Copyright and use of this thesis

This thesis must be used in accordance with the provisions of the Copyright Act 1968.

Reproduction of material protected by copyright may be an infringement of copyright and copyright owners may be entitled to take legal action against persons who infringe their copyright.

Section 51 (2) of the Copyright Act permits an authorized officer of a university library or archives to provide a copy (by communication or otherwise) of an unpublished thesis kept in the library or archives, to a person who satisfies the authorized officer that he or she requires the reproduction for the purposes of research or study.

The Copyright Act grants the creator of a work a number of moral rights, specifically the right of attribution, the right against false attribution and the right of integrity.

You may infringe the author's moral rights if you:

- fail to acknowledge the author of this thesis if you quote sections from the work
- attribute this thesis to another author
- subject this thesis to derogatory treatment which may prejudice the author's reputation

For further information contact the University's Copyright Service.
sydney.edu.au/copyright
CFD SIMULATION OF TAYLOR FLOW IN MICROCHANNELS: METHOD DEVELOPMENT AND VALIDATION

A thesis submitted in fulfilment of the requirements for the degree of MASTER OF PHILOSOPHY

By

Azadeh Najafi Asadollahi

March 2012
Declaration

I hereby declare that the work presented in this thesis is solely my own work and that, to the best of my knowledge, the work is original except where otherwise indicated by reference to other authors. No part of this work has been submitted for any other degree or diploma.

Azadeh Najafi Asadollahi
March, 2012
Abstract

In the past several decades, two-phase gas-liquid flow in micro-devices have been used in a large number of engineering applications, such as high heat-flux compact heat exchangers, catalyst coating in capillaries, cooling systems in micro-electronics or supercomputers. This wide range of applications has generated a large number of studies of the flow and heat transfer characteristics for gas-liquid flow in microchannels. Taylor flow is a dominant gas-liquid flow regime which occurs in a wide range of operating conditions in microchannels. It consists of elongated, bullet-shaped gas bubbles, separated by liquid slugs. The gas bubble occupies most of the tube cross-section and there is only a thin liquid film connecting two liquid slugs. The complex flow structure and heat transfer behaviour of Taylor flow in microchannels presents a computational challenge to comprehensively understand this flow. This thesis makes an important contribution to the mentioned area by using computational fluid dynamics (CFD) to model Taylor flow.

Taylor flow has been modelled in the literature applying different computational approaches. Fully-developed Taylor flow can be modelled as an unsteady, periodic flow in the laboratory frame of reference, but is steady in a frame of reference moving with the bubble. Recently several researchers have used the laboratory frame of reference methodology to simulate Taylor flow, but the gas bubble and liquid slug lengths have not been controlled. It is now understood that the effect of the liquid slug length and gas hold-up on the heat transfer in Taylor flow is essential and significant. Moreover, some researchers have modelled fully-developed Taylor flow in a single unit cell in a reference frame moving with the bubble velocity. These studies have been focused only on the hydrodynamics of fully-developed Taylor flow and there is no heat transfer investigation using this methodology. Therefore, in this work two approaches (defined as the stationary domain method and moving domain method) are developed to generate gas bubbles and liquid slugs with controlled lengths to allow a direct comparison between the simulation results and experimental data for both the hydrodynamics and heat transfer in Taylor
flow. In both methodologies the ANSYS Fluent software is used for the CFD modelling of Taylor flow and the volume of fluid (VOF) method is applied for capturing the gas-liquid interface.

Two techniques are used for the modelling of the flow field and heat transfer for a case with a Reynolds number of 713, Capillary number of 0.004 and void fraction of 0.366 for water/nitrogen flow and the results obtained for both methods were compared. The hydrodynamics results including the gas bubble and liquid slug lengths, the film thicknesses and the bubble velocities were in very good agreement. There were only small differences between the wall shear stresses obtained from the two methods in the bubble film region and the liquid slug region, which can be explained by the velocity gradients not being wrapped in the moving domain method. However, the average Nusselt number obtained for the stationary domain ($N_{uav} = 11.4$) was larger than that of the moving domain ($N_{uav} = 10.8$) due to thermally fully-developed flow not being obtained even after the generation of 14 bubbles in the stationary domain method. The required computational mesh size and time for the moving domain method (38 hours) was much less than that of the stationary domain method (1460 hours). Overall, the hydrodynamic and heat transfer results obtained from the two approaches were found to be very similar, giving confidence in the implementation of both methods.

Fully-developed flow and heat transfer results obtained from the simulations using the moving domain method were compared with the experimental data of Leung et al. (2010, 2012) for water/nitrogen and ethylene-glycol/nitrogen systems. The cases simulated covered a wide Reynolds number range of 475-1189 (water/nitrogen), 22-40 (ethylene-glycol/nitrogen) and a Capillary number range of 0.0026-0.0066 (water/nitrogen), 0.0850-0.1570 (ethylene-glycol/nitrogen).

For the two systems studied, water/nitrogen and ethylene-glycol/nitrogen, the simulated results obtained for the homogenous void fraction ($\beta$), the gas bubble and liquid slug lengths, the bubble velocity were in good agreement with the experiments. Also, the simulated liquid film thicknesses were very close to the experimental results with the
average difference being 3.3% for the ethylene-glycol/nitrogen system. However, the location of the gas and liquid interface could not be detected in experimental photographic images for the water/nitrogen system, so the film thickness from the simulations were compared with those from the Suo and Griffith (1964) correlation (established for a stationary and constant thickness film) using the experimental bubble velocity and mixture velocity and the values calculated were 15-42% higher than the CFD results.

There were no experimental data from Leung et al. (2010; 2012) for the pressure drop, so the ratio of the simulated pressure drops to that of the liquid-only flow (normalized pressure drop) were compared with the Kreutzer et al. (2005) correlation (developed for long slugs and thin liquid films). The results obtained from the simulation and correlation gave average differences of 16% and 6% for the water/nitrogen and ethylene-glycol/nitrogen systems, respectively.

In the experimental work, oscillations at the bubble tail shape were observed from the mixture velocity of 0.42 m s\(^{-1}\) for the water/nitrogen system. This unsteady behaviour was also found in the modelling at a mixture velocity of 0.53 m s\(^{-1}\), having a Reynolds number of 1189, but there were still good agreement between the simulated and experimental results for the bubble and slug lengths, the liquid film thickness and the bubble velocity. The normalized pressure drop was the only parameter which showed a significant effect of the variation of the bubble tail shape versus time, with the simulated normalized pressure drop oscillating in the range 0.2 to 2.1 for a case with \(U_{TP} = 0.53\) m s\(^{-1}\) and \(\beta = 0.49\).

The comparison of the fully-developed heat transfer results between the CFD and experiments were made based on a constant wall heat flux boundary condition for the water/nitrogen and ethylene-glycol/nitrogen systems. There was good agreement between the ratio of the average Nusselt number to that of the liquid-only flow (normalized average Nusselt number) from the simulations and experiments for both of liquid phases used. All of the cases for the water/nitrogen system with \(U_{TP} = 0.21\) m s\(^{-1}\) and 0.32 m s\(^{-1}\) and most of the cases studied for the ethylene-glycol/nitrogen system fell in the
experimental uncertainty range. The simulated and experimental normalized Nusselt numbers ($Nu^*$) gave the maximum difference of around 30% when the mixture velocity reached $0.53 \text{ m s}^{-1}$ for the water/nitrogen system and the unsteady fluctuations at the bubble tail shape appeared in the modelling and experiments.

The effect of the mixture velocity and homogenous void fraction on the simulated $Nu^*$ were studied. The simulated results showed that $Nu^*$ was approximately constant with increasing mixture velocity from $0.21 \text{ m s}^{-1}$ to $0.42 \text{ m s}^{-1}$ for the water/nitrogen system which was different from the behaviour of the experimental $Nu^*$, which increased with increasing mixture velocity. The trend of the simulated and experimental $Nu^*$ versus mixture velocity were similar for the ethylene-glycol/nitrogen system and they increased with increasing the mixture velocity, although the enhancement of the experimental $Nu^*$ was more significant than that of the simulations. For both of the systems, water/nitrogen and ethylene-glycol/nitrogen, the normalized Nusselt numbers from the simulations and experiments increased with increasing homogenous void fraction until a maximum value was reached which varied for different mixture velocities, then it decreased on further increasing the homogenous void fraction. This is explained by the effect of two contributions to heat transfer. There are more complete fluid recirculation cycles in shorter liquid slugs which enhances the heat transfer but the effect of the liquid flow rate reduction becomes stronger than the previous effect as the homogenous void fraction is increased further.

In summary, in this thesis two methods for the CFD modelling of Taylor flow in microchannels were developed to generate gas bubbles and liquid slugs with controlled lengths. Therefore, a meaningful comparison of the hydrodynamics and heat transfer results between the simulations and experiments was possible for a wide range of Reynolds and Capillary numbers to obtain a deeper understanding of the flow field and heat transfer in Taylor flow.
Acknowledgements

This research project would not have been possible without the support of many people.

