume-of-fluid or front-tracking methods, values for the volume fraction can take any value between zero and one in a mixture model because the two phases are assumed to be freely interpenetrating. Due to this assumption, it follows that in each cell, the temperature and pressure are assumed to be the same for the two phases. Certain, non-equilibrium effects such as surface tension forces are not modeled. The mixture model consists of a set of six conservation equations, and a constitutive relation for the slip velocity. Additionally, a heat diffusion equation is solved in the solid domains of the vaporizer to capture the heat flow paths around the fluid.
4.1.1 Mass and Momentum Conservation

The continuity equation governs the conservation of mass for the entire mixture (both liquid and vapor), and takes the form \[28]\:

\[
\frac{\partial}{\partial t} \left( \alpha_L \rho_L + \alpha_V \rho_V \right) + \nabla \cdot \left( \alpha_L \rho_L \vec{v}_m + \alpha_V \rho_V \vec{v}_m \right) = S_m
\]

(4-1)

where \( \alpha_V \) is the vapor volume fraction (also called the void fraction) and \( \alpha_L \) is the liquid volume fraction (\( \alpha_L = 1 - \alpha_V \)). By defining the density of the mixture as,

\[
\rho_m = \alpha_L \rho_L + \alpha_V \rho_V ,
\]

(4-2)

it follows that the mass-averaged mixture velocity, \( \vec{v}_m \), can be calculated as,

\[
\vec{v}_m = \frac{1}{\rho_m} \left( \alpha_L \rho_L \vec{v}_L + \alpha_V \rho_V \vec{v}_V \right).
\]

(4-3)

where both the liquid and the vapor are assumed to have constant densities. The term on the right hand side of Eq. (4-1), \( \dot{m}_m \), represents a volumetric mass source \[28\]. In the ALFA\(^2\) model, this source term is always set equal to zero, as it defines a source of mixture mass.

The momentum conservation law is obtained by summing momentum contributions over both phases \[28\],

\[
\frac{\partial}{\partial t} \left( \rho_m \vec{v}_m \right) + \nabla \cdot \left( \rho_m \vec{v}_m \vec{v}_m \right) = -\nabla P
\]

\[
+ \nabla \cdot \left[ \mu_m \left( \nabla \vec{v}_m + \nabla \vec{v}_m^T \right) \right] + \rho_m g + \nabla \cdot \left( \vec{v}_{dr} \vec{v}_{dr} \rho_m \right)
\]

(4-4)

where \( \mu_m = \alpha_L \mu_L + \alpha_V \mu_V \) is the mixture dynamic viscosity, and \( \vec{v}_{dr} \) is the drift velocity of the vapor, defined as the velocity of the vapor phase relative to the mixture velocity. Figure 4-1 graphically illustrates how these different velocities are defined.

In the case of single phase flow (\( \alpha_V = 1 \) or \( \alpha_V = 0 \)), the mixture momentum equation reduces to a Navier-Stokes equation. The last term in Eq. (4-4) represents a diffusive stress due
to phase slip that accounts for interaction between phases. Based on the definition of the drift velocity, it does not cancel out when the averaging between phases.

![Image of velocity terms](image-url)

**Figure 4-1.** Illustration of differences between different velocity terms.

Because the solution to the momentum equation is the mixture velocity, the individual phasic velocities are unknown. Therefore, a direct calculation of the drift and slip velocity is not possible. However, constitutive relationships have been developed to determine the slip velocity [29]. In the FVM, an algebraic formulation for the slip velocity is employed assuming a shared pressure field between phases (i.e. local equilibrium) [28]. The form of the slip velocity is given by,

\[
\vec{v}_{VL} = \vec{v}_V - \vec{v}_L = \frac{\tau}{f_{\text{drag}}} \left( \rho_V - \rho_m \right) \ddot{a}_V
\]  

where \( f_{\text{drag}} \) is a drag function that determines the interaction between the phases, \( \ddot{a}_V \) is the acceleration of the vapor, and \( \tau \) is the characteristic relaxation time of a bubble defined as [29]

\[
\tau = \frac{\rho_v d_b^2}{18 \mu_L}.
\]  

The relaxation time of a particle is a measure of how quickly a bubble reacts to the flow field (i.e. comes to mechanical equilibrium within the flow field).

The acceleration vector of the vapor bubble, \( \ddot{a}_V \), is given by [28],

\[
\ddot{a}_V = \ddot{g} - (\vec{v}_m \cdot \nabla) \vec{v}_m - \frac{\partial \vec{v}_m}{\partial t}
\]  

The drag function \( f_{\text{drag}} \) is dependent only on the local flow Reynolds number [30].

For \( \text{Re} \leq 1000 \)

\[
f_{\text{drag}} = 1 + 0.15 \text{Re}^{0.687} \quad (4-8)
\]

For \( \text{Re} > 1000 \)

\[
f_{\text{drag}} = 0.0183 \text{Re} \quad (4-9)
\]

Together, Eqs. (4-5) – (4-8) define the slip velocity between the two phases. The drift velocity and the slip velocity are then connected by the relationship,

\[
\vec{v}_{\text{dr}} = \vec{v}_{VL} - x\vec{v}_{VL} = (1 - x) \vec{v}_{VL} \quad (4-10)
\]

This definition of the drift velocity in Eq. (4-10) corresponds to the illustration in Figure 4-1. The vapor mass fraction \( x \) is defined in each cell as,

\[
x = \frac{\alpha_v \rho_v}{\rho_m} \quad (4-11)
\]

Note that if the slip velocity is zero, the model reduces to a local homogenous flow. That is, while the velocity of both phases will be equal to the mixture velocity \textit{within each element}, the mixture velocity can vary within different elements throughout the fluid domain (in momentum boundary layers, for example).

### 4.1.2 Energy Conservation

The energy equation is also defined for the mixture of phases [28],

\[
\frac{\partial}{\partial t} \left( \alpha_L \rho_L E_L + \alpha_v \rho_v E_v \right) + \nabla \cdot \left[ \alpha_L (\rho_L E_L + P) \vec{v}_L + \alpha_v (\rho_v E_v + P) \vec{v}_V \right] = \nabla (k \nabla T) + S_E
\]

(4-12)

where (by ignoring compressibility effects), the total phase energies can be expressed as,
\[ E_L = h_L, \quad E_V = h_V \]  
\[ (4-13) \]

where \( h \) is the sensible enthalpy of phase, \( j \), defined as,

\[ h_j = \int_{T_{ref}}^{T} c_{p,j} dT \]  
\[ (4-14) \]

Note that the pressure field in Eq. (4-12) does not contain a subscript. This is a direct consequence of the mixture model, as the pressure field is assumed to be shared by both phases. Therefore, no interfacial pressure gradients exist that can support bubble growth or collapse. In Eq. (4-14), the reference temperature used to calculate the enthalpy of each fluid is 298.15 K. The source term on the right hand side of Eq. (4-12) \( S_E \) includes all volumetric heat sources [W/m\(^3\)]. In the present application, this term represents a sink/source that accounts for the latent enthalpy of vaporization during the phase change.

### 4.1.3 Vapor Continuity

Along with the conservation equations for mass, momentum, and energy, an additional equation is required when implementing the mixture model. This equation enforces continuity of the vapor phase and is expressed as [28],

\[ \frac{\partial}{\partial t} (\alpha_v \rho_v) + \nabla \cdot (\alpha_v \rho_v \vec{v}_m) + \nabla \cdot (\alpha_v \rho_v \vec{v}_{dr}) = R_v \]  
\[ (4-15) \]

Because \( \vec{v}_v \) is not explicitly known, the drift velocity term is included to implicitly calculate the vapor velocity through the relationship: \( \vec{v}_m + \vec{v}_{dr} = \vec{v}_m + \vec{v}_v - \vec{v}_m = \vec{v}_v \). The term on the right-hand side \( R_v \) represents a source term that defines the phase change mechanism and controls the change in vapor volume fraction during vaporization or condensation. When solving
the system of equations, the constraint $\Sigma \alpha_i = 1$ is enforced so that only one phase continuity equation needs to be solved [28].

4.1.4 Source Terms

Because local thermal equilibrium is assumed between the two phases, the source terms that model the vaporization of liquid and condensation of vapor serve only to conserve the mass and energy of the liquid-vapor mixture. The source terms used to model the phase change are taken from a phenomenological model developed by Lee [31]. These source terms are driven by the local cell temperature, $T$.

For $T \geq T_{SAT}$ (boiling),

$$R_L = -\lambda (1 - \alpha_v) \rho_L \frac{|T - T_{SAT}|}{T_{SAT}} \tag{4-16}$$

$$R_V = \lambda (1 - \alpha_v) \rho_L \frac{|T - T_{SAT}|}{T_{SAT}} \tag{4-17}$$

For $T < T_{SAT}$ (condensation),

$$R_L = \lambda \alpha_v \rho_v \frac{|T - T_{SAT}|}{T_{SAT}} \tag{4-18}$$

$$R_V = -\lambda \alpha_v \rho_v \frac{|T - T_{SAT}|}{T_{SAT}} \tag{4-19}$$

and,

$$S_E = -R_V h_{fg} \tag{4-20}$$

The vapor source terms have units of $[\text{kg-m}^{-3}\cdot\text{s}^{-1}]$, the energy source term has units of $[\text{kW-m}^{-3}]$, so the relaxation parameter $\lambda$ therefore carries units of $[\text{s}^{-1}]$. The actual value of this relaxation parameter can be varied during a simulation to affect the speed of convergence and/or
increase the stability of the solution. Choosing the optimum value for $\lambda$ at a given point in the solution sequence takes a trial and error approach and is obtained by balancing vapor advection with vapor conversion. Too high of a relaxation parameter will emphasize the vapor source in Eq. (4-15) and the cell will essentially become flooded (with vapor) without advecting it away. If the relaxation factor is too small, the vapor will advect out of a cell without being sufficiently replenished by an equal amount of vapor conversion (required for steady-state). The end result in the former case is an extremely unstable solution, and for the latter, a solution that is prohibitively slow to converge. In practice, the value for $\lambda$ is varied throughout the simulation, and taken to be the highest possible value while still providing a reasonably stable solution. Because this relaxation factor is only applied to the source term, $\lambda$ is set equal to one during the final iterations to ensure proper proportioning between the terms in the vapor continuity equation.

The vapor volume fraction is included in Eqs. (4-16)-(4-19) for two reasons. First, these terms act as a switch that mathematically sets the value of the source term to zero when appropriate. For example, an element containing only superheated vapor will have a cell temperature greater than the saturation temperature. However, because the element is already full vapor ($\alpha_v = 1$) the second term in Eq. (4-17) ensures that the source term will equal zero because there physically cannot be any creation of vapor. Using this “switch” eliminates the additional time-consuming logical operations that would otherwise be required. The condensation source terms works similarly.

Secondly, these terms act as a secondary relaxation multiplier that sweeps from one to zero based on the current value of the cell’s void fraction. As $\alpha_v$ approaches one, these terms
reduce the overall magnitude of the vapor/liquid sources to help prevent any overshoot as the cell approaches a single-phase element.

The fluid density (either liquid or vapor) is also included in the source terms as a scaling factor. While not used in the calculations, liquid source terms are shown above to reinforce that any vapor mass generation corresponds physically to an equal amount of liquid mass depletion. These scaling terms are required to account for the fact that a larger volume of vapor is created relative to liquid for a fixed amount of converted mass because the density of the vapor is much less than the liquid.

The last term represents the driving potential of the source function, and is simply the normalized vapor superheat (for the boiling sources) or liquid subcooling (for condensation). The higher the cell temperature, the greater the driving potential, and hence, more energy is available for the creation of vapor in that cell. The absolute value of the cell superheat is used because the direction of mass conversion is already defined by the negative signs in Eq. (4-16) and (4-19).

Finally, Eq. (4-20) is just the statement that the energy absorbed (or released) is equal to the latent heat corresponding to the volumetric vapor mass source. The negative sign indicates that this energy term is a heat sink when vapor is generated (i.e. the mass flux term, $R_v$, is positive), and is a heat source when vapor is condensed.

### 4.1.5 Conjugate Heat Transfer

The vaporizer model includes conjugate heat transfer to the surrounding channel to better account for the heat flow paths around the channel. In these solid regions, the energy transport equation is defined by [18],
where $\rho$ is the density of the solid, $h$ is the sensible enthalpy defined in Eq. (4-14), $k$ is the isotropic thermal conductivity, and $S$ allows for heat sources within the solid. In the vaporizer assembly, the materials comprising the solid domain are tungsten for the cathode body, and molybdenum for the two flanges.

For this application, there is no convection or energy generation and the system is at steady-state. Therefore, Eq. (4-21) reduces to a pure heat diffusion equation, $\nabla \cdot (k \nabla T) = 0$, where only conduction through the solid is present. This equation has parabolic characteristics, making it stable numerically and allowing for a relaxation of the meshing requirements. At the interfaces between the solid and fluid domains, the conditions along the domain boundaries are not known a priori. Therefore, these conjugated interfaces are set to enforce continuity (in terms of energy transport) between the fluid and the solid. Boundary conditions on the other external walls of the solid domain are the same used in the network model (Figure 3-12).

### 4.2 Problem Simplifications

Several factors contribute to making the full conjugated two-phase FVM difficult to converge and expensive to solve numerically. Fundamentally, these factors all stem from computational limitations with respect to memory and processing capability. In decreasing order of severity, these factors are:

1. Large density ratio between phases $(\rho_\text{L} / \rho_\text{V})$.
2. Sharp angles and corners due to the helical channel pitch.
3. Inherent three dimensionality of domain.
4. Extremely large aspect ratio of the fluid channel $(L / d_\text{H})$. 
5. Small size of the fluid domain relative to the solid domain.

The first three issues indirectly cause problems with overall model size, either by causing stability issues or by driving the selection of certain mesh element types. The last two issues directly affect the physical size of the computational domain, and therefore lead to excessive meshing requirements. Each of these issues will be discussed further in the following sections.

