2020

The effect of Prandtl number on heat transfer in the wake of a highly confined bubble in a large-aspect-ratio minichannel

Joseah K. Amai

Follow this and additional works at: https://louis.uah.edu/uah-theses

Recommended Citation
https://louis.uah.edu/uah-theses/334

This Thesis is brought to you for free and open access by the UAH Electronic Theses and Dissertations at LOUIS. It has been accepted for inclusion in Theses by an authorized administrator of LOUIS.
THE EFFECT OF PRANDTL NUMBER ON HEAT TRANSFER IN THE
WAKE OF A HIGHLY CONFINED BUBBLE IN A
LARGE-ASPECT-RATIO MINICHANNEL

by

JOSEAH K. AMAI

A THESIS

Submitted in partial fulfilment of the requirements
for the degree of Master of Science
in
The Department of Mechanical and Aerospace Engineering
to
The School of Graduate Studies
of
The University of Alabama in Huntsville

HUNTSVILLE, ALABAMA
2020
In presenting this thesis in partial fulfillment of the requirements for a master’s degree from The University of Alabama in Huntsville, I agree that the Library of this University shall make it freely available for inspection. I further agree that permission for extensive copying for scholarly purposes may be granted by my advisor or, in his absence, by the Chair of the Department or the Dean of the School of Graduate Studies. It is also understood that due recognition shall be given to me and to The University of Alabama in Huntsville in any scholarly use which may be made of any material in this thesis.

[Student Signature]
04/08/2020
(Student Signature) (Date)
THESIS APPROVAL FORM

Submitted by Joseah Amai in partial fulfillment of the requirements for the degree of Master of Science in Engineering with an option in Mechanical Engineering and accepted on behalf of the Faculty of the School of Graduate Studies by the thesis committee.

We, the undersigned members of the Graduate Faculty of The University of Alabama in Huntsville, certify that we have advised and/or supervised the candidate on the work described in this thesis. We further certify that we have reviewed the thesis manuscript and approve it in partial fulfillment of the requirements for the degree of Master of Science in Engineering with an option in Mechanical Engineering.

Dr. D. Keith Hollingsworth 04/08/2020 Committee Chair

Dr. Sarma Rani

Dr. Guangsheng Zhang 04/08/2020

Dr. D. Keith Hollingsworth 04/08/2020 Department Chair

Dr. Shankar Mahalingam 04/08/2020 College Dean

Dr. David Berkowitz 04/08/2020 Graduate Dean
ABSTRACT

The School of Graduate Studies
The University of Alabama in Huntsville

Degree Master of Science in Engineering College/Dept. Engineering/Mechanical and Aerospace Engineering

Name of Candidate Joseah K. Amai

Title The Effect of Prandtl Number on Heat Transfer in the Wake of a Highly Confined Bubble in a Large-Aspect-Ratio Minichannel

A numerical study of the effect of Prandtl number on the heat transfer in the wake of a highly confined bubble in a large-aspect-ratio minichannel was performed. The channel height was $1.25 \text{ mm}$, and the precursor liquid flow was laminar at channel Reynolds numbers of 169 and 338. The upper channel was modeled as an electrically heated metal foil, and the lower wall was adiabatic. The simulation was performed using ANSYS Fluent, with the volume-of-fluid method was used to determine the phase boundary. A Lagrangian reference frame translating at the average bubble velocity was used to simulate a channel of arbitrary length. The resulting numerical domain was $30 \text{ mm}$ in the streamwise direction and $20 \text{ mm}$ in the spanwise direction.

This study adds a data set at Prandtl number of 6 to the results obtained by a previous study in which the numerical model as built, qualified, and executed for Prandtl numbers of 12 and 1. An investigation of the near-field of the bubble rear wake was performed using the expanded data set. A new Nusselt number correlation for the near-field heat transfer was proposed. A graphical examination of the velocity components and fluid temperature in the near-field region was performed to document evidence of dynamical structures near the bubble. The velocity components at specific locations supported the presence of dynamical structures which may operate to enhance the heat transfer rate. The material properties of the heated upper channel wall were altered to examine the effect of the thermal boundary condition imposed by the foil on the heat transfer in the bubble wake. Both the specific heat and thermal conductivity of the foil were reduced by a factor of three. The evolution of the wall-to-bulk temperature difference and the local wall heat flux were substantially changed. The resulting Nusselt number was largely unchanged at Prandtl number of 12, but was increased at Prandtl number of 6.

Abstract Approval: Committee Chair
Department Chair
Graduate Dean

David Berkowitz
DEDICATION

To my Parents, Uncle and Aunt.
ACKNOWLEDGEMENTS

First, I would like to express my deepest gratitude to my committee chair, Dr. Keith Hollingsworth, for his invaluable guidance and instruction during the course of this research. His vast experience in the field of heat transfer ensured that he could seamlessly relate high level concepts in the subject to everyday learning, making it more adventurous and exciting. His personal concern, thoughtful care and dedicated attention ensured a healthy balance between every-day life and research. Secondly, I would like to thank my committee members, Dr. Sama Rani and Dr. Guangsheng Zhang for their acceptance to be a part of this study. Their insightful ideas made it possible to produce the best possible version of this work. I would also like to thank Dr. John Willard for his introduction and induction to this research, and for his eagerness to help whenever he was needed. A word of thank to Mr. Euan Holton for always ensuring that the working station was in excellent working condition to run the simulations. Lastly, I would like to thank to my family and friends for their immense support and encouragement during the course of this research. To my parents, uncle and aunt, without whom this would never have been possible, I am forever grateful for all you did. God bless you!
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Chapter 1 Introduction</th>
<th>.............................................................</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Motivation for the Present Study</td>
<td>..................................................................................</td>
<td>3</td>
</tr>
<tr>
<td>Chapter 2 Literature Review</td>
<td>..................................................................................</td>
<td>6</td>
</tr>
<tr>
<td>Sliding Bubble Dynamics and Heat Transfer</td>
<td>........................................................................</td>
<td>7</td>
</tr>
<tr>
<td>Heat Transfer Models of Sliding Bubbles</td>
<td>........................................................................</td>
<td>12</td>
</tr>
<tr>
<td>Numerical Simulation of Sliding Bubble Flows</td>
<td>.................................................................</td>
<td>16</td>
</tr>
<tr>
<td>Flows due to Bouyancy</td>
<td>.................................................................................</td>
<td>16</td>
</tr>
<tr>
<td>Pool Boiling</td>
<td>....................................................................................</td>
<td>17</td>
</tr>
<tr>
<td>Horizontal Flows</td>
<td>....................................................................................</td>
<td>18</td>
</tr>
<tr>
<td>Numerical Methods</td>
<td>....................................................................................</td>
<td>21</td>
</tr>
<tr>
<td>Interface Tracking Methods</td>
<td>...........................................................................</td>
<td>22</td>
</tr>
<tr>
<td>Surface Tension Modelling</td>
<td>...........................................................................</td>
<td>25</td>
</tr>
<tr>
<td>Pressure-Velocity Coupling</td>
<td>...........................................................................</td>
<td>25</td>
</tr>
<tr>
<td>Summary and Directions for the Present Work</td>
<td>....................................................................</td>
<td>26</td>
</tr>
<tr>
<td>Chapter 3 Methodology</td>
<td>....................................................................................</td>
<td>27</td>
</tr>
<tr>
<td>Numerical Methods</td>
<td>....................................................................................</td>
<td>29</td>
</tr>
<tr>
<td>Fundamental Equations</td>
<td>..................................................................................</td>
<td>29</td>
</tr>
<tr>
<td>Model Implementation</td>
<td>..................................................................................</td>
<td>30</td>
</tr>
</tbody>
</table>

vii
Solver Setup..............................................................................................................31
Computational Domain ..........................................................................................33
Computational and Boundary Conditions .............................................................34
Solution Initialization ............................................................................................39
Parameter Space .....................................................................................................39

Chapter 4 Results and Discussion .......................................................................41
Validation ..................................................................................................................42
  Single Phase Test Case .........................................................................................42
The Case of \( Pr = 6 \) ..............................................................................................45
  Reduction of Data ..................................................................................................46
  Definition of Terms ................................................................................................47
  Thermal Response ..................................................................................................49
The Near Field Region ............................................................................................53
  Velocity and Temperature Profiles of the Near-Field ........................................58
Case of \( D_b/H = 3.3, \ Re = 169, \ and \ Pr = 6 \) ......................................................60
Case of \( D_b/H = 3.3, \ Re = 169, \ and \ Pr = 12 \) ......................................................66
Near-field Nusselt Number Correlation .................................................................71
Effect of changing the thermal boundary condition on the heated wall .............75

Chapter 5 Summary and Conclusions ..................................................................82
Conclusions ..............................................................................................................83
List of Figures

**Figure 1.1.** Image of a highly confined bubbly flow in a channel with spacing of 1.27 mm and width of 23 mm from Ozer (2010). .......................................................... 3

**Figure 2.1.** Bubble rendering used by Natesh (2012). The volume of fraction of cells is represented by the color code, with green corresponding to 0.5, which is the interface location. ........................................................................................................... 21

**Figure 3.1.** Two-dimensional VOF Method for phase interface tracking (Willard 2017). .................................................................................................................................................. 28

**Figure 3.2.** Implementation of the Lagrangian framework in the model (Willard, 2017). .................................................................................................................................................. 31

**Figure 3.3.** Illustration of the computational domain (not to scale). (Willard 2017). ..... 33

**Figure 4.1.** Temperature distribution of the heated wall and the bulk fluid along the channel for low Re, $D_b/H = 2.0$ $Pr = 12$ at $t^* = 1$.................................................................................................................................. 43

**Figure 4.2.** Temperature distribution of the heated wall and the bulk fluid along the channel for low Re, $D_b/H = 2.0$ $Pr = 6$ at $t^* = 1$.................................................................................................................................. 44

**Figure 4.3.** Heat flux and $Nu$ distribution along the heated wall center-line for low Re, $D_b/H = 2.0$ $Pr = 12$ at $t^* = 1$.................................................................................................................................. 44

**Figure 4.4.** Heat flux and $Nu$ distribution along the heated wall center-line for low Re, $D_b/H = 2.0$ $Pr = 6$ at $t^* = 1$.................................................................................................................................. 45

**Figure 4.5.** Convection coefficient versus distance behind bubble for all three Pr........ 50

**Figure 4.6.** $Nu$ versus $x*$ plotted with log-log axes for the three Pr. The precursor value is marked with a horizontal line. .................................................................................................................. 52

**Figure 4.7.** $Nu$ versus $x_b$ in the near field region for the three Pr cases. .......................53
Figure 4.8. $Nu$ versus $x^*$ plotted with log-log axes for the cases $Pr = 6$ and $Pr = 12$ with low $Re$. ...............................................................

Figure 4.9. $Nu$ versus $x^*$ plotted with log-log axes for the measured and predicted values in the channel.................................................................

Figure 4.10. Temperature in the fluid phases at planes located at for $D_b/H = 3.3$ from Willard (2017). The bubble surface is also shown in gray, the diameter of the bubble at the location of the section plane as well as the contact line where the bubble meets the plate can be seen. Overlaid on the temperature field are representations of prevailing velocity vectors at that location. The $R$, $G$, and $B$ locations are indicated.................................

Figure 4.11. Velocity and Temperature Profiles through the depth of the channel at $x_b = 1$ mm. (a), (b) and (c) are the $u$-, $v$- and $w$-velocity components respectively, while (d) is the temperature profile. Case of Case of $D_b/H = 3.3$, $Re = 169$, and $Pr = 6$. .................

Figure 4.12. Velocity and Temperature Profiles through the depth of the channel at $x_b = 2$ mm. (a), (b) and (c) are the $u$-, $v$- and $w$-components of velocity, respectively, (d) is the temperature profile. Case of Case of $D_b/H = 3.3$, $Re = 169$, and $Pr = 6$. .........................

Figure 4.13. Velocity and Temperature Profiles through the depth of the channel at $x_b = 5$ mm. (a), (b) and (c) are the $u$-, $v$- and $w$-velocity, respectively, (d) is the temperature profile. ..........................................................

Figure 4.14. Velocity and Temperature Profiles through the depth of the channel at $x_b = 1$ mm. (a), (b) and (c) are the $u$-, $v$- and $w$-components of velocity, respectively, (d) is the temperature profile. Case of Case of $D_b/H = 3.3$, $Re = 169$, and $Pr = 12$. .................
Figure 4.15. Velocity and Temperature Profiles through the depth of the channel at $x_b = 2$ mm. (a), (b) and (c) are the $u$-, $v$- and $w$-velocity components respectively, (d) is the temperature profile. Case of Case of $D_b/H = 3.3$, $Re = 169$, and $Pr = 12$. ..................69

Figure 4.16. Velocity and Temperature Profiles through the depth of the channel at $x_b = 5$ mm. (a), (b) and (c) are the $u$-, $v$- and $w$-velocity components respectively, (d) is the temperature profile. Case of Case of $D_b/H = 3.3$, $Re = 169$, and $Pr = 12$. ..................70

Figure 4.17 Convection coefficient vs $x_b/D_b$ behind bubble for all three $Pr$. .................71

Figure 4.18. $N u_x$ vs $Re_{xb}$ in the near field region .......................................................73

Figure 4.19. $N u_x$ vs $Re_{xb}$ for Equations 4.11 and 4.13. .............................................75

Figure 4.20. Heated foil temperature vs $x$ for $Pr = 12$ $Re = 169$ for base and altered foil properties with no bubble present..........................................................79

Figure 4.21. $Nu$ vs $x$ for $Pr = 12$ $Re = 169$ for base and altered foil properties with no bubble present..........................................................80

Figure 4.22. $T_w - T_b$ vs $x_b$ for base and altered foil material properties for $D_b/H = 2.0$, $Re = 169$, and $k$ and $C_p$ reduced by a factor of three. ..........................80

Figure 4.23. Heat flux vs $x_b$ for base and altered foil material properties for $D_b/H = 2.0$, $Re = 169$, and $k$ and $C_p$ reduced by a factor of three. ..........................81

Figure 4.24. $Nu$ vs $x_b$ for base and altered foil material properties for $D_b/H = 2.0$, $Re = 169$, and $k$ and $C_p$ reduced by a factor of three.................................81
List of Tables

Table 3.1. Summary of the simulated cases for Pr study.............................40
Table 4.1. Summary of the Near Field Cases Studied.................................60
Table 4.2. Deviation from Correlation in Figure 1.17.......................73
Table 4.3. Runs used to Investigate Boundary Condition Sensitivity........77
Table A.1. Thermophysical Properties of Materials.................................86
Table B.1. Table of Thermal Boundary Conditions taken from Willard (2017) 87
Table B.2. Table of Flow Boundary Conditions taken from Willard (2017) ...90
### Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_p$</td>
<td>Specific heat</td>
<td>$J/(kg\ {}^\circ C)$</td>
</tr>
<tr>
<td>$D_h$</td>
<td>Channel hydraulic diameter, $2HW/(H + W)$</td>
<td>$mm$</td>
</tr>
<tr>
<td>$D_b$</td>
<td>Bubble diameter</td>
<td>$mm$</td>
</tr>
<tr>
<td>$dt$</td>
<td>Time step Size</td>
<td>$sec$</td>
</tr>
<tr>
<td>$h$</td>
<td>heat transfer coefficient</td>
<td>$W/(m^2-K)$</td>
</tr>
<tr>
<td>$h_x$</td>
<td>spanwise-average, time-averaged heat transfer coefficient</td>
<td>$W/(m^2-K)$</td>
</tr>
<tr>
<td>$H$</td>
<td>Channel height</td>
<td>$mm$</td>
</tr>
<tr>
<td>$k$</td>
<td>Thermal conductivity</td>
<td>$W/(m-K)$</td>
</tr>
<tr>
<td>$k_f$</td>
<td>Thermal conductivity of liquid</td>
<td>$W/(m-K)$</td>
</tr>
<tr>
<td>$k_p$</td>
<td>Thermal conductivity of plate</td>
<td>$W/(m-K)$</td>
</tr>
<tr>
<td>$L$</td>
<td>Channel length</td>
<td>$mm$</td>
</tr>
</tbody>
</table>
\( Nu \)  
Nusselt number based on channel hydraulic diameter, \( h_c \, D_H / k_f \)

\( Nu_c \)  
Nusselt number in the bubble wake based on distance behind the bubble, \( h_c \, x_b / k_f \)

\( Nu_0 \)  
Theoretical fully developed single-phase channel Nusselt number

\( Pe \)  
Peclet Number \( \equiv Re \, Pr \)

\( Pr \)  
Prandtl number \( \equiv v/\alpha \)

\( Re \)  
Channel Reynolds number, \( \equiv u \, D_h / \nu \)

\( Re_{bubble} \)  
Bubble Reynolds number \( \equiv u_A \, D_h / \nu \)

\( Re_{xb} \)  
Reynolds number based on distance behind the bubble \( \equiv u_A \, x_b / \nu \)

\( \dot{Q} \)  
Volumetric generation in the plate \( W/m^3 \)

\( q_{w} \)  
Wall heat flux \( W/m^2 \)