I wish to express my deepest gratitude to my supervisors, Professors David F. Fletcher and Brian S. Haynes who were abundantly helpful and offered invaluable assistance, support and guidance. I would have been lost without their knowledge and ideas. Throughout my thesis-writing period David provided encouragement, good teaching and good company.

The work was supported through a University of Sydney Postgraduate Award (UPA). The financial support of the Heatric Division of Meggitt (UK) is also gratefully acknowledged.

I would like to thank Dr. Raghvendra Gupta for teaching me lots of new things about Taylor flow. I am also especially grateful to Sharon Leung for sharing her experimental data which were so helpful on this research. I also thank my many student colleagues and Marta Kremky for providing a nice environment in which to learn.

It is pleasure to thank my friend Mahsa Saidan for her encouragement through the difficult times.

I wish to thank my amazing sisters, Bahareh and Pooneh for all of the constant emotional supports and assistance.

I am heartily thankful to my unique parents, Faegh and Maliheh Najafi, for their unconditional love and support throughout all of the stages in my life. I am deeply indebted to them for their dedications and faith in me.
I am extremely grateful to my wonderful husband, Amir who is everything to me for his patience, understanding, support and love. His encouragement and support were infinite in helping me to achieve my dreams.

I dedicate this thesis to my parents and Amir.
# Table of Contents

Declaration................................................................................................................................... i  
Abstract....................................................................................................................................... ii  
Acknowledgements................................................................................................................... vi  
Table of Contents....................................................................................................................... x  
List of Tables ........................................................................................................................... xii  
List of Figures ........................................................................................................................... xiii  
Nomenclature.......................................................................................................................... xix  
Publications Arising from this Thesis............................................................................... xixii  

1. Introduction........................................................................................................................... 1  
   1.1 Background ....................................................................................................................... 1  
   1.2 Objectives ........................................................................................................................ 3  
   1.3 Thesis Structure ............................................................................................................... 3  
2. Literature Review .................................................................................................................. 5  
   2.1 Taylor Flow Regime ........................................................................................................ 5  
   2.2 Numerical Techniques .................................................................................................... 7  
   2.3 Modeling of Taylor Flow ................................................................................................. 7  
   2.4 Heat Transfer Studies of Taylor Flow ......................................................................... 13  
   2.5 Summary ......................................................................................................................... 16  
3. Approaches to CFD Modelling of Taylor Flow ................................................................... 18  
   3.1 Mathematical Model ...................................................................................................... 18  
      3.1.1 Governing Equations ............................................................................................. 18  
      3.1.2 Boundary and Initial Conditions ........................................................................... 20  
   3.2 CFD Methodology ......................................................................................................... 21  
      3.2.1 Stationary Domain ................................................................................................. 21  
      3.2.2 Moving Domain ..................................................................................................... 26  
   3.3 Numerical Methods ........................................................................................................ 30  
   3.4 Verification Results and Discussion ............................................................................. 31  
      3.4.1 Comparison of the Stationary Domain and Moving Domain Results ............ 31  
   3.5 Summary ........................................................................................................................ 44  
4. Validation of Simulation Results for Hydrodynamics....................................................... 46
List of Tables

Table 3.1: The gas and liquid properties used in the simulations ......................................... 31
Table 4.1: Properties of the gas and the liquids used in the simulations .............................. 47
Table 4.2: The homogenous void fraction obtained from the simulations for the water/nitrogen system. The cases denoted by a star were simulated using constant properties at a temperature of 21°C. ................................................................................................. 53
Table 4.3: The homogenous void fraction obtained from the simulations for the ethylene-glycol/nitrogen system ................................................................. 54
Table 4.4: Comparison of the experimental and simulated gas bubble and liquid slug lengths for the water/nitrogen system. The cases denoted by a star were simulated using constant properties evaluated at a temperature of 21°C................................................................. 56
Table 4.5: Comparison of the experimental and simulated gas bubble and liquid slug lengths for the ethylene-glycol/nitrogen system ................................................................. 57
Table 4.6: Comparison of the liquid film thickness between the simulations and experiments for the ethylene-glycol/nitrogen system ................................................................. 64
Table 5.1: Comparison of the experimental and simulated normalized average Nusselt numbers for the water/nitrogen system .................................................................................. 79
Table 5.2: Comparison of the experimental and simulated normalized average Nusselt numbers for the ethylene-glycol/nitrogen system ................................................................. 84
List of Figures

Figure 1.1: Typical images of flow patterns of air/water system in a circular tube of 3 mm diameter observed by Liu et al. (2005). (a) Bubbly flow, (b) slug-bubbly flow, (c) Taylor flow, (d) churn flow and (e) annular flow................................................................................................................................. 1

Figure 1.2: The recirculation zone in the liquid slug of Taylor flow. Adapted from Angeli and Gavrilidis (2008)................................................................................................................................. 2

Figure 2.1: Flow regime map developed by Liu and Wang (2008) for a 2.37 mm diameter circular tube. The solid lines represent the boundaries of flow regime transitions. .............. 6

Figure 2.2: The initial bubble shape and boundary conditions used in the modelling of Taha and Cui (2006). ........................................................................................................................................ 8

Figure 2.3: Bubble shapes (bold solid lines) and the recirculation patterns (dashed lines) are shown for $\varepsilon G = 0.2$ and $-dp/dx = 850$ MPa m$^{-1}$ using different computational domains: (a) $L_x/R = 2$, (b) $L_x/R = 3$, (c) $L_x/R = 8$. Taken from the work of Fukagata et al. (2007). ................................................................................................................................. 10

Figure 2.4: The geometry and boundary conditions used in the work of Shao et al. (2008). This figure is taken from Shao et al. (2008). ................................................................................................. 11

Figure 2.5: The geometry and boundary conditions used by Gupta et al. (2009a). Taken from Gupta et al. (2009).................................................................................................................. 12

Figure 2.6: A contour plot of temperature (K) for a fully-developed unit cell from the work of Gupta et al. (2010a). The constant wall heat flux boundary condition was used. The bubble interface and the boundaries of the unit cell are shown by a black line and two black vertical lines, respectively................................................................. 14

Figure 3.1: Schematic of the bubble generation process for one time period (a) when a bubble and the liquid surrounding it enter the channel (b) when only liquid enters the channel .................................................................................................................. 23

Figure 3.2: Volume fraction plot for fluids of water and nitrogen for cases with: (a) no liquid layer present at the inlet, (b) with a liquid layer thickness calculated using Bretherton’s correlation ($R = 0.25$ mm, $R_{in} = 0.24$ mm), and (c) a thicker liquid layer ($R$
In Figure 3.2(b), the liquid film is shown in an enlarged plot. The gas and liquid are coloured blue and red, respectively.

Figure 3.3: Volume fraction plot for a case using ethylene-glycol and nitrogen as the fluids, in which there is no liquid layer present at the inlet ($R_{in} = 0.25$ mm). The nitrogen and ethylene-glycol are coloured blue and red, respectively.

Figure 3.4: Schematic of the geometry and boundary conditions used in the dynamic mesh method.

Figure 3.5: Velocity profiles with respect to the dynamic mesh at the periodic interfaces. The direction of temperature wrapping is shown.

Figure 3.6: Comparison of the bubble shape obtained from the moving domain method for different grid sizes.

Figure 3.7: Comparison of the bubble shape obtained from the moving (dashed line) and stationary domain (solid line) methods.

Figure 3.8: Normalised velocity profile in the middle of the liquid slug for the moving (dashed line) and stationary domain (solid line) approaches. The parabolic velocity profile for the fully-developed laminar flow is also shown by a dotted line.

Figure 3.9: Local wall shear stress in the axial direction. In part (b), the scale has been changed to show the small differences in the calculated wall shear stress in the moving and fixed domain methods. The solid lines are for the stationary domain while the dashed lines are for the moving domain. The location of the bubble is also shown by a dotted line.

Figure 3.10: Variation of the bulk and wall temperatures for the stationary domain. The dashed straight line shows the temperature increment for a homogeneously mixed two-phase flow. The location of the bubbles is also shown by dotted lines.

Figure 3.11: Heat transfer development analysis for the fixed domain approach and a constant wall heat flux boundary condition. (a) Volume fraction plot for the fourth to seventh cells, and (b) Radial distribution of the normalised temperature increment across the most thermally-developed unit cells.

Figure 3.12: Variation of the bulk and wall temperatures for the unit cell of the moving domain and the stationary domain. The moving domain temperatures are shown by dashed lines. A constant heat flux thermal boundary condition is applied at the walls.
Figure 3.13: Variation of the local Nusselt number for the unit cell of the moving domain and the stationary domain (sixth unit cell). The moving domain results are shown with a dashed line. A constant wall heat flux is set at the wall. In part (b), the scale of the local Nusselt number is restricted to focus on the slug region. The location of the bubble is also shown by a dotted line.