4.2.1 Cathode Partitioning

The density ratio between the liquid and vapor phase determines how the vapor volume fraction (void fraction) evolves with respect to vapor mass fraction (quality). For phases with equal densities ($\rho_L / \rho_V = 1$), vapor quality varies linearly with the void fraction. Increasing the density ratio, however, causes large increases in the void fraction at the onset of boiling. This is because a given amount of converted mass requires a larger associated volume as the mass density of the vapor phase is decreased.

![Figure 4-2. The void fraction plotted versus vapor quality for a range of density ratios.](image)

This effect is exaggerated as the density ratio is increased, where even a small increase in the vapor quality results in an almost discrete jump in the void fraction. This effect was
discussed in Section 4.8.2, and is illustrated in Figure 4-2 using Chisholm’s slip correlation defined by Eq. (3-44). As a point of reference, at atmospheric pressure and the saturation temperature, the density ratio ($\rho_L / \rho_V$) for lithium is 7660. At 20 kPa, this ratio increases to 33,000.

Because the mixture properties have a strong dependence on the void fraction, it follows that a large gradient in the fluid mixture properties will result as liquid begins to be converted into vapor (i.e. at the onset of vaporization). The most severe consequence of the large density ratio is that the vapor source dominates the advection term in the vapor mass conservation law of Eq. (4-15). To better balance these terms, an extremely fine mesh is required to resolve the large gradients in the flowfield and maintain solution stability in this region.

Further complicating this issue is that the location of this velocity jump is not known a priori. This onset region is not spatially fixed within the domain; it varies throughout the simulation. Because the vapor source is temperature-driven and the temperatures are the highest at the channel exits, the onset of boiling exhibits the general trend of starting at the channel exit and then retreating back as the simulation progresses. This severely restricts any effort related to concentrating cell density in the expected boiling onset region. Potential approaches to addressing this issue, such as adaptive mesh refinement, will be discussed as future work.

In addition to complications stemming from the unfavorable density ratio, the inherent three dimensionality of the cathode severely restricts any simplifications based on planer or axial symmetry. This is caused by the particularly complex geometry of the pre-channel region, with asymmetries associated with the location of the feedtube, the off-center inlet plenum, and the heater flange. In the channel region, the helical geometry is inherently three-dimensional.
These geometric characteristics force an approach that can segregate these problematic areas so that they can be resolved independently. This leads to an approach of partitioning the cathode assembly as shown in Figure 4-3. By treating the boiling onset and pre-channel regions separately, four distinct segments are created; the pre-channel region, a liquid region, a small region of boiling onset, and the two-phase region. The helical lines sketched over the surface of the cathode body highlight the boundaries of each individual fluid channel and its surrounding solid.

Figure 4-3. Top: The complete cathode assembly. Bottom: The cathode divided into subregions for computational purposes.
The prechannel includes the feedtube inlet, the heater flange, and the inlet plenum. This region also includes a small section of all four channels so that the pressure drop through the flow restriction is captured. The liquid region extends from the outlet of the prechannel to a location upstream of where the onset of boiling occurs. While somewhat arbitrary, this location is taken as a point where the fluid is one degree less than the saturation temperature. Also included in the liquid region is a segment of the mounting flange that will capture the effect of the conduction heat sink through this component.

The boiling onset region is a small section that covers only 1/16th of a helical revolution, using the outlet conditions of the liquid region as inlet conditions. A small section is all that is required because the steep gradient in the volume fraction persists for only a small distance. This boiling region can then be heavily meshed to capture the flow details in this complex flow region. The remaining segment is the region that includes the majority of the two-phase flow and the single-phase vapor region that exists at the channel exit. Because the volume fraction changes little after the initial jump at boiling onset, the mesh density in this section can be relaxed somewhat.

### 4.2.2 Geometry Modifications

In addition to cathode partitioning, the cathode assembly was further modified to reduce the complexity of the geometry. The objective for these modifications was to eliminate any features of the vaporizer tube that did not contribute significantly to the overall flow field, but could cause meshing difficulties. Figure 4-4 shows the modifications made to the cathode assembly, with the solid components removed for clarity. The left side of the figure shows the original fluid domain. This domain forms the actual geometry that the lithium will be forced to
flow through. However, several features of this domain are sources of poor elements; indicated by circles in the figure. These problem areas included the acute angles at the entrance and exit of the helical channels, as well as the narrow downward facing step located along the inside diameter of the feedtube. This step is created due to the mismatching bores of the feedtube as it mates with the cathode tube. The right side of Figure 4-4 illustrates how the geometry was modified to eliminate these problem areas. The step was removed in favor of a constant area feedtube, and the sharp corners were filled in. In addition to these modifications, all fillets, chamfers, and rounds were removed.

**Figure 4-4.** *Left:* The circled areas indicate geometric features that are sources of poor elements. *Right:* The modified vaporizer geometry renders the domain more CFD-friendly.
4.2.3 Channel Symmetry

Even with the cathode partitioning approach, the computational model remained prohibitively large. Therefore, simplifications based on symmetry were explored. Note in Figure 4-3 that by isolating the helical channel section from the complex geometry of the pre-channel section, a relatively regular geometry results. Geometrically, each of the four helical leads is identical to each other lead, with each of the four inlets being shifted successively by 90°. By assuming that the inlet temperatures of each channel is equal and that the total mass flow rate is uniformly distributed between each of the four fluid channels, each will be thermally identical. From this observation, it is apparent that a complex helical symmetry exists in cathode body that can be exploited to reduce the overall size of the FVM. While the internal lithium flow does not necessarily exhibit any symmetry, the cathode body exhibits a helical symmetry with respect to the outer surfaces of its neighboring channels. The validity of this assumption as it relates to the uniform distribution of the total mass flow and the similarity of the channel inlet temperatures will be evaluated in Section 4.6.

Figure 4-5 illustrates of the handling of the boundary conditions, where channel #1 is the region of interest. This channel is bracketed by channel #2 on its downstream side and by channel #4 on its upstream side. Consider the heat flux normal to the downstream surface of channel #1 at point A, \( q_A'' \). The location of point B is the corresponding contact point on the upstream side of channel #2. The heat flux at point B, \( q_B'' \), should have an outward normal heat flux equal and opposite of that of point A, i.e. \( q_B'' = -q_A'' \). Because the thermal solution is assumed identical for all channels, the heat flow value at the corresponding axial location \( z \) of channel 2 (point C) should be exactly equal to \( q_B'' \). Therefore, the relationship between the heat fluxes at A and C is,
where \( z \) is the axial coordinate. Thermally, point \( C \) and \( B \) (and point \( A \) and \( D \)) are identical because these two points are the same distance from the channel inlet (along the channel coordinate, i.e. \( s_1 = s_2 \)), and they are located along the same channel surface (downstream face, upstream face, etc.). If the inlet conditions (temperatures and mass flow rates) are equal for each channel, the thermal solution should be expected to be identical as well. For example, the onset of boiling will not occur at different locations along the fluid path of each individual channel.

Equation (4-22) defines the relationship between heat fluxes at points \( A \) and \( C \). Following similar logic, the relationship between the temperatures at points \( A \) and \( C \) is simply,

\[
(\frac{q_A}{q_C})_z = -(\frac{q_A}{q_C})_z = -(T_A)_z = (T_C)_z
\]  

(4-23)

---

Figure 4-5. *Left:* Illustration of the helical symmetry of the cathode assembly. *Right:* Azimuthal cross-section of the cathode body across \( z \) showing relevant points.
The practical implication is that the heat flux along the upstream face will be equal and opposite to the heat flux along the downstream face at every axial location. Although not time-dependent, these boundaries can be thought of as a type of periodic condition, where the boundary condition along one face is dependent on the boundary conditions along some other face. Because the thermal boundary conditions along these “internal” surfaces are not known, an iterative solution is required.

![Figure 4-6. Simplified vaporizer domain shown as modeled.](image)

Using this concept of helical symmetry, only a single channel of the cathode assembly needs to be explicitly modeled. Although it requires complex handling of the boundary conditions, the bulk effect of the entire cathode assembly can be implicitly modeled by solving for only a single channel. The three simplifications described above are enabling in the sense that they reduce the overall size of the FVM (in terms of physical volume as well as in meshing requirements) to within the limitations of available computational resources.
The conjugate mixture model is applied to the resulting simplified geometry as shown in Figure 4-6. The pre-channel region is modeled in its entirety because no assumptions with respect to symmetry are made, thus allowing for evaluation of the assumption of thermally identical inlet conditions. The liquid region extends from the prechannel domain to boiling onset, including a portion of the mounting flange equal to 1/4 of the total volume (based on a four channel vaporizer design) to account for parasitic heat losses to the actively cooled bus bar.

The boiling onset region separates the liquid region from the two-phase region, and consists of a small channel segment that contains the development of the vaporization process. No solid channel surrounds this fluid volume because the conjugate problem is prohibitive due to the dense mesh required at boiling onset. In the absence of conjugated heat transfer in this short area, heat fluxes into the fluid are obtained iteratively as discussed below. The remainder of the vaporizer domain contains the majority of the two-phase flow, as well as the vapor phase after dryout. By using the geometry from Figure 4-6, the vaporizer model is made tractable, but requires careful handling of the boundary conditions to produce a self-consistent solution.

4.3 Mesh Description

Because of the additional physics of multiphase systems, stable convergence of the iterates can be substantially more difficult than single-phase fluid flows. The additional equations, as well as the source terms that capture the effects of mass conversion during vaporization further complicate the system. Because of this inherent instability, mesh generation is a non-trivial process. In the present study, iterate stability was observed to be highly sensitive to a number of parameters, including element type, element size, and element quality.
Aside from the physics of the phase change, the primary difficulty in discretizing the cathode assembly is the severe aspect ratio of the fluid channel in terms of path length and hydraulic diameter ($L/d_H \sim 1000$). When unraveled, the length of the fluid channel is greater than one meter, while the channel height is only $a = 7.5 \times 10^{-4} \text{ m} (0.75 \text{ mm})$. This channel height drives the selection of very small elements that, in general, need to be propagated along the path length of the fluid channel. Considerations of overall model size and associated solution times make it necessary to optimize the mesh over the computational domain. Because of the extreme aspect ratio of the vaporizer tube geometry, as well as the complex boundary structures, finding a sufficient node density while maintaining computational efficiency required trial and error tuning.

To make the overall model size more tractable, the meshing strategy involved the use of sizing functions to concentrate grid points closer to the domain boundaries. These sizing functions are placed in regions of the flow field where steep gradients in velocity, temperature, and vapor growth are expected (i.e. within the momentum, thermal, and vapor boundary layers along the channel walls). In regions not constrained by the sizing functions, the mesh can be stretched to effectively coarsen the grid in regions of the flow where the only minimal gradients are expected to exist, such as the solid domain and the core of the fully developed single phase flow. By increasing element density only where required, the sizing functions help to minimize the total number of elements required to resolve a channel cross-section; thus providing fewer elements to be propagated over the length of the channel. By implementing this approach, overall mesh sizes can be reduced without significant loss of information.

In practice, however, the amount of stretching is limited, as excessive coarsening in only a single direction (in the streamwise direction, for example) can result in poorly sized/shaped
elements. Of particular concern are spire-type elements, shown in Figure 4-7a, where the point of the spire is oriented along a flow streamline in the vaporizer channel. The two metrics used to define element quality in the present study are illustrated in Figure 4-7b; the aspect ratio and measurement of the elements dihedral angles.

![Figure 4-7. (a) Partial listing of poor quality tetrahedral elements [38] and (b) definition for element quality.]

The aspect ratio criteria is calculated by inscribing a circle of radius $r$ within the element and circumscribing a circle of radius $R$ passing through each bounding node. As the ratio $R/r$ increases, element quality decreases. The measurement of dihedral angles between an element’s surfaces is also used as a figure or merit for element quality. As a dihedral angle $\beta$ approaches $\pi$, the tetrahedral element degrades in quality.

While many additional metrics exist for defining the quality of a tetrahedral element, all poor elements share similar characteristics. In general, any element with a small volume but large relative nodal displacements will cause local errors in the derivatives of the flow variables. Also shown in top and bottom rows of Figure 4-7a is that the vertices of the elements are nearly
linear or nearly planar, respectively. This causes increases in the local condition numbers\(^1\) of these elements [38] which has the undesired consequence of magnifying errors in the system. In all cases, these conditions destroy solution accuracy and stability.

For the cathode assembly geometry, the solid domain is significantly larger than the fluid domain. To address this issue, this relatively large solid domain was divided into four subregions as illustrated in Figure 4-8. Creating these extra surfaces allow for greater flexibility in the meshing scheme by creating additional edges for the placement of sizing functions. Therefore, cells can be concentrated near the fluid channel and stretched as they move radially outward. However, because of the sharp angles that these subdomains introduce, care must be taken to ensure that poor quality elements do not result from the sizing functions placed along the solid-solid interface.

---

\(^{1}\) A condition number is a property of a matrix associated with the measure of the posedness of an equation system. Low condition numbers are associated with well-posed problems. Large condition numbers are an indication that a problem is ill-posed, which leads to error magnification and overall system sensitivity (i.e. a small change in input produces a large change in output).
Dividing the large solid domain into subregions also has the favorable result of converting the large solid domain into individual mappable subregions. In general, a mappable volume is one that can be transformed such that the mesh represents a logical cube. In order to represent a logical cube, the following criteria must be met:

- There must be 8 corner nodes of the volume.
- Each corner node must be connected to each of the three other corner nodes via a logical row of nodes.

A logical row of nodes is a group of points that can be characterized as a straight line for discretization purposes. For example, the helical curves of the fluid channel are logical rows because they can be mathematically “unraveled” and straightened using a mapping function, subdivided into elements, and then remapped into the original helix while retaining the position of the nodal locations.