\( t \)  
Simulation time \( sec \)
\begin{align*}
T & \quad \text{Local liquid temperature} \quad K \\
T_{ini} & \quad \text{Initial wall temperature} \quad K \\
T_b & \quad \text{Local bulk mean temperature of liquid} \quad K \\
T_w & \quad \text{Local wall temperature} \quad K \\
u & \quad \text{Local Liquid Velocity} \quad \text{mm/sec} \\
u_p & \quad \text{Plate velocity (Lagrangian frame)} \quad \text{mm/sec} \\
u_b & \quad \text{Bubble velocity (laboratory frame)} \quad \text{mm/sec} \\
u_l & \quad \text{Average liquid velocity} \quad \text{mm/sec} \\
u_c & \quad \text{Centerline liquid velocity} \quad \text{mm/sec} \\
u_A & \quad \text{Relative centerline velocity} \equiv u_c - u_b \quad \text{mm/sec} \\
u^* & \quad \text{Dimensionless streamwise velocity} \equiv \frac{u(z)-u_b}{u_c-u_p} \quad \text{dimensionless} \\
W & \quad \text{Channel width} \quad \text{mm} \\
x & \quad \text{Coordinate in streamwise direction} \quad \text{mm} \\
\end{align*}
\( x_b \) Streamwise distance behind bubble \( \text{mm} \)

\( x^* \) Dimensionless position behind bubble \( \text{dimensionless} \)

\[
\equiv \frac{x_b}{D_h} \left( \frac{H_b}{H} \right)^m \left( \frac{Re_{bubble}}{Pr} \right)^n
\]

\( y \) Coordinate in spanwise direction \( \text{mm} \)

\( z \) Coordinate in the channel height \( \text{mm} \)

direction

**Greek**

\( \alpha \) Molecular thermal Diffusivity \( m/s^2 \)

\( \delta_p \) Plate Thickness \( \mu m \)

\( \theta \) Dimensionless Temperature \( \equiv \frac{T(x)-T_b}{T_{ini}-T_b} \)

\( \mu \) Absolute Viscosity \( Pa \cdot sec \)

\( \nu \) Kinematic Viscosity \( m/s^2 \)

\( \rho \) Density \( kg/m^3 \)

\( \rho_f \) Liquid Density \( kg/m^3 \)
\begin{align*}
\rho_g & \quad \text{Gas Density} & \quad \text{kg/m}^3 \\
\rho_p & \quad \text{Plate Density} & \quad \text{kg/m}^3
\end{align*}
Chapter 1

Introduction

The study of two-phase flow in narrow channels is an important subject in the field of heat transfer. An understanding of the techniques involved provides an effective approach to the cooling of high-heat flux systems. Practical examples of these processes include the removal of the internally generated energy from densely-packed electronic packages, components in high-power laser weapon systems, heat exchangers in nuclear fission reactors, among others. The ability to cool these systems using two-phase flows with a highly confined vapor phase provides an interesting design approach because the bubble-driven mixing mechanism of these flows yields enhanced convection between the near wall and the bulk fluid, with an accompanying energy transfer through evaporation. In this type of flow, caution has to be exercised to ensure that the mixing-cup temperature, or bulk fluid temperature, does not approach, or equal, the saturation temperature of the liquid. When this happens, there is a rapid bubble growth which can result in ‘dry-out’ regions on the heated foil. These regions would essentially decrease the heat transfer rate which would result in a rapid increase in the local surface temperature. This problem is mitigated by the use of subcooled bubble flow where the mixing-cup temperature is maintained below the saturation temperature. The overall effect of subcooled bubbly flows in a confined geometry where the upper wall is heated, and the lower wall is not, is the movement of
low-temperature fluid from lower portions of the channel towards the heated plate. This mixing is caused by the flow patterns that form due to bubble passage. This fluid movement is responsible for enhanced temperature gradient at the wall, which in turn yields an enhanced average heat transfer coefficient.

Two-phase flows are described and classified based on the size of the containing geometry together with the nature of the driving force. A geometrical classification based on the hydraulic diameter divides the flow types into three major categories; conventional (large ducts), minichannels, and microchannels. A significant number of studies have been done for two-phase flows in microchannels (hydraulic diameters under 500 μm). Fewer studies have been done in minichannels (hydraulic diameters near 1 mm); however, this range has been the focus of much of the work done by this research group. The two primary driving forces in two-phase flows in minichannels are pressure, where liquid velocity drives bubble motion in horizontal channels, and buoyancy, where gravity is responsible for an upward bubble motion. Minichannels (those with channel spacing near 1 mm and a cross-channel width an order of magnitude larger) offer an appealing range for technological applications of two-phase cooling. These channels provide adequate confinement of the bubble which results in a higher heat transfer enhancement during bubble motion along the heated wall. They also exhibit a desirable span of two-phase taxonomy which yields a variety of flow fields. Figure 1.1 shows an example of naturally nucleated bubbles in a channel with spacing of 1.27 mm from the work of Ozer (2010). By comparison, the smaller microchannels (with round or square cross sections) tend to exhibit instability in flow and a difficulty in maintaining bubbly flow regimes.
Pressure-driven flows in minichannels are of great research interest because of the flexibility and ease of controlling the flow speeds as compared to a reliance on gravity. This blend of proficiency of control and elevated heat transfer makes pressure-driven bubbly flows in minichannels of special engineering significance. This engineering interest has motivated studies in this type of flow, which has largely taken experimental approaches and focused more on the bubble fields and the resulting heat transfer coefficient produced by the fields. A recent study by Willard (2017) explored and demonstrated computational simulations of the heat transfer enhancement produced by a single confined bubble in a high-aspect-ratio minichannel, and detailed the flow structures responsible for this enhancement. The work presented here seeks to widen the scope of those simulations by, expanding the Prandtl number range of the work, taking a keener look into the near field of the bubble flow regime, and varying the material properties of the heated surface to explore the effect of the thermal boundary condition of the simulation results.

Motivation for the Present Study

The study of bubbly flows and their effect on heat transfer has been a subject of great interest over several decades. These studies document the substantial enhancement in heat
transfer on the surfaces along which the bubbles slide. In the recent past, there has been a more deliberate effort to investigate and document the nature of the mechanisms responsible for this elevated heat transfer. Two particular studies from this research group that immediately precede the current work exemplify this effort. One, the experimental work by Albahloul et al. (2014, 2015) documented the enhancement in heat transfer in horizontal minichannels as a result of injecting air bubbles into the channel. He determined that these injected bubbles were as effective as the naturally nucleated vapor bubbles in increasing the average heat transfer coefficient in the channel. The key conclusion from this experimental investigation was that the localized mixing of the liquid due to bubble passage was a greater contributor to the heat transfer increase than either vapor production due to nucleation and bubble growth. This conclusion was a key foundation to the numerical work by Willard et al. (2017, 2018), which sought to investigate the precise nature of the mixing. He performed a numerical simulation of the channel used by Albahloul using Ansys Fluent™. Willard’s work simulated a single highly confined bubble in a laminar flow through a rectangular minichannel with a heated upper wall. His findings demonstrated the presence of a complex set of fluid structures in the near field which produce the fluid-mixing responsible for the enhanced heat transfer in the bubble wake. The key structures observed were a pair of twin vortices that moved the liquid from the lower channel regions towards the upper heated plate. Also found were a pair of lateral jets on either side of the bubble that served to provide a secondary enhancement effect.

Motivated by these recent findings, this thesis seeks to further the scope of the work by Willard while providing a first look into modeling the near-field heat transfer. This study
uses the same numerical model in Ansys Fluent™ as used by Willard (2017). The present work seeks to expand Willard’s study to include the following elements:

- A set of simulations and analysis for a liquid $Pr = 6$ for both low and high $Re$. This data set expands the work of Willard who produced simulations of $Pr = 12$ and $Pr = 1$.

- An improved characterization of the near-field region by the introduction of a new near-field Nusselt number correlation and by an analysis of the local behavior of velocity and fluid temperature.

- An investigation of the effect of changing the thermal boundary condition on the heated channel wall so as to simulate a wall with the capacity to more rapidly change its temperature in response to the bubble passage. This will be done by changing the specific heat and conductivity of the heated foil. The net effect will be to simulate a “thermally thinner” surface without changing the actual surface thickness.
Chapter 2

Literature Review

The study of bubbly flows in millimeter-scale channels dates to the late 1960’s. During that time, buoyancy-driven bubbly flows have received most of the focus, with pressure-driven flows receiving comparatively less attention. In addition, numerical simulations of pressure-driven flows in millimeter-scale channels, did not exist until recently. However, the findings of the preceding experimental work provide the background and direction for numerical work which can deepen the understanding of the heat transfer mechanisms involved.

Numerical simulations of multi-phase flows in relatively unconfined geometries are now well-established. The simulations have been useful in investigating the physics of these flows, as well as establishing models for tracking the vapor-liquid interfaces around the bubbles. The best method for the simulation of the phase interface has been a contentious issue. A number of methods have been developed, and each has varying advantages over the other.

From the foregoing introduction, this section presents a concise review of the relevant experimental and numerical work dealing with the heat transfer enhancement due to sliding bubbles in minichannels. The motivation for the present work is presented at the end of this section.
Sliding Bubble Dynamics and Heat Transfer

The foundational concepts of two-phase flows in millimeter and micro-scale channels have been comprehensively reviewed by Kandlikar (2002). He proposed a categorization of channels based on their hydraulic diameter, $D_h$, into three classes; conventional channels with $D_h \geq 3 \text{ mm}$, minichannels having $200 \mu m \leq D_h < 3 \text{ mm}$, and microchannels having $10 \mu m \leq D_h < 200 \mu m$. In addition to this classification, two-phase channel flows are differentiated based on the local liquid temperature. The flow is saturated when the mixing-cup temperature is equal to the saturation temperature, and when that temperature is below saturation, the flow is subcooled.

The earliest study of the enhancement mechanisms produced by bubbly flows that produce bubbles sliding along or near heated surfaces was first published by Ishibashi and Nishikawa (1969). Their undertakings were an attempt to identify and predict the principal mechanism of the enhanced heat transfer for a bubbly flow in a vertical cylindrical annulus with a 3 mm gap. They observed that the motion of the vapor bubbles on the heated surface could be associated with the increase in heat transfer coefficient. A notable conclusion from this work outlined that 70% of the energy transmitted from the surface was taken up by the liquid phase through convection, and not through the process of phase change. This finding challenged the popular belief that the increased heat transfer in two-phase flows is directly related to vapor production. The work proposed a steady-state Nusselt Number ($Nu$) correlation for the coalesced bubble region, in addition to providing a novel attempt to model the enhanced heat transfer of the confined sliding bubbles.

The multi-tube bundles by Nakashima (1978) and those by Cornwell et al. (1980) went further to highlight the relationship between the creation of bubbles in one location (lower
tubes) and their corresponding heat transfer enhancement from their sliding motion in other locations (upper tubes). These works suggested that this enhancement was caused by the evaporation of the liquid microlayer between the bubbles and the heated surface. The bubbles were produced from the nucleation sites or from the main stem approach flow in the lower tubes and they slid their way to the upper tubes. The heat transfer coefficient in the upper tubes was observed to be four times higher than those of the lower ones. The microlayer evaporation phenomenon was also observed as the cause for the enhancement by Cornwell and Schuller (1982). An experimental work by Cooper and Lloyd (1969) demonstrated an actual existence of the liquid microlayer, which was also confirmed by Koffman and Plesset (1983). However, the latter’s conclusion did not fully credit evaporation as the primary cause of the enhanced heat transfer.

Significant contributions to the field were made by the Monde group (1981-1995). Kusuda and Monde (1981) developed an experimental and theoretical study of heat transfer as a result of the motion of confined sliding bubbles in narrow vertical spaces of 1 – 5 mm. The setup had one side of the channel heated while the others were insulated. Bubbles were introduced to slide on the heated surface at a frequency of 0.02 – 3 Hz, for a bubble length of 30 mm. The temperature changes on the walls were recorded by use of thermocouples. They observed that there was a significant temperature drop behind a sliding bubble and that the heat transfer was a function of the bubble injection period and the thickness of the boundary layer. More work on the relationship between bubble passage frequency and heat transfer was published by Monde et al. (1988). Their conclusions indicated that there was an enhanced heat transfer with the increase in bubble injection frequency. Monde and Mitsutake (1989) carried out another study using smaller bubbles and utilizing the
geometry developed previously. The findings of this work laid bare the shortcomings of
the previous model because the results were about three times higher than the experimental
results.

Refrigerants were used as a working fluid in boiling experiments conducted by Houston
and Cornwell (1996). Vertical columns with horizontal tubes were used to for this study,
and the effect of sliding bubbles on heat transfer enhancement was observed. The findings
of this work provided evidence to discredit evaporation as the fundamental reason for the
enhanced heat transfer. This notwithstanding, evaporation was still maintained as a
significant percentage of the energy transferred, though its portion was conclusively found
to be less than that from convective transport. Therefore, the work concluded that the
convective mechanisms from bubble motion was more responsible for the heat transfer
enhancement as compared to the evaporation in the microlayer.

The effect of channel size on the heat transfer was studied by Lie and Lin (2006). The
setup had a horizontal annular duct (gap size 1 and 2 mm) with subcooled R-134a as a
working fluid. They observed that decreasing the gap size increased the heat transfer
coefficient. These results were backed up by Soupremanien et al. (2011) where they
observed that for high heat fluxes, the heat transfer coefficient increased with increasing
channel aspect ratio.

The individual phenomenon of heat transfer enhancement behind sliding bubbles was
explored further by Thorncroft and Klausner (1999). Their work outlined single-phase
convection, phase change and bubble growth, and bubble sliding as the three constituents
of heat transfer. Turbulence as a result of sliding bubble motion together with latent heat
were touted as the causes of sliding heat transfer. These studies were fundamental in the
development of the narrative that the enhancement in heat transfer in high-aspect-ratio minichannels could be a result of sliding bubble motion. The primary concentration of a study from the current research group, by Ozer et al. (2010, 2011, 2012) was this concept of high-aspect-ratio channels in steady response to fields of highly confined sliding bubbles. In this work, liquid crystal thermography was used as a way to measure temperature on the heated channel surface. In addition, this work made an attempt to document the bubble size corresponding to the fields of bubbles. Ozer also introduced a conduction model for the heat transfer behind a bubble, motivated by the findings of Ishibashi and Nishikawa (1969). Ozer observed that the heat transfer enhancement of flow in the two-phase region was 2-5 times the convective heat transfer of the single phase.

Albahloul et al. (2014, 2015) also from the current group, sought to advance on the ideas developed by Ozer. They made an effort to investigate the effects of sliding bubbles on enhancement of heat transfer when there is no change of phase. Their work replaced the vapor bubbles with air bubbles that were injected into a subcooled fluid domain comprising of Novec-649. The choice of air bubbles eliminated the influence of phase such that the measurement of enhanced heat transfer was only attributed to the presence of the bubble. Their findings confirmed that there was a heat transfer enhancement of 3-5 times that of single-phase heat transfer, which was firmly credited to the sliding bubbles since there was minimal phase change in this highly subcooled flow. A follow-up research by Albahloul and Hollingsworth (2014) reported that there exists a strong relationship between the bubble diameter and bubble speed with respect to the speed of the liquid flow in the channel. They observed that the bubble speed is lower than the centerline liquid speed. This phenomenon is responsible for the formation of complex flow structures around the
bubble, for instance momentum and temperature wakes that extend both in the front and rear side of the bubble. Albahloul (2015) captured the first set of in situ images of sliding bubbles acquired from inside a horizontal minichannel. The images displayed the lateral edges of the bubble and they augmented the plan view images taken through a transparent lower channel wall as done by Ozer. From these two simultaneous views of the bubble, a relationship between the bubble diameter and bubble confinement (the ratio of bubble height to channel height) was developed. This new addition indicated that the bubble diameter could be adopted as a more inclusive parameter in the description of bubble morphology.

Willard (2017) sought to advance on Albahloul’s work by performing a numerical investigation of the flow details and convective mechanisms that are necessarily unseen in Albahloul’s experiments. The difficulty in isolating these mechanisms inside of the experimental domain prompted the move to computer simulations. Willard used ANSYS Fluent to perform a low-Reynolds-Number full simulation of a single highly confined bubble moving through a laminar-flow rectangular minichannel that is heated through its upper wall. A channel of an arbitrary length was simulated by use of a Lagrangian formulation where in the numerical domain followed the bubble. A simulation of this type was unprecedented in the literature of this subject at the time of Willard’s work. He observed that present in the flow field near the bubble are a set complex fluid structures that generate the mixing action in the fluid, hence the enhancement in the heat transfer in the bubble wake. In addition to these structures, a pair of lateral jets on the bubble sides were observed. A secondary enhancement of heat transfer was credited to these lateral jets. Lastly, Willard was able to identify and classify three regions of the rear wake that
exhibited an enhancement in heat transfer. The first region described active mixing produced by near-field structures. A power-law decay was exhibited by the Nusselt number in the second region, while the third region exhibited an asymptotic return to the precursor value. In his work, Willard also included a two-dimensional reduced-order numerical model of the heat transfer in the wake. The boundary conditions were applied such that near-field mixing behind the bubble was captured. The results obtained from this simple model supported the identification of the critical elements of the wake heat transfer.

**Heat Transfer Models of Sliding Bubbles**

A heat transfer model due to sliding bubbles near a heated surface was first proposed by Ishibashi and Nishikawa (1966). They observed that the dominant mechanism for the enhanced heat transfer behind the sliding bubble was the mixing process triggered by bubble passage. This mixing process was assumed to produce a stagnant, isothermal fluid region which they treated as a semi-infinite solid. Further assumptions included in the model were a constant temperature during the transient process and a region of isothermal fluid produced instantly. The results of the model when compared to the experimental measurements were found to underestimate the heat flux. To correct for this deficiency, they introduced an evaporation term to the model.