Figure 4.1: Experimental images of bubble heads and tails with $\beta = 0.5$ for the water/nitrogen system at (a) different mixture velocities. Taken from Leung et al. (2012). In parts (b) and (c), the transient bubble tails are shown for $U_{TP} = 0.42 \text{ m s}^{-1}$ and $U_{TP} = 0.53 \text{ m s}^{-1}$, respectively.

Figure 4.2: The effect of the mixture velocity ($U_{TP}$) on the bubble velocity ($U_B$) obtained from CFD simulations (○) and the experiments (□) for the water/nitrogen system. The correlation of Liu et al. (2005) is shown by a dashed line.

Figure 4.3: The effect of the mixture velocity ($U_{TP}$) on the bubble velocity ($U_B$) obtained from CFD simulations (○) and the experiments (□) for the ethylene-glycol/nitrogen system. The correlation of Liu et al. (2005) is shown by a dashed line.

Figure 4.4: The effect of the homogenous void fraction ($\beta$) on the normalized bubble, slug and unit cell lengths ($L/d$) for the water/nitrogen system with $U_{TP} = 0.32 \text{ m s}^{-1}$.

Figure 4.5: The effect of the homogenous void fraction ($\beta$) on the normalized bubble, slug and unit cell lengths ($L/d$) for the ethylene-glycol/nitrogen system with the mixture velocity value of $0.20 \text{ m s}^{-1}$.

Figure 4.6: The effect of Capillary number ($Ca$) on the non-dimensional film thickness ($\delta_f/R$) for the water/nitrogen system for an experimental homogenous void fraction ($\beta$) of 0.4. The symbols (□) and (△) are used for film thicknesses calculated from equation (4.3) using the experimental and simulated bubble velocities ($U_B$), respectively. The symbol (○) is used for the results obtained directly from the simulations. The Aussillous and Quere (2000) correlation is also shown by a dashed line.

Figure 4.7: The inertial effect on the liquid film thickness. Taken from the work of de Ryck, (2002).

Figure 4.8: The effect of the variation of the bubble tail shape on the non-dimensional film thickness ($\delta_f/R$) for $U_{TP} = 0.53 \text{ m s}^{-1}$ and $\beta = 0.4$ for the water/nitrogen system.
Figure 4.9: The effect of the Capillary number \( (Ca) \) on the non-dimensional film thickness \( (\delta_f/R) \) for the ethylene-glycol/nitrogen system. The symbols (□) and (○) are used for the experimental and simulation results, respectively. Values from the Aussillous and Quere, (2000) correlation are shown by a dashed line.

Figure 4.10: Comparison of the bubble shapes obtained from the simulations and experiments for the ethylene-glycol/nitrogen system with \( U_{TP} = 0.20 \text{ m s}^{-1} \). In part (a), the effect of the homogenous void fractions \( (\beta) \) on the bubble shapes is shown. In part (b), the bubble shapes for case 6, with \( U_{TP} = 0.20 \text{ m s}^{-1} \) and \( \beta = 0.68 \), are plotted. The simulation results are shown by solid red lines.

Figure 4.11: The effect of the mixture velocity \( (U_{TP}) \) on the bubble shape for the experimental homogenous void fractions \( (\beta) \) of ~0.55 for the ethylene-glycol/nitrogen system. The simulation results are shown by solid red lines.

Figure 4.12: The effect of the homogenous void fraction \( (\beta) \) on the normalized pressure drop \( (\phi) \) at \( U_{TP} = 0.32 \text{ m s}^{-1} \) for the water/nitrogen system. The normalized pressure drops calculated from the equation (4.6) are also shown by (□) symbol.

Figure 4.13: The effect of the homogenous void fraction \( (\beta) \) on the normalized pressure drop \( (\phi) \) at \( U_{TP} = 0.20 \text{ m s}^{-1} \) for the ethylene-glycol/nitrogen system. The normalized pressure drops calculated from equations (4.6) and (4.7) are also shown by the (□) and (△) symbols, respectively.

Figure 4.14: The effect of the mixture velocity \( (U_{TP}) \) on the normalized pressure drop \( (\phi) \) for the homogenous void fraction of ~0.5 for the water/nitrogen system. The normalized pressure drops calculated from the equation (4.6) are also shown by the (□) symbol.

Figure 4.15: The effect of the mixture velocity \( (U_{TP}) \) on the normalized pressure drop \( (\phi) \) for the homogenous void fraction of ~0.58 for the ethylene-glycol/nitrogen system. The normalized pressure drops calculated from equation (4.6) and (4.7) are also shown by the (□) and (△) symbols, respectively.

Figure 5.1: Schematic of the experimental setup for the water/nitrogen system. Taken from the work of Leung et al. (2010). \( T_n (n = 1, 2, \ldots, 10) \) shows the thermocouples used for measuring the temperature of the copper blocks.

Figure 5.2: Schematic of the heat transfer mechanisms in Taylor flow. Taken from the work of Leung et al. (2012).
Figure 5.3: Normalized average Nusselt number ($Nu^*$) versus the homogenous void fraction ($\beta$) for the water/nitrogen system for different mixture velocities of (a) 0.21 m s$^{-1}$, (b) 0.32 m s$^{-1}$ (c) 0.42 m s$^{-1}$ (Δ) and 0.53 m s$^{-1}$ (◊). The experimental and simulation results are shown by solid and open symbols, respectively................................................81

Figure 5.4: The simulated $Nu^*$ versus time for case 15 of the water/nitrogen system with $U_{TP} = 0.53$ m s$^{-1}$ and $\beta = 0.39$. ................................................................................................82

Figure 5.5: The experimental results for the normalized local Nusselt number at the centre of each block from 2 to 9 are shown by (Δ) symbol for the water/nitrogen system with $U_{TP} = 0.53$ m s$^{-1}$ and $\beta = 0.4$. The mean experimental $Nu^*$ is also shown by a dashed line............................................................................................................................................83

Figure 5.6: Normalized average Nusselt number ($Nu^*$) versus the homogenous void fraction ($\beta$) for the ethylene-glycol/nitrogen system for different mixture velocities of (a) 0.20 m s$^{-1}$ (○), (b) 0.29 m s$^{-1}$ (□) and 0.37 m s$^{-1}$ (Δ). The experimental and simulated results are shown by solid and open symbols, respectively................................................85

Figure 5.7: The simulated $Nu^*$ versus time for case 6 of the ethylene-glycol/nitrogen system with $U_{TP} = 0.20$ m s$^{-1}$ and $\beta = 0.67$ at 30°C temperature........................................86

Figure 5.8: The simulated $Nu^*$ are shown for case 6 of the ethylene-glycol/nitrogen system with $U_{TP} = 0.20$ m s$^{-1}$ and $\beta = 0.67$ at 23.6°C and 30°C temperatures. Error bars representing two standard deviations in the variation of the mean Nusselt number are also shown........................................................................................................................................88

Figure 5.9: The effect of the mixture velocity ($U_{TP}$) on the normalized Nusselt number for the water/nitrogen system. The experimental and simulated results are shown by solid and open symbols, respectively. The mean value of the simulated $Nu^*$ over different mixture velocities is also shown by a dashed line..............................................................................................................89

Figure 5.10: The experimental results for the normalized local Nusselt number at the centre of blocks 2 to 9 are shown by (Δ) symbol for the water/nitrogen system with $U_{TP} = 0.42$ m s$^{-1}$ and $\beta = 0.5$. The mean experimental $Nu^*$ is also shown by a dashed line. ......90

Figure 5.11: The effect of the mixture velocity ($U_{TP}$) on the normalized Nusselt number for $\beta \sim 0.55$ for the ethylene-glycol/nitrogen system. The experimental and simulation results are shown by solid and open symbols, respectively. The symbols (○) and (Δ) are
used for \( L_s/d \) and \( Nu^* \), respectively. The results obtained for a case with \( U_{TP} = 0.53 \text{ m s}^{-1} \) and \( \beta = 0.56 \) are also shown in red.

Figure 5.12: The effect of the homogenous void fraction (\( \beta \)) on the normalized Nusselt number for the water/nitrogen system. The experimental and simulated results are shown by solid and open symbols, respectively. The symbols (○) and (Δ) are used for the mixture velocities of 0.21 m s\(^{-1}\) and 0.42 m s\(^{-1}\), respectively.