A more succinct statement of the above criteria is that the volume should contain six sides, each of which are themselves rendered mappable by correct specification of its vertices. Based on this criterion, it follows that the fluid domain and each individual solid subdomains are able to be mapped. Prior to subdividing as shown in Figure 4-8, the solid domain did not meet the requirements of a mappable volume.

Mapped grids are ideal for the present study because it enables the usage of hexahedral elements. Mapped, hexahedral elements are preferred over unstructured tetrahedral elements\(^2\) for a number of reasons. Because a single hexahedron consists (geometrically) of two

\[^2\text{ An unstructured mesh is one in which each element does not share the same topology (i.e. does not share the same number of neighbors). Therefore, the banded structure of the matrix will in general be larger because any one element may communicate with any number of additional elements. Typically, unstructured volume meshes consist of tetrahedral volume elements with triangular faces.}\]
tetrahedrons, implementing hexahedral elements in the mesh will decrease in the total number of elements required to discretize the vaporizer domain. In addition, mapped hexahedral elements share a consistent topology in terms of the quantity of neighboring elements. This characteristic of mapped grips results in a significant reduction in system bandwidth. This has non-trivial implications on computation time because it directly decreases the storage requirement for the banded matrix and decreases the number of operations (i.e. multiplications) at each iteration. Additionally, structured grids are more accurate, more stable, and require less memory to generate than their unstructured counterparts.

Table 4-1 summarizes the statistics and schemes used to generate the mesh in all regions. In total, the overall mesh size is 5,880,145 elements. In the pre-channel region, the complex geometry does not permit the usage of boundary-fitted mapped grid generation. Therefore, the mesh in the pre-channel domain is generated using an unstructured hybrid-tetrahedral approach. A hybrid-tetrahedral mesh is one that consists primarily of tetrahedral elements, except for areas where hexahedral, pyramid, or wedge elements are either required or appropriate.

Physically, the largest individual domain using the partitioned cathode approach is the two-phase region. This region also contains the location of several key engineering quantities of interest such as the dryout point, the exit temperature, and the exit velocity. These observations drive the mesh in the two-phase to nearly twice the size (from an element number standpoint) of any other region. To resolve the steep gradients expected in the boiling onset region, an extremely dense mesh is required, with 385,000 elements packed within only 1/16 of a helical revolution of the fluid channel.

In the pre-channel region, the mesh size was not constrained by computational limitations, but rather a balance between sufficient node density and element quality. For
example, available resources permitted a much finer mesh in the fluid domain of the pre-channel. Doing so, however, created constraints on the surface meshes where the solid domain interfaces with the fluid domain. This limited the usage of sizing functions off these areas. Therefore, when implementing the unstructured hybrid-tetrahedral meshing scheme in the pre-channel region of the cathode assembly, it was observed that a mesh with less elements of higher quality consistently outperformed a more dense mesh.

<table>
<thead>
<tr>
<th>Region</th>
<th>Domain</th>
<th>Scheme</th>
<th>Elements</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-Channel Region</td>
<td>Solid</td>
<td>Unstructured Tet/Hybrid</td>
<td>244,902</td>
<td>282,875</td>
</tr>
<tr>
<td></td>
<td>Fluid</td>
<td></td>
<td>37,973</td>
<td></td>
</tr>
<tr>
<td>Liquid Region</td>
<td>Solid</td>
<td>Mapped Hexahedral</td>
<td>1,358,746</td>
<td>1,748,746</td>
</tr>
<tr>
<td></td>
<td>Fluid</td>
<td></td>
<td>390,000</td>
<td></td>
</tr>
<tr>
<td>Boiling Onset Regiona</td>
<td>Fluid</td>
<td>Mapped Hexahedral</td>
<td>385,000</td>
<td>385,000</td>
</tr>
<tr>
<td>Two-Phase Region</td>
<td>Solid</td>
<td>Mapped Hexahedral</td>
<td>2,810,244</td>
<td>3,463,524</td>
</tr>
<tr>
<td></td>
<td>Fluid</td>
<td></td>
<td>653,280</td>
<td></td>
</tr>
</tbody>
</table>

Table 4-1. Meshing statistics and schemes for the FVM.

4.4 Boundary Conditions

4.4.1 Thermal Boundary Conditions

Channel Regions

Due to the partitioning approach presented in Section 4.2, the challenge of implementing the FVM becomes breaking the problem into computationally tractable segments while maintaining self-consistency with respect to the boundary conditions. In general, the boundary conditions used in the mixture model are consistent with those applied in the network model.
from Figure 3-12. However, the single-channel approach requires additional handling of the boundary conditions on what would otherwise be surfaces internal to the cathode assembly. These walls are labeled in the shortened helical segment of Figure 4-9, and include faces coplanar to the fluid inlet and outlet, as well as upstream and downstream facing surfaces of the channel walls. In the following discussion, these four channel surfaces are referred to as the internal walls, and the corresponding boundary conditions are referred to as internal boundary conditions.

![Diagram of helical channel faces](image)

**Figure 4-9.** Portion of the helical channel defining the nomenclature of the faces in the solid domain.

In the absence of net axial or net azimuthal heat flow, these surfaces would be represented by adiabatic boundaries. However, because these heat flow components are not neglected, the boundary conditions on these four “internal” walls are not known a priori. Based on the thermally identical channel assumption, however, the net heat flux in the azimuthal direction (through the domain boundaries) should be nearly zero. On the inlet and outlet faces of Figure 4-9, the outward normal vectors are nearly aligned with the azimuthal direction. The small resulting heat fluxes on these two faces coupled with the small surface areas (relative to the
other surfaces of the domain) should cause the contribution of these heat fluxes to be insignificant in the overall solution. This is a reasonable assumption if the pitch is much smaller than the circumference of the cathode. For the baseline ALFA\textsuperscript{2} geometry, this ratio is approximately 0.061. The larger the ratio between the pitch and the circumference, the more this face will deviate from that of a true azimuthal facing surface.

For the upstream and downstream faces, however, this is not case. These surface areas represent the largest boundaries of the solid domain, and the outward normal vectors are nearly aligned with the axial direction. Potentially large heat fluxes are expected in the axial direction in the regions near the heat sunk flanges, as suggested by the results of the network model. Because these internal boundary conditions are not known a priori, they must be determined iteratively. In reference to the face designations from Figure 4-9, the procedure developed for obtaining the boundary conditions is:

1. Apply initial estimate for the upstream and downstream walls.
2. Solve the FVM until converged.
3. Store and plot temperature profile of upstream and downstream walls.
4. Check for boundary condition agreement between the two internal walls.
5. Obtain updated guess for internal boundary.
6. Divide the updated guess into reasonable segments for curve fitting.
7. Implement the updated boundary condition in the simulation.
8. Re-solve the FVM until converged.
9. Repeat steps 3-9 for boundary heat fluxes along internal walls.

In all simulations, the initial estimate used in step 1 is an adiabatic boundary condition. Therefore, after completion of step 2, the results are similar to that of the network model in the sense that the heat flow across domain boundaries is only in the radial direction. Even with
adiabatic boundaries, however, axial heat transfer is still allowed within the domain. The convergence of each intermediate case is assessed by monitoring several solution parameters, including the inlet pressure, outlet temperature and void fraction, and equation residuals. When the change in these parameters at successive iterations flatten, the case is considered to be converged, and the temperature solution along the internal walls are stored and imported into Matlab for processing as illustrated in Figure 4-10.

Note that because a Neumann boundary condition is applied to the internal walls in step 1, the wall temperatures are unconstrained and are therefore allowed to float. Within the framework of the single-channel (thermally identical) approach, the simulation will be converged if the two temperature profiles perfectly overlap. That is, the statement of Eq. (4-23) will be satisfied. Otherwise, the two surface temperature profiles are averaged at each axial location as shown in Figure 4-10b to become the updated “guess” for the internal boundary conditions. This averaging assumes that $T = T(z)$ only, which is considered reasonable because the axial temperature gradient (between the inlet and outlet) is order of magnitudes greater than the radial temperature gradient (between the inner and outer surfaces).

![Figure 4-10. Illustration of the iterative solution of the internal boundary conditions](image-url)

(a) Plot wall boundaries  
(b) Calculate Average  
(c) Define curve fits  

Axial Coordinate, $z$
The updated boundary condition is then divided into a number of segments and represented by a series of linear curves as shown in Figure 4-10c. These boundary conditions are then applied to the internal walls of the channel via a user-defined function (UDF). A UDF is an auxiliary script that is written in C programming language and interpreted into the FLUENT interface to extend the capabilities of the application.

The case is then re-solved using the updated boundary condition. During this simulation, the surface heat fluxes are now the unconstrained values due to the application of the temperature (Dirichlet) boundaries. After convergence is determined, a process similar to that described for the wall temperature profile is performed for the surface heat fluxes. The major difference is that the downstream heat flux should be equal in magnitude and opposite in direction to the heat flux through the upstream wall as stated Eq. (4-22), with a “positive” flux being defined by the inward pointing normal. Accordingly, when processing the flux profiles in step 3, a sign change needs to be included to evaluate the agreement of the internal walls. Another sign change in step 7 provides the correct profile for linear curve fits.

Final convergence of the single-channel solution is assessed by monitoring both exit temperature of the domain and the agreement between the updated boundary conditions. If successive boundary condition iterations (as described in the process above) show less than a five percent change or if the fluid temperature at the channel outlet no longer shows sensitivity to updates of the boundary conditions, the simulation is aborted and the single-channel solution stored. In all cases considered, only three to four manual iterations were required for the outlet fluid temperature to converge.

In the segment of the channel region containing the onset of boiling, an extremely fine mesh was required to resolve the large flow gradients. For this reason, implementation of
conjugate heat transfer problem is computationally expensive. As a practical matter, this boiling onset region is only 1/16 of a full channel revolution. In terms of power, this channel segment consumes 19.8 W of the total power required based on the uniform heat flux applied to the inner cathode surface. Over the four leads, the approximately 80 W consumed corresponds to less than two percent of the total supplied power (3.56 kW). Therefore, the boundary conditions applied to the walls of this small fluid domain are estimated by linearly interpolating between the heat flux solutions at the exit of the liquid region and the beginning of the two-phase region. This estimation is reasonable considering the relatively small amount of energy consumed by this small channel segment.

As suggested by the results of the network model (Figure 3-14), the temperature of the cathode assembly in the region of the two-phase flow is expected to be relatively fixed. Because the fluid mixture remains stable at the saturation temperature in this region, no net axial temperature gradient exists to drive axial heat transfer. Therefore, adiabatic boundary conditions are used along all internal walls in the two-phase region of Figure 4-3. Note that only net axial heat flow is neglected; energy can still flow in all directions within the fluid and solid domains. This is important, because it provides a heat flow path around the channel to better capture radiation losses from the outer surface of the cathode.

While the argument for this assumption is made on physical basis, it is also implemented due to practical considerations. Making this assumption significantly simplifies the solution process for the FVM of the two-phase region by eliminating the process of iterative calculation of the internal boundary conditions as described above. This is especially important in the two-phase region where a single intermediate case (in step 1, for example) can take over two months to fully converge.
The remaining two surfaces of the channel segments are the inner and outer cathode surfaces. To simulate radiation heat transfer from the graphite heater, a uniform heat flux of 184,000 W/m² is applied over the inner surface. Over the 22.88 cm cathode length, this corresponds to 3.56 kW, which was estimated by the network model to be the baseline ALFA² heater power (for the preliminary design arrangement operating at 100 K of vapor superheat). On all outer cathode surfaces, radiative boundary conditions are applied. Upstream of the mounting flange, the outer surface of the channel emits to a 300 K far field ambient enclosure. Downstream of this flange, the outer surface emits to the concentric anode at 600 K. These radiation boundary conditions are consistent with those applied in the network model and summarized in Figure 3-12.

Pre-Channel Region

The radiation views in the pre-channel are more complex than those in the channel region due to the feedtube and the heater flange. In this region, rather than explicitly using view factors that would require additional user-defined functions, an area-weighted average for the view temperatures was used. For example, the radiation views from the downstream side of the heater flange are illustrated in Figure 4-11, where \( L_f \) is the distance between the flanges, and \( L_s \) is the length of the slanted side. From the heater flange, the total area of the viewed surfaces is \( A_r = A_1 + A_2 + A_3 \). The value used for the view temperature is calculated using,

\[
T_{\text{VIEW}} = \frac{A_1 T_1 + A_2 T_2 + A_3 T_3}{A_r}
\]

where \( T_1, T_2, \) and \( T_3 \) are mean surface temperatures of each viewed cathode surface. While this approach is only approximate, it does account for emission to multiple surfaces at different
temperatures while eliminating the additional expense required for rigorous view factor calculations. Each other radiating surface in the pre-channel region is handled similarly in the simulation.

Because no symmetry assumptions are made, the only internal boundary present in the pre-channel region is at the interface to the channel segment. Coupling at this interface is important because parasitic heat losses through the mounting flange in the liquid region can affect the thermal solution of the pre-channel region. This is accomplished by manually iterating on the boundary conditions (similar to the process described above for the channel segments) over these two surfaces until agreement is obtained.

Figure 4-11. Calculation of the view temperatures in the pre-channel region

\[ A_3 = \pi L_s (r_{MF,O} + r_{HF,O}) \]
\[ A_2 = \pi (r_{MF,O}^2 - r_{C,O}^2) \]
\[ A_1 = 2\pi L_F r_{C,O} \]

4.4.2 Hydrodynamic Boundary Conditions

Operating Pressure and Density

In implementing the FVM, a constant vapor density was used. This was done for two reasons. First, stability issues stemming from the unfavorable phase density ratio (see Section 4.2.1) were compounded when an equation of state of implemented. Large gradients in the void
fraction were coupled to the pressure field causing erratic behavior in the mixture velocity (counter-current flow, for example). The simulations exhibited a high degree of sensitivity to this coupling. Secondly, within the framework of cathode partitioning, a constant vapor density eliminates the need for additional time-consuming manual iterations on pressure boundary conditions that would be required to ensure consistency between the different vaporizer segments.