As a follow-on to the work of Ishibashi and Nishikawa, Kusuda and Monde (1981) explained the enhancement in heat transfer due to sliding bubbles that sweep away the thermal boundary layer. This was a conduction model which included a thin liquid microlayer between the bubble and the heated plate. Even though their work was an advancement of the model proposed by Ishibashi and Nishikawa by adopting varying wall temperature, they still maintained the assumption of an isothermal fluid region behind the
bubble and ignored the background advection flow. Monde (1988) developed on the previous model of Kusuda et al. (1981) to explore the effects of the liquid microlayer thickness on heat transfer enhancement. This investigation was done for both constant-heat-flux and constant temperature boundary conditions. Their findings indicated a departing of the model results from the experimental ones. To rectify this, Monde et al. (1989) suggested a model that included latent heat of vaporization in the microlayer. However, they observed that for low heat fluxes the model results agreed with the experimental results of Monde et al. (1989), while at higher heat fluxes, the model underestimated the experimental results. They suggested that this discrepancy could have been a result of the ignored background advective flow.

A number of models have been proposed to explain and classify the heat transfer into regions based on the dominant mechanisms near a bubble. Warrier and Dhir (2006) reviewed many of these models and settled on two primarily distinct types – models based on empirical correlations, and mechanistic models. The correlation-based studies by Zeitoun (1994)), partitioned the heat transfer region into two main zones. Evaporation heat transfer dominates the first zone while “agitation heat flux” is included as a third component of heat transfer in the second zone. On other hand, the mechanistic models identified several distinct regions in the flow. The variation in these regions were dependent on the stage of evolution in bubble growth. Basu et al. (2005) proposed models which subdivide the heat transfer into separate parts. Their models take into account the single-phase heat transfer, the heat transfer by evaporation and the transient heat transfer due to bubble passage.
The model developed by Basu et al. (2005) examined two different segments of the heat transfer process. Their model suggested in regions of slight bubble growth, convective mixing due to bubble passage accounted for the primary heat transfer enhancement in the single-phase background. Following this region of zero or little bubble growth is a zone where the bubble significantly grows and ultimately rises from the nucleating surface. The heat transfer in this second region was modeled by the evaporation heat flux, the forced convection heat flux and a component of heat conduction in the rear side of the bubble. They worked on the assumption that bubble movement interfered with the boundary layer, with a region of cold liquid filling the region that has just been left evacuated by the bubble. This component of transient conduction was modeled as a semi-infinite solid with a constant temperature boundary condition. The results of this model were validated in a part II of this work by Basu et al. (2005) which compared the findings to those of experimental work on vertical flows.

The observations made by Ozer et al. (2010-2012) of bubbles sliding through horizontal minichannels was augmented by their adaptation of the transient conduction model based on the one proposed by Ishibashi and Nishikawa (1969), and Monde et al. (1981, 1989). Ozer et al. included the thermal response of the heated foil surface into the bubble heat transfer model. It also incorporated a “mixed length” over which the precursor enthalpy profile is integrated to determine the initial conditions. The mixed length was based on the assumption that a column of the liquid behind the bubble and near wall was uniformly mixed such that it could be modeled as a uniform temperature region at the corresponding velocity-weighted mean temperature. Previous models had assumed a mixing of the entire channel height. The mixed length was left as a free parameter which could be correlated
with other observed parameters like the bubble diameter. Since this model by Ozer et al. (2010-2012) was developed for a series of bubbles, it was used repetitively as a new bubble traversed a location in the channel. The temperature profile produced by the previous bubble was used as an initial condition for the new bubble. The findings of their work were validated against the measurements of Ozer et al. (2011, 2012) for subcooled boiling by correlating the required mixed length to match the observed heat transfer to observables like average bubble diameter or bubble frequency.

The work by Albahloul et al. (2015) extended the transient conduction model of Ozer et al. (2010-2012) with the inclusion of the background advective transport term to capture the heat transfer rate resulting from precursor single-phase flow. Similar to the previous work, this model demonstrated a steady state response of the system due to the passage of a series of bubbles. The findings of this work were validated against the experimental results by Albahloul et al. (2014, 2015) for injected air bubbles.

These endeavors to model the heat transfer due to sliding bubbles have cemented the idea that the enhancement in heat transfer is substantially caused by the mechanisms associated with mixing as a result of bubble passage. Albahloul (2015) was able to demonstrate in his work that the measured enhanced effect was directly caused by the mixing effect. Most recently, Willard (2017) investigated the details of this mixing by use of computer simulations. He observed that in the near field are a set of complex fluid structures that produce mixing which is responsible for the enhanced heat transfer in the wake of the bubble. His results were for a single bubble passage and could not replicate the results of Ozer and Albahloul for the passage of continuous bubble fields.
Numerical Simulation of Sliding Bubble Flows

Detailed computational simulations of two-phase flows present challenges that are specific to a particular problem. Detailed simulations of bubbles in limited geometries and at low Reynolds Numbers are now possible with commercial computational fluid dynamics (CFD) software. While a large amount of study has gone into buoyancy-driven flows and pool boiling, there were few, if any, relevant studies published prior to Willard’s work which focused on horizontal pressure-driven bubbly flows. However, some work on the simulation of multiphase flows in other geometries exists.

Flows due to Bouyancy

Dhir et al. (2007) developed a simulation for the growth of bubbles and the corresponding heat transfer as the bubbles traversed an inclined plate. The bubble interface simulation was done using the level-set method. Dhir and his graduates have applied the level-set method to several problems in boiling and two-phase flows. He remains the biggest proponent of this method in academic research. For their simulation, the plate was set to incline at 75° from the vertical, and the bubbles slid over a heated surface that was maintained at a constant temperature. They noted a significant increase in the temperature gradient behind the sliding bubble, which was believed to propel the enhanced heat transfer in the bubble wake.

Senthil Kumar (2009) developed two-dimensional simulations for sliding air bubbles rising along an inclined plate. His approach to model the bubbles using the volume-of-fluid (VOF) method was very crucial in arriving at key conclusions. One, the VOF method made it possible to predict the bubble dynamics and interaction with the inclined plate. The effects of buoyancy and surface tension were modeled well using the VOF method. The
overall results of this simulation agreed with the experimental findings, with the possibility to improve on its accuracy by incorporating the conduction in the third dimension. The discrepancy observed in the results was greatest for the temperature of the heated surface. The enhancement in heat transfer was attributed to vortex shedding due to the action of the sliding bubble in the simulation.

Akhtar et al. (2015) developed a three-dimensional simulation using the VOF method for bubbles impacting and sliding along an inclined heated surface. The simulation accuracy at the interface was enhanced by use of a localized grid refinement. In this work, bubbles were generated at a location away from the wall prior to letting them to rise and impact on the inclined plate. Their findings concluded that the methods adopted in the simulation were capable of producing results that were consistent with experimental results of similar bubbles.

Zu et al. (2011) used the VOF method in ANSYS Fluent to perform a three-dimensional simulation of a bubble sliding along an inclined surface. This work did not focus on the generation of the bubble, instead, it injected a pre-formed bubble onto the surface and assigned it a growth rate similar to the experimental one, by use of a code. These authors observed vortices near the bubble which they concluded were responsible for the increased mixing hence the enhanced heat transfer.

Pool Boiling

There are many numerical simulations of pool boiling spanning several decades. Some of the more recent developments are reviewed here. Mukherjee and Dhir (2004) sought to investigate the consequence of bubble merger and the corresponding heat transfer in pool boiling by use of the level set method. Their conclusions indicated that bubble merger
resulted in enhanced heat transfer. Mukherjee et al. (2007) investigated single bubble nucleation in pool boiling, with a focus on the effect of the contact angle on bubble growth. They also used the level set method to simulate bubble growth on a horizontal heated surface.

Kunkelmann and Stephan (2009) used the VOF method coupled with the level-set method to simulate pool boiling. They observed that the coupling of the two methods resulted in accurate prediction of the bubble interface and eliminated the need for a local mesh refinement near the nucleation site. They also noted that the amount of increased heat transfer was proportional to the size of the bubble simulated, which could be attributed to more mixing by larger bubbles and vice versa. Kunkelmann et al. (2012) used this VOF method again to investigate the interaction of the three phases at the heated surface. The noteworthy conclusion of this work was that transient conduction in the liquid after bubble passage is the primary heat transfer enhancement mechanism.

Son and Dhir (1999) modified the level-set method to model phases interfaces in their investigation of growth and departure of vapor bubbles in pool boiling. Their findings from the numerical simulation were consistent with those from experimental measurements. They noted that the evaporation of the microlayer was responsible for about 20% of the overall heat transfer on the surface.

*Horizontal Flows*

Fukagata et al. (2007) performed a two-dimensional simulation of a 10 μm axisymmetric radius tube. The two phases comprised of water and air, with their interface being tracked using the level set method. It was noted that the local Nusselt number
increased significantly near the bubble, which was attributed to the liquid circulation near the bubble.

Taylor flow bubbles were numerically simulated by Gupta et. al. (2010). Theirs was a two-dimensional simulation in a 0.5 mm horizontal channel. This work provided useful insights into the effectiveness of the interface tracking techniques in producing equivalent results with respect to bubble size and shape. The work also reported a 2.5-times increase in the heat transfer for single phase flow, and a presence of remarkable mixing behind the bubble.

The effect of gas bubbles embedded in fluid slugs sliding in a microchannel was investigated by He et al. (2010). Simulation of the bubble interface was achieved by use of a phase-field technique. Their work indicated an enhanced mixing in the fluid slug due to bubble presence, which had the overall effect of a 2.4-times heat transfer enhancement as compare to that of single-phase flow.

Asadollahi et al. (2011) conducted another Taylor flow simulation to investigate the interaction of water and nitrogen bubbles. This was a two-dimensional flow with phase interface tracking achieved by use of the VOF method. A moving reference frame was assigned to the flow domain with the bubble remaining stationary. The boundary conditions were set for the inflow and the outflow velocities such that the bubble remained stationary as the walls slid past it. The main aim of this work was to determine the efficiency of the choice of a reference frame. Simulations were conducted to compare the results from a small moving reference frame and those from a large fixed reference frame domain. They discovered that a moving reference frame demanded less computational resources as
compared to a fixed one. In addition, the moving reference frame had no impact on the accuracy of the results, while also allowing for longer and larger domains to be simulated.

Talimi et al. (2012) investigated the effect of bubbles on pressure drop by simulating bubbles in slug flows. This was a two-dimensional simulation in microtubes using the VOF method with a moving reference frame to obtain flows of arbitrary length. They also developed a similar model for a stationary numerical domain. A comparison of the results from the two simulations indicated an overestimation of the pressure drop in the moving reference frame due to high wall shear stress.

The precursor work to Willard was done by Natesh (2012). He made an effort to numerically model a single bubble in a horizontal channel using Ansys Fluent. This work was devised to augment the experimental study by Ozer et al. (2012). Like Willard, he did not include phase change in his model. To address the heavy computational demand in the simulation, Natesh (2012) also adopted a moving reference frame with the bubble being held stationary in the channel. The bubble was initialized as a cylinder that spanned the height of the channel and maintained its shape throughout the VOF solution process. This constraint (no evolution in bubble shape) vastly reduced the computational resources required and is a major restriction in comparison to Willard’s later work. A rendering of this bubble can be seen in Fig. 1.
The findings of Natesh also demonstrated that even with no phase change, there was a significant heat transfer enhancement in the minichannel due to the presence of the bubble. In addition, he observed that the temperature of the heated surface reduced significantly long after a bubble had departed. His work also demonstrated a reasonable accuracy of the effect of the bubble in the “mixed length” model of transient conduction.

**Numerical Methods**

Computational fluid dynamics (CFD) is a collection of techniques that are used to construct numerical models of fluid flows. Commercial solvers are ever more widely used because they now have advanced and well-proven methods for discretizing the Navier-Stokes equations, and they package many helpful additional models together such as Volume of Fluid for phase boundaries and various turbulence model for turbulent flows. However, there remains a wide debate on the use of some of these methods, for instance
the most efficient method to track an interface when modeling the motion of phase interface locations.

**Interface Tracking Methods**

Over the years, a number of methods have been proposed to model the physics of two-phase flows. Kharangate and Mudawar (2017) in their review covered several studies dealing with numerical simulations of boiling and condensation. They explored several methods that can be used to model multiphase flows. The significant take away from their work is that no one particular method of interface tracking preferred in all cases.

- **Level-Set Method**

  This is a wide-spread and renowned method used in handling of interfaces that was initially proposed by M. Sussman et al. (1994). The method employs a global function which has a value that defines the distance to the interface. The phases are then defined by the method as positive or negative based on the value on the distance function. In addition, the interface itself exists at the position where the value of the location function is naught. Since the location of the interface is known implicitly, this method makes it easy to account for complicated surfaces. However, this implicit nature can easily fail to account for the conservation of mass when the directions involved are not parallel to the coordinate axes. Therefore, one must exercise caution when using this method since it also doesn’t have the ability to evaluate fluid properties in multi-fluid cells.
• *Front Tracking*

Tryggvason et al. (2001) and Unverdi and Tryggvason (1992) developed a method of front tracking which utilizes a pair of numerical grids. The technique is designed for interface tracking between phases. This method utilizes an Eulerian grid to evaluate the fundamental fluid equations in the phases, and a moving grid to track the phase interface. During the solution, the interface advection equation is solved using the velocities at the interface location, which is then used to solve for the motion in the interface. For merging bubbles or droplets, this method falls short because of the unique nature of the grid deployed to track the interface. This can only be addressed by adding special implementations to the solution. Nevertheless, the method is significantly useful in simulation of swarms of bubbles where there is no amalgamation taking place.

• *Volume of Fluid*

The primary work by Hirt et al. (1981) forms the basis for the Volume-of-Fluid method. This technique assigns a color function (equal to volume fraction) ranging from 0 to 1 to the existing phases. In this method, 0 is indicative of a cell with entirely one phase while 1 represents a cell with only the other phase. Therefore, a color code between 0 and 1 implies the existence of a cell with both phases. These types of cells have their properties evaluated by weighting the average depending on the color value. One limitation of this method is that it lacks the capacity to construct an interface. It only generates cells that have properties which lie between the phases. To address this, several other techniques have been proposed for interface construction.

Hirt and Nicholas (1981) also attempted to develop a technique that doesn’t necessarily construct an interface. The method assigns a color function to the cells with values between
0 and 1. This rudimentary technique is known as the donor-acceptor scheme. Ubbink and Issa (1999) also proposed a method that could be used to model sharp interfaces, unlike the donor-acceptor, without requiring interface construction. Their method, referred to as the compressive interface capturing scheme for arbitrary meshes (CISCAM) was used with no interface construction in simulations.

Youngs (1982) first developed the widely adopted interface construction method referred to as the piecewise linear interface construction (PLIC). The method generates surface vectors, using the information in the color value of the adjacent cells, to represent the cell interface. Several repetitions of this phenomenon leads to an infinitely thin representation of the interface plane, which when combined, results in a full interface. Although the method is considered powerful, it is computationally expensive because of the need to solve for the plane in every interface cell.

High Resolution Interface Construction technique was first introduced by Muzafirija et al. (1998). The method integrates a solver discretization scheme and the interface construction technique to provide a high-quality phase interface and permit a straightforward use of the upwind discretization schemes.

A study by Sussman et al. (2000), advanced by Enright et al. (2002), detailed a method that sought to address the shortcomings of the level-set and the VOF methods by integrating them. This method solved the color function and tracked phases by use of the VOF advection equation, then solve for the distance values using the level-set advection equation. Sun et al. (2010) modified this version by solving for the advection equation only for the volume-of-fluid, and then solving the level-set interface location function using geometric interpolations. The interface would then be constructed by use of the distance
obtained from the level-set solution in a manner that is similar to other VOF interface construction techniques.

**Surface Tension Modelling**

Brackbill et al. (1992) proposed the Continuum Surface Force (CSF) model for the representation of surface tension between fluid phases. The method integrates the surface tension force into the momentum equations then solve for the force present in each phase interface cell. Surface tension would then be calculated using the curvature of the interface surface. The normal vector of the surface was evaluated from the solver or computed from the color value of the cell for the VOF surfaces. This method was successful in modelling wall adhesion and it replaced the normal vectors with a vector that was determined by the wall contact angle.

To eliminate the challenges associated with the CSF model and their computational expenses in calculating the surface curvature, Lafaurie et al. (1994) developed the Continuum Surface Stress model. Compared to the CSF model, this new method is conservative which made it useful for flows with variable surface tension and sharp edges.

**Pressure-Velocity Coupling**

Patankar and Spalding (1972) proposed the SIMPLE (Semi-implicit Method for Pressure Linked Equations) algorithm which essentially couples pressure and velocity. This is the mainstream solver process used by the Fluent™ solver. The algorithm is executed repeatedly through several time-steps (iterations) until the solution converges, i.e., the correction factors become negligible. A simplified version of the process involves assembling values from the previous time-steps, computing gradients in the field, solving for momentum equations, solving for pressure-correction equation having relaxation terms,
correcting velocities and pressures, returning to solve the momentum equations again unless the corrections are negligible, advancing to the next step. Although other techniques have been developed for iterative processes, the SIMPLE method forms the foundation for the iterative solver. Its limitation is the associated computational expense and time taken to converge.