Figure 5.13: The effect of the homogenous void fraction (\( \beta \)) on the normalized Nusselt number for the ethylene-glycol/nitrogen system. The experimental and simulated results are shown by solid and open symbols, respectively. The symbols (○) and (Δ) are used for the mixture velocities of 0.20 m s\(^{-1}\) and 0.37 m s\(^{-1}\), respectively. The result obtained for a case with \( U_{TP} = 0.20 \text{ m s}^{-1} \) and \( \beta = 0.87 \) is shown in red.
Nomenclature

List of symbols

\( A \) \hspace{1em} \text{Area (m}^2\text{)}
\( c_p \) \hspace{1em} \text{Specific heat capacity (J kg}^{-1}\text{K}^{-1}\text{)}
\( Ca \) \hspace{1em} \text{Capillary number (}\mu_LU_{TP}/\sigma\text{)}
\( d \) \hspace{1em} \text{Diameter of the channel (m)}
\( k \) \hspace{1em} \text{Thermal conductivity (Wm}^{-1}\text{K}^{-1}\text{)}
\( L \) \hspace{1em} \text{Length (m)}
\( n \) \hspace{1em} \text{Unit normal vector}
\( Nu_j \) \hspace{1em} \text{Experimental local Nusselt number of block j}
\( Nu^* \) \hspace{1em} \text{Normalised Nusselt number (}\text{Nu}/\text{Nu}_{LO}\text{)}
\( Nu_x \) \hspace{1em} \text{Local Nusselt number in simulation}
\( P \) \hspace{1em} \text{Pressure (Pa)}
\( \Delta P \) \hspace{1em} \text{Pressure drop (Pa)}
\( \Delta P_{cap} \) \hspace{1em} \text{Pressure drop over the bubble cap (Pa)}
\( q \) \hspace{1em} \text{Heat flux (W m}^{-2}\text{)}
\( R \) \hspace{1em} \text{Channel radius (m)}
\( r \) \hspace{1em} \text{Radial coordinate (m)}
\( Re \) \hspace{1em} \text{Two-phase (liquid-only) Reynolds number (}\text{U}_{TP}\rho_Ld/\mu_L\text{)}
\( T \) \hspace{1em} \text{Temperature (K)}
\( t \) \hspace{1em} \text{Time (s)}
\( t^* \) \hspace{1em} \text{Non-dimensional flow time (}\text{t}/\text{t}_{uc}\text{)}
\( t_G^* \) \hspace{1em} \text{Non-dimensional gas flow time (}\text{t}_G/\text{t}_{uc}\text{)} \text{ U_B \hspace{1em} Bubble velocity (m s}^{-1}\text{)}
\( U_G \) \hspace{1em} \text{Gas superficial velocity (m s}^{-1}\text{)}
\( U_L \) \hspace{1em} \text{Liquid superficial velocity (m s}^{-1}\text{)}
\( U_{TP} \) \hspace{1em} \text{Mixture velocity (}\text{U}_G+\text{U}_L\text{) (m s}^{-1}\text{)}
\( v \) \hspace{1em} \text{Velocity vector (m s}^{-1}\text{)}
$u_x$ Axial velocity (m s$^{-1}$)

$v_z$ Radial velocity (m s$^{-1}$)

$x$ Axial coordinate (m)

**Greek symbols**

$\alpha$ Volume fraction

$\beta$ Homogeneous void fraction ($U_G / U_T$)

$\delta(r)$ Dirac-delta function

$\delta_F$ Film thickness (m)

$\epsilon_G$ Volume fraction of gas in a unit cell (also termed the void fraction and gas hold-up)

$\kappa$ Curvature of the interface (m$^{-1}$)

$\mu$ Dynamic viscosity (kg m$^{-1}$ s$^{-1}$)

$\rho$ Density (kg m$^{-3}$)

$\sigma$ Surface tension coefficient (Nm$^{-1}$)

$\tau_{rx}$ Wall shear stress ($= -\mu \frac{\partial v_x}{\partial r}$) (N m$^{-2}$)

$\xi$ Non-dimensional radius ($= r/R$)

**Subscripts**

$av$ Average value for a unit cell

$B$ Bubble

$b$ Bulk mean

$cyl$ Cylindrical

$F$ Film

$G$ Gas

$i$ Interface

$in$ Inlet

$j$ Experimental block number

$L$ Liquid

$out$ Outlet

$S$ Slug
$UC$  Unit cell

$w$  Wall

$XS$  Cross-section
Publications Arising from this Thesis

Journal papers


Conference presentations

Chapter 1

Introduction

1.1 Background

Gas-liquid flow in microchannels has been a very active research area in the last few decades because of its occurrence in nature and its use in micro-structured devices being developed across a range of industries. These applications include gas-liquid reactions in microreactors (Günther et al., 2004), catalyst coating in capillaries (Kolb and Cerro, 1991), boiling flow in electronics cooling (Thome, 2004), the refrigeration industry, flow in blood vessels and respiratory airways (Zheng et al., 2007) and flow in porous media.

Several researchers have applied different methods of experimental visualization to understand the flow characteristics of adiabatic gas-liquid two-phase flow in micro tubes (Liu et al, 2005). In Fig. 1.1, taken from Liu et al. (2005), typical images of two-phase flow patterns observed in a vertical circular channel of 3 mm diameter are shown for an air/water system. Five main flow regimes which occur in gas-liquid flow in microchannels are identified in this figure and they are named bubbly flow, slug-bubbly flow, Taylor flow, churn flow and annular flow.

![Figure 1.1: Typical images of flow patterns of air/water system in a circular tube of 3 mm diameter observed by Liu et al. (2005). (a) Bubbly flow, (b) slug-bubbly flow, (c) Taylor flow, (d) churn flow and (e) annular flow.](image-url)

Figure 1.1: Typical images of flow patterns of air/water system in a circular tube of 3 mm diameter observed by Liu et al. (2005). (a) Bubbly flow, (b) slug-bubbly flow, (c) Taylor flow, (d) churn flow and (e) annular flow.
The slug or Taylor flow regime occurs across a wide range of gas and liquid flow rates in microchannels (Triplett et al., 1999; Zhao and Bi, 2001). Taylor flow is characterized by the regular and periodic flow of gas bubbles with a diameter which is almost that of the channel that are separated from each other by the liquid slugs (Fig. 1.1c). This flow has two interesting features: the existence of internal circulations in the liquid slug and a thin liquid film around the gas bubbles, as shown in Fig. 1.2. Thus Taylor flow provides efficient heat and mass transfer and reduces the axial mixing significantly. As a result, Taylor flow is important for a number of industrial applications, such as compact heat exchangers, microfiltration and multiphase reactors. Numerous papers have been published on the slug flow topic (Taylor, 1961; Thulasidas et al, 1997; Kreutzer et al, 2005b;).

Figure 1.2: The recirculation zone in the liquid slug of Taylor flow. Adapted from Angeli and Gavriilidis (2008).

A large number of experimental investigations have been carried out using non-intrusive visualisation techniques, such as micro-PIV, to measure Taylor flow characteristics and understand the flow behaviour. These studies have included measurements of pressure drop, bubble velocity, bubble and slug lengths and liquid film thickness (Suo and Griffith, 1964; Thulasidas et al, 1997) and some works have been done on the heat transfer (Oliver and Wright, 1964; Walsh et al, 2010). Obtaining detailed and local data, especially for heat transfer, for slug flow from the experiments is difficult and complicated due to the intermittent behavior of this flow. Thus, numerical simulations such as computational fluid dynamics (CFD) play an important role to understand more
details of the hydrodynamics and heat transfer phenomena in Taylor flow which cannot be scrutinized by applying existing experimental tools.

However, the numerical modeling results are acceptable only after validating them against experimental data or developed correlations from the literature. The need to validate the computational results with experimental data can be explained by the fact that the simplification of the theoretical equations and the solution method used by the simulation software may affect the validity of the results. In addition, all of the essential phenomena influencing the process in reality may not be reflected by the specified models used in the software.

1.2 Objectives

Therefore, the intention of this study is to develop an efficient validated methodology for the CFD modeling of Taylor flow. It is also aimed to validate both the hydrodynamics and heat transfer results from the simulations with experimental data, so the simulation technique can be trusted to provide a comprehensive understanding of flow and heat transfer in Taylor flow.

1.3 Thesis Structure

The remaining Chapters of this thesis are summarized below:

- Chapter 2 (Literature review): presents a review on the relevant literature that has been studied on the non-boiling, gas-liquid Taylor flow in microchannels to set the stage of this study in this topic.

- Chapter 3 (Approaches to CFD modeling of Taylor flow): describes and benchmarks two methodologies developed for the modeling of Taylor flow in the laboratory frame of reference (stationary domain method) and in a frame of reference moving with the bubble velocity (moving domain method). Both methods are implemented in ANSYS-Fluent by using the volume-of-fluid (VOF) method for capturing the gas-liquid interface. In the two methods, bubble and slug lengths can be simulated with prescribed values.
• Chapter 4 (Validation of simulation results for hydrodynamics): validates the hydrodynamics results obtained from the CFD modeling using the moving domain method against experimental results for two systems consisting of water/nitrogen and ethylene-glycol/nitrogen.

• Chapter 5 (Validation of modeling results for heat transfer): compares the simulated and experimental heat transfer results obtained for the cases described in Chapter 4 for the same systems. In addition, the effects of changing the mixture velocity and the homogenous void fraction on the heat transfer results from the simulations are studied.