In previous studies by Ferreira and Delcroix [32] and Cassady [7,33] that focused heavily on the physics within the thermionic emitter, the lithium flow was observed to be sonic at the exit of the emitter channel. The implication of this result is that thermal choking of the lithium flow is not expected to occur within the vaporizer channel. Therefore, the operating pressure within the vaporizer channel for each flow rate is chosen to be as low as possible while not allowing choking.

When using a constant vapor density, the largest Mach number will occur in the vapor region at the dryout point. From this location the vapor velocity will remain constant, but the static temperature of the flow will increase, thereby increasing the local speed of sound and decreasing the Mach number. For a given operating pressure and its corresponding saturation temperature, an operating vapor density is calculated. Then, by enforcing continuity, a flow velocity and a Mach number can be obtained. This Mach number dependence on pressure (at dryout) is shown in Figure 4-12. Lowering the pressures beyond the point where \( M = 1 \) will result in a thermally choked flow within the vaporizer channel.

In the present study, the pressure used to calculate the vapor density is referred to as the operating pressure and is defined as the minimum allowable pressure that will enforce subsonic flow throughout the channel. Because the maximum Mach number occurs at dryout, the
saturation pressure and the operating pressure are equal. To calculate the operating vapor density, the saturation temperature and operating pressure is used. There is coupling between these values, however, because \( M_{\text{max}} = M_{\text{max}}(T_{\text{sat}}, P_{\text{sat}}) \). That is, both the saturation pressure and operating pressure are functions of the saturation temperature. Therefore, these values are solved iteratively for each flow rate to obtain a consistent set of saturation conditions (temperature and pressure) that ensures subsonic flow.

![Figure 4-12. The operating pressure shown as a function of the Mach number at the dryout point.](image)

**Bubble Diameter**

In order to model the interaction between the liquid and vapor phases, a characteristic bubble diameter needs to be defined [see Eq. (4-5) and (4-6)]. In reality, a number of factors contribute to the actual size of a bubble, including interfacial pressure gradients, Jakob number (ratio of sensible and latent heat absorbed), surface tension, and wall adhesion forces. However, to ensure self-consistency with the equilibrium assumptions of the mixture model (Section 4.1), a characteristic bubble diameter \( d_b \) needs to be chosen sufficiently small to enforce a low particle relaxation time, \( \tau \).
The behavior of a bubble in a flow field can be described by considering the particle Stokes number,

\[ St = \frac{\tau}{\tau_c} \quad (4-24) \]

where,

\[ \tau_c = \frac{L_c}{U_c} \quad (4-25) \]

is the characteristic time scale of the system. The characteristic length \( L_c \) of the channel is given by the radius of curvature of the helical channel, and the average mixture velocity is taken to be the characteristic velocity \( U_c \). To ensure that a bubble will follow the flow streamlines, the Stokes number should satisfy the condition \( St \ll 1 \), implying a negligible reaction time for the bubble to reach equilibrium with the flow. Enforcing the Stokes number constraint, a relationship for a representative bubble diameter can be obtained using the above statements. In terms of the geometric variables already introduced, Eq. (4-7) can be rearranged,

\[ d_b \ll \left[ \frac{18 \mu_v \left( r_{c,\ell} + t_1 - \frac{a}{2} \right)}{\rho_v \| \vec{v}_m \|_{avg}} \right]^{\frac{1}{2}} = C \left[ \frac{18 \mu_v \left( r_{c,\ell} + t_1 - \frac{a}{2} \right)}{\rho_v \| \vec{v}_m \|_{avg}} \right]^{\frac{1}{2}} \quad (4-26) \]

Note that bubble diameter is inversely proportional to the mixture velocity, which varies (locally) from a minimum at the onset of boiling (\( \vec{v}_m = \vec{v}_L \)) to a maximum at the dryout point (\( \vec{v}_m = \vec{v}_y \)). To satisfy the Stokes number condition throughout the entire domain, the minimum (local) bubble size is used as the maximum allowable (global) bubble diameter. This bubble diameter is obtained directly from Eq. (4-26) using a mixture velocity evaluated in the limit of \( \alpha_v = 1 \).
\[ \|\vec{v}_m\| = \|\vec{v}_v\| = \frac{G}{\rho_v}. \quad (4-27) \]

Note that when Eq. (4-27) is substituted into Eq. (4-26), the characteristic bubble diameter becomes a function only of constant parameters. While each flow rate will result in different bubble sizes through a dependence on the mass flux \( G = \dot{m} / A \), the bubble size will be invariant to the operating pressure used in the FVM.

While only the global maximum allowable bubble diameter is of interest here, we see that the local bubble diameter will vary throughout the two-phase flow portion of the flow by the factor \( \rho_m / \rho_v \). In this context, the maximum allowable bubble diameter corresponds to the point of boiling onset where the density ratio is the largest, \( \rho_m / \rho_v = \rho_L / \rho_v \). The local bubble diameter at this point will be much greater than at dryout, which is consistent with the interpretation of the flow patterns discussed in Section 3.2.

In general, larger bubble diameters correspond to an increase in the interaction between the two phases. At low void fractions where wedging flows dominate, liquid plugs form between individual vapor wedges. These liquid plugs interact so fully with the vapor wedges that they move at nearly the same velocity (i.e. the two-phase flow is homogeneous). The extremely large bubbles predicted by Eq. (4-26) at boiling onset would naturally enforce this homogeneous flow at low void fractions. This does not necessarily correspond to a physical representation of true bubble dynamics, but rather a consequence of applying the equilibrium assumptions in the mixture model.

Evaluating Eq. (4-26) using properties for lithium at the saturation temperature (Table 4-2) and the ALFA\(^2\) channel geometry for a flow rate of 80 mg/s, an upper bound of \( d_b < 1.9 \text{ mm} \) is obtained. This maximum allowable bubble diameter is larger than the channel
height (0.75 mm), which is consistent with the wedging and slug flow patterns discussed in Section 3.2. In these flow regimes, characteristic bubble diameters are larger than the channel because they are elongated in the flow direction.

The mixture model, however, does not allow for bubbles this large because they are not likely to be at equilibrium with the flow field surrounding it. To self-consistently implement the mixture model, the $C$ multiplier in Eq. (4-26) is chosen such that the Stokes number condition is satisfied. Therefore, in all simulations, the characteristic bubble diameter was chosen to be an order of magnitude less than that of the maximum allowed by setting $C = 0.1$. The resulting hydrodynamic conditions are summarized in Table 4-2. The operating pressures used in the FVM are of the same order as predictions made by Cassady of the pressure at the inlet of the emitter channel [7].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>40 mg/s</th>
<th>80 mg/s</th>
<th>120 mg/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_{SAT}$, kPa</td>
<td>5.20</td>
<td>10.66</td>
<td>16.24</td>
</tr>
<tr>
<td>$T_{SAT}$, K</td>
<td>1271</td>
<td>1336</td>
<td>1377</td>
</tr>
<tr>
<td>$\rho_v$, kg/m$^3$</td>
<td>$3.4 \times 10^{-3}$</td>
<td>$6.7 \times 10^{-3}$</td>
<td>$9.8 \times 10^{-3}$</td>
</tr>
<tr>
<td>$d_b$, mm</td>
<td>0.26</td>
<td>0.19</td>
<td>0.15</td>
</tr>
</tbody>
</table>

### 4.5 Solution Methodology

The framework for obtaining the final solution of the cathode assembly using the single-channel approach is shown in Figure 4-13. This flowchart summarizes all individual FVM processes, including the manual iterative loops required to ensure consistency of the boundary conditions between the multiple regions of the cathode. For a given mass flow rate, the segments are solved sequentially in the flow direction, beginning with the pre-channel region and ending
with the two-phase region. The circled “S” symbol in the flowchart of Figure 4-13 is the notation for the general finite volume process summarized in Figure 4-14. This methodology was completed for each of the three flow rates investigated in the present study.

Figure 4-13. Flowchart of the solution process for the entire domain using the single channel approach.
The inlet conditions for the pre-channel region are the inlet temperature and the total mass flow rate (40, 80, or 120 mg/s). The mass flow rate and fluid temperature is averaged over each of the four channel exits at the outlet of the prechannel region. These averages are applied as the inlet conditions for the liquid region. Because the outlet conditions between each individual region are not known a priori, neutral exit conditions are used to resolve the flow field\(^3\). These neutral boundaries do not substantially influence the flow variables at the exit, so the solution at the exit of one region can be directly linked to the inlet conditions for the next region.

After solving for the liquid region, the “internal” boundaries of the channel are checked for agreement, and boundary condition profiles updated (if required) using the process described in Section 4.4.1 and summarized in Figure 4-10. These walls exhibit reasonable convergence (less than 5 percent difference) after 3-4 manual iterations.

The liquid and prechannel regions are also coupled due to the contact areas on the outlet face of the prechannel and the inlet face of the liquid region. This coupling allows for the conduction heat sink (located on the mounting flange in the liquid region) to extract energy out of the prechannel region. If the temperatures (or heat fluxes) are different along these two faces, the average value is calculated and applied as the updated boundary condition. Changing this boundary condition has no significant effect on the internal walls of the liquid region, eliminating the need for any additional treatment of the profiles along these surfaces.

---

\(^3\) Neutral boundary conditions are used to model flow exits where details of the flow (velocities, pressures, etc.) are not known a priori. These types of boundaries are appropriate for fully developed flows, because they assume a zero normal gradient for the flow variables (except pressure) [28].
The inlet conditions to the boiling region are mass flow rate per channel and an inlet temperature taken to be one degree less than the saturation temperature. Both of these values are known a priori. Therefore, the only boundary condition that couples the boiling onset region with the liquid region is the heat flux across each portion of the solid-fluid interface.

Because conjugate heat transfer is not included in the densely meshed boiling onset region, the heat fluxes along the channel boundaries must be interpolated between the liquid and two-phase regions. To develop the interpolating function, the heat flux is stored at the outlet of the liquid region and the inlet of the two-phase region. In addition, because the segments are solved sequentially in the flow direction, the wall heat fluxes are not available from the two-phase region when initially solving for the boiling onset region. In this instance, the wall fluxes from the liquid region are applied as constant heat flux boundaries and updated once the two-phase region has been resolved.

For the inlet conditions for the two-phase region, the phasic mass flow rates at the outlet of the boiling onset region are used. Because axial and azimuthal heat flow is neglected, adiabatic conditions are placed on along all “internal” boundaries. This region contains several results of engineering interest, such as the dryout point and exit temperature.

Finally, after solving for the two-phase region, a complete solution for the vaporizer is obtained. To generate this approximate, initial solution, a first-order linear upwinding scheme was implemented using a first-order accurate pressure scheme. To check for self-consistency between all of the individual regions, the entire solution is repeated one additional time. For this final sweep over each segment, upwinding was performed using the Quadratic Interpolation for Convective Kinetics (QUICK) with a second-order pressure scheme. The QUICK formulation is third-order accurate which helps to mitigate the unfavorable effect of artificial diffusion that can
occur when using low-order upwinding schemes. This is especially useful in the boiling regions, where steep gradients in the vapor fraction need to be resolved.

The basic structure of the Fluent solver is shown in Figure 4-14, including the implementation of user-defined functions (UDF). In all segments, temperature-dependent fluid properties (listed in Appendix A) are calculated. In the boiling onset and two-phase regions, UDF’s that govern the vapor sources are applied, with an associated volumetric energy sink. In the liquid region, a UDF is used to apply a heat flux (or temperature) profile along the upstream and downstream faces. These UDFs are developed in C programming language and interpreted in the FLUENT simulation.

In cells with incompressible phases, the velocity and pressure are coupled using the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm. This algorithm provides an estimate for the pressure field, $P^*$, for use in calculating the estimated velocities ($u^*$, $v^*$, $w^*$).
\(v^*, w^*\) in the momentum equation. Because these velocities are based on a guess for the pressure field, they will not necessarily satisfy the continuity equation. Using the continuity equation, a pressure correction is calculated and applied to the initial pressure guess. This updated pressure is then used to re-solve the momentum equation. This process is repeated until convergence is reached. Because of the nature of the iterations, this type of method is called a predictor-corrector scheme.

In general, due to the vapor and energy sources, multiphase simulations are difficult to converge. Therefore, the concept of relaxation factors are used in generating the value of the iterates at successive iterations. A relaxation factor \(\omega\) is implemented in the following form,

\[
\Theta_k = \Theta_{i-1} + \omega (\Theta_i - \Theta_{i-1})
\]

where \(\Theta\) represents any flow variable. The subscripts “\(i\)” and “\(i-1\)” refer to the calculated solutions at the current and previous iterations, respectively. The subscript “\(k\)” refers to the actual value stored as the solution at the current iteration. For a relaxation factor equal to one, the stored solution is identical to the calculated solution. If a relaxation factor is greater than unity, the process is called successive over-relaxation, which can be used to accelerate convergence. If the factor is less than unity (as they are in the present study), the process is called under-relaxation.

For critical equations, such as the pressure-velocity coupling equation, the under-relaxation factors are kept small \((\omega < 0.5)\) throughout the simulation. In the boiling onset region, for example, the pressure field is very sensitive to changes in the void fraction due to the large resulting changes in the flow properties. Using a small under-relaxation factor for the pressure in this region will ensure that these otherwise rapid fluctuations will occur over several iterations. This allows for the velocity and pressure fields to react more gradually to changes in
the fluid properties of the mixture, thus enabling the system to reach equilibrium in a more stable (but more lengthy) manner.

Because they result in lower velocity magnitudes and smaller flow gradients in general, low flow rates tend to be more stable than higher flow rates. Because of this increased stability, the simulations were solved parametrically in the sense that the inlet mass flow rate was gradually increased from the lowest flow rate (40 mg/s) to the other flow rates of interest (80 mg/s and 120 mg/s). Too large of an increase between flow rates, or transitioning from high to low flow rates, was observed to have the effect of reversing flow at the outlet (which destroys solution stability). This method also helped to reduce simulation time by providing a solution for a low flow rate to become a reasonably accurate initial estimate for each successively higher flow rate.