Doormal and Raithby (1984) modified the SIMPLE algorithm to SIMPLEC. They sought to simplify the correction step by employing a simpler formulation of the term. More refinement of the SIMPLE method was suggested by Patankar (1980) whereby the correction was only limited to the velocity field. Ubbink and Issa (1999) proposed the PISO method which transforms the correction step into an iteration. This included sub-iterative process had the overall effect of reducing the number of iterations performed by the solver, hence saving on solution time.

**Summary and Directions for the Present Work**

There is now considerable experimental data regarding heat transfer enhancement due to sliding bubbles, including those in confined channels. Willard’s computational study has been done to help visualize these structures, and to document how they contribute to the active mixing of the fluid in the near field region. While this simulation was revelatory in understanding these structures and their actual location in the channel, there remained a need to expand the range of Willard’s results by filling in obvious gaps and exploring “future work” items left undone. This thesis explores three items considered of immediate interest as listed in the Motivation section of Chapter 1.
Chapter 3

Methodology

Presented in this chapter is the numerical framework used to model the problem and to carry out the simulations. The description of the model of the liquid-vapor interface is first presented. Following this is a brief account of the numerical schemes for the spatial and temporal discretization of the Navier-Stokes, the Volume of Fluid (VoF) surface tracking model and the energy equations, the interpolation near the liquid-vapor interface, together with the pressure interpolation have also been provided. First, a description of the numerical method is presented as directly adapted from Willard (2017). This gives a clear outline on the numerical schemes that have been deployed to discretize the Navier-Stokes equations both in the spatial and temporal domains. The computational domain is described, and a concise review of the relevant boundary conditions is provided.

The Volume of Fluid Method

The volume of fluid (VOF) model is a popular computational modelling method used where two immiscible and incompressible fluids interact. This method, developed by Hirt and Nichols (1981), ensures the volume of a fluid in a domain of interest is conserved, and does not allow for any form of mixing, physical or chemical. The cells in a domain are defined such that a volume fraction indicative of the amount of each fluid in the cell is outlined. During the simulation, the model determines the net mass flux for each fluid in a
cell of interest, together with the corresponding volume fraction. A simplified version of a VOF model is shown in Fig. 3.1.

![Figure 3.1](image)

**Figure 3.1.** Two-dimensional VOF Method for phase interface tracking (Willard 2017).

An interface between the phases allows information about momentum to be interchanged. The construction of that interface is a necessary part of a phase-change model. Following Willard (2017), the present work has selected the “plane-based full geometric reconstruction” available in Fluent as the interface reconstruction model. This interface model uses the volume fraction information of the adjacent cell with its own volume fraction to solve for a plane in each non-unity, non-zero volume fraction cell. The overall effect of this interface reconstruction is an appearance of a large number of flat surfaces which commutatively combine to form a phase interface. Pressure matching is used to determine the pressure interaction of the fluids. This makes it possible to include surface tension and pressure gradients so that the two fluids to respond to each other.
Numerical Methods

The numerical setup of this work was a follow-on of the work of Willard (2017). The problem was setup based on a Lagrangian framework to bypass the computational challenges associated with this kind of a problem. This framework made it possible to simulate an arbitrary long channel by use of a relatively small computation domain, which moved with respect to an approximately stationary bubble.

Fundamental Equations

The continuity equation, the incompressible Navier-Stokes equation and the energy equations are the governing equations for these simulations, and the foundation on which the Fluent solver is built. The forces due to surface tension and gravity have been included because they are significant in the present problem. Equations 3.1 and 3.2 present the forms of the continuity and momentum equations solved in this problem respectively (Willard 2017),

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \quad \text{Eq. 3.1}
\]

\[
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{uu}) = -\nabla p + \nabla \cdot (\mu (\nabla \mathbf{u} + \nabla^T \mathbf{u})) + F_{sv} + \rho \mathbf{g}. \quad \text{Eq. 3.2}
\]

The equation 3.1 consists of the velocity vector, \( \mathbf{u} \), the density, \( \rho \), the viscosity, \( \mu \), acceleration due to gravity, \( \mathbf{g} \), and the force due to surface tension that occurs during phase
reconstruction, \( F_{sv} \). This surface tension force is adapted from the work of Brackbill et al. (1992), and is defined thus,

\[
F_{sv} = \sigma k \hat{n}.
\]

Eq. 3.3

This expression for the surface tension model has the surface tension coefficient, \( \sigma \), the surface curvature, \( k \), and the normal vector, \( \hat{n} \). The model is the basis for the pressure coupling at the gas and liquid interfaces so that the surfaces can interact with each other as the simulation progresses.

The energy equation used to model this problem is given by

\[
\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho E) = \nabla \cdot (k \nabla T) + \dot{Q}.
\]

Eq. 3.4

In Eq. 3.4, \( E \) is the internal energy per unit mass, \( T \) the temperature, \( k \) the thermal conductivity and \( \dot{Q} \) the internal energy generation per unit volume. During the simulation, the internal generation term in the foil upper surface is the main source of thermal energy.

Model Implementation

Fluent\textsuperscript{TM} is a commercial computational fluids dynamics software that has the ability to simulate multi-phase flows using the VOF method. The software uses a finite-volume-based solver. This work uses a Lagrangian reference frame attached to the bubble as it traverses down the channel. In this frame, the model is simulating an infinitely long channel as shown in Fig. 3.2.
Figure 3.2. Implementation of the Lagrangian framework in the model (Willard, 2017).

The primary purpose for this approach is to cut down on the immense required computer memory and time for simulating a moving bubble in a stationary channel. The result of this approach is that the bubble moves through a streamwise distance that is significantly longer than the computational domain. In addition to this, the approximately stationary location of the bubble simplifies the analysis in terms of comparisons of data for different time realizations. Another requirement of this coordinate system is that the upper heated wall that forms the solid domain should move during the simulation.

Solver Setup

The domain was created, and the appropriate meshing done using the tools provided in ANSYS Fluent. The Design Modeler was used to model the solid domain while the Meshing tool was used to develop and generate the required mesh for the problem. The work adapted the use of hexahedral shapes for meshing because of their ability to minimize on the errors associated with the VOF model. The solid and the liquid subdomains were assigned different size of meshes in the vertical direction. This was necessitated by the requirement of the cells in the fluid domain to change independently to allow for the determination of the grid independence. To avoid for any interpolation-based coupling
from one domain to another, the mesh size was maintained in the streamwise and spanwise direction. One of the sources of numerical errors when using VOF is high aspect ratios. To avoid this in the solid domain, the thin plate was meshed with a single cell in the z-direction. This ensures that the cell aspect ratio in this domain remains the same for all grid sizes. This approach to meshing replicates the shell-conduction model in Fluent™ in which case the solver develops a single layer of cells over the fluid domain to create an arbitrary solid domain. However, the shell-conduction model does not yield useful or detailed data for the arbitrary solid surface, thus the need to model a single layer of an actual meshed solid in this work.

The modeled domain complete with meshes was imported to Ansys Fluent for further setup. To articulate the accompanying physics during the simulation, the appropriate models were selected for the problem, on the graphics user interface (GUI). The key models checked included the heat transfer, flow, solid motion (Moving reference Frame) and Volume of Fluid. Heat and flow models made it possible to model the convective flow. Appropriate property materials for the solid domain were input manually while those of the fluid domain were added using the user defined database files that were loaded from the compiler. The boundary conditions at the interfaces, and at the leading and following walls, both for the solid and liquid subdomains were also added using user-defined functions (UDFs). The VOF model allowed for the modeling of an air bubble inside of the liquid domain. This was done during the initialization of the problem by use of the ‘patching’ technique in Fluent. The solid model made was used to simulate the sliding action of the solid plate above the bubble.
Computational Domain

The primary focus of the present study was to expand the scope of the work done by Willard. (2017). As a consequence, the size of the computational domain size remains unchanged. The domain is $1.25 \, \text{mm} \times 20 \, \text{mm} \times 30 \, \text{mm}$ long. These dimensions have the blessings of the experimental work by Ozer and Albahloul et al.: $1.0 - 1.48 \, \text{mm} \times 23 \, \text{mm} \times 357 \, \text{mm}$. In comparison to the experimental work, the computational domain moved through a greater distance in the streamwise direction. The domain was set up such that it moved at bubble speed while the bubble remained stationary with respect to the moving domain.

![Figure 3.3](image-url) Illustration of the computational domain (not to scale). (Willard 2017).

The computational domain comprised of two subdomains; the solid domain made up of the thin metal plate at the top of the channel, and a fluid subdomain that comprises of two fluid phases.
Computational and Boundary Conditions

The simulation domain utilized the Lagrangian reference frame. The nature of this frame enables the mimicking of the laboratory perspective where the moving bubble is continuously sliding into new and undisturbed region of the channel. Temperature boundary conditions are used to model the movement of the domain into the downstream direction where the mixing cup temperature is continually rising. The inlet boundary, ahead of the bubble, is referred to as ‘leading’ boundary, and represents the undisturbed portion of the channel, while the trailing boundary is chosen as ‘following’, to represent the channel portion already passed by the bubble. A characteristic velocity profile for channel flow is implemented during the simulation. Because of the Lagrangian frame at work, a Lagrangian shift occurs where the velocity crosses a zero and flows in the opposite direction.

A heat source is provided to the model through electrical heating in the solid subdomain. This forms the heated boundary condition in the simulation that is also moving. Material properties of Hastealloy foil (Hastealloy C-276: \( \rho = 8970 \text{ kg/m}^3 \), \( C_p = 397 \text{ j/kg-K} \) and \( k = 8.9 \text{ W/m-K} \)) used in experiments were assigned to the solver to simulate this solid. The thickness of the foil was chosen such it represented a single layer of cells to avoid high aspect ratios, and reduce complexities associated with determination of the grid independence in the domain. The top surface of this is an adiabatic boundary, while the internal generation (at a value selected to replicate values in the range of the experiments) is set at 25.6 \( mW/mm^3 \) in each cell. The thermal boundary conditions of the leading and following edges of the domain are implemented uniquely. The following edge is an adiabatic boundary condition to avoid excess heat flux resulting from an overestimated
boundary temperature. On the other hand, the leading edge is assigned a thermal boundary condition using the user defined functions that is loaded to the solver before the iterations begin. This boundary specifies the foil temperature at a Eularian location of the leading edge with respect to the current time as the bubble moves,

\[ T(t) = T_{ini} + \left( \frac{q_w}{\rho_f C_{pf} u_t} \right) (L + u_p t) \]

Eq. 3.5

As the bubble slides into new, undisturbed streamwise locations, the temperature of the plate at the leading boundary increases at it does in a uniform-flux channel wall. To achieve this in Fluent, a custom C-code developed by Willard (2017) and introduced into Fluent as a user-defined function calculates the Eularian location of the domain and assigns the correct plate temperature based on the present simulation time. For the wetted surface of the heated plate (the lower boundary of the solid domain) that is in contact with the fluid domain is coupled to the adjacent fluid domain cells through an interface that equates temperature and heat flux. This is a feature that is provided within Fluent and it ensures that the iterations of the coupled interfaces result in consistent temperature on both sides.

Liquid and gas phases make up the fluid subdomain. The liquid phase simulates the properties of 3M Novec 649™. This engineered fluid is a refrigerant produced by 3M which has attractive boiling temperatures for experiments that have thermochromic liquid crystals as temperature sensors. Properties of atmospheric air have been used to simulate the gas phase. For very low temperatures, such experiments do not record any significant change in phase. To be consistent with the existing work on subcooled flows by Albahloul
and the computational work by Willard, this study simulated the air/liquid interface with no change in phase.

The Volume of Fluid (VOF) module provided within Fluent™ was used to model the air/liquid phase interactions. This was done in addition to selecting the “full geometric reconstruction” of the interface. The reconstruction uses the volume fraction of the current cell and the volume fraction of the adjacent cell to develop a planar surface through the cell which is a representation of the interface. This is done efficiently in Fluent such that an accurate representation of the interface is obtained without the need to expensive computational resources. The resulting surfaces from full geometric reconstruction are not smooth, but the still present a better model of the force balance compared to other known models. To model the surface tension at the fluid interface, the continuum method (Brackbill et al. (1992)) was utilized. The solid/liquid/gas contact was maintained at 12 degrees, which is the same value used by Willard in his work. This choice was motivated by his observation of the front-on photographs of different sizes of bubbles that were taken by Albahloul (2015). This provided a physical model that is consistent with the previous computational and experimental work.

The boundaries at the channel sides were developed to model a central section of a wider channel. The locations of these boundaries are selected to minimize their influence on the bubble dynamics and the heat transfer near the bubble. The boundaries have no penetration and zero shear. In the laboratory frame, upper and lower wall of the foil are stationary. This is achieved in the simulation by assigning no-slip boundaries at the top and bottom of the fluid, with a speed that is equal and opposite to the Lagrangian velocity of the domain. The upper (heated plate) and lower (adiabatic) walls also move at a speed
that is equal and opposite to the Lagrangian velocity of the domain. For the leading and following boundaries of the fluid domain, the streamwise velocity is given by

\[ u(z) = 6 \, u_l \left\{ \left( \frac{z}{H} \right) - \left( \frac{z}{H} \right)^2 \right\} - u_p \]  

Eq. 3.6

The velocity profile of the liquid is a fully developed, pressure driven, channel flow that is parabolic in the z-axis, for both the leading and following boundary conditions. A proper representation of the velocity in these two boundaries is critical in achieving an accurate mass balance in the simulated system.

The thermal boundaries at the side walls and the bottom of the fluid are adiabatic. A thick sheet of Plexiglas used in the experiments as the lower channel surface is simulated as adiabatic and moving toward the bubble at the bubble speed as is the upper heated surface. Simulation of the leading and following thermal boundaries is done through a representation of the Lagrangian domain motion. By using the temperature of a parallel-planes channel with one adiabatic and one heated wall, the temperature of the leading thermal boundary can be determined by

\[ T(t, z) = T_{ini} + \left\{ \frac{q_w}{\rho_f \, C_{pf} \, u_l \, H} \right\} * (L + u_p t) \]  

\[ - \left\{ \left( \frac{q_w \, D_H}{k_f} \right) * \left[ \frac{1}{2} \left(1 - \frac{z}{H}\right)^4 - \left(1 - \frac{z}{H}\right)^3 + \left(1 - \frac{z}{H}\right) \right] \right\} \]  

Eq. 3.7

With the advancing simulation time, there is an increase in the enthalpy at this boundary, which is accompanied by a rise in the fluid temperature. The overall import of
the leading boundary for both the plate and fluid domain is the determination of the temperature for the computational domain, and they remain unaffected by the values computed by the solver. On the contrary, the following thermal boundary is assigned a profile that is computed dynamically as the simulation progresses. Information from the previous time step is utilized to determine the values for the current boundary. The outcome of this dynamic calculation enables the fitting of a fully developed temperature profile to the plate temperature at any spanwise \((y)\) location defined by the boundary below,

\[
T(y, t, z) = T_{\text{wall}}(y) + \left\{ \left( \frac{q_w}{\rho_f C_{pf} u_l H} \right) \ast (L + u_p t) \right\} \\
- \left\{ \left( \frac{q_w}{k_f} \frac{D_h}{2} \right) \ast \left[ \frac{1}{2} \left( 1 - \frac{z}{H} \right)^4 - \left( 1 - \frac{z}{H} \right)^3 + \left( 1 - \frac{z}{H} \right) \right] \right\}
\]

Eq. 3.8

As mentioned earlier, the \(T_{\text{wall}}(y)\) represents the temperature at the trailing boundary of the heated plate computed from the previous time step. A comparison of this wall temperature value with that of an actual wall temperature of a corresponding location in an undisturbed channel flow would yield some difference. This is due to the fact that in most cases the temperature wake caused by bubble passage extends back to the end of the domain. This informs the reason behind the usage of this custom \(T_{\text{wall}}(y)\) boundary condition. A table of the boundary conditions for the simulation is shown in Appendix B, and a table of the solver settings is shown in Appendix C. These appendices were taken verbatim from Willard (2017) and included here for completeness.
Solution Initialization

Before the start of calculations, the variables of interest (velocity and temperature) have to be initialized in the flow field in the entire domain. The initial time, \( t_0 \), in the Lagrangian frame is determined by computing the values for an arbitrary zero point in the Eulerian frame using the equations for the flow and thermal fields. This yields a similar solution to a fully developed channel with an initial temperature condition. Using the patching tool, a cylinder with a height equivalent to that of the channel and a volume equivalent to that of the desired bubble is patched into the fluid domain. As the solution progresses, the Fluent solver computes a bubble shape consistent with the forces on the bubble at that bubble volume. Solutions show that the bubble shape stabilizes quickly and long before the thermal field reaches a quasi-steady distribution. Before the solver performs any calculation, it makes one initial pass that assigns fluid properties to the corresponding air and liquid phases in the appropriate cells and also constructing the phase interface. Once this VOF initialization is done, the solution will proceed as outlined earlier in this chapter for the VOF model.

Parameter Space

One of the key objectives of this study was to expand the parameter space of the immediate preceding study by Willard. This was done by simulating for three bubble volumes at two different liquid flow rates with the same thermal conductivity. The thermal conductivity was selected such that it produced a Prandtl number of 6, roughly half-way between 12 and 1, the values computed by Willard (2017). In consistency with the previous work, there is no phase change in the model, thus the vapor volume is constant. For an accurate comparison of the current data with the results from the previous work, the flow
rates were maintained such that the resulting average liquid speeds were 27 mm/s \((Re = 169)\) and 54 mm/s \((Re = 338)\) and the same bubble diameters (and initial bubble volumes) were used. Table 3.1 gives the relevant values.