• Chapter 6 (Conclusions and recommendations): presents the conclusions of this study followed by some suggestions for future work.
Chapter 2

Literature Review

This chapter provides a review of literature concerning non-boiling, two-phase gas-liquid Taylor flow in microchannels. The purpose is to identify the research questions that need to be answered and to set the objectives of this thesis.

2.1 Taylor Flow Regime

Gas and liquid can distribute in a channel in different ways, giving various flow regimes or flow patterns. Many experimental studies have been performed to investigate such flow regime maps based on the gas and liquid properties, flow rates and channel geometries, for example, Suo and Griffith (1964), Barnea et al. (1983), Damianides and Westwater (1988), Barajas and Panton (1993), Fukano et al. (1993; 2007), Mishima and Hibiki (1993; 1996), Tripplet et al. (1999), Coleman and Garimella (1999), Bao et al. (1994; 2000), Zhao and Bi (2001a), Chen et al. (2002), Pehlivan et al. (2006), Liu and Wang (2008), Saisorn and Wongwises (2008) and Pohorecki et al. (2008).

The flow regime map of Liu and Wang (2008) for air and water as the working fluids in a vertical upward round tube of 2.37 mm inner diameter is shown in Fig. 2.1. In this figure the gas superficial velocity ($U_g$) and liquid superficial velocity ($U_L$) are used as the horizontal and vertical coordinates, respectively. The solid lines determine the boundaries of the observed flow patterns. It can be seen that the flow pattern map is divided into four major regions, covering bubbly flow, Taylor flow, churn flow and annular flow. The two remaining flow patterns, which are named slug-bubbly flow and bubble-train slug flow, occupy only small regions of the flow pattern map.
2. Literature Review

Figure 2.1: Flow regime map developed by Liu and Wang (2008) for a 2.37 mm diameter circular tube. The solid lines represent the boundaries of flow regime transitions.

Slug flow (also called Taylor, plug, segmented, intermittent and bubble-train flows) is one of the most interesting flow patterns amongst the two-phase flow regimes. It can be obtained by increasing the gas flow rate to move above the bubbly flow regime, so the flow changes to distorted bubbles and then to bullet-shaped bubbles. The industrial applications of Taylor flow include catalyst coating (Kolb and Cerro, 1991), compact heat exchangers and enhanced oil recovery process (Schwartz et al., 1986) means that this regime has been extensively investigated because of its special configuration.

Taylor flow is composed of liquid slugs which are separated by gas bubbles, with a thin liquid film that separates the bubbles from the wall and forms the only connection between slugs. These characteristics prevent back-mixing by reducing the axial mass transfer in the liquid, whilst enhancing radial mass transfer (Thulasidas et al., 1999; Salman et al., 2006). Other advantages of this flow pattern are the increase in the mass and heat transfer rates between the liquid and wall, by the recirculating flow in the liquid slugs (Bercić and Pintar, 1997). Many studies of the hydrodynamics, heat and mass transfer in the Taylor flow regime in microchannels have been performed (Fairbrother...
and Stubbs, 1935; Bretherton, 1961; Prothero and Burton, 1961; Taylor, 1961; Suo and Griffith, 1964; Hughmark, 1965; Oliver and Hoon, 1968; Horvath et al., 1973; Pedersen and Hovarth, 1981; Chen, 1986; Schwartz et al., 1986; Monde et al., 1989; Edvinsson and Irandoust, 1996; Giavedoni and Saita, 1997; Aussillous and Quéré, 2000; Heiszwolf et al., 2001; Gupta et al., 2009). These works have been reviewed and summarised recently by Kreutzer et al., (2005a), Angeli and Gavriilidis, (2008) and Gupta et al., (2010b).

2.2 Numerical Techniques

Many investigators have used experimental and analytical methods to study the flow field and heat transfer of slug flow. With the improvement in digital computers during the last few decades, the numerical (computational) approach has been added to the other methods. There are several numerical techniques such as Computational Fluid Dynamics (CFD) based on Navier-Stokes equations and lattice Boltzmann methods (LBM) based on the molecular description of fluids which have been used to model gas-liquid two phase flow in microchannels. In this work, CFD approach is applied to model slug flow in circular microchannels to obtain a greater understanding of the hydrodynamics and heat transfer occurring in this regime. Several methods, for example, boundary integral (Tsai and Miksis, 1994), finite-element (Edvinsson and Irandoust, 1996) and interface capturing (Taha and Cui, 2004) have been developed to model slug flow with CFD approach. As the gas-liquid interface is well-defined in Taylor flow, interface capturing/tracking methods, such as volume-of-fluid (VOF) (Hirt and Nichols, 1981), level-set (Sussman et al., 1994), marker points (Unverdi and Tryggvason, 1992) and phase-field (Anderson et al., 1998) can be used for Taylor flow modelling. This study uses the VOF method as implemented in ANSYS Fluent (ANSYS, 2010) to capture the gas-liquid interface. The equations used in the VOF method are given later in Chapter 3.

2.3 Modeling of Taylor Flow

There are different computational approaches for modelling Taylor flow. As discussed in Gupta et al. (2010b), fully-developed Taylor flow is an unsteady, periodic flow in the
laboratory frame of reference, but is steady in a frame of reference moving with the bubble. Below a review of the methods used by previous researchers to simulate the hydrodynamics of Taylor flow is presented.

Taha and Cui (2006) studied the motion of single Taylor bubbles in vertical tubes. Fluent was used for the two-dimensional, axisymmetric simulations with the volume of fluid method applied to capture the gas-liquid interface. They modelled the flow in a unit cell, comprising a bubble and the adjacent half slugs, by moving the wall with the bubble rise velocity in the opposite direction to the flow. Figure 2.2 taken from their study shows the initial bubble shape and boundary conditions used for the modelling of the slug flow. For the initial bubble shape, they used a cylinder joined to a hemisphere of the same radius.

A fully-developed single phase velocity profile with an average velocity equal to the difference of the bubble velocity and the superficial liquid velocity was specified at the inlet and an outflow condition was used for the outlet boundary condition. In their work, the bubble velocity could not be set from the start of the simulation run as it was a part of the results, so they had to adjust it until the bubble stopped to move in the axial direction. Another point about the method of Taha and Cui (2006) is to use a fully-developed velocity profile at the inlet boundary condition which is not an accurate assumption for the short liquid slugs. As expected, the computed results indicated that the liquid viscosity and surface tension affect the bubble shape and the degree of bluntness of the head of the bubble increased with a decrease of the viscosity. The results obtained from the simulations for the bubble velocity in different tube diameters and liquid properties
were in good agreement with the experimental results in the literature (Griffith and Wallis, 1961; White and Beardmore, 1962; Nicklin et al., 1962).

It should be noted that the above boundary condition is in fact incorrect as at any location in the tube the mean velocity is equal to the two phase velocity, the sum of the liquid and gas superficial velocities (see equation (5) of Suo and Griffith (1964)). Therefore Taha and Cui's simulations apply to a different liquid superficial velocity than they believed they were using.

Fukagata et al. (2007) performed two-dimensional, axisymmetric numerical simulations of two-phase flow, involving air and water, in a 20 μm diameter tube. The flow conditions were set to be similar to those used by Serizawa et al. (2002) in their experimental work. The level-set method was selected for capturing the gas-liquid interface. In their work, the flow was modelled in the laboratory frame of reference by using periodic boundary conditions at both ends of a specified computational domain. Different values for the gas void fraction ($\varepsilon_G = 0.2 - 0.6$) and pressure gradient ($-\frac{dp}{dx} = 85 - 3000$ MPa m$^{-1}$) were used as input values.

The computed relationship between the gas void fraction and the homogenous void fraction ($\beta$) was in good agreement with the Armand and Treschev (1946) correlation which was developed for macro-sized flow, but there was a small discrepancy between the computational results and the experimental results of Serizawa et al. (2002). The difference was argued to be caused by the wall condition next to the gas bubble which may be dry or wet in the experiments. They also found the length of the computational domain affects the flow pattern significantly. Figure 2.3, taken from their work, shows that the bubble shape and the recirculation pattern changed when the bubble period (the computational domain length) was changed for the constant gas void fraction and pressure drop. Therefore, they noted the importance of having exactly the same conditions as used in experiments when validating the numerical results.
Akbar and Ghiaasiaan (2006) modelled Taylor flow in a two-dimensional, axisymmetric circular capillary having a diameter of 1 mm. The volume-of-fluid option of ANSYS Fluent was used for tracking the gas-liquid interface. They used a dynamic mesh to move the computational domain containing one gas bubble and two halves of liquid slugs with the bubble velocity, so the bubble remained stationary with respect to the computational domain. The initial bubble shape, comprising a cylinder and two hemispheres at its ends, was set in the centre of the domain. They used periodic boundary conditions for the axial and radial velocities and a constant pressure of atmospheric was set at the outlet. In their paper, the methodology used to implement the boundary conditions was not clear. Therefore, the PhD thesis of Akbar (2004) was also studied to understand the method used in the work of Akbar and Ghiaasiaan (2006).