Stability and convergence were assessed by monitoring the residuals of the flow variables, as well as by monitoring key values in the domain. For example, in the boiling onset region, large gradients in the void fraction forced the usage of small relaxation factors ($\omega < 0.5$). Because small relaxation factors can have the undesired effect of artificially reducing residuals (thus giving the appearance of a converged solution), the exit mass flow rate and void fraction were also monitored to help confirm that convergence of the simulation was obtained.
4.6 Results and Discussion

4.6.1 ALFA$^2$ Baseline

The evolution of the volume fraction at the onset of boiling is shown in the contour plots of Figure 4-15, which represent a slice down the helix midline. Consistent with the results from the network model and Figure 4-2, a steep increase in the void fraction is observed in the flow direction. For all flow rates, the characteristic “bullet” shape of a vapor slug (discussed in Section 3.2) is seen as low density vapor is advected into the core of the flow. Also, the apex of the cone is pressed towards the outer channel wall, with the severity of the pinching decreasing as flow rate increases. This observation suggests that the primary cause for this pinching is due to the large amount of vapor production along the inner channel wall, rather than by a centrifugal force (which would result in greater pinching at higher flow rates and higher velocities).

Figure 4-15. Contours of void fraction at the onset of boiling.
Several cross-sectional slices are shown in Figure 4-16 for the 80 mg/s case. The cross-sections shown are perpendicular to the path coordinate, \( s \), and the location of each cross-sectional slice is reported as the distance from the beginning of boiling onset, \( \Delta s_{ob} \). While vapor production occurs primarily along the inner channel wall, it is seen that the short side walls of the channel also contribute to the formation of vapor. The void fraction is lowest along the upper channel because this wall tends to extract energy from the flow, leaving little excess for the production of vapor. Also, the vapor is seen to form a film along the channel walls. The consequence of this low conductivity vapor film (\( k_V = 0.17 \ \text{Wm}^{-1}\text{K}^{-1} \)) is that it reduces the amount of energy that can be absorbed into the fluid.

Figure 4-16. Evolution of void fraction across channel cross-sections at onset of boiling for 80 mg/s flow rate. Distance, \( \Delta s_{ob} \), is measured from the onset of boiling.
The effect of this decrease in heat transfer between the solid channel and the two-phase mixture is shown in Figure 4-17 (taken from the two-phase region for the 80 mg/s case). The coupled domains have been highlighted with black outlines for clarity. Because of the high concentration of low conductivity vapor along the inner surface of the channel, a localized region of increased temperature occurs directly below the channel. This hot spot drives the heat to flow around the channel where the thermal (conduction) resistance is significantly lower. In fact, the heat flow through the solid channel walls is seen to actually curl around to the upper portion of the fluid domain.

![Figure 4-17. Cross-section of the temperature contours in the fluid and solid domain.](image)

Consistent with network model predictions, the radial temperature gradient across the cathode assembly is relatively small. From Figure 4-17, the inner and outer surfaces of the cathode have a minimum temperature of 1427 K and a maximum temperature of 1444 K, respectively. Even with the localized effect of the vapor film, the cathode is observed to support a total radial temperature gradient of only 17 K in the two-phase region of the vaporizer.
This radial gradient increases significantly in areas surrounding the actively cooled mounting flange. Over the outer half of the flange, a 500 K boundary condition is applied to simulate the mounting surface of the cathode as shown in Figure 4-18. In these areas, the radial temperature gradient increases to 86 K. Capturing the effects of axial heat transfer in this region is important because it tends to drive heat towards the mounting flange. This decreases thermal efficiency by increasing parasitic heat loses through the actively cooled mounting surface of the flange.

The magnitude of this axial heat flux through the upstream and downstream walls in the region near the flange is shown in Figure 4-19, where “positive” values are directed inward into the volume. The origin of the $x$-axis corresponds to the beginning of the channel region. Along
the entire length of the liquid region, a large axial heat flux is directed towards center of the flange thickness ( \( z \approx 1.75 \) cm from the start of the channel). To the left of the flange, the heat flows into the upstream facing surface and out of the downstream facing surface of the channel. At the flange midpoint, there is zero axial heat transfer, as heat energy flows only in the radial direction out of the cathode body. The magnitude of the axial heat transfer is greatest near the beginning and ends of the flanges. Note also that the axial component of the heat flux obeys the constraint posed in Eq. (4-22). This is a direct consequence of the manual iteration process on these “internal” boundary conditions. The effect of these heat fluxes was neglected in the network model, and therefore represents a major refinement of the FVM approach.

Figure 4-19. The calculated axial heat flux along the downstream wall
Figure 4-20 shows the contours of mixture velocity magnitude at the exit of the short boiling onset region, which corresponds to the location $\Delta s_{ob} = 11.08$ mm (from Figure 4-16). Clearly visible are two high velocity “cores” in the two inner corners of the flow. Note that these two cores are each adjacent to two “hot” channel walls. Between these two cores, the fluid is adjacent to a single hot wall, but surrounded by cooler fluid everywhere else. The resulting large vapor production in these two lower corners (as can be seen in Figure 4-16) drives the flow to high velocities.

![Velocity Contours](image)

**Figure 4-20.** Contours of velocity magnitude at exit of boiling onset region.

![Pressure Contours](image)

**Figure 4-21.** Contours of relative static pressure at exit of boiling onset region.

Figure 4-21 shows the static pressure contours that correspond to the velocity field in Figure 4-20. These pressure field throughout the domain is reported relative to a designated reference pressure. Because a constant vapor density was used in conjunction with neutral outflow conditions, the value and location of this reference pressure is completely arbitrary. The negative pressures listed in legend of Figure 4-21 are a consequence of placing the reference
pressure location in the lower right hand corner of the cross-section. Also, it is seen that the outer portion of the channel contains the largest pressures; a result suggested by vapor volume fraction contours of Figure 4-15. Overall for the 80 mg/s case, however, a relatively small pressure variation (<0.14 Pa) is observed across the entire cross-section.

The results for the prechannel are shown in the next two figures. In Figure 4-22, the temperature contours are shown across an axial cross-section through the feedtube. Increased temperatures are observed in the area opposite of the feedtube. This is caused by the increased residence time of the liquid lithium in the inlet plenum. The left side of the figure shows an enlargement of the fluid-solid interface, along with an expanded view of the isotherms.

![Contours of Static Temperature, K](image)

Figure 4-22. Temperature contours of cross-section in the prechannel region for the 80 mg/s flow rate.

The pressure contours in the prechannel region are shown in Figure 4-23. The trends are identical for all three flow rates, with the only difference being the total pressure drops through this region. The pressure drop through this region is on the order of a 1 Pa, and can therefore be
considered negligible relative to the large pressure drop through the remaining vaporizer channel, as will be discussed in the next section.

4.6.2 Performance Sensitivity

The next three figures present model results for the ALFA$^2$ baseline case for flow rates of 40 mg/s, 80 mg/s, and 120 mg/s. As with the network model, these curves begin at the location corresponding to the inlet to the helical channels ($x=3.44$ cm). To generate these plots, the flow variables are averaged along the axial coordinate. Because of the pitch of the helical channel, this process is slightly different than averaging the flow across a constant value of the path coordinate $s$ and then subsequently converting this path variable to an axial location. By averaging across a constant value of the axial coordinate, the points used have a variation in the streamwise direction of $\sim 13$ mm. While averaging was done in this manner for convenience, the averaging method employed is not expected to change the results or the observed trends in any appreciable way.

Unlike the network model that assumed a one-dimensional flow, the FVM includes full three-dimensional physics. Evidence of this is seen in Figure 4-24, where the fluid temperature
during boiling does not remain exactly at the saturation temperature, but rather slightly (but steadily) increases. This reason for these elevated temperatures are that portions of the domain have become fully vaporized \( \alpha_v = 1 \) and can thus support superheating. Referring to the void fraction contours of Figure 4-16, for example, it is seen that the fluid along the inner surface of the channel will vaporize more quickly than fluid along the outer surface. Even though the average void fraction across the cross-section will be less then unity, the fluid along the inner surface (when fully vaporized) will begin to superheat.

In Figure 4-24, the knee in the fluid temperature curves prior to superheating indicate that the lithium flow has been fully vaporized (determined mathematically as the point where the void fraction increases above 0.99999). The saturation temperature for each flow rate is taken from Table 4-2 for different operating pressures. The dashed line represents the location of the channel exit for the ALFA\(^2\) baseline geometry \( x = 22.68 \) cm. For the nominal 80 mg/s baseline case, the dryout point occurs at \( z = 17.6 \) cm. Decreasing the total flow rate by half (40 mg/s), the dryout point decreases to \( z = 13.1 \) cm. Note that while the 40 mg/s and 80 mg/s cases both fully vaporize and superheat, the 120 mg/s case never reaches its dryout point. This propellant flow rate will therefore expel liquid lithium into the outlet plenum (and multichannel hollow cathode). This result is consistent with the network model that predicts a dryout point of \( z = 23.1 \) cm, which is slightly longer than the modeled channel length. In addition, it is seen that when the flow reaches the dryout point, the temperature of the vapor increases rapidly. The slight leveling off at the end of these curves in a numerical artifact of the post-processing when plotting.

Figure 4-25 shows the void fraction for all three flow rates. As predicted by Figure 4-2, the void fraction quickly increases from zero to nearly one. This is a consequence of the
unfavorable density ratio and the primary reason for partitioning the cathode into computational segments.

Figure 4-24. Temperature profiles as calculated by the FVM for the three flow rates tested.

Figure 4-25. Void fraction for the three flow rates tested.
Convergence of the simulation was assessed by evaluating the residuals and checking that the dryout point location no longer changed with subsequent iterations. In addition, the exit mass flow rate was one of the parameters that is monitored throughout the simulation to assess the convergence. As mentioned earlier, the vaporizer was partitioned into four segments or regions which were solved separately. These four segments were a prechannel region (containing the inlet tube), a liquid only region, a boiling onset region, and a two phase region. Mass was “conserved” in all domains except the two-phase region, where it was found to differ from the inlet mass flow rate by up to a factor of five. The cause for this mass deficiency is believed to be the result of the extremely high density ratio, which ranges from 40,714 for the 120 mg/s flow rate to 117,352 for the 40 mg/s flow rate. As seen in Figure 4-2, where the void fraction is plotted as a function of vapor quality (ratio of vapor to total mass), density ratios this large will result in an extremely large jump in the void fraction at the onset of boiling.

The fact that the void fraction is extremely close to one throughout this segment, as seen in Figure 4-25, can be problematic. In a numerical context, this can have severe consequences because numerical round-off and truncation errors may exceed the actual gradient in the void fraction. Because $\alpha_v$ is the parameter used for mixture weighting (as discussed in Section 4.1), computational precision may not be sufficient for the mixture model to conserve mass when $0.99999 < \alpha_v < 1$. A significant portion of two-phase flow persists after the point where this range is met which can also lead to an accumulation of errors. This explanation was subsequently checked using less severe density ratios ($1 < \rho_L/\rho_v < 500$). These simulations converged successfully and conserved mass at the outlet, thus appearing to confirm the explanation for the mass deficiency described above. This suggests that decreasing the density ratio will decreases the mass deficiency at the channel exit.
Figure 4-26. Pressure drop across vaporizer channels for three flow rates tested. Pressure is relative to the channel exit.

The pressure drop across the channels for each flow rate is shown in Figure 4-26, where the pressures are measured relative to the channel exit. Because the mixture model formulation neglects all non-equilibrium effects, the FVM will tend to underpredict the pressure drop in the two-phase region. This effect is exacerbated in microchannel flow due to the increased importance of surface tension effects (see Section 2.1).

A steep increase in the pressure gradient at dryout is seen in Figure 4-26, which is caused by increases in the local Reynolds number as the flow becomes vaporized. The low density vapor flow moves at a higher velocity causing a significant pressure gradient $\Delta P$. Safety margin, in terms of the amount of vapor superheat actually required to insure against expelling liquid lithium, needs to be determined so that excessive pressure drops do not result from the additional channel length of vapor flow.

In Figure 4-27, the ability of the prechannel inlet plenum to uniformly distribute the total mass flow is assessed, where the mass flow through the $i$-th channel is shown normalized with the mass average as $\dot{m}_i / \dot{m}_{AVG}$, where the $\dot{m}_{AVG} = \dot{m} / (\# \text{channels})$. For a perfectly
distributed flow, the non-dimensional flow rates should equal one. The corresponding outlet
numbers are listed on the inset of the figure, and are numbered clockwise from the direction of
the feedtube. Although reasonably well distributed, the channel closest to the feedtube shows a
higher percentage of the total flow. As a point of reference for the 80 mg/s case, the maximum
difference between channel flow rates is 4.3 mg/s. As the flow rate is increased, however, the
inlet plenum becomes more effective at uniformly distributing the total mass between channels.

![Graph showing flow rates and temperatures]

Figure 4-27. Assessment of the outlet temperatures and the distribution of total mass
flow rate through pre-channel region.

Fluid temperatures at each channel outlet are also shown in Figure 4-27 normalized by
the average of the outlet temperatures. Higher temperatures occur in the channel opposite of the
feedtube due to the increase in residence time of the flow through the inlet plenum. Even with these elevated temperatures, the maximum temperature difference between fluid outlets for the 80 mg/s case is 26 K. These results validate the thermally-identical channel assumption made in Section 4.2.3 when exploiting the helical symmetry of the vaporizer channels. However, because channel 1 will process the largest flow rate at the lowest inlet temperature, using average values may result in slight underpredictions of the dryout point in this channel (while overpredicting the dryout point in channel 3).