Table 3.1. Summary of the simulated cases for \(Pr\) study

<table>
<thead>
<tr>
<th>Nominal (D_b/H)</th>
<th>(Re)</th>
<th>Average Liquid Speed (mm/s)</th>
<th>(Pr)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>169</td>
<td>27</td>
<td>6</td>
</tr>
<tr>
<td>2.0</td>
<td>169</td>
<td>27</td>
<td>6</td>
</tr>
<tr>
<td>3.3</td>
<td>169</td>
<td>27</td>
<td>6</td>
</tr>
<tr>
<td>1.1</td>
<td>338</td>
<td>54</td>
<td>6</td>
</tr>
<tr>
<td>2.0</td>
<td>338</td>
<td>54</td>
<td>6</td>
</tr>
<tr>
<td>3.3</td>
<td>338</td>
<td>54</td>
<td>6</td>
</tr>
</tbody>
</table>

A key consideration in the simulation is the attempt to replicate the diameter ratios, \(D_b/H\), observed in experimental work done previously. The nominal values of 1.1, 2.0 and 3.3 that were used in the previous computation work were also adopted for use in the present study. For each of the initial bubble volumes the corresponding value of \(D_b/H\) were the same for each liquid flow rate. These ratios are determined by taking the average bubble width in the streamwise and spanwise direction once the bubble is at the quasi-steady state. The values obtained are useful in investigating the effect of bubble confinement on the response of the system.
Chapter 4

Results and Discussion

This chapter presents the results from the simulations and analysis for a liquid $Pr = 6$ for both low and high $Re$, which is an expansion of the work by Willard who produced the simulations of $Pr = 12$ and $Pr = 1$. The chapter also presents the findings of the improved and expanded characterization of the near field region through the introduction of a new near-field Nusselt number correlation and the analysis of the local behavior of velocity and fluid temperature. Lastly, the chapter discusses the effect of changing the thermal boundary condition on the heated channel wall to simulate a wall with the capacity to change its temperature rapidly in response to bubble passage.

To build an argument for these objectives, the results are first validated against a single-phase case to ensure that the results of the simulations are in agreement with the predicted results for a known flow. This foundation is then used as a reference for the discussion of the results from the simulation of an expanded parameter space for an arbitrary fluid ($Pr = 6$), in the second section of this chapter. The third section takes a closer look into the activities in the near-field region of the fluid domain, with an accompanying characterization of the velocity and temperature profiles. This is aimed at establishing the causal relationships, between the observed spatial pattern of surface heat flux and the flow structures that were observed by Willard. Also, a new Nusselt number correlation is developed to predict the key heat transfer parameters in this region, in terms of the distance
behind the bubble. The last section of this chapter will provide an in-depth discussion of the effect of changing the thermal boundary condition of the heated channel, which in essence simulates a wall with the capacity to rapidly change its temperature in response to bubble passage. This is achieved by changing the material properties (specific heat capacity, $C_p$, and the molecular thermal conductivity, $k$) of the heated wall and observing the corresponding changes in temperature.

Validation

Single Phase Test Case

To validate the results obtained from the Fluent\textsuperscript{TM} model, simulations were run for the precursor values for the cases $Pr = 6$ and $Pr = 12$ using the low $Re$ and the diameter ratio $D_b/H = 2.0$. For these cases, the secondary (vapor) phase in the VOF model was turned off such that the formulation was only solving for the primary (liquid) phase. To obtain a meaningful time scale for the computation, a dimensionless time is defined in Eq. 4.1,

$$t^* = \frac{t}{t_p}$$  \hspace{1cm} \text{Eq. 4.1}

Where $t$ is the flow time, $t_p$ is the time it takes a point on the heated foil to traverse from one end to the other of the computational domain (30 mm) in the moving reference frame (which is identical to the time taken for the bubble to traverse a length of the channel that is equivalent to the domain length).

The plot of the wall temperature and mixing cup temperature for the case $Pr = 12$ and $Pr = 6$ at $t^* = 1$ is shown in Figs. 4.1 and 4.2 respectively. It can be noted that the slope of both plots are equal and uniform through the length of the channel, which is the expected
outcome in a channel with a uniform surface heat flux boundary condition. The mixing cup (fluid bulk) temperature is defined by,

$$ T_m = \frac{\int u_b(z) \ast T(x, z) \, dz}{\int u_b(z) \, dz}. $$

Eq. 4.2

The variation of the mixing cup temperature along the channel is not a standard output of the report function in Fluent™ post-processor. This implies that there is a need for a custom function within the Fluent™ that generates a product of the local fluid velocity ($u$) and the local fluid temperature ($T$). The custom field is then integrated in the $y$-$z$ plane to obtain a value for each local fluid velocity, on each surface, beginning from the channel inlet. The ratio of each of the value from the custom field function value to the corresponding integral local fluid velocity yields the mixing cup temperature at a streamwise position.

![Figure 4.1](image_url)  

**Figure 4.1.** Temperature distribution of the heated wall and the bulk fluid along the channel for low $Re$, $D_bH = 2.0$ $Pr = 12$ at $t^* = 1$.  

43
Figure 4.2. Temperature distribution of the heated wall and the bulk fluid along the channel for low $Re$, $D_b/H = 2.0$ $Pr = 6$ at $t^* = 1$.

Figure 4.3. Heat flux and $Nu$ distribution along the heated wall center-line for low $Re$, $D_b/H = 2.0$ $Pr = 12$ at $t^* = 1$. 
Figure 4.4. Heat flux and $Nu$ distribution along the heated wall center-line for low $Re, \frac{D_b}{H} = 2.0$ $Pr = 6$ at $t^* = 1$.

The grid and temperature convergence of the model were not primarily studied since there were no fundamental changes in the model at hand. However, the previous validation was done to ensure that when this specific test case was run on a Windows OS, having being previously solved on a Linux OS, the few changes that were made to the algorithm did not affect the expected results. This was then used as a background to explore confidently the scope of the work that was outlined for this study as outlined in the subsequent sections.

The Case of $Pr = 6$

From previous studies, it was clearly noted that a change in $Pr$, which is a direct result of changing the liquid thermal conductivity resulted in the biggest change in the Nusselt number. It was therefore the goal of this author to expand the study into this phenomenon,
to piece up the gaps that were left by Willard’s work, following some of his recommendations. Consequent to this, a Prandtl number that was approximately half that of the Novec-649 liquid was selected for use in this study, i.e. $Pr = 6$. The desire to obtain a set of data that would approximately fill the gap between the previously studied Prandt numbers, informed this selection. The results obtained for a fluid with $Pr = 6$ were analyzed and compared to those from Novec-649, which is an actual fluid with a $Pr = 12$, in addition to those of a fluid with thermal conductivity such that its $Pr = 1$.

**Reduction of Data**

Generally, these computations in Fluent™ generates several gigabytes of data. As a result, there was a need to narrow in on the data that could reduce the size of data to be analyzed, without affecting in any way the overall results of the simulation at hand. As suggested in the time-dependence studies by Natesh (2012) and Willard (2017), the established and widely-accepted transients in the system were observed to be present in the system until after $t^* = 1$. This was found to be the case in this study, therefore this previous guideline for a quasi-steady, solution free of starting transients, was also adopted here.

With the elimination of these starting transients, it was also noted that bubble movement relative to the average bubble motion, both laterally and sometimes in the direction of flow, always remained. The degree of these movements varied with respect to the bubble size, with the smallest bubbles recording the highest lateral movements. The choice of the region behind the bubble to analyze the plate thermal response was made difficult by these movements. This matter was mitigated by selecting the data for the time period in which the bubble was located in the channel center with the wake directly behind it. Considering the results from these computations are quasi-steady in nature, the
investigation of the near-field temperature and momentum measurements are reasonably representative of those present at any other time. Such a representation was obtained by averaging three realizations taken for $t^* > 1$ separated by nonrepeating time intervals. The resulting data sets provided the spatial distribution of the surface temperature and surface heat flux utilized in the analysis. The last mile in this data reduction process was another spatial averaging in the spanwise direction, over the spanwise extent of the bubble wake. The outcome of this averaging in the y-direction is a single value of a particular quantity (wall temperature or surface heat flux) at that x-location.

Definition of Terms

A number of important terms that will be used repeatedly in the analysis and the subsequent discussion are hereby defined. The selection includes both the dimensional and the non-dimensional values that will be used to characterize the heat transfer behavior in the simulation. From the temporal averaging in three irregularly displaced time realizations, to the lateral averaging in the lateral (y) direction to obtain a single value in each x-location in the bubble wake, the resulting data was used to calculate a heat transfer coefficient defined thus

$$h_x \equiv \frac{q''_w (x)}{T_w(x) - T_m(x)}.$$  \hspace{1cm} \text{Eq. 4.3}

Where $T_w$ is the plate surface temperature, $T_m$ is the mixing cup temperature (as defined earlier in equation 4.2), and $q''_w$ is the surface heat flux at that particular x-location.

Two types of Reynolds numbers were adopted for this analysis. The first one is the standard channel Reynolds number, defined using the average liquid velocity in a channel cross section and the hydraulic diameter,
\[ Re \equiv \frac{u_l \cdot D_H}{v}. \quad \text{Eq. 4.4} \]

The second Reynolds number is adapted from the work of Willard (2017), which is defined such that the characteristic speed is the difference between the actual bubble speed and the maximum bubble speed, while the characteristic length is the bubble diameter

\[ Re_{bubble} \equiv \frac{u_\Delta \cdot D_b}{v}. \quad \text{Eq. 4.5} \]

where

\[ u_\Delta \equiv u_{CL} - u_b. \quad \text{Eq. 4.6} \]

The relative velocity of the bubble, \( u_\Delta \), is defined with respect to the centerline liquid velocity. The range of values of the \( Re_{bubble} \) obtained, 60-150, is consistent to those of a laminar flow around an obstruction which in this case is represented by the bubble with a diameter, \( D_b \).

To correlate the recovery of the thermal wake behind the bubble, Willard proposed a dimensionless group similar to the group that characterizes the dimensionless \( x \)-location in developing laminar duct flows. Willard’s version includes the effect of bubble confinement, the bubble Reynolds number, and separate exponents for both the Reynolds and Prandtl numbers. The resulting definition is given in Eq. 4.7,

\[ x^* \equiv \frac{(x_b)}{P{\tau}(Re_{bubble})^n (H_b)^m}. \quad \text{Eq. 4.7} \]
This equation captures three length scales: channel spacing, $H$ or $D_H$, and the bubble scales, $H_b$, the bubble height, and $D_b$, the bubble diameter. In this expression, $H_b/H$ is the bubble confinement such that a value of 1 represents a fully confined bubble. This dependence on confinement has been clearly demonstrated in the work of Albahloul (2015), Ozer (2010) and adopted by Willard (2017). Therefore, the definition that Willard proposed formed the basis to test the consistency of the results, but it also became a subject of scrutiny in this study with respect to the near field region (2 mm) behind the bubble.

Thermal Response

The heat transfer coefficient in the bubble wake was analyzed for the new $Pr = 6$ data for both the low $Re$ and high $Re$ cases in the same manner as was done by Willard for $Pr = 12$ and $Pr = 1$. First, the convection coefficient, $h_x$, in the bubble’s trailing wake is computed and plotted against the distance behind the bubble, $x_b$. The averaging process described above was employed. Figure 4.5 shows the results for all three $Pr$. From this log axis plot, it can be noted that inside of the trailing wake, there is a similar development in the dimensional $h_x$, for $Pr = 6$ and $Pr = 12$. The data for $Pr = 1$ lie significantly higher and have a shallower trend.
Figure 4.5. Convection coefficient versus distance behind bubble for all three $Pr$.

From a general perspective, the curves demonstrate a consistent shape which resembles a power law relationship that approximates $h \sim x^{-1/2}$ for the heat transfer and the distance immediately behind the bubble. The midfield region, however, show some departure from this approximation. This calls for a dimensionless casting of these results based on

$$Nu_x \equiv \frac{h_x D_b}{k_f}.$$  

Eq. 4.8

Willard suggested a need to consider the dependence on bubble size and speed, the level of confinement, and the liquid flow speed. These requirements led to a dimensionless streamwise wake position given in Eq. 4.7. Willard found the values of the exponents given in Eq. 4.9 for a reasonable fit for $Nu_x$ in the midfield region,
\[ x^* \equiv \frac{(x_b/D_H)}{Pr(Re_{bubble})^{0.45}} \left(\frac{H_b}{H}\right)^{0.6} \]  

Eq. 4.9

It was of interest in this work to determine if data for \( Pr = 6 \) supported those constants. Figure 4.6 is a graph of \( Nu_x \) vs. \( x^* \) for all three \( Pr \). The curves in Fig. 4.6 depict the improvement in collapse of the data found by Willard, and the \( Pr = 6 \) data follow the \( Pr = 12 \) data. Note that the \( Pr = 1 \) data are low at lower \( x^* \) but track at higher \( x^* \). The only exception from the new data set is the larger bubble, \( D_b/H = 3.3 \), running at the lower \( Re \) (channel) of 169. The reason for this deviation still remains unclear from the findings of this work, but it may be attributed to the differences occasioned by the reorganization of the fluid after the bubble passage compared to other bubble volumes. This has been explored at length in a different section of this chapter.
Figure 4.6 shows that for the mid and far wake, $Nu_x$ depends approximately on $(x^*)^{-1/2}$. Further downstream, the curves approach the precursor value of $Nu_0 = 5.4$. In addition, it can also be noted from Fig. 4.6 that the thermal response can be split approximately into three main parts. The first region is the near field which is immediately after the bubble passage, $x^* < 0.001$, where $Nu_x$ is fairly uniform. This indicates the region where the fluid is being re-introduced to the plate right after the bubble passage and the near-field structures are actively involved in fluid mixing. A detailed examination of this region follows in the next subsections of this chapter.

The second section region of the bubble wake in Fig. 4.6 shows a consistency with the previous findings from Willard’s work. This region depicts a power law relationship where $Nu_x$ closely follows $(x^*)^{-1/2}$. This region is right at the end of the mixing region behind the bubble where flow is reorganizing to form a developed channel flow.
The third region of the thermal plate response demonstrates a return to the precursor values. This region begins where the curves deviate from the power law and start to asymptotically approach the precursor value. This behavior is unclear for the higher Pr data (6 & 12) because the curves terminate at the end of the domain before the trend is clear, but the value of the Pr =1 data is that they complete the picture of the return to channel flow heat transfer.

The Near Field Region

The preceding discussion follows Willard and adds new data that support his findings. An area left relatively unexplored by Willard is the details of the near wake. Figure 4.7 presents the general linear plot of $N_u$ vs. $x_b$ for the first 3 mm of the wake.

![Figure 4.7](image)

**Figure 4.7.** $Nu$ versus $x_b$ in the near field region for the three $Pr$ cases.
From Fig. 4.7 there is a clustering of the plots based on the $Pr$. Therefore, it is evident that the $Pr$ remains important in the region very near the bubble. It can also be noted that for all the cases, at $x_b > 2\ mm$, the change in $Nu$ with respect to the distance behind the bubble is significantly reduced. This observation is part of what prompted the present study to explore further into the region within $2\ mm$ of the bubble where the mid-region collapse does not hold. The selection of this ‘near-field’ region is backed by the general trend of the curves in the region $0 \leq x < 3\ mm$ behind the bubble on Figs. 4.5 and 4.6. In Fig. 4.5, the change in the heat transfer coefficient is rapid in this region, before all the curves converge together into a power decay region, which was classified by Willard to represent the mid-field behind the bubble.

In addition, a visual investigation of the $Nu$ plots in Fig. 4.5 reveal that some curves, in the range of $1 \leq x < 4\ mm$, drop below the common power-law behavior seen in the majority of the cases. For example, the cases with $D_b/H = 3.3$ for the low $Re$ (169) at $Pr = 6$ and $Pr = 12$ clearly show a departure from other curves in this section, and thus are identified for a more in-depth study. Figure 4.8 shows the decluttered plot with these selected cases.
The corresponding curves for $D_{b}/H = 2.0$ do not show this behavior and are shown in the graph for comparison. They are also the bubbles with the most detailed study of their flow characteristics and thermal responses. The dip in the three curves shown compared to the more common behavior was a feature in some of the $Pr = 12$ curves from Willard’s work, and it was an issue that was never resolved by him. The first effort here in the near-field will be to offer an explanation of this behavior.