They simulated some experimental data of Laborie et al. (1999) and a few data of Liu et al. (2005). Overall, the simulation results were in good agreement with the experimental data. The simulation bubble velocities were close to the prediction of the Liu et al. (2005) correlation. They also showed that the simulated values of the film thicknesses were within a factor of ~2 with the Aussillous and Quéré (2000) correlation which is proposed for negligible inertia conditions.
Shao et al. (2008) used CFX 4.3 to study the formation of Taylor bubbles for a two-dimensional, axisymmetric geometry of a 1 mm diameter capillary using the VOF method. The length of the computational domain was three times the tube diameter and the grid was refined near the wall to capture the liquid film. A co-flow configuration was applied at the inlet for modelling the flow in a stationary computational domain. According to Fig. 2.4, taken from their paper, it can be seen that the gas phase was fed from a nozzle in the centre of the main channel, while the liquid phase entered around the gas nozzle.

![Diagram of the geometry and boundary conditions used in the work of Shao et al. (2008).](image)

Figure 2.4: The geometry and boundary conditions used in the work of Shao et al. (2008). This figure is taken from Shao et al. (2008).

Air was used as the gas phase and three liquids including water, octane and “semi-octane” (a hypothetical fluid with the density and viscosity similar to water, but it has the same surface tension as octane) were used as working fluids. It was found that the bubble volume increased with an increase in the gas superficial velocity and a decrease in the liquid superficial velocity. The results were also in good agreement with the experimental data of Amador et al. (2004). The use of the three mentioned liquids showed much more dependence of the bubble size on surface tension in comparison with the viscosity and density for the specific conditions considered. The simulated dimensionless bubble lengths were compared with different literature correlations, and there was better agreement between the numerical bubble sizes and the correlations which contain the phase fraction or the ratio of superficial phase velocities.
Gupta et al. (2009) modelled two-dimensional, axisymmetric, transient, two-phase Taylor flow in a circular channel having a diameter of 0.5 mm and a length of 10 diameters. The ANSYS Fluent commercial CFD software was used for that simulation by using the VOF method to capture the gas-liquid interface. The fluids used were air and water, and based on the constant properties of the working fluids the Capillary and Reynolds number considered were 0.006 and 280, respectively. In order to have a sufficiently fine near wall mesh to capture the liquid film, with a uniform square grid in the region of gas-liquid interface, a grid having uniform square elements of size 5 μm in the core and rectangular elements (2.4 μm × 5 μm) in the 12 μm thick film region was used for the simulations. Figure 2.5 taken from their work shows a schematic of the geometry and boundary conditions applied. As the modelling was performed in the laboratory frame of reference, so the gas and liquid superficial velocities and the gas and liquid volume fractions were specified at the inlet. In the methodology used in their work, there is no control on the lengths of the gas bubble and the liquid slug. However, the results of the simulations including film thickness, gas hold-up, bubble velocity and pressure drop were found to be in good agreement with literature correlations. The meshing strategy could capture the liquid film successfully and minimize the errors in surface tension modelling.

![Figure 2.5: The geometry and boundary conditions used by Gupta et al. (2009a). Taken from Gupta et al. (2009).](image)

He et al. (2010) studied Taylor flow numerically in a circular channel of 600 μm diameter, which was chosen based on the experimental work of Hayashi et al. (2007). The phase-field method was used to capture the gas-liquid interface. In their work, the flow was assumed to be periodic and axisymmetric, so they simulated a two-dimensional computational domain, including one gas bubble and one liquid slug, using periodic boundary conditions. Fluids of air and water were used. The non-dimensional length of
the computational domain and the mixture velocity were changed in the range of 3-15 and 0.03-1.5 m s\(^{-1}\), respectively. They showed that the simulation homogenous void fractions were in good agreement with the experimental data of Hayashi et al. (2007) and the Armand correlation which is developed for macro-channels.

They also compared the simulation film thicknesses obtained with the experimental results of Han and Shikazono (2008) and the deviations were significant at low Capillary numbers. They reported that the deviations observed may be caused by the difference of the bubble lengths. However, it can be seen in Fig. 4 of their paper that surprisingly the film thicknesses obtained from their simulations remain constant when the Capillary number was varied from 7.1\times10^{-4} to 8.6\times10^{-3}. In addition, the film thicknesses are inconsistent with the experimental results of Han and Shikazono (2008) and those measured from their Fig. 4 are inconsistent with those presented in their Fig. 12.

### 2.4 Heat Transfer Studies of Taylor Flow

Several researchers have studied heat transfer of non-boiling gas-liquid Taylor flow in microchannels numerically and experimentally. In this section, a brief review of the computational and experimental studies on Taylor flow heat transfer is presented.

Fukagata et al. (2007) simulated the flow and heat transfer in a periodic computational domain for a circular channel of diameter 20 μm. The constant wall heat flux was used for the thermal boundary condition. They observed the Nusselt number in Taylor flow is larger than that in a liquid single-phase. This enhancement was explained by the existence of the recirculation zone in the liquid slug. It was also found that a stronger recirculation flow in the liquid slug increases the Nusselt number.

Gupta et al. (2010a) studied fully-developed flow and heat transfer using ANSYS Fluent for a cylindrical domain having a 0.5 mm diameter and the length of 40 diameters. They used two boundary conditions, namely constant wall heat flux and constant wall temperature. They reported that it took 40 hours simulation time for the hydrodynamics
and heat transfer modelling of the channel with $15d$ length which was not long enough to reach to the fully-developed heat transfer. Therefore, they used a computational domain with a length of $40d$ and it was meshed using the strategy presented in the previous work of Gupta et al. (2009).

Overall, the results showed that the average Nusselt number for both boundary conditions was around 2.5 times that of the equivalent single-phase, laminar and fully-developed Nusselt number. Another finding from their study was that the Nusselt number remained constant at around 4.8 when the mixture velocity was varied over the range 0.3-1 m s$^{-1}$ for the constant wall heat flux boundary condition. The variation of the Nusselt number versus the homogeneous void fraction was also studied and the results showed that the Nusselt number decreased from 14.4 to 8.3 when the homogenous void fraction increased from 0.3 to 0.7. In all of their simulations, the value of the Nusselt number obtained was higher than the liquid-only flow and it was explained by the creation of radial flow at the nose and the tail of the bubbles and the existence of the recirculation zone in the liquid slug. In Fig. 2.6, taken from their paper, they showed the liquid slug recirculation effect on the temperature field for a constant wall heat flux boundary condition by plotting a contour of temperature in a fully-developed unit cell.

![Figure 2.6: A contour plot of temperature (K) for a fully-developed unit cell from the work of Gupta et al. (2010a). The constant wall heat flux boundary condition was used. The bubble interface and the boundaries of the unit cell are shown by a black line and two black vertical lines, respectively.](image-url)
Walsh et al. (2010) studied heat transfer in air-water Taylor flow in a capillary with a 1.5 mm diameter experimentally. They investigated the slug length effect on the heat transfer for a constant wall heat flux condition. In their paper, there are important inconsistency between their Figs. 7 and 8 which present the data and its analysis. Firstly, they confuse the void fraction \( e \) with the homogenous void fraction \( \beta \). Secondly, the \( e \) value quoted in the captions of Figs. 7 and 8 should be 0.67 and not 0.33 as given.

They reported that the Nusselt number increases with decreasing liquid slug length for a constant homogenous void fraction and mixture velocity. They explained their findings by the fact that the longer slug length has a velocity profile closer to a parabolic profile (related to the single-phase fully-developed velocity profile). However, they noted the same explanation in their previous work (Walsh et al., 2009) and it has been criticised by Gupta et al. (2010a) that this is not completely true, as the hydrodynamics of the developing flow in the liquid slug of gas-liquid Taylor flow is certainly different from the liquid-only flow. Gupta et al. (2010a) mentioned two reasons for the increase of Nusselt number in the slug flow: (1) the development of the flow in the liquid slug is made faster by the developed liquid film entering into the liquid slug, and (2) the radial velocities around the nose of the bubble cause convective heat transfer which is more important for the shorter liquid slugs.

Mehdizadeh et al. (2011) studied numerically the heat transfer characteristics of Taylor flow for a constant wall heat flux boundary condition. They simulated the experimental data of Walsh et al. (2010) for a circular channel of 2 mm external diameter including the solid wall thickness of 0.25 mm. In their study, they claim the liquid film was captured successfully employing a dynamic mesh adaption method with VOF method for capturing the gas-liquid interface. It is noted that they used this strategy in their previous work (Mehdizadeh et al., 2009) and they were able to obtain the same results as in the work of Gupta et al. (2009) by applying 96% less mesh than that they used. However, it is evident that there are extremely large spurious currents at the interface resulting from their non-uniform mesh size. Mehdizadeh et al. (2011) claimed that the effect of the liquid film on the overall heat transfer is significant, so the existence of the liquid film
should not be ignored in the modelling of Taylor flow. However, they confused the existence of high Nusselt numbers in the bubble region with this region being important for heat transfer. They also found that the average Nusselt number is independent of the wall material and wall thickness, while the local Nusselt number is affected these parameters.