4.6.3 Comparison with Network Model

The next two figures directly compare the results of the network model with the results of the FVM. The temperature profile predictions for both modeling approaches are shown in Figure 4-28 for the 80 mg/s case, where the largest area of disagreement occurs in the region near the heat sunk mounting flange. This result is expected because the large axial component of heat flux in this region (as shown in Figure 4-19) was neglected in the development of the network model.

From an engineering standpoint, however, several key results compare favorably between the two models, including exit temperature and the location of the dryout point as shown in Table 4-3. For the 40 mg/s and 80 mg/s cases, a maximum disagreement of 13.0 percent and 4.2 percent is seen in the dryout point and exit temperature, respectively. For the 120 mg/s case, both models predict that the flow never fully dries out for the ALFA$^2$ baseline length of 22.88 cm. Therefore, the exit temperatures calculated by each modeling approach represents the temperature of the liquid-vapor mixture at channel exit.
Figure 4-28. Comparison between the network model and FVM predictions for the fluid temperature profile for the 80 mg/s case.

Figure 4-29 shows a comparison of the void fraction profile between the two models for the 80 mg/s case. The zero location on the $x$-axis corresponds to the location of boiling onset. While the two profiles evolve in slightly different manners, the overall result at the outlet of the boiling onset region are in good agreement. As a point of comparison, the FVM predicts an outlet void fraction of 0.936, while the network model result is 0.965.

This result suggests that while axial heat transfer has a large effect on the shape of the fluid temperature curve, the overall thermal efficiency of the vaporizer is reasonably captured using the approximations of the network model. This is an important result, as the network model is much faster and easier to implement than the FVM. In the present study, the FVM required several weeks to converge, while the network model only takes several minutes.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Network Model</th>
<th>Finite-Volume Mixture Model</th>
<th>Disagreement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dryout Point&lt;sup&gt;a&lt;/sup&gt;, cm</td>
<td>N/A</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>40 mg/s</td>
<td>10.7</td>
<td>12.3</td>
<td>13.0 %</td>
</tr>
<tr>
<td>80 mg/s</td>
<td>16.7</td>
<td>17.6</td>
<td>5.1 %</td>
</tr>
<tr>
<td>120 mg/s</td>
<td>N/A</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>Exit Temperature, K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>40 mg/s</td>
<td>1948</td>
<td>1872</td>
<td>4.1 %</td>
</tr>
<tr>
<td>80 mg/s</td>
<td>1866</td>
<td>1791</td>
<td>4.2 %</td>
</tr>
<tr>
<td>120 mg/s</td>
<td>1377</td>
<td>1432</td>
<td>3.8 %</td>
</tr>
</tbody>
</table>

<sup>a</sup> At 120 mg/s, both models predict that the flow does not dryout for the baseline ALFA<sup>2</sup> baseline length of 22.88 cm.

Figure 4-29. Comparison between the network model and FVM void fraction predictions for the 80 mg/s case.
5 CONCLUSIONS

5.1 Network Model

The network model is based on a resistive network which assumes one-dimensional (radial) heat flow through the lithium vaporizer. This model is characterized as first-order because it does not include axial conduction through the structure nor the details of the fluid dynamics (such as boundary layers). Despite these simplifications, the model reveals sensitivity of the vaporizer performance to several design parameters including cathode tube emissivity, mass flow rate, vapor superheat, and number of channels. An interesting result obtained by the network model is that there exists an optimal design point (cathode length) that minimizes power consumption and maximizes thermal efficiency.

Of the parameters considered, the vaporizer had the greatest sensitivity to the cathode tube emissivity. This is not surprising given that this represents the dominant energy flow path out of the vaporizer in this model. These results suggest that to the limited extent that emissivity can be controlled in the design (through surface finish, chemical treatments, growth of oxide layers, etc.), the emissivity should be as low as tolerable in order to minimize the heater power required for a cold-start. Results obtained from the thermal model for the overall thruster, including the arc heat flux, should help in determining the range of allowable values for the emissivity during operation. Another strong driver of vaporizer performance is the heat sunk mounting flange, which caused, in the case modeled, the recondensation of lithium vapor and sudden decreases in the temperature of the cathode assembly (including the fluid). While increasing the thermal resistance to this flange would be favorable (through a thermal choke, insulating ring, etc.), this approach may not be practical, as the mounting flange will need to be able to conduct several thousand amperes of current from the bus bar to the cathode.
The minimum power required to vaporize a fluid between two thermodynamic states scales linearly with mass flow rate. This trend was evident in our results. There are second order effects as well, since increasing the flow rate will increase the Reynolds number and enhance convective heat transfer. This will have the effect of slightly changing the shape of the vaporizer length-power curves. When we investigated the effect of using multiple channels (between one and six) to vaporize the lithium, we observed the same trend. Fewer channels for the same mass flow rate have the effect of increasing the Reynolds number (and the convective heat transfer) in a given lead/channel, resulting in only minor efficiency gains. These results suggest that the choice of whether to use multiple channels should be based on considerations such as improving uniformity of flow into the plenum or reducing risk of blockage rather than improving thermal efficiency.

Also investigated was the performance sensitivity of the vaporizer to the amount of vapor superheating. This parameter is driven by the desire to maintain some margin between the calculated dryout point and the channel exit to minimize the risk of liquid expulsion or condensation of lithium vapor in the plenum during cold-starts. The baseline case assumed a vapor superheat of 100 K, which corresponds to a thermal efficiency of approximately 56 percent. At a superheat of 200 K, the efficiency drops to 55 percent, but falls to less than 48 percent at 300 K. This is a direct consequence of the fact the vapor is nearly in thermal equilibrium with the channel walls and there is very little temperature difference to drive the heat flow into the fluid. This penalty for margin can be mitigated somewhat through the use of low-emissivity radiation shielding that was shown to significantly decrease the thermal efficiency penalty at higher exit temperatures. Further analysis of flow instabilities which could lead to
oscillation of the dryout point along the channel should provide some information as to how much margin is really necessary.

5.2 Finite-Volume Model

A finite volume model of the vaporizer was implemented using a single-fluid mixture model that included conjugated heat transfer to the surrounding solid channel. Three flow rates were examined in detail; 40 mg/s, 80 mg/s, and 120 mg/s. While showing localized disagreement in the region near the mounting flange, reasonable agreement in the overall solution was obtained with the network model, with maximum disagreements of 13.0 percent in predicting the dryout point and 4.2 percent in predicting the exit temperature.

For a fixed power and length, increasing the propellant flow rate (which may be desired to change thruster performance, for example) will have practical limits. For the 120 mg/s flow rate, the flow did not dry out and would thus expel liquid lithium into the outlet plenum. This result was in agreement with the network model. During cold-starts, this will have serious consequences for an operating thruster. However, if some margin is designed into the baseline operating point, a reasonable range of flow rates can be accommodated as suggested by the relatively small change in the dryout point from 17.6 cm to 12.3 cm for 80 mg/s and 40 mg/s case, respectively.

Several hot spots were identified in the cathode design. One such location was in the prechannel region in the area opposite the feedtube. This is caused by a longer residence time of the lithium because it needs to flow a greater distance before entering into the helical channels. Another hot spot was captured in the two-phase flow region of the solid channel. Low conductivity vapor has the tendency to coat the channel walls and “insulate” the fluid. This
causes a decrease in the heat flux into the lithium that results in higher temperatures along the inner portion of the channel wall. Another consequence of this hot spot is that it causes the heat to conduct around the channel due to the lower resistance to heat flow through the solid channel wall.

Overall, the inlet plenum in the prechannel region does an adequate job at uniformly distributing the total flow into the four channels. However, at 80 mg/s, the channel located closest to the feedtube needs to process 12.5% more lithium than a uniformly distributed flow. Therefore, using an average mass flow rate may underpredict the dryout point.

5.3 Observations

Based on the insights gained from the results obtained by the modeling efforts discussed in this work, some general recommendations for increasing the thermal performance of the vaporizer are presented. The design recommendations presented below are driven by considerations of vaporizer performance, and may not be feasible to implement in an actual thruster design. These recommendations are based on observations from the results presented.

1. **Cathode Outer Diameter Should be as Small as Possible**

Decreasing the diameter of the outer surface of the cathode body decreases the radiation losses out from the surface. Regardless of the length of the vaporizer, a consistent result from the modeling work is that reducing the outer area of the cathode increases thermal efficiency by presenting a smaller area for radiation losses.

Concurrent with this recommendation is to make the mounting flanges as large in diameter as possible while maintaining the same bus bar contact area. Additionally, this contact
area should be as small as required by considerations of electrical current carrying capacity. Coupled with a small outer cathode diameter, a large flange will increase the distance of the low thermal resistance path between the lithium fluid and the heat sink. Also, the smaller the cathode body, the more choked the heat flow will be out of the cathode tube and into the cool mounting flange (i.e. it will need to pinch into a smaller area). In this context, the necking of the cathode between the cathode tip and the channel exit (seen in Figure 1-2) is recommended, and should be exploited further if materials and manufacturing capability permit.

2. **Cross-Sectional Channel Dimensions Should be as Small as Possible**

Small cross-sectional channel dimensions have the effect of increasing the heat transfer into the fluid, both by increasing the Reynolds number of the flow and by forcing microchannel flow patterns. These flow patterns trap the high-conductivity liquid against the channel walls, thereby allowing more energy to be absorbed into the fluid (as opposed to lower conductivity vapor films that can insulate the fluid). Therefore, microchannel flow is recommended to minimize the power consumed during vaporization. Section 2.1 can be referred to for microchannel flow criteria. Note, however, that microchannels can also have the effect of significantly increasing the pressure drop through the vaporizer due to the increased mass flux (Figure 3-23). In practice, the channel dimensions will likely be determined by balancing the enhancement of heat transfer and maintaining reasonable feed system pressures.

Based on the network model sensitivity studies (Section 3.8.3), however, it may be possible to accomplish both of these requirements. A cathode design using small channel dimensions will enhance the heat transfer. A cathode design including many channels (i.e. a vaporizer tube with multiple starts) will decrease the pressure drop. Note that as the number of
channels is increased, the pitch of each individual helical channel will become larger. Therefore, by using many channels, the helical channels will need to complete less revolutions over the length of the vaporizer, and in the limit will correspond to a straight channel from the inlet plenum to the outlet plenum.

This concept could be extended as well. In the current ALFA\textsuperscript{2} design, the channel geometry permits heat to bypass the fluid by flowing around through the solid, directly into the cathode tube. Ideally, one would want all of the supplied energy to be forced to flow into the fluid. To reduce this heat flow path, an extension of the concept introduced in the previous paragraph could be applied. In the limit of using the maximum number of channels, each individual channel would be a straight axial groove that is small in cross-section and extremely close to its neighboring channels. If manufacturing capability permits, the thin walls can remain, serving almost in the capacity of a fin, maximizing fluid-to-surface contact area.

Alternatively, removing these thin “fins” completely would transform the small individual channels into a singular annular channel that extends the entire length of the vaporizer that is geometrically similar to the inlet plenum. A thin annular ring will still provide the benefits of a microchannel (because the gap size is still small), and may help minimize pressure fluctuations because there are a fewer number of channel walls to support transient phenomena, including nucleation/bubble growth and intermittent flooding, as well as wave instabilities that exist in the trapped liquid film that coats channel surfaces. Also, the annular ring will transform the flow dynamics from that of a quasi-one-dimensional flow to a full three-dimensional flow, which will allow the confined bubbles (and the associated pressure fluctuations) to expand laterally as well as along the path coordinate. The annular channel also has the advantage of
eliminating the low-resistance (conduction) heat flow path around the channel walls, which may increase the overall thermal efficiency of the vaporizer.

3. Feedtube and Inlet Plenum Relocation

In the pre-channel area, the area upstream of the inlet plenum ($L_{NC}$ from Figure 3-1) should be made as small as possible. This region adds no value to vaporizer performance and directly decreases the optimal thermal efficiency of the vaporizer by adding a heat flow path around the fluid. Also, the feedtube should be located as close as possible to the back of the inlet plenum (towards the heater flange), similar to the MAI design (see Figure 2-1, for example). Doing so will reduce the flow stagnation currently observed in the upstream end of the inlet plenum and potentially result in a more uniform distribution of the total mass flux between channels.
6  FUTURE WORK

6.1 Transient Modeling

While the focus of the present study was to understand the steady-state thermal-fluid behavior of the vaporizer, many characteristics of a two-phase vaporizing flow are inherently unstable and transient. Among others, these phenomena include wave structures in the liquid film and intermittent flooding, as well as bubble growth and coalescence. To better capture the physics of these flow structures, the present work can be extended to the development of a transient simulation. Understanding the nature of these phenomena will help to provide a better characterization of fluctuations in the total pressure drop across a vaporizer channel. A direct consequence of these pressure drop fluctuations are variations in the total mass flow rate of propellant, thereby resulting in unsteady performance characteristics of the thruster.

The mixture model implemented in this study is not applicable to this type of transient modeling because of the equilibrium conditions inherent in developing the governing equations. Applying the mixture model in a time-dependant framework would provide details only of the initial start up period. While this information may be helpful, more valuable insight could be gained by investigating the transient effects of non-equilibrium disturbances in the system caused by surface tension effects, heat load fluctuations, and bubble dynamics. Insights into the nature of any pressure fluctuations are likely to result in strategies to attenuate such disturbances. These strategies can range from simple modifications to inlet/outlet plenum geometry to more complex measures, such as mechanical pressure regulating valves.

While volume-of-fluid (VOF) or front-tracking schemes would be able to capture the details of bubble interaction, these methods are unlikely to be implemented on full-scale domains due to prohibitively large computational requirements. Typically, these methods are limited to
domains on the order of several bubble diameters. The transient problem, therefore, will likely require a parallel approach.