Very near the bubble, the heat transfer is responding to a steep slope in near-wall temperature as well-mixed cooler liquid floods the surface near the departing phase front. As time goes on, read in this problem as $x$ distance increasing, this process gives way to a growing thermal boundary layer as the flow reorganizes. Finally, an organized channel flow is reestablished in the very far field. The rate this sequence progresses is a function
of $Pr$ so that for $Pr = 12$, the organized channel flow has yet to be fully realized by the end of the computational domain. To model the evolution from the flood-response near-field region to the reorganizing-boundary-layer region, we borrow an idea from the field of mixed convection heat transfer: the interaction of two competing heat transfer mechanisms. The classic treatment when two competing mechanism are present is to create a nonlinear combination of the $Nu$ created by each mechanism:

$$ (Nu - Nu_0)^n = (Nu_1 - Nu_0)^n + (Nu_2 - Nu_0)^n. $$  \ \ \ \ \text{Eq. 4.10}

where the subscripts 1 and 2 refer to each competing mechanism and the exponent, $n$, is a matter of traditional data analysis, usually equal to 3 or 4. We will use $n = 3$ here. The term $Nu_0$ in this analysis will be the channel-flow baseline $Nu$. That aspect can be ignored here. \textit{The analysis presented here is purely notional in that the underlying $Nu$ response curves are created to demonstrate the idea and are not computed from any mechanism-based analysis.} Figure 4.8 shows data for the case of $D_b/H = 3.3$, $Re = 169$, and $Pr = 12$. Also shown are two presumed process curves. The red curve, $Nu_1$, is an imagined near-field process that was created by a fit to the data just behind the bubble and then allowed to decay as a power law. The blue curve, $Nu_2$, is an imagined far-field process that was fit to the far-field data and allowed to plateau in strength in the mid-field region. These curves were combined using Eq. 4.10 with $n = 3$. 

56
This exercise demonstrates that two heat transfer mechanisms, one in the very near field of the bubble and one in the mid field, can, if combined in a traditional nonlinear weighting common to mixed-mechanism problems, produce a dip in $Nu$ that tracks a measured example from this study. This presents the possibility that these dips (which are repeatable when the runs are repeated) are caused by some cases having a delay between these two mechanisms while other cases do not. Following this argument, the curves for the plots of $D_b/H = 3.3$ in Figs. 4.5 and 4.8 for low $Re$ can be defined into three sections: the near-field in the range of $0 < x_b < 3 \text{ mm}$, the mid-field in the range of $3 < x_b < 20 \text{ mm}$, and the return to channel flow behavior for $x_b > 20 \text{ mm}$.
*Velocity and Temperature Profiles of the Near-Field*

The near-field region of the bubble wake covers approximately $0 < x_b < 3 \text{ mm}$. This is a region dominated by a complex set of vortical structures observed by Willard including a pair of columnar structures with axes in the $z$ direction sitting immediately behind the bubble. The structures immediately behind the bubble were observed by Willard to move cooler fluid from the bottom of the channel toward the heated top plate in the near field. This motion was associated with the heat transfer enhancement in the rear wake. Figure 4.10 from Willard (2017) is his illustration of the effect and general placement of the structures he found. In the figure the rear wake is to the left of the image and the cold fluid is blue. The columnar structures are indicated by circles in these $x$-$y$ planes.

The present study seeks to make follow-on observations by observing graphs of the velocity components and the temperature across the channel in the $z$ direction. Several locations in the rear wake will be chosen. This set of observations are unique to the current study and were not presented in Willard (2017).
Figure 4.10. Temperature in the fluid phases at plane located at \(0.75H\) for \(D_b/H = 3.3\) from Willard (2017). The bubble surface is also shown in gray, the diameter of the bubble at the location of the section plane as well as the contact line where the bubble meets the plate can be seen. Overlaid on the temperature field are representations of prevailing velocity vectors at that location. The \(R\), \(G\), and \(B\) locations are indicated.

Two cases were selected for this study: \(D_b/H = 3.3\) running at low \(Re\) for \(Pr = 6\) and \(Pr = 12\). These are the cases with the dip in \(Nu\) in the mid-field wake. Three streamwise locations behind the bubble were identified, \(x_b = 1, 2, \text{ and } 5 \text{ mm}\). The first point is well inside the near field and the others are spaced out such that the last point is sufficiently outside of the near-field region. At each value of \(x_b\) there will be three locations in the span: the midplane, \(B\), of the wake located at \(y = B \approx 100 \text{ mm}\), and the lateral edges of the extent of the bubble, \(y = B - D_b/2\), and \(y = B + D_b/2\). A summary of these selected cases is shown in Table 4.1. For each case the \(u\)-, \(v\)- and \(w\)-velocity components and the
temperature measurements are graphed and discussed. The ‘R’, ‘B’ and ‘G’ designations in the graphs have been used to represent the three spanwise locations. ‘B’ corresponds to the mid-line \( y = B \), ‘R’ is at \( y = B - D_b/2 \) and ‘G’ is at \( y = B + D_b/2 \) as indicated in Fig. 4.10.

**Table 4.1.** Summary of the Near Field Cases Studied

<table>
<thead>
<tr>
<th>Nominal ( D_b/H )</th>
<th>( Re )</th>
<th>Average Liquid Speed (mm/s)</th>
<th>( Pr )</th>
<th>( x_b ) (mm)</th>
<th>Offset from midline (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.3</td>
<td>169</td>
<td>27</td>
<td>6</td>
<td>1, 2, 5</td>
<td>(-D_b/2, 0, +D_b/2)</td>
</tr>
<tr>
<td>3.3</td>
<td>169</td>
<td>27</td>
<td>12</td>
<td>1, 2, 5</td>
<td>(-D_b/2, 0, +D_b/2)</td>
</tr>
</tbody>
</table>

*Case of \( D_b/H = 3.3, Re = 169, and Pr = 6 \)*

Figures 4.11 through 4.13 shows the velocity components and the corresponding temperature profiles at the three \( x_b \) locations, respectively. It is helpful to compare each component across the three figures as well as component-to-component within one figure. The \( u \)-velocity plot in Fig. 4.11 shows that at \( x_b = 1 \) mm \((x_b/D_b = 0.30)\), the streamwise velocity at the centerline behind the bubble is well-mixed and almost stagnate in the laboratory coordinates \((-27 \text{ mm/s} \text{ on the graph})\). The \( u \)-velocity at the wake extremes, \( R \) and \( G \), is near the expected parabolic profile. The \( v \)-velocity components, at \( R \) and \( G \), a bubble radius away from the midline, provide an approximately mirrored image of each other as seen in Fig. 4.11 (b). The oscillation across \( v = 0 \) in each curve indicates a pair of vortical structures oriented with axes in the \( x \)-direction such that they create a common-flow inward toward the wake centerline immediately below the heated surface \((z = 1.25 \text{ \)}
mm) and near the channel bottom \((z = 0)\). There is a common-flow diverging from the wake centerline near the midline of the channel height. The \(v\)-velocity at the wake centerline location is likely to be instantaneously reflective of one side or the other, in this moment, the \(R\) curve. If the source of the common flow into the wake is cooler fluid from the center of the channel, then this graph may indicate a heat transfer enhancement mechanism not identified by Willard. The \(w\)-velocity component indicates a small velocity away from the heated plate (negative \(w\)) and a small positive velocity near the lower surface. The \(w\)-response is consistent with the pair of vortices indicated in the \(v\) curves that sweep fluid past the upper surface and then create a common flow down (negative \(w\)) near the upper surface at the wake centerline and a common-flow up near the lower wall. The temperature profile across the channel, Fig. 4.11 \((d)\), at the \(B\) position is consistent with a well-mixed fluid column creating a sharp temperature gradient near the upper heated wall. This is consistent with the well-mixed \(u\) profile in Fig. 4.11 \((a)\). The high slope at the upper wall is consistent with a large heat transfer coefficient at the wake centerline. The flanking positions show some departure from the expected channel-flow shape but also show the lower slopes at the upper wall indicative of a lower heat transfer rate.

The profiles of the velocity components and the temperature at \(x_b = 2\) \(mm\) \((x_b/D_b = 0.60)\), are shown in Fig. 4.12. Figure 4.12 \((a)\) shows that the \(u\)-velocity component is more disturbed at the lateral extents of the wake, but beginning recovery at the centerline. The temperature curves in Fig. 4.12 \((d)\) show a rather surprising degree of recovery for the \(B\) curve at the centerline; however, the slope at the upper heated wall has increased at the lateral extents. Both findings indicate that the momentum and thermal wakes are now wider (in \(y\)) at these positions. The \(v\)- and \(w\)-velocity components now
indicate a different story than at the $x = 1 \text{ mm}$ location. The $v$ curves indicate a general
divergence (negative velocity at $R$ and positive at $G$) from the wake centerline. The $w$
curves now show a velocity upward and against the heated upper surface at high values of
$z$, and downward near the lower surface. These curves are consistent with Willard’s
findings of a pair of columnar vortices aligned in $z$ and rotating so that fluid is pushed into
the wake (negative $x$) and away from the bubble. We can note that the $u$-velocity is lower
(larger negative values) at $B$ than at the lateral positions. He also found this motion to be
associated with positive $w$ near the upper surface as fluid was pushed toward the upper
surface. A pair of columnar vortices would produce a $v$-component flow away from the
centerline if they were situated between the rear surface of the bubble and the measurement
location.

The profiles of the velocity components and the temperature at $x_b = 5 \text{ mm}$
($x_b/D_b = 1.51$), are shown in Fig. 4.13. For these locations, the $u$-component of the
velocity has recovered to channel-flow parabolic, and the $v$ and $w$ components are
approximately zero. This clearly indicates that the momentum field has recovered from
the passage of the bubble by 1.5 bubble diameters from the trailing edge of the bubble.
Therefore, the momentum transport and the corresponding thermal response assumed to be
present in the near-field of the bubble has ended – a finding consistent with the divisions
proposed in the previous section ($0 < x_b < 3 \text{ mm}$ for the near-field). The temperature curves
show a lower temperature at the center of the channel than at the lower adiabatic surface.
This finding indicates that the temperature field (at $Pr = 6$) has yet to recover as we would
expect from Willard’s documentation of a long thermal wake that (at $Pr = 12$) extends far
beyond the extent of the computational domain.
Figure 4.11. Velocity and Temperature Profiles through the depth of the channel at $x_b = 1\ mm$. (a), (b) and (c) are the $u$-, $v$- and $w$-velocity components respectively, while (d) is the temperature profile.

Case of Case of $D_b/H = 3.3$, $Re = 169$, and $Pr = 6$. 
Figure 4.12. Velocity and Temperature Profiles through the depth of the channel at $x_b = 2$ mm. (a), (b) and (c) are the $u$-, $v$- and $w$-components of velocity, respectively, (d) is the temperature profile. Case of Case of $D_b/H = 3.3$, $Re = 169$, and $Pr = 6$. 
Figure 4.13. Velocity and Temperature Profiles through the depth of the channel at $x_b = 5$ mm. (a), (b) and (c) are the $u$-, $v$- and $w$-velocity, respectively, (d) is the temperature profile.
Case of \(D_b/H = 3.3, \ Re = 169, \ and \ Pr = 12\)

This section provides results of a form similar to those of the \(Pr = 6\) in the previous section. Figures 4.14 through 4.16 shows the velocity components and the corresponding temperature profiles at the three \(x_b\) locations. The elements that have changed are (1) \(Pr = 12\), (2) this is a different computational run of the same bubble volume, and (3) the data are extracted at a different computational time and therefore at a different position in the channel. Elements (1) and (3) are important in that the values of the temperature at the wall and through the fluid will be different than for the run at \(Pr = 6\). None of these elements should create a difference in the momentum field except for moment-to-moment variations expected of this unsteady flow.

The velocity component plots in Fig. 4.14 are similar, as expected, to those in Fig. 4.11 at \(x_b = 1 \ mm (x_b/D_b = 0.30)\). The \(u\)-velocity repeats quite well, and the \(v\)- and \(w\)-components, while not replicating values, again support the fluid structure argument made for the previous case. The temperature profile across the channel, Fig. 4.14 \((d)\), shows even sharper gradients in all three profiles and higher temperature differences from the upper wall to the channel bulk flow. Both of these differences are expected of the lower fluid conductivity used to create the higher \(Pr\).

The profiles of the velocity components and the temperature at \(x_b = 2 \ mm (x_b/D_b = 0.60)\), are shown in Fig. 4.15. Again, the velocity profiles are similar to the previous case at these positions, and the same vortical-structure arguments are supported. The temperature curves in Fig. 4.15 \((d)\) show a somewhat larger variation across the wake than in Figure 4.12 \((d)\). This is expected as the driving temperature difference is larger, and the thermal wake recovers more slowly (less recovery at a given value of \(x_b\)) at the
higher $Pr$. The profiles of the velocity components and the temperature at $x_b = 5 \, mm$ ($x_b/D_b = 1.51$), are shown in Fig. 4.16. Again, the momentum field has largely recovered to channel-flow values, but the temperature field remains “noisy” with the highest near-wall gradient seen at the wake centerline (curve $B$). These findings are consistent with Willard’s documentation of a long thermal wake that at $Pr = 12$ which extends far beyond the extent of the computational domain.

From this exercise, we may conclude the following:

(1) An investigation of the velocity components at nine positions in the rear wake supports the idea that for the largest bubble in this study, the “near-field” of the wake, where dynamical structures rearrange the momentum and thermal fields, terminates between $x = 2 \, mm$ and $x = 5 \, mm$.

(2) The data support the idea of a complex system of dynamical structures very near the bubble surface. It can be argued that these structures operate to enhance the heat transfer rate in the near field of the rear wake. The data at $x = 2 \, mm$ support the $z$-axis columnar structures discussed as a dominate feature by Willard.

(3) The momentum field returns to channel-flow values quickly ($x_b/D_b \leq 1.5$), but the diffusion of thermal energy across the channel is a much slower process for the $Pr$ simulated.
Figure 4.14. Velocity and Temperature Profiles through the depth of the channel at $x_b = 1 \text{ mm}$. (a), (b) and (c) are the $u$, $v$- and $w$-components of velocity, respectively, (d) is the temperature profile.

Case of $D_b/H = 3.3$, $Re = 169$, and $Pr = 12$. 

68
Figure 4.15. Velocity and Temperature Profiles through the depth of the channel at $x_h = 2$ mm. (a), (b) and (c) are the $u$-, $v$- and $w$-velocity components respectively, (d) is the temperature profile. Case of Case of $D_d/H = 3.3$, $Re = 169$, and $Pr = 12$. 
Figure 4.16. Velocity and Temperature Profiles through the depth of the channel at $x_h = 5 \text{ mm}$. (a), (b) and (c) are the $u$-, $v$- and $w$-velocity components respectively, (d) is the temperature profile. Case of Case of $D_h/H = 3.3$, $Re = 169$, and $Pr = 12$. 
Near-field Nusselt Number Correlation

Willard (2017) proposed a dimensionless group (Eq. 4.9) to collapse the $Nu$ data in the rear wake. Figure 4.6 shows that the collapse is a generally reasonable one for all three $Pr$ studied, but it does not work well in the near field. This is not surprising as it is based on laminar diffusion in a developing channel flow and does not account for the presence of momentum effects in the near-field region. This section proposes a new near-field $Nu$ correlation. This region was defined based on the observation of $h_x$ very close to the bubble. Fig. 4.17 shows the behavior of $h_x$ versus $x_b/D_b$ in the bubble wake. For $x_b/D_b \leq 1.5$, the data do not collapse in Fig. 4.17 or in the near field of Fig. 4.6 when plotted against $x^*$. However, in Fig. 4.6 the data do collapse well in the mid- and far-fields which are generally located at $x_b/D_b \geq 1.5$ in Fig. 4.17.

Figure 4.17 Convection coefficient vs $x_b/D_b$ behind bubble for all three $Pr$. 

71
The region \(x_b/D_b \leq 1.5\) in Fig. 4.17 was selected as the near-field for the purpose of creating a near-field correlation. A number of approaches were considered. The most effective approach was one where we assume \(Nu_x\) to be a function of \(Pr\), confinement in the form of \(D_b/H\), and an \(x_b\)-based Reynolds number that uses the relative velocity between the bubble and the centerline channel velocity,

\[
Nu_x = 0.25 Pr^{0.5} \left(\frac{D_b}{H}\right)^{0.25} Re_{xb}^{0.66}
\]

Eq. 4.11

where the Reynolds number, \(Re_{xb}\), is based on the velocity difference between the undisturbed liquid centerline velocity, \(u_{cl} = 1.5 u_t\), and the bubble velocity, \(u_b\), and the distance behind the bubble, \(x_b\),

\[
Re_{xb} = \frac{(u_{cl} - u_b) x_b}{v}.
\]

Eq. 4.12

A fit to the data in the range of \(x_b/D_b \leq 1.5\), or \(x_b\) typically less than 5 mm, for each bubble across all three \(Pr\) gave the leading constant. The result is shown in Fig. 4.17. The collapse to this power-law formulation is shown in Table 4.2. Percent deviations are quoted for the span of the entire grouping. Note that the correlation collapses data across all \(Pr\) to very low \(x_b\) including the \(Pr = 1\) data. The worst agreement is at \(Re_{xb} = 10\) where one of the \(Pr = 12\) datasets is 72% over the correlation value. The agreement for the other positions quoted is quite good.
Figure 4.18. \( Nu_x \) vs \( Re_{xb} \) in the near field region

Table 4.2. Deviation from Correlation in Figure 1.17

<table>
<thead>
<tr>
<th>( Re_{xb} )</th>
<th>2.0</th>
<th>10.0</th>
<th>50.0</th>
<th>80.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>% Deviation of lowest value</td>
<td>26%</td>
<td>13%</td>
<td>13%</td>
<td>20%</td>
</tr>
<tr>
<td>% Deviation of highest value</td>
<td>27%</td>
<td>72%</td>
<td>35%</td>
<td>23%</td>
</tr>
</tbody>
</table>

It is instructive to compare the resulting correlation, Eq. 4.11, with the \( Nu_x \) correlation for the standard laminar boundary layer developing on a uniform-flux surface,
\[ Nu_x = 0.453 \, Pr^{0.33} \, Re_{xb}^{0.50} \]  \hspace{1cm} \text{Eq. 4.13}

where \( x_b \) is taken to be the distance from the initiation of the boundary layer. Both correlations have \( x_b \) on both sides of the equation, given the definition of \( Nu_x \). In Eq. 4.11, the net dependence of the heat transfer coefficient, \( h \), on \( x_b \) is \( x_b^{-0.33} \), a slower decay than in Eq. 4.13, the standard laminar flat plate boundary layer \( (x_b^{-0.50}) \). This slower decay may well be expected given the presence of three-dimensional flow structures discussed in the previous section. Figure 4.18 shows a comparison of the resulting power-law curves for \( Pr = 6 \) and \( Pr = 12 \) for a bubble with \( D_b/H = 2 \). While the \( Nu_x \) values at a given \( Pr \) are similar in magnitude, the flow in the near field of these bubbles does produce a somewhat higher \( Nu_x \) than we see in laminar boundary layer flow.
Effect of changing the thermal boundary condition on the heated wall

The final investigation in this thesis concerns the effect on the rear wake heat transfer and temperature recovery due to a change in the thermal boundary condition imposed by the heated upper plate. The plate in this study, following the laboratory facility, is a 75 \( \mu m \)-thick nickel alloy foil. The concept in the selection of this surface was to provide a laboratory approximation to a uniform-flux surface. This laboratory approximation is achieved by selecting a thin surface of low thermal conductivity. The surface is required to be an electrical conductor with minimal change in conductivity with temperature, and to be stiff enough to handle without creasing. These five constraints lead to the 75 \( \mu m \)-thick nickel alloy foil. However, it was well-established by Ozer et al. (2010-2012) prior to the time that Willard began his simulations that the heat capacity of the foil surface was
allowing it to supply a very non-uniform heat flux in the high-$h$ near-field of the bubble. For this reason, Willard’s simulations, and those of the present study, model the thermal dynamics of the plate as part of the overall simulation. An item of interest left uninvestigated by Willard is the sensitivity of his results for $h(x)$ to a change in the plate construction – most particularly its replacement with a thinner plate.