Leung et al. (2012) investigated heat transfer in Taylor flow experimentally for the constant wall heat flux boundary condition. In their work, a vertical tube with a 2 mm diameter was used with vertical upward flow. They studied a wide range of Capillary numbers from 0.001 to 0.190 by using water, ethylene-glycol and 50 wt% ethylene-glycol/water mixture as the liquid phases. According to their findings, the Nusselt number for two phase flow increases with decreasing liquid slug lengths and this was explained by increasing the number of recirculation cycles in the shorter liquid slugs. It was observed that the Nusselt number increases with increasing mixture velocity. They also reported that the Nusselt number first increases with an increase in the homogenous void fraction until it reaches a maximum point which changes based on the the liquid phase fluid and then it decreases. In addition, they found that the heat transfer in Taylor flow depends on the liquid film thickness, the size of the recirculation regions in the liquid slug and the recirculation number. The works of Leung et al. (2010, 2012) have been used to validate the computational results of this thesis. More details of their works are reported in Chapters 4 and 5.

2.5 Summary

A number of investigations have been undertaken on the hydrodynamics and heat transfer of Taylor flow in microchannels. However, through a review of current literature, it is found that there are still some gaps in the literature for further work:

1. In many studies of the modeling Taylor flow using the laboratory frame of reference methodology, there is no control of the gas bubble length and the liquid slug length. It is now well-known that the heat transfer in Taylor flow depends on the length of the liquid slug and the gas hold-up. Therefore a comparison of the CFD simulation results
with the experimental data requires the bubble and slug lengths to be the same as those obtained from experiments.

2. Simulations of heat transfer using a laboratory frame of reference are too computationally costly to make detailed parametric studies.

3. Some researchers modelled fully-developed Taylor flow in a single unit cell in a reference frame moving with the bubble velocity. There is no clear explanation about the methodology used in the literature. In addition, these works have been studied only the hydrodynamics of fully-developed Taylor flow and heat transfer has not been investigated.

4. There are few works on CFD modelling of Taylor flow for which the computational results have been validated by experimental data hydrodynamically and thermally based on the same condition for wide ranges of Capillary numbers and Reynolds numbers.

In this study, the flow field and heat transfer of Taylor flow is investigated numerically with the aim of developing an improved computational methodology and validating it against data from a wide range of experimental conditions. In the next chapter, the equations and numerical methodologies used throughout this thesis to model Taylor flow are described.
Chapter 3

Approaches to CFD Modelling of Taylor Flow

In this chapter, two approaches developed to model Taylor flow are described and benchmarked. The first, and conceptually simplest, is to generate bubbles and slugs in a long tube by using a time-dependent boundary condition (referred to as a stationary domain here onwards). In the second method, the flow and heat transfer in a single unit cell, consisting of a bubble surrounded by liquid slugs, is solved in a frame of reference moving with the bubble velocity (referred to as a moving domain here onwards). Both methods were implemented in ANSYS Fluent.

3.1 Mathematical Model

As mentioned in Chapter 2, the VOF method was used in this study as implemented in ANSYS Fluent (ANSYS, 2010) to capture the gas-liquid interface. This is a proven technique for interface tracking in which an advection equation is solved for the volume fraction of one of the phases, giving the location of the interface.

3.1.1 Governing Equations

In interface-capturing methods, a “single-fluid” formulation is applied throughout the computational domain in which common velocity and pressure fields are shared amongst the phases. The conservation equations for mass, momentum and energy (equations (3.1)-(3.3)) are solved together with a volume fraction advection equation (equation (3.4)) to capture the gas-liquid interface.

Continuity:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \quad (3.1)
\]
3. Approaches to CFD Modelling of Taylor Flow

Momentum:

\[ \frac{\partial (\rho v)}{\partial t} + \nabla \cdot (\rho v \otimes v) = -\nabla p + \nabla \cdot (\mu (\nabla v + \nabla v^T)) + F_{sv} + g \]  \hspace{1cm} (3.2)

Energy:

\[ \frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho vh) = \nabla \cdot (k\nabla T) \]  \hspace{1cm} (3.3)

Volume fraction:

\[ \frac{\partial \alpha}{\partial t} + v \cdot \nabla \alpha = 0 \]  \hspace{1cm} (3.4)

The bulk properties, such as density, viscosity and thermal conductivity are determined from the volume fraction weighted-average of the properties of the two fluids. The arithmetic mean is the default in ANSYS Fluent for calculating all mixture properties. However, as discussed in Patankar [1980], use of harmonic averaging provides a better representation of the interfacial flux when the two fluids have very different transport properties. Both averages for calculating the mixture viscosity were used, as given in equations (3.5) and (3.6), to investigate the effect of this assumption. Harmonic averaging of the viscosity was not straightforward in ANSYS Fluent but was achieved using the procedure described in Appendix A via the user-defined function listed in Appendix B.

Arithmetic mean:

\[ \mu = \alpha_G \mu_G + (1 - \alpha_G) \mu_L \]  \hspace{1cm} (3.5)

Harmonic mean:

\[ \mu = \left( \frac{\alpha_G}{\mu_G} + \frac{1 - \alpha_G}{\mu_L} \right)^{-1} \]  \hspace{1cm} (3.6)
Simulations for an ethylene-glycol/nitrogen system showed that the use of a harmonic mean for the viscosity instead of an arithmetic mean did not produce any noticeable change in the bubble shape and other hydrodynamic characteristics of the flow. Therefore, given that the studied system had a higher viscosity difference than the air/water system, arithmetic averaging is used for all the future calculations.

The surface tension force is included in the momentum equation via the continuum surface force (CSF) model proposed by Brackbill \textit{et al.} (1992) and is represented as follows:

\[ F_{sv} = \sigma \kappa \delta (r - r_{int}) \mathbf{n} \quad (3.7) \]

\section*{3.1.2 Boundary and Initial Conditions}

The boundary and initial conditions for flow used in the two modelling approaches are different and are discussed in detail below. A no-slip boundary condition and a constant wall heat flux (H2) boundary condition were applied at the wall.

In the case of a stationary domain a time-dependent, periodic inlet boundary condition having a uniform velocity equal to the mixture velocity and a temperature equal to 300 K was employed. The volume fractions specified at the inlet were chosen such that in a given time period the volume of gas and liquid entering the channel corresponded with the desired volume of the gas bubble and liquid slug, respectively. A gauge pressure of 0 Pa was applied at the outlet boundary. Initially the computational domain had only liquid phase present, with a uniform velocity equal to the mixture velocity and a temperature of 300 K.

In the moving domain case, periodic boundary conditions, in which the velocity and modified temperature profiles were wrapped from the outlet to the inlet, were employed. A constant liquid flow rate was specified at the inlet. Initially, the computational domain contained a cylindrical gas bubble the size of which gives the desired gas volume
fraction. A parabolic velocity profile having an average velocity equal to the mixture velocity was applied at the inlet.

3.2 CFD Methodology

Two-dimensional, axi-symmetric, transient simulations to model vertical upward Taylor flow in a microchannel were performed using the commercial CFD package ANSYS Fluent 12.1.4 (ANSYS, 2010). The details specific to the stationary and moving domain approaches are presented in Sections 3.2.1 and 3.2.2, respectively.

Recall that the aim of this work is to simulate a fully-developed unit cell of length $L_{UC}$, having a bubble length (the distance between the nose and tail at the centreline) of $L_B$, with bubbles separated by a nominal slug length (the distance between the bubble tail to the following nose at the centreline) of $L_S$. In the stationary domain approach a computational bubble train is generated with a known two-phase velocity ($U_{TP}$) that produces fully-developed bubble and slug lengths close to the specified values. The void fraction is an output of the simulation depending on the homogeneous void fraction used. In the moving domain simulations the inputs are similar, except that it is the void fraction that is specified and the homogeneous void fraction is an output of the simulation.