In this context, one advantage of two-phase microchannel flows is that any individual vapor wedge will interact with only two additional vapor wedges. Therefore, high-fidelity VOF or front-tracking simulations can be developed that capture the detailed interactions between neighboring pairs of vapor wedges. Information gained from such modeling approaches can then be extended to a two-fluid mixture model. While remaining tractable for a full-scale domain, this two-fluid modeling approach is more amenable to the inclusion of non-equilibrium effects because field equations are solved for each individual phase (rather than the mixture of phases).

Regardless of the solution methodology employed, the cathode partitioning framework and the manual iteration process described in Section 4.4.1 will not support a transient simulation. To capture the effects of unsteady phenomena, all regions and segments of the vaporizer assembly will need to be directly coupled at each iteration. Combining all of the vaporizer domains while maintaining computationally feasibility, in terms of model size and solution times, will be a challenge.

### 6.2 Adaptive Grid Refinement

The primary driver behind partitioning the cathode into several small segments was the short region located at the onset of boiling, where the steep increase in the vapor volume fraction cause sharp gradients in the flow field. The objective of the partitioning approach was to be able to isolate and resolve this problematic region by increasing element density. To be self-consistent within this framework, extensive handling, tracking, and patching of the boundary conditions is required. To eliminate this need, an adaptive grid refinement (AGR) algorithm can
be implemented based on some set parameters (such as volume fraction gradient or relative magnitude of the vapor source terms). Localized grid refinement in these areas can be used to track this boiling onset region as it drifts back through the domain. Also, unlike the partitioning approach, AGR will support a transient modeling approach.

6.3 Pressure Dependence

In the present study, the vapor density was calculated using a global operating pressure. Thus the density was invariant to local changes in pressure. The saturation temperature was calculated similarly. This simplification was made to aid the stability and convergence properties of the simulations, as well as to eliminate additional manual patching of pressure boundary conditions at the outlet of each vaporizer segment. Each of these issues can be minimized somewhat by using the results of present work to provide a reasonably accurate initial estimate of the pressure across the vaporizer channel. Therefore, a straightforward extension would be to include pressure dependent saturation temperatures and usage of an equation of state for the vapor density. This approach would also allow for experimentation with pressure-driven evaporation models, in favor of the temperature-driven source terms used in the present study.

6.4 Experimental Work

To better validate the modeling results, experimental measurements of temperature (inlet and outlet temperature, for example), mass flow rate, and pressure drop are required. Based on this experimental data, the physics occurring in the vaporizer can be better understood, and the models can be analyzed to identify areas of improvement. This will provide a basis of comparison that can be used to further refine the network model.
### APPENDIX A: MATERIAL PROPERTIES

All temperature units are in degrees K, and pressure units are Pa. Units for the calculated property are included in brackets where applicable.

#### Tungsten

**Emissivity**  
\[
\varepsilon = -2.6858 \times 10^{-2} + \left(1.819696 \times 10^{-2}\right)T - \left(2.1946163 \times 10^{-8}\right)T^2
\]

**Thermal Conductivity**  
\[
k = \left(-1.97490 \times 10^{-8}\right)T^3 + \left(8.86502 \times 10^{-5}\right)T^2 - \left(0.147765\right)T + 198.053
\]

#### Liquid Lithium

**Thermal Conductivity**  
\[
k = 9.59 + \left(4.55 \times 10^{-3}\right)(T_F - 273.15)
\]

**Density**  
\[
\rho = 1000 \left\{0.515 - \left[1.01 \times 10^{-4} (T_F - 473.15)\right]\right\}
\]

**Viscosity**  
\[
\mu = \left(3.89881 \times 10^{-10}\right)T_F^2 - \left(1.17037 \times 10^{-6}\right)T_F + 1.01388 \times 10^{-3}
\]

**Specific Heat**  
For \(T_F \leq 873.15\) K  
\[
c_p = \left(1.33490 \times 10^{-3}\right)T_F^2 - \left(23.36107\right)T_F + 5185.4404
\]

For \(T_F > 873.15\) K
\[ c_p = 4163.08 \quad \left[ \frac{J}{kgK} \right] \]

**Surface Tension***

\[ \sigma = \left( -1.60 \times 10^{-4} \right) T_p + 0.47298 \quad \left[ \frac{N}{m} \right] \]

**Saturation Temperature***

\[ T_{SAT} = \frac{8143}{\sqrt{8 - \log_{10}(133.3P_{SAT})}} \quad [K] \]

* Least-squares curve fit to data from Yih [34]

** Least-squares curve fit to data from Jeppson [35]

*** Equation adapted from Jeppson [35]
APPENDIX B: RAW SOURCE CODE

Network Model Matlab Script

```matlab
% simulation parameters
mdot_tot = 8e-6;       % Mass flow rate (kg/s)
flux = 184000;         % Baseline Heater flux (W/m2)
dq_i = 0.1;            % Power into element (W)
leads = 4;             % Number of leads, (thread paths)
mdot = mdot_tot/leads; % Mass flow per channel (mg/s)
P_op = 101300;         % Operating pressure (Pa)

% boundary conditions
T_super = 5;           % Amount of superheat (K)
Tfo = 500;             % Inlet fluid temp (degrees K)
Tg = 1000;             % Initial guess for outer wall temp (K)
Tinf = 300;            % Ambient temperature (K)
Tb = 1000;             % Temperature of the back end isolator plate(K)
T_plate = 500;         % Bus bar temperature (K)
T_bn = 600;            % Boron-nitride temperature
Tsat = 1377;           % Saturation temp of lithium (degree K)
T_cathode = Tsat - 5;  % Estimate of downstream cathode temp (K)

% toggle switches (1 is on, 0 is off)
Flange_Toggle = 0;     % stores temperature history across profile
Error_Toggle = 0;       % Calculates/stores upper bound error estimate
Plot_Toggle = 1;        % Plot results
Boiling_Toggle = 0;     % Plot boiling stats (quality, vapor fraction)
Flux_Toggle = 0;        % Calculates and plots flux data
Temp_Toggle = 1;        % Plots temperature history
Flow_Props = 0;         % Stores and plots flow props (Nu,h,rho,etc)
Pressure_Toggle = 0;    % Calculates and plots the pressure profile
PC = 1;                 % PC=1 Standard friction;  PC=2 Microchannel

% geometry(all units in m or m^2)
a = 0.75e-3;            % Channel depth
b = 2.5e-3;            % Channel width AXIALLY in x
c = 2.5e-3;            % Thread spacing AXIALLY in x
pitch = leads*(b+c);  % Thread pitch
t1 = 3.0e-3;           % Shneck thickness
t2 = 8.9e-3;           % Cathode tube thickness
D = 27e-3;             % ID of schneck (~OD of heater)
Dcl= D+t1+t1-a;        % Diameter of fluid centerline
C = pi*Dcl;            % Circum. at fluid centerline
Dc = D+(2*(t1+t2));    % Cathode tube OD (non-flared end)
Da = 190e-3;           % ID of anode
c_prime = c*cos(atan(pitch/C)); % Corrected thread spacing
b_prime = b*cos(atan(pitch/C)); % Corrected channel width
Dh = (4*a*b_prime)/(a+a+(2*b_prime)); % Hydraulic diameter of channel
```

Note: The text can be represented as plain text using the provided MATLAB script.
\[ Ac = a \times b_{\text{prime}}; \]  
% Cross-sectional channel area
\[ ds = dq_1/(\text{flux} \times (b_{\text{prime}} + c_{\text{prime}})); \]  
% Calculated grid spacing
\[ dA = (b_{\text{prime}} + c_{\text{prime}}) \times ds; \]  
% Area element (along s-coordinate)
\[ s_{\text{rev}} = \sqrt{(C^2 + (\text{pitch}^2));} \]  
% Distance along s per revolution
\[ L_{\text{nc}} = 14.20e-3; \]  
% Extra, unused heated length (m)
\[ L_p = 20.10e-3; \]  
% Length of pool section (m)
\[ L_{\text{pre}} = L_{\text{nc}} + L_p; \]  
% Total pre-spiral channel length
\[ rf1 = 50e-3; \]  
% Radius of heater flange
\[ rf2 = 68.58e-3; \]  
% Radius of cathode mounting flange
\[ rce = 29.6e-3; \]  
% Radius of flared end of cathode
\[ tf1 = 6e-3; \]  
% Thickness of both flanges
\[ fs = 12.5e-3; \]  
% X-coordinate start of mounting flange
\[ fe = fs + tf1; \]  
% X-coordinate end of the flange
\[ Hu = 46e-3; \]  
% Distance between flanges
\[ Hd = 246.80e-3; \]  
% Distance between flanges

% Other constants
\[ SB = 5.67032e-8; \]  
% Stefan-Boltzmann constant (W/m²K⁴)
\[ h_{\text{fg}} = 19.5e6; \]  
% Latent heat of vap. (J/kg)
\[ km = 90; \]  
% Thermal conduct. of moly (W/mK)
\[ kv = 0.1070; \]  
% Thermal conduct. of V-lith (W/mK)
\[ cp_v = 7794; \]  
% Heat capacity of V-lith (J/kgK)
\[ mu_v = 1.471e-5; \]  
% Viscosity of Vap. lithium
\[ dHm = \text{mdot} \times h_{\text{fg}}; \]  
% Latent heat thru phase change (W)
\[ G = \text{mdot} / Ac; \]  
% Mass flux in channel (kg/s m²)
\[ Rc = 7e-6; \]  
% Contact resistance (m²-K/W)
\[ R = 1197.8; \]  
% Gas constant for lithium (J/kgK)

% Solving for the temperature distribution in the PRE-channel region---
\[ [Y, h1] = \text{sectionA} (\text{flux}, Tfo, Tb, Tg, L_{\text{nc}}, L_p, tf1, D, tl, t2, rf1, \text{mdot_tot}, \ldots \)
Tinf, a, b, c, Rc); \]
\[ \text{Tf} = Y(14); \quad Tws = Y(4); \quad Twb = Y(3); \quad Twt = Y(8); \quad T_hf = Y(12); \]
\[ Twl = Tws; \quad Twct1 = Twt; \quad Twcb1 = Twb; \quad Twol = Y(9); \quad \text{Tflange} = Y(12); \]

% Initializing some variables--------------------------
\[ i = 0; \quad s = 0; \quad \text{quality} = 0; \quad \text{lhc} = 0; \quad \text{Tend} = \text{Tsat} + \text{T_super}; \quad x = 0; \quad d = 0; \quad dP\_tot = 0; \]
\[ f\_\text{ind} = 0; \quad \text{Prop\_Temp} = \text{Twol}; \quad \text{flag} = 0; \quad \text{Tg1} = 1400; \quad \text{Tg2} = 1300; \quad \text{Tg3} = 1200; \]
\[ \text{index} = 0; \]

% Integrating---------
\[ \text{while Tf} \leq \text{Tend} \]
\[ i = i + 1; \]
\[ \text{fluid\_T}(i) = \text{Tf}; \]
\[ \text{if} \ x < \text{fs} \]
\[ T_{\text{an}} = \text{Tinf}; \quad \text{E2} = 1; \]
\[ \text{else} \]
\[ T_{\text{an}} = \text{T_bn}; \quad \% \text{Boron-nitride temp} \sim 600K \]
\[ \text{E2} = 0.70; \quad \% \text{Radiation to boron-nitride insulator} \]
\[ \text{end} \]
\[ [E, k, cp, rho1, mu1] = \text{material\_props} (\text{Prop\_Temp}); \quad \% \text{Material props} \]
rhov = P_op/(R*Tf); % Density of V-lith
emm = (1/E)+((Dc/da)*((1/E2)-1)); % concentric emmitters
hr = SB*da/emm; % Radiation htc
R1 = (t1-a)/(k*da); % Resistance schneck
R2 = (a/2)/(k*c_prime*ds); % Resistance bottom
R2c = R2+(Rc/(c_prime*ds)); % Resistance upper
R3 = t2/(k*da); % Resistance outer
Y=0; RHS=0; AB=0;

if Tf <= Tsat
    h=h1_calc(Tf,Tws,Twb,Dh,G,a,b_prime,cp);
    if (x > fs) & (x <= fe) % under the flange?
        [AB, RHS, hrad] = get_flange_system(1,h,Tf,leads,pitch,SB,...
            dq_i,R1,R2c,R2,R3,b_prime,a,ds,Rc,da,mdot,s_rev,Hd,Hu,...
            Two1,T_hf,rf2,D,dc,rc,rf1,tfl);
        Y = solve_flange(Tg,AB,RHS,hrad,Tg1,Tg2,Tg3);
    else
        [AB, RHS] = get_AB(1,h,Tf,dq_i,R1,R2c,R2,R3,b_prime,a,ds,...
            da,mdot,cp,T_an,cp_v);
        Y = solve_temp(Tg,AB,RHS);
    end
    av = quality;
    lhc=0; flag=0;
    dP = get_dP2(ds,Dh,rhol,rhov,Ac,G,mu_l,mu_v,av,avm,PC,a,...
        b_prime,quality);
elseif quality <= 1
    if flag == 0;
        quality = (mdot*cp*(Tf-Tsat))/dHm;
        flag = 1;
    end
    h=h2_calc(Tf,quality,G,Dh,kv);
    if (x > fs) & (x <= fe)
        [AB, RHS, hrad] = get_flange_system(2,h,Tf,leads,pitch,SB,...
            dq_i,R1,R2c,R2,R3,b_prime,a,ds,Rc,da,mdot,s_rev,Hd,Hu,...
            Two1,T_hf,rf2,D,dc,rc,rf1,tfl);
        Y = solve_flange(Tg,AB,RHS,hrad,Tg1,Tg2,Tg3);
    else
        [AB, RHS] = get_AB(2,h,Tf,dq_i,R1,R2c,R2,R3,b_prime,a,ds,...
            da,mdot,cp,T_an,cp_v);
        Y = solve_temp(Tg,AB,RHS);
    end
end

lh=Y(7); % Heat absorbed into the fluid
lhc = lhc + lh; % Integrating dq_h
quality = lhc/dHm; % Mass fraction or quality
avm = av; % Storing volume fraction at last step
Cs = (1-(quality*(1-(rhol/rhov))))^(0.5); % Chisholm slip ratio
av = 1/(1+(Cs*((1-quality)/quality)*(rhov/rhol))); % Void fraction

if (quality < 0) % Loop to correct for recondensation
    quality = 0;
    av = 0;
    Tf = Tsat;
    flag = 0;
end
dP = get_dP2(ds,Dh,rhol,rhov,Ac,G,mu_l,mu_v,av,avm,PC,a,...
b_prime, quality));
else

    h = h3_calc(Tf);  % htc for superheated region

    [AB, RHS] = get_AB(3, h, Tf, dq_i, R1, R2c, R2, R3, b_prime, a, ds,...
                     dA, mdot, cp_T_an, cp_v);
    Y = solve_temp(Tg, AB, RHS);
    av = quality;
    dP = get_dP2(ds, Dh, rhol, rhov, Ac, G, mu_l, mu_v, av, avm, PC, a,...
                  b_prime, quality);
end

if Flange_Toggle == 1

    if (x > fs) & (x <= fe)  % Storing flange flux and temp. history
        f_ind = f_ind + 1;
        fl_location(f_ind) = x + L_pre;  fl_history(f_ind, 1) = Y(9);
        fl_history(f_ind, 2) = Y(10);  fl_history(f_ind, 3) = Y(11);
        fl_history(f_ind, 4) = Y(12);  oa(f_ind) = Y(13);  ob(f_ind) = Y(14);
        oc(f_ind) = Y(15);  od(f_ind) = Y(16);  oe(f_ind) = Y(17);  ot(f_ind) = Y(6);
    end
end