In the present work, we avoid changing the computational mesh as required to install a thinner plate as this necessitates a major rebuild of the mesh. Instead, we investigate the outcome of changing two important parameters in the direction of a thinner plate. The first is lateral energy diffusion, the ability of the plate to supply energy from one location to neighboring locations where the $h$ is higher. Lateral diffusion scales with the conductivity-thickness product $(k\delta)$, so a reduction of conductivity, $k$, takes us in the direction of a reduction in thickness, $\delta$. The second is the ability of the plate to supply a large flux demand by depleting the storage of thermal energy $\frac{\partial}{\partial t} \left( \rho \delta AC_p T \right)$ at a location. That ability scales with the heat capacity of the plate and therefore the specific heat, $C_p$. A reduction in $C_p$ also takes us in the direction of a thinner plate. Both these properties appear in the thermal diffusivity, $\alpha \equiv k/\rho C_p$, the parameter that controls the rate at which a change in temperature may propagate through the material. Therefore, we can change the $k$ and $C_p$ so that to hold $\alpha$ constant or to change $\alpha$. We envision a plate of 1/3 the thickness of the actual plate, so we selectively reduce the $k$ and $C_p$ to 1/3 the actual values. For this exercise we use the base case without a bubble present at $Pr = 12$, and the case of $D_b/H = 2.0$ with $Pr = 6$ and $Pr = 12$. Table 4.3 lists the selections for which cases were run. The base values for the foil plate are $k = 8.9 \text{ W/mK}$, $C_p = 397 \text{ J/kg K}$, and $\alpha = 2.5 \times 10^{-6} \text{ m}^2/\text{s}$.
Table 4.3. Runs used to Investigate Boundary Condition Sensitivity

<table>
<thead>
<tr>
<th>Description</th>
<th>$Pr$</th>
<th>$k$</th>
<th>$C_p$</th>
<th>$\alpha$</th>
</tr>
</thead>
<tbody>
<tr>
<td>No bubble, base values</td>
<td>12</td>
<td>base</td>
<td>base</td>
<td>base</td>
</tr>
<tr>
<td>No bubble, base values/3</td>
<td>12</td>
<td>base/3</td>
<td>base/3</td>
<td>base</td>
</tr>
<tr>
<td>No bubble, $k/3$, base $C_p$</td>
<td>12</td>
<td>base/3</td>
<td>base</td>
<td>base/3</td>
</tr>
<tr>
<td>No bubble, $C_p/3$, base $k$</td>
<td>12</td>
<td>base</td>
<td>base/3</td>
<td>3-base</td>
</tr>
<tr>
<td>$D_b/H = 2.0$</td>
<td>12</td>
<td>base</td>
<td>base</td>
<td>base</td>
</tr>
<tr>
<td>$D_b/H = 2.0$</td>
<td>6</td>
<td>base</td>
<td>base</td>
<td>base</td>
</tr>
<tr>
<td>$D_b/H = 2.0$</td>
<td>12</td>
<td>base/3</td>
<td>base/3</td>
<td>base</td>
</tr>
<tr>
<td>$D_b/H = 2.0$</td>
<td>6</td>
<td>base/3</td>
<td>base/3</td>
<td>base</td>
</tr>
</tbody>
</table>

First, we establish that changing the plate properties did not affect the ability to compute an undisturbed channel flow, and that the correct properties (in this case $k$ in $Nu$) is being used in the analysis. Figure 4.19 shows the foil temperature computed for $Pr = 12$ and $Re = 169$ for the base and the altered foil properties with no bubble present. Figure 4.20 show the $Nu$ for those cases. In both cases, the data were acquired along a line of cells at $y = 100 \text{ mm}$ and averaged over three instances in time. The wall temperature tracks the original base case for the altered properties. There is a small dip in $Nu$ with lower plate conductivity immediately adjacent to the boundary condition upstream of the bubble, otherwise all the baseline simulations agree with theory.

In Figs. 21 through 23, the baseline data were averaged in $y$ for rows within $\pm D/2$ of the centerline of the wake and averaged over three instances in time. This is the procedure used to present the $Nu$ data discussed earlier in the chapter. The curves with the altered foil properties were spatially averaged but are single-time realizations. Figure 4.21 shows the driving temperature difference, $T_w - T_b$, vs. $x_b$ for base and altered foil material.
properties. Particularly, the case of $D_b/H = 2.0$, $Re = 169$ and $k$ and $Cp$ reduced by a factor of three. This case preserves the base value of the foil diffusivity. Results are shown for both fluid $Pr$. As would be expected, the “thermally thinner” foil sees a deeper temperature reduction directly behind the bubble where the heat transfer coefficient is high. The effect is large: $1/3$ or more of the base temperature difference. The energy demand must be satisfied by a larger temperature drop due to the lower specific heat, and the reduced conductivity restricts the ability of lateral heat flow within the plate to replace that debt at any streamwise location within the wake. The result is two-fold: a larger initial drop in foil temperature and a slower recovery with increasing $x_b$. The effect of fluid $Pr$ is also to lengthen the thermal wake, and so the $Pr = 12$ case sees this initial deeper drop sustained for a longer distance than in the $Pr = 6$ case. The $Pr = 6$ case shows an interesting behavior: by $x_b \approx 11$, the initial temperature drop has resolved into an exceedingly slow recovery that approximately parallels the base case for that $Pr$. We can attribute this to the greatly reduced lateral heat flow inside the altered foil. Figure 4.22 shows the heat flux for these cases graphed against $x_b$. For both $Pr$, we see reductions in heat flux for the altered foil properties. By approximately $x_b = 15 \text{ mm}$ all curves return to the background value supplied by the electric energy dissipation; however, the heat flux for both curves with the altered plate properties returns earlier. The heat flux for the altered properties and $Pr = 6$ returns to the background value first, again in keeping with reduced internal lateral heat flow. Finally, the $Nu$ versus $x_b$ for these cases is shown in Figure 4.23. We see that the reduction in the local driving temperature difference is generally offset by a reduction in the local heat flux so that the heat transfer coefficient and resulting $Nu$ are not greatly changed. The two $Pr = 12$ curves agree as well as could be expected for data from two
different runs and channel streamwise positions. The two $Pr = 6$ curves show a larger disagreement, and the curve for the altered foil properties lies above the base curve and does not show a convincing return to the channel baseline $Nu$ by the exit of the computational domain. This appears to be a systemic response to the boundary condition change at $Pr = 6$ and not entirely a difference between the base case and one randomly chosen realization in a separate run.

The overall conclusion to this exercise is that a factor of three reduction in the “thermal thickness” of the heated foil surface makes no quantitative difference in the $Nu$ response in the rear wake at $Pr = 12$, but may raise $Nu$ and significantly delay wake recovery at $Pr = 6$. There are large differences in the driving temperature difference and in the heat flux; these tend to be offsetting, but not entirely so.

![Figure 4.20. Heated foil temperature vs $x$ for $Pr = 12$ $Re = 169$ for base and altered foil properties with no bubble present.](image)
Figure 4.21. Nu vs x for Pr = 12 Re = 169 for base and altered foil properties with no bubble present.

Figure 4.22. $T_w - T_b$ vs. $x_b$ for base and altered foil material properties for $D_b/H = 2.0$, $Re = 169$, and $k$ and $C_p$ reduced by a factor of three.
Figure 4.23. Heat flux vs $x_b$ for base and altered foil material properties for $D_b/H = 2.0$, $Re = 169$, and $k$ and $C_p$ reduced by a factor of three.

Figure 4.24. $Nu$ vs $x_b$ for base and altered foil material properties for $D_b/H = 2.0$, $Re = 169$, and $k$ and $C_p$ reduced by a factor of three.
Chapter 5

Summary and Conclusions

Presented in this work is the numerical study of the effect of Prandtl number on heat transfer in the wake of a highly confined bubble in a large-aspect-ratio minichannel. The bubble traverses a horizontal channel whose primary phases of the fluid domain is an engineered fluid, Novec-649, while the secondary phase is an atmospheric air bubble. The two phases are allowed to interact with no phase change. The upper surface of the channel is comprised of a nickel alloy foil with uniform internal energy generation via ohmic heating. The lower surface is adiabatic.

This study is achieved through a direct numerical simulation of an unsteady low-Reynolds number model problem to increase the parameter space of the previous study (Willard, 2017) by introducing simulations with $Pr = 6$. The simulations are performed using the commercial finite-volume solver in the software ANSYS Fluent. The simulation solves the unsteady momentum and temperature fields in the fluid and in the heated metal foil as a single bubble traverses the rectangular minichannel. To achieve this, the liquid vapor-interface was tracked using the Volume-of-Fluid method native to the solver. The model is built using a moving reference frame, also known as a Lagrangian formulation, that moves in the streamwise direction at the average bubble speed so as to hold the bubble approximately stationary during the computation, while the channel translates past the
bubble at a speed equivalent to the computed average bubble speed. The consequence of this reference frame is the ability to model a channel with an arbitrary length while using a relatively small computational domain and computer resources.

The study expanded the existing parameter space by analyzing the results of a simulation for a liquid $Pr = 6$, for the two $Re$ and the three bubble volumes used in the previous study. An improved and expanded characterization of the near-field region was done through the introduction of a new near-field $Nu$ correlation, and also by analysis of the local behavior of the velocity components and fluid temperature at key locations in the rear wake. A significant change in the thermal boundary condition imposed by the heated upper wall was simulated to evaluate the effect on the $Nu$ in the wake due to a large reduction in the heat capacity and thermal conductivity of the upper wall.

Conclusions

The key results and conclusions of this work include;

- One complete simulation group at liquid $Pr = 6$ was done to provide a bridge between the values of $Pr = 12$ and $Pr = 1$ that were done in the initial work. The group included three bubble volumes and two $Re$ as in the initial work. The resulting momentum and were analyzed to document the thermal response in the rear wake for each run. The preceding study had found a collapse of $Nu$ in the mid and far wake to a dimensionless streamwise position, $x^*$. However, the wake in $Pr = 1$ data recovered so rapidly that it was questionable as to the support that data set provided the proposed $x^*$. The new $Pr = 6$ data supports the proposed $Nu = Nu(x^*)$, 

and supplies a much-needed bridging data set with a similar development in $x^*$ such that the three Prandtl number cases are consistent.

- An investigation of the velocity components at nine positions in the rear wake supports the idea that for the largest bubble in this study, the “near-field” of the wake, where dynamical structures rearrange the momentum and thermal fields, terminates between $x = 2 \text{ mm}$ and $x = 5 \text{ mm}$. The momentum field returns to channel-flow values quickly ($x_b/D_b \leq 1.5$), but the diffusion of thermal energy across the channel is a much slower process for the two $Pr$ investigated. The data support the idea of a complex system of dynamical structures very near the bubble surface. It can be argued that these structures operate to enhance the heat transfer rate in the near field of the rear wake. The data at $x = 2 \text{ mm}$ support the $z$-axis columnar structures discussed as a dominate feature by Willard.

- Changing the thermal boundary condition created by the heated foil upper channel surface to a thermally thinner surface makes no quantitative difference in the $Nu$ response in the rear wake at $Pr = 12$, but may raise $Nu$ and significantly delay wake recovery at $Pr = 6$. There are large differences in the driving temperature difference and in the heat flux, but these appear to be offsetting for $Pr = 12$ but less so for $Pr = 6$. The altered foil properties (reductions in specific heat and conductivity by a factor of three) produce a surface that is closer to the ideal of a uniform flux, but we see that the heat transfer mechanisms very near the bubble are still capable of producing an order of magnitude excursion in heat flux.
Recommendations for Future Work

- The time dependent data of the near field can be further analyzed with techniques not attempted in this work. This would provide more insight into the operation and stability of the fluid structures in the near field region of the fluid domain.

- A follow-on study can be done for a full simulation of a similar model with a thinner plate. This will be useful in providing a more conclusive insight on the effect of such a model, with the actual geometric dimensions altered while keeping the material properties constant.

- The current study still focused on the modelling of a single bubble in the fluid domain. Future studies can attempt to investigate the interactions of a number of bubbles and the corresponding heat transfer enhancement.

- The present simulation models a single bubble in a horizontal channel that is pressure-driven with no phase change taking place. The most important, and quite substantial challenge, will be the inclusion of phase change into the model.
Appendix A

Material Properties

Table A.1. Thermophysical Properties of Materials

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Hastealloy® C-276</th>
<th>3M™ Novec-649</th>
<th>Atmospheric Air</th>
</tr>
</thead>
<tbody>
<tr>
<td>Absolute Viscosity $(\mu) \times 10^{-4} \text{ kg/m-s}$</td>
<td>N/A</td>
<td>6.4</td>
<td>0.179</td>
</tr>
<tr>
<td>Density $(\rho) \text{ kg/m}^3$</td>
<td>8970</td>
<td>1600</td>
<td>1.225</td>
</tr>
<tr>
<td>Kinematic Viscosity $(\nu) \times 10^{-7} \text{ m}^2/\text{s}$</td>
<td>N/A</td>
<td>4.0</td>
<td>146.1</td>
</tr>
<tr>
<td>Specific Heat Capacity $(Cp) \text{ J/kg-K}$</td>
<td>397</td>
<td>1103</td>
<td>1006</td>
</tr>
<tr>
<td>Surface Tension $(\sigma) \text{ N/m}$</td>
<td>N/A</td>
<td>0.0108 (with Air)</td>
<td>N/A</td>
</tr>
<tr>
<td>Thermal Conductivity $(k) \text{ W/m-K}$</td>
<td>8.9</td>
<td>0.706 ($Pr = 1$)</td>
<td>0.0242</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.118 ($Pr = 6$)</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.059 ($Pr = 12$)</td>
<td></td>
</tr>
</tbody>
</table>
### Table B.1. Table of Thermal Boundary Conditions taken from Willard (2017)

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Plate Leading Edge</strong></td>
<td>( T(t) = T_{ini} + \left{ \frac{q^*}{p_f C_{pf} u_l H} \right} * (L + u_p t) )</td>
</tr>
<tr>
<td><strong>Plate Following Edge</strong></td>
<td>( \frac{\partial T}{\partial x_{x=0}} = 0 )</td>
</tr>
<tr>
<td><strong>Plate Side Walls</strong></td>
<td>( \frac{\partial T}{\partial y_{y=0,w}} = 0 )</td>
</tr>
<tr>
<td><strong>Top of Plate</strong></td>
<td>( \frac{\partial T}{\partial z_{z=H+t_p}} = 0 )</td>
</tr>
<tr>
<td>Plate Fluid Interface</td>
<td>$T_{\text{plate}} = T_{\text{fluid}}$</td>
</tr>
<tr>
<td>-----------------------</td>
<td>--------------------------------------</td>
</tr>
<tr>
<td>Channel Side Walls</td>
<td>$\frac{\partial T}{\partial y,_{y=0,w}} = 0$</td>
</tr>
<tr>
<td>Channel Leading Edge</td>
<td>$T(t, z) = T_{\text{ini}} + \left{ \left( \frac{q_w}{\rho_f C_{pf}} \frac{u_H}{H} \right) + \left( L + u_p t \right) \right}$ $- \left{ \left( \frac{q_w D_H}{2k_f} \right) \right} ^ {\frac{1}{2}} \left( 1 - \frac{z}{H} \right) ^ 4 - \left( 1 - \frac{z}{H} \right) ^ 3 + \left( 1 - \frac{z}{H} \right) \right}$</td>
</tr>
<tr>
<td>Channel Following Edge</td>
<td>$T(y, t, z) = T_{\text{wall}}(y)$ $+ \left{ \left( \frac{q_w}{\rho_f C_{pf}} \frac{u_H}{H} \right) \right} \left( L + u_p t \right) \right} $ $- \left{ \left( \frac{q_w D_H}{2k_f} \right) \right} ^ {\frac{1}{2}} \left( 1 - \frac{z}{H} \right) ^ 4 - \left( 1 - \frac{z}{H} \right) ^ 3 + \left( 1 - \frac{z}{H} \right) \right}$</td>
</tr>
<tr>
<td>Channel Bottom</td>
<td></td>
</tr>
<tr>
<td>----------------</td>
<td></td>
</tr>
<tr>
<td><img src="image1.png" alt="Diagram" /></td>
<td></td>
</tr>
<tr>
<td>$\frac{\partial T}{\partial z_{x=0}} = 0$</td>
<td></td>
</tr>
</tbody>
</table>
Table B.2. Table of Flow Boundary Conditions taken from Willard (2017)

<table>
<thead>
<tr>
<th>Condition</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Channel Leading Edge</strong></td>
<td>( u(z) = 6 , u_l \left( \frac{z}{H} \right) - \left( \frac{z}{H} \right)^2 - u_p )</td>
</tr>
<tr>
<td><strong>Channel Side Walls</strong></td>
<td>( \frac{\partial \tau}{\partial y}_{y=0,W} = 0 )</td>
</tr>
<tr>
<td><strong>Channel Following Edge</strong></td>
<td>( u(z) = 6 , u_l \left( \frac{z}{H} \right) - \left( \frac{z}{H} \right)^2 - u_p )</td>
</tr>
<tr>
<td><strong>Channel Bottom</strong></td>
<td>( u(z = 0) = -u_p )</td>
</tr>
<tr>
<td><strong>Plate Fluid Interface</strong></td>
<td>( u(z = H) = -u_p )</td>
</tr>
</tbody>
</table>
Appendix C

Summary of Solver Settings

The following is a summary list of solver settings used within Ansys Fluent to produce the simulations in this work. This list outlines the non-standard settings used by the solver. This listing is taken from Willard (2017).

- **Multi-Phase Model Settings**
  - Volume of Fluid Method
    - Explicit Formulation
    - Sharp Interface Modeling
    - Implicit Body Force Formulation
    - Volume Fraction Cutoff = $10^{-6}$
    - Sub-Timestep CFL = 0.25
    - Continuum Surface Force Surface tension
    - Full Geometric Reconstruction (Modified PLIC) of interface

- **Boundary and Cell Zone Settings**
  - Plate Domain
    - Volumetric Energy Generation Source Term
    - Moving Reference Frame Formulation
      - Appropriate Lagrangian Velocity for bubble being simulated
  - Plate – Fluid Interface
    - Node Matched Boundary
    - Thermal Coupling enabled

- **Discretization and solver algorithm**
  - SIMPLE algorithm for pressure-velocity coupling
  - Least Squares Cell Based Gradient Discretization
  - PRESTO Pressure Discretization
o Second Order Upwind Momentum Discretization
o Second Order Upwind Energy Discretization
o First Order Implicit Transient Time Stepping
o Adaptive Time Stepping
  ▪ Flow CFL max = 1
Listed below is a set of user defined functions developed by Willard (2017) that used to set the Fluent solver to run. The implementation and initialization of these boundary conditions is discussed in Chapter 3.

```c
#include "udf.h"

// Channel Dimensional Constants
#define FLOW_CHANNEL_HEIGHT 0.00125
#define FLOW_CHANNEL_WIDTH 0.02
#define FLOW_CHANNEL_LENGTH 0.03
#define PLATE_THICKNESS 0.000075
#define MEAN_FLOW_VELOCITY 0.027
#define LAGRANGIAN_VELOCITY 0.029
#define CELL_SIZE_X 0.000125
#define CELL_SIZE_Y 0.000125
#define CELL_SIZE_Z 0.00003125

// Channel Thermal Constants
#define PLATE_INITIAL_TEMPERATURE 327.0
#define PLATE_OVERALL_NUSSELT_NUMBER 5.4
#define PLATE_HEAT_FLUX 1920

// Material Property Constants
#define LIQUID_SPECIFIC_HEAT 1103.0
#define LIQUID_THERMAL_CONDUCTIVITY 0.059
#define LIQUID_DENSITY 1600.0
#define LIQUID_PRANDTL 12.0

// Global Variables
int previousTimeStep = 0;
int NUMBER_CELLS_X = FLOW_CHANNEL_LENGTH / CELL_SIZE_X;
int NUMBER_CELLS_Y = FLOW_CHANNEL_WIDTH / CELL_SIZE_Y;
```
int NUMBER CELLS Z = FLOW CHANNEL HEIGHT / CELL SIZE Z;

int have wall Temperature = 0;

architecture Velocity Boundaries Leading and Following
define profile (Boundary VelocityProfile, t, i)
{
    real position[ND ND]; /* holds the position vector */
    real x; /* holds x-location */
    real y; /* holds y-location */
    real z; /* holds z-location */
    real time; /* holds flow time */
    real velocity; /* holds centroid velocity */
    real hstar; /* holds non-dimensional z-location */
    face_t f; /* holds current boundary face */
    time = CURRENT TIME; /* gets flow time from solver */

begin_f_loop(f, t) /* begins loop of all faces in boundary */
{
    F CENTROID(position, f, t); /* gets face location from solver */
    x = position[0];
    y = position[1];
    z = position[2];
    hstar = (z / FLOW CHANNEL HEIGHT); /* calculates non-dimensional height */
    velocity = (6 * MEAN FLOW VELOCITY * (hstar - (hstar * hstar))) - LAGRANGIAN VELOCITY; /* calculates centroid velocity for given location */
    F PROFILE(f, t, i) = velocity; /* returns value to solver */
}
end_f_loop(f, t)

architecture Temperature Boundary for Leading Edge (Precursor)
define profile (Boundary TemperatureProfile, t, i)
{
    real position[ND ND]; /* holds the position vector */
    real x; /* holds x-location */
    real y; /* holds y-location */
    real z; /* holds z-location */
    real time; /* holds flow time */
    real temperature = 0.0; /* holds centroid temperature */
    real hstar temp; /* holds non-dimensional z-location */
    real hydraulic Diameter; /* holds hydraulic diameter */
    face_t f; /* holds current boundary face */
    time = CURRENT TIME; /* gets flow time from solver */

begin_f_loop(f, t) /* begins loop of all faces in boundary */
\{ 
  \text{F\_CENTROID}(position,f,t); 
  x = position[0] + (time*LAGRANGIAN\_VELOCITY); /* sets lagrangian position */ 
  y = position[1]; 
  z = position[2]; 
  hstar_temp = (1 - (z / FLOW\_CHANNEL\_HEIGHT)); /* calculates non-dimensional height */ 
  \text{hydraulicDiameter} = (2 * FLOW\_CHANNEL\_HEIGHT); /* calculates hydraulic diameter */ 
  \text{if}(hstar_temp > 0)\{ /* Checks if current location is in fluid or plate*/ 
    \text{temperature} = \text{PLATE\_INITIAL\_TEMPERATURE} + (((
        \text{PLATE\_HEAT\_FLUX} / (LIQUID\_DENSITY * LIQUID\_SPECIFIC\_HEAT * MEAN\_FLOW\_VELOCITY * FLOW\_CHANNEL\_HEIGHT)) * (x)) - ((\text{PLATE\_HEAT\_FLUX} * (\text{hydraulicDiameter} / 2)) / (LIQUID\_THERMAL\_CONDUCTIVITY)) * ((0.5 * (hstar_temp * hstar_temp * hstar_temp * hstar_temp)) - (hstar_temp * hstar_temp * hstar_temp) + (hstar_temp))); 
  \} 
  \text{else if}(hstar_temp <= 0)\{ /*if face is on the plate*/ 
    \text{temperature} = \text{PLATE\_INITIAL\_TEMPERATURE} + (\text{PLATE\_HEAT\_FLUX} / (LIQUID\_DENSITY * LIQUID\_SPECIFIC\_HEAT * MEAN\_FLOW\_VELOCITY * FLOW\_CHANNEL\_HEIGHT)) * x; 
  \} 
  \text{F\_PROFILE}(f,t,i) = \text{temperature}; /* returns value to solver */ 
\} 
end_f_loop(f,t) 
\} /* Temperature Boundary for Following Boundary (Curve Fit) */

\text{DEFINE\_PROFILE}(\text{Following\_TemperatureProfile},t,i) \{ 
  \text{real} \text{position[ND\_ND]}; /* holds the position vector */ 
  \text{real} \text{cellPosition[ND\_ND]}; /* Cell Position Vector*/ 
  \text{real} x; /* holds x-location */ 
  \text{real} y; /* holds y-location */ 
  \text{real} z; /* holds z-location */ 
  \text{real} \text{time}; /* holds flow time */ 
  \text{real} \text{temperature} = 0.0; /* holds centroid temperature */ 
  \text{real} hstar_temp; /* holds non-dimensional z-location */ 
  \text{real} \text{hydraulicDiameter}; /* holds hydraulic diameter */ 
  \text{int} \text{timeStepNumber} = 0; /* holds current time step */ 
  \text{real} \text{wallTemperature[NUMBER\_CELLS\_Y]}; /* holds array of wall temperature values*/ 
  \text{int} \text{validWallTemperature[NUMBER\_CELLS\_Y]}; /* holds array of flags denoting if solver has data for a cell*/ 
  \text{int} \text{iter}; /* holds iterator for loops */ 
\}
face_t f;  /* holds current boundary face */
cell_t c;  /* holds current cell */
Thread *c_thread;  /* cell thread pointer */
Domain *domain;  /* domain pointer */
time = CURRENT_TIME;  /* gets flow time from solver */
timeStepNumber = N_TIME;  /* gets time step from solver */

for(iter=0; iter<NUMBER_CELLS_Y; iter++){
    /* Initialize valid data flag array */
    validWallTemperature[iter] = 0;
}

if(timeStepNumber > 1){ /* check solver is past initialization */
    domain = Get_Domain(4);  /* assigns domain id for plate domain (4) */
    thread_loop_c(c_thread, domain){ /* begins loop of cell threads */
        begin_c_loop(c,c_thread){ /* begins loop of cells for thread */
            if(Data_Valid_P()){ /* asks solver if it has valid data */
                C_CENTROID(cellPosition,c,c_thread);  /* gets current cell location */
                if(cellPosition[0] <= ((0.0) + (CELL_SIZE_X/2.0)) &&
                    cellPosition[2] >= FLOW_CHANNEL_HEIGHT){ /* checks if cell is on following plate boundary */
                    int cellNumberY = (int)(cellPosition[1] / CELL_SIZE_Y);  /* calculate relative position of wall cell */
                    wallTemperature[cellNumberY] = C_T(c,c_thread);  /* add temperature value to array */
                    validWallTemperature[cellNumberY] = 1;  /* flag data as valid */
                }
            }
        }
    }
}

begin_f_loop(f,t){ /* begins loop of all faces in boundary */
    F_CENTROID(position,f,t);  /* get face location */
    x = position[0];
}
\[ y = \text{position}[1]; \]
\[ z = \text{position}[2]; \]

\[ \text{int cellNumberY} = (\text{int})(\text{position}[1] / \text{CELL\_SIZE\_Y}); \]
/* calculate relative position of boundary face */

\[ \text{hstar_temp} = (1 - (z / \text{FLOW\_CHANNEL\_HEIGHT})); \]
/* calculates non-dimensional height */

\[ \text{hydraulicDiameter} = (2 * \text{FLOW\_CHANNEL\_HEIGHT}); \]
/* calculates hydraulic diameter */

if( validWallTemperature[cellNumberY] == 1){
/* checks for valid flag */

\[ \text{real currentWallTemperature} = \text{wallTemperature}[\text{cellNumberY}]; \]
/* get temperature for current cell */

\[ \text{temperature} = \text{currentWallTemperature} - (((\text{PLATE\_HEAT\_FLUX} * \\
(\text{hydraulicDiameter} / 2)) / (\text{LIQUID\_THERMAL\_CONDUCTIVITY})) * \\
((0.5 * (\text{hstar_temp} * \text{hstar_temp} * \text{hstar_temp} * \\
\text{hstar_temp})) - (\text{hstar_temp} * \text{hstar_temp} * \text{hstar_temp}) + \\
(\text{hstar_temp}))); \]

} else{
/* lack of valid data reverts to precursor values */

if(\text{hstar_temp} > 0){
\[ \text{temperature} = \text{PLATE\_INITIAL\_TEMPERATURE} + (((\text{PLATE\_HEAT\_FLUX} * \\
(\text{MEAN\_FLOW\_VELOCITY} * \text{FLOW\_CHANNEL\_HEIGHT})) * (x)) - (((\text{PLATE\_HEAT\_FLUX} * \\
(\text{hydraulicDiameter} / 2)) / (\text{LIQUID\_THERMAL\_CONDUCTIVITY})) * \\
((0.5 * (\text{hstar_temp} * \text{hstar_temp} * \text{hstar_temp} * \\
\text{hstar_temp})) - (\text{hstar_temp} * \text{hstar_temp} * \text{hstar_temp}) + \\
(\text{hstar_temp}))); \]

} else if(\text{hstar_temp} \leq 0){
\[ \text{temperature} = \text{PLATE\_INITIAL\_TEMPERATURE} + ((\text{PLATE\_HEAT\_FLUX} / \\
(\text{LIQUID\_DENSITY} * \text{LIQUID\_SPECIFIC\_HEAT} * \\
\text{MEAN\_FLOW\_VELOCITY} * \text{FLOW\_CHANNEL\_HEIGHT})) * x) ; \]

}

\[ \text{F\_PROFILE}(f,t,i) = \text{temperature}; \]
/* returns value to solver */

} 

end_f_loop(f,t)
/********************************************************************************
Temperature and Velocity Initialization of Domains
***********************************************************************************/
DEFINE_INIT(Initialize_Domains, domain)
{
    Thread *t; /* holds cell thread pointer */
    Thread **pt; /* holds array of thread pointers */
    cell_t c; /* holds current cell */
    face_t f; /* holds current face */
    real position[ND_ND]; /* holds position vector */
    real x; /* holds x location */
    real y; /* holds y location */
    real z; /* holds z location */
    real temperature; /* holds temperature */
    real velocity; /* holds velocity */
    real hstar_temp; /* holds non-dimensional height */
    real hstar_veloc; /* holds non-dimensional height */
    real hydraulicDiameter = (2 * FLOW_CHANNEL_HEIGHT); /* holds and calculates hydraulic diameter */

    thread_loop_c (t,domain){
        /* begins loop of cell threads in domain*/
        if (!FLUID_THREAD_P(t)) /* checks if cells are solid */{ /* else */
            begin_c_loop_all(c,t){
                /* loop over all cells in thread */
                C_CENTROID(position,c,t);
                x = position[0];

                temperature = PLATE_INTIAL_TEMPERATURE +
                ((PLATE_HEAT_FLUX / (LIQUID_DENSITY *
                LIQUID_SPECIFIC_HEAT * MEAN_FLOW_VELOCITY
                * FLOW_CHANNEL_HEIGHT)) * x);

                C_T(c,t) = temperature;
            }
            end_c_loop_all(c,t)
            }
            else if (FLUID_THREAD_P(t)){ /* checks for fluid cells */
                begin_c_loop_all(c,t){
                    /* loop over all cells in thread */
                    C_CENTROID(position,c,t);
                    x = position[0];
                    z = position[2];
                    y = position[1];

                    Hstar_temp = (1 - (z / FLOW_CHANNEL_HEIGHT));
                    /* calculate non-dimensional height */
                    temperature = PLATE_INTIAL_TEMPERATURE +
                    ((PLATE_HEAT_FLUX / (LIQUID_DENSITY *
                    LIQUID_SPECIFIC_HEAT * MEAN_FLOW_VELOCITY
                    * FLOW_CHANNEL_HEIGHT)) * (x)) -
                    ((PLATE_HEAT_FLUX * (hydraulicDiameter /
                    2)) / (LIQUID_THERMAL_CONDUCTIVITY)) *
                    ((0.5 * (hstar_temp * hstar_temp *
                    hstar_temp * hstar_temp)) - (hstar_temp *
                    hstar_temp * hstar_temp)) - (hstar_temp *
                    hstar_temp * hstar_temp))

            } /* else */
        } /* else */
    } /* thread_loop_c */
hstar_temp * hstar_temp) +
(hstar_temp));

C_T(c,t) = temperature; /* set temperature */

hstar_veloc = (z / FLOW_CHANNEL_HEIGHT);
/* calculate non-dimensional height */

velocity = (6 * MEAN_FLOW_VELOCITY * (
    hstar_veloc - ( hstar_veloc * hstar_veloc )
)) - LAGRANGIAN_VELOCITY;

C_U(c,t) = velocity; /* set velocity */
}
end_c_loop_all(c,t)
}
References


Oncel, A. F., (2011). *Heat Transfer Caused by the Controlled Production of Sliding Vapor Bubbles in a Laminar Subcooled Flow in a Narrow Channel*. (PhD Dissertation), University of Houston, Houston, Texas, USA.