Twb = Y(2);
Tws = Y(3);
Twt = Y(4);

if Flow_Props == 1

    NuN(i) = Nu;  % Nusselt Number
    H(i) = h;  % Convection Coefficient
end

if Boiling_Toggle == 1

    Q(i) = quality;
    AV(i) = av;

    if (quality > 0) & (quality < 1)
        d = d + 1;
        [Ggl, Glp] = get_Baker_map(G, quality, rhov, rhol, Tf);
        By(d) = Ggl;  Bx(d) = Glp;
        Vg(d) = (G*quality)/rhov;
        Vl(d) = (G*(1-quality))/rhol;
        [Eo(d) We_LS(d) We_VS(d) Re_LS(d) Re_VS(d)] = Eotvos_Calc(Tf, ...
                                                                    rhov, rhol, Dh, G, quality, mu_v);
        bx(d) = (pitch/s_rev)*s;
        slip_ratio(d) = Cs;
    end
end

if Flux_Toggle == 1

    dq_o(i) = Y(6);
    dq_h(i) = Y(7);
    flux_o2(i) = (Twt - Y(5))/(R3*dA);
end

if Temp_Toggle == 1

    Tw_i(i) = Y(1);  Tw_cb(i) = Twb;  Tw(i) = Tws;  Tw_ct(i) = Twt;  Tw_o(i) = Y(5);
end

dP_array(i) = dP;
dP_tot = dP_tot + dP;  dP = 0;
Prop_Temp = Y(5);
S(i)=s;
Tf=Tf+Y(8);
Tg=Y(5);       % Outer wall temp. is guess for next element
s=s+ds;       % Marching along the channel
x = (pitch/s_rev)*s;
X(i) = x;

if Error_Toggle == 1
    condition=cond(AB);
    error1(i)=condition*length(AB)*eps / (1 - condition*length(AB)*eps);
    error2(i)=condition*eps;
    error3(i)=(condition*1.01*...
        ((length(AB)^3)+(3*length(AB)*length(AB)))*23*eps)... /((length(AB)^3)+(3*length(AB)*length(AB)))*23*eps));
end

smax=max(S);
xmax=(smax/s_rev)*pitch;
n = length(S);
A_fl = (pi*D*xmax);       % Fluid heating area
A_pre = (pi*D*L_pre);    % Unused channel area
L = xmax+L_pre;            % Total heater/channel length
P = flux*(A_fl+A_pre);    % Total heater power
Pmin = leads*({mdot*cp*(Tsat-Tfo)}... % Minimum Power
            +{mdot*h_fg}+{mdot*cp_v*(Tend-Tsat)});

% Print out power and length to screen
fprintf(1,'Length is %1.3f cm \n', (xmax + L_pre)*100)
fprintf(1,'Power is %1.3f kW \n',P/1000)
fprintf(1,'Minimum power is %1.3f kW \n',Pmin/1000)
fprintf(1,'Thermal efficiency is %1.3f percent \n',100*Pmin/P)
fprintf(1,'Total pressure drop is %1.3f kPa \n',dP_tot/1000)

if Plot_Toggle == 1
    %-----PLOTS/SECONDARY CALCULATIONS-------------------------------------
    X=X+L_pre;
xstart = X(1)*100;     xend = (xmax+L_pre)*100;

    if Temp_Toggle == 1
        figure(1)
        plot(X*100,fluid_T,'k',X*100,Tw_o,'b',X*100,Tw_i,'r','linewdth',2)
        legend('Fluid','Cathode O.D.','Shneck I.D.','Location','SouthEast')
        title('Temperature History')
        xlabel('Axial Position, cm')
        ylabel('Temperature, K')
        axis([0 xend 0.95*min(fluid_T) 1.05*Tw_i(n)])
        grid on

        figure(2)
        plot(X*100,fluid_T,'k',X*100,Tw,'b',X*100,Tw_ct,'r',X*100,...
            Tw_cb,'g','linewdth',2)
        legend('Fluid','Side Wall','Top Wall','Bottom Wall','Location',...
            'SouthEast')
        title('Temperature History')

end
xlabel('Axial Position, cm')
ylabel('Temperature, C')
axis([0 xend 0.95*min(fluid_T) 1.05*Tw_i(n)])
grid on
end
if Flux_Toggle == 1
  % Calculating the fluxes for for inputs into PETM and CFD models
  % ---Pre-channel region
  k = 116; % Avg value for t.c. in pre-chan.
  flux_g1 = (Tw1-Twct1)/((.5*a/k)+Rc);
  flux_o1 = 0.25*SB.*((Tw1^4)-(T_an^4));
  flux_o1b = (Twct1-Tw1).*k./t2;
  flux_fs1 = (Tw1-fluid_T(1))*h1;
  flux_fb1 = (Twcb1-fluid_T(1))*h1;
  flux_ft1 = (Twct1-fluid_T(1))*h1;

  % ---Channel region
  k = 107; % Avg value for t.c. in channel
  flux_g1 = (Tw1-Twct1)/((.5*a/k)+Rc);
  flux_fs2 = (Tw-fluid_T).*H;
  flux_fb2 = (Tw_cb-fluid_T).*H;
  flux_ft2 = (Tw_ct-fluid_T).*H;
  Q_OUT_flange = (Tflange-Tb)/Rc;

  figure(3)
  plot(X*100,dq_o,'r',X*100,dq_h,'b',X*100,dq_i,'m','linewidth',2)
  legend('q_o','q_f','q_i','Location','SouthEast')
  title('Power Profile Throughout Fluid Region')
  xlabel('Vap. Length from Start of Fluid Region (X-direction), cm')
  ylabel('Power, W')
  axis([0 xend 0 1.02*dq_i])
  grid on

  figure(4)
  plot(X*100,flux_o2/1000,'r',X*100,flux_fs2/1000,'b',X*100,...
       flux_fb2/1000,'m',X*100,flux_ft2/1000,'g','linewidth',2)
  legend('q"_{OUT}','q"_{FV}','q"_{FH_B}','q"_{FH_T}',...'
        'Location','SouthEast')
  title('Flux History')
  xlabel('Vaporizer Length, cm')
  ylabel('Heat Flux, kW/m^2')
  grid on
end
if Boiling_Toggle == 1
  figure(5)
  plot(X*100,Q,'b',X*100,AV,'r','linewidth',2)
  legend('Quality','Void Fraction','Location','SouthEast')
  xlabel('Axial Position, cm')
  ylabel('Quality')
  axis([0 xend 0 1.02*max(AV)])
  grid on

  figure(6)
  loglog(Bx,By,'r','linewidth',3)
ylabel('Gg/lambda')
xlabel('Gl*psi')
title('Baker Map for Flow Regimes')
grid on

figure(7)
plot(bx*100,Eo,'linewidt',2)
legend('Eotvos Number << 1 = Microchannel','Location','NorthEast')
xlabel('Axial Position, cm')
ylabel('Eotvos Number')
grid on

i=0; mul = 50;
for i = 1:floor(length(Vg)/mul)
    ip = i*mul;
    VL(i) = Vl(ip);
    VG(i) = Vg(ip);
end

figure(8)
loglog(VG,VL,'bo','linewidt',1)
ylabel('V_L (m/s)')
xlabel('V_G (m/s)')
title('Cubaud-Ho Map for Flow Regimes')
grid on

end

if Flange_Toggle == 1
    figure(9)
    plot(fl_location*100,fl_history(:,1),'r',fl_location*100,...
         fl_history(:,2),'b',fl_location*100,fl_history(:,3),'g',...
         fl_location*100,fl_history(:,4),'m','linewidt',2)
    legend('Flange ID','Flange, lower 1/3','Flange, upper 1/3',...
           'Flange O.D.','Location','SouthOutside','Orientation',...
           'horizontal')
xlabel('X location, cm')
ylabel('Flange Temperatures')
grid on

    figure(10)
    plot(fl_location*100,oa,'r',fl_location*100,ob,'b',...
         fl_location*100,oc,'g',fl_location*100,od,'m',...
         fl_location*100,oe,'k',fl_location*100,ot,'y','linewidt',2)
    legend('a','b','c','d','e','t','Location','SouthOutside',...
           'Orientation','horizontal')
xlabel('X location, cm')
ylabel('Heat Flow')
grid on
end

if Flow_Props == 1
    figure(11)
    plot(X*100,velocity,'b','linewidt',2)
    legend('Velocity','Location','SouthEast')
xlabel('Axial Position, cm')
ylabel('Velocity, m/s')
grid on

end
if Error_Toggle == 1
    figure(12)
    plot([1:1:length(Tw)],error1,'b',[1:1:length(Tw)],error2,'r',...  
          [1:1:length(Tw)],error3,'m','linewd',2)
    title('Upper Bound on Errors')
end

if Pressure_Toggle == 1
    n = 0; n = length(dP_array);
    pressure = zeros(n,1); P_op = 0; % gage pressure
    for i=1:n
        p_ind = n - (i-1);
        P_op = P_op + dP_array(p_ind);
        pressure(p_ind) = P_op;
    end
    figure(13)
    plot(X*100,pressure./1000,'r','linewd',2)
    legend('Pressure','Location','SouthEast')
    xlabel('Axial Position, cm')
    ylabel('Gage Pressure, Pa')
    grid on
end

end

Fluent UDF C# Script

#include "udf.h"
#include "sg_mphase.h"
define T_SAT 1336
#define LAT_HT 196.e5
#define C 1

DEFINE_SOURCE(liq_src_lit, cell, pri_th, dS, eqn)
{
    Thread *mix_th, *sec_th;
    real m_dot_l;
    mix_th = THREAD_SUPER_THREAD(pri_th);
    sec_th = THREAD_SUB_THREAD(mix_th, 1);
    if(C_T(cell, mix_th)>=T_SAT){
        m_dot_l = -C*C_VOF(cell, pri_th)*C_R(cell, pri_th)*
                 fabs(C_T(cell, pri_th) - T_SAT)/T_SAT;
        dS[eqn] = -C*C_R(cell, pri_th)*
                 fabs(C_T(cell, pri_th) - T_SAT)/T_SAT;
    } else {
        m_dot_l = C*C_VOF(cell, sec_th)*C_R(cell, sec_th)*
                 fabs(T_SAT-C_T(cell,mix_th))/T_SAT;
    }
}
dS[eqn] = 0.;

return m_dot_1;

DEFINE_SOURCE(vap_src_lit, cell, sec_th, dS, eqn)
{
    Thread * mix_th, *pri_th;
    real m_dot_v;

    mix_th = THREAD_SUPER_THREAD(sec_th);
    pri_th = THREAD_SUB_THREAD(mix_th, 0);

    if(C_T(cell, mix_th)>=T_SAT){
        m_dot_v = C*C_VOF(cell, pri_th)*C_R(cell, pri_th)*
                   fabs(C_T(cell, mix_th) - T_SAT)/T_SAT;
        dS[eqn] = 0.;
    }
    else {
        m_dot_v = -C*C_VOF(cell, sec_th)*C_R(cell, sec_th)*
                   fabs(T_SAT-C_T(cell,mix_th))/T_SAT;
        dS[eqn] = -C*C_R(cell, sec_th)*
                   fabs(C_T(cell, sec_th) - T_SAT)/T_SAT;
    }

    return m_dot_v;
}

DEFINE_SOURCE(enrg_src_lit, cell, mix_th, dS, eqn)
{
    Thread *pri_th, *sec_th;
    real m_dot;

    pri_th = THREAD_SUB_THREAD(mix_th, 0);
    sec_th = THREAD_SUB_THREAD(mix_th, 1);

    if(C_T(cell, mix_th)>=T_SAT){
        m_dot = -C*C_VOF(cell, pri_th)*C_R(cell, pri_th)*
                fabs(C_T(cell, pri_th) - T_SAT)/T_SAT;
        dS[eqn] = -C*C_VOF(cell, pri_th)*C_R(cell, pri_th)/T_SAT;
    }
    else {
        m_dot = C*C_VOF(cell, sec_th)*C_R(cell, sec_th)*
                fabs(T_SAT-C_T(cell,mix_th))/T_SAT;
        dS[eqn] = -C*C_VOF(cell, sec_th)*C_R(cell, sec_th)/T_SAT;
    }

    return LAT_HT*m_dot;
}
REFERENCES


Space Technology and Applications International Forum (STAIF), Feb. 11-14, 2001, Albuquerque, NM.