3.2.1 Stationary Domain

The CFD methodology in the stationary domain simulations is similar to that employed in Gupta et al. (2009; 2010a). The methodology has been extended to generate gas bubbles and liquid slugs of pre-determined lengths by employing a time-dependent inlet boundary condition. Taylor flow simulations in the stationary domain can be performed by specifying the gas and liquid superficial velocities ($U_G, U_L$) and the gas and liquid volume fractions at the inlet, when a time-independent inlet boundary condition is used. The mixture velocity ($U_{TP}$) is defined at the inlet boundary and the bubble velocity ($U_B$) is obtained as part of the CFD solution. The homogenous void fraction is varied over the time required to produce a unit cell ($t_{UC}$) in such a manner that the time-averaged homogenous void fraction gives the desired void fraction in a unit cell.
3.2.1.1 Inlet Boundary Condition

A set of transient, periodic inlet boundary conditions were developed that allowed controlled amounts of gas and liquid to enter the computational domain. These were applied via the UDF given in Appendix C. A constant velocity \((U_{TP})\) and temperature are set at the inlet for all time. Oliver and Hoon (1968) controlled the bubble length in their experiments by setting a controlled amount of air and liquid to enter the channel alternatively. A similar approach has been adopted in the simulations of the stationary domain method. The amount of liquid and gas entering the channel, having only liquid present initially, in a time period \((t_{UC} = L_{UC}/U_{TP})\) is controlled to give the desired liquid (slug and film) and gas (bubble) volumes in a unit cell (a single bubble and a liquid slug). A schematic of the inlet conditions is shown in Fig. 3.1. In one time period, first a gas core with a thin annular liquid film is allowed to enter the domain until the gas volume reaches that of a gas bubble \((0 < t < t_G)\) and then only liquid enters the inlet until a liquid volume equal to that of the slug has entered \((t_G < t < t_{UC})\). The same process is repeated periodically to define a train of bubbles and slugs. It is noteworthy that the wall contact angle is not required in this simulation as the bubble generation process is not modelled in detail and there is no gas-liquid-wall interaction in the problem.
Figure 3.1: Schematic of the bubble generation process for one time period (a) when a bubble and the liquid surrounding it enter the channel (b) when only liquid enters the channel.

Here it should be explained that the simple approach of injecting gas across the entire inlet over a certain time period and then liquid, allowing a Taylor bubble to form does require modelling of the gas-liquid-wall contact and does not necessarily capture the liquid film separating the bubble, as seen in Fig. 3.2(a) for water and nitrogen as working fluids. The use of an annular liquid film surrounding the gas core at the inlet ensured the existence of the liquid film between the bubble and channel wall (see Fig. 3.2(b)). The thickness of this annular liquid layer can be chosen arbitrarily, as the true thickness of the film separating the gas bubble and wall is obtained from the simulations and is independent of the inlet liquid layer thickness. However, the use of a liquid film that is too thick can cause the undesired effect of gas bubble break-off that is too early (see Fig. 3.2(c)). Therefore, the Bretherton (1961) correlation, given in equation (3.8), was used to set the thickness of the annular liquid layer at the inlet \((R - R_{in})\), via

\[
(R - R_{in})/R = 1.34 \, Ca^{2/3}
\]  

(3.8)
The use of this equation for the inlet liquid layer thickness calculation always resulted in the production of just one bubble from each period of gas injection.

Figure 3.2: Volume fraction plot for fluids of water and nitrogen for cases with: (a) no liquid layer present at the inlet, (b) with a liquid layer thickness calculated using Bretherton's correlation \( R = 0.25 \text{ mm}, R_m = 0.24 \text{ mm} \), and (c) a thicker liquid layer \( R = 0.25 \text{ mm}, R_m = 0.22 \text{ mm} \). In Figure 3.2(b), the liquid film is shown in an enlarged plot. The gas and liquid are coloured blue and red, respectively.

Figure 3.3: Volume fraction plot for a case using ethylene-glycol and nitrogen as the fluids, in which there is no liquid layer present at the inlet \( R_m = 0.25 \text{ mm} \). The nitrogen and ethylene-glycol are coloured blue and red, respectively.

In the case of a highly viscous fluid, such as ethylene-glycol \( (\mu \sim 0.17 \text{ Pa.s}) \), it was possible to inject gas across the entire inlet. As shown in Figure 3.3, in such cases a liquid film remains at the wall throughout the domain and it was not necessary to impose this at the inlet boundary. The criteria that determine whether or not this minor simplification can be invoked were not investigated and it is recommended that this be treated on a case-by-case basis.
The required time for creating the bubble \((t_G)\) is obtained from an approximate gas volume balance, and is given by:

\[
t_G = \frac{L_{cy}l}{U_{TP}}
\]  

(3.9)

where \(L_{cy}l\) is the length of a cylindrical bubble of radius \(R_{in}\) and having the same volume as that of the gas bubble.

An approximate liquid volume balance for the slug gives the slug creation time, \((t_L)\), of:

\[
t_L = \frac{L_{UC} - L_{cy}l}{U_{TP}}
\]  

(3.10)

The total time \((t_{UC})\) for creating one unit cell (one bubble and one slug) is given by:

\[
t_{UC} = t_G + t_L = \frac{L_{UC}}{U_{TP}}
\]  

(3.11)

The times \((t_{UC}, t_G, t_L)\) from the above equations can be used to generate a single gas bubble and liquid slug. To create multiple bubbles, non-dimensional bubble and unit cell creation times, given by \(t_G^*\) and \(t^*\), respectively, are defined:

\[
t_G^* = \frac{t_G}{t_{UC}}
\]  

(3.12)

\[
t^* = \frac{t}{t_{UC}} - \text{int}\left( \frac{t}{t_{UC}} \right)
\]  

(3.13)

where "int" is a function that returns the integer part of its argument. The calculated values of \(t_G^*\) and \(t^*\) from equations (3.12) and (3.13) can be used as the first estimates at the beginning of the simulation. For cases where experimental data were available i more sophisticated estimates of the gas and liquid injection times were tried, for example, by
assuming a bubble shape that had elliptical ends rather than making it cylindrical. However, these practices were found to give only a marginally better first estimate for the bubble and slug lengths. Instead the injection times were modified, assuming linear variations of the bubble/slug lengths with the gas/liquid injection time, during the creation of subsequent bubbles and slugs to achieve the desired lengths.

This methodology provides a tool to study simultaneous development of flow and heat transfer together with, for a tube of sufficient length, fully-developed Taylor flow. However, a direct comparison of computed developing flow with experimental data for developing flow is only possible when the bubble generation mechanism is the same as that in the experiments.

3.2.2 Moving Domain

The implementation of the model in a moving domain is more complicated than that for a stationary domain and has often been poorly explained in previous work. Here the approach is described in detail so that it is clear what assumptions are made.

3.2.2.1 Background

Some researchers have modelled two-phase Taylor flow in a single unit cell by moving the channel wall at a velocity equal to that of the bubble but in a direction opposite to the flow (Taha and Cui, 2004; Wang and Liu, 2008) thus keeping the bubble stationary. In these simulations the assumption that the flow at the upstream and downstream boundaries is fully-developed was applied, simplifying the modelling.

Akbar (2004) and Akbar and Ghiaasian (2006) used a dynamic mesh moving with the bubble velocity to keep the bubble stationary with respect to the computational mesh. They used periodic boundary conditions to model the hydrodynamics of fully-developed Taylor flow. A similar methodology has been implemented in this thesis to model fully-developed Taylor flow and has been also extended to model heat transfer for a constant wall heat flux (H2) boundary condition.
The dynamic mesh model in ANSYS Fluent can be used to model flows where the shape of the domain is changing with time due to motion of the domain boundaries or when the domain is moving with a specified velocity. Since in this work the computational domain is moved in the streamwise direction without any change in its shape, a ‘rigid body’ treatment was used, where the rigid body moved with a velocity equal to that of the bubble. The mesh is updated at each time step using a dynamic layering method which removes layers of cells adjacent to the inlet boundary and adds them at the outlet boundary. The fluid velocity when using a dynamic mesh has two components: the velocity of the dynamic mesh (bubble velocity in our case) and the fluid velocity in a frame of reference moving with the bubble.

As the bubble velocity is not known a priori, it is calculated at the beginning of each time-step using equation (3.14) via the user-defined function (UDF) given in Appendix D. A gas volume-fraction-weighted axial velocity is used and is calculated via:

\[ U_B = \frac{\int_{x}^{x+L_C} \int_{0}^{R} \alpha_G u_x(2\pi r) dr dx'}{\int_{x}^{x+L_C} \int_{0}^{R} \alpha_G (2\pi r) dr dx'} \]  

(3.14)

As the bubble always remains inside the domain, the volume of gas specified initially remains unchanged. Therefore this approach requires the correct volume of gas in a unit cell, i.e. the void fraction \( \varepsilon_G \), to be specified initially. At the start of a simulation a cylindrical bubble shape is set with a volume equal to the gas void fraction \( \varepsilon_G \) times the unit cell volume. The bubble shape is an outcome of the simulation, evolving to a time-independent shape. A fully-developed single phase parabolic velocity profile, with the average velocity corresponding to the mixture velocity \( U_{TP} \), is set in the entire domain for initialising the flow. Thus the void fraction \( \varepsilon_G \) and the mixture velocity \( U_{TP} \) are defined initially and the bubble velocity \( U_B \) and homogeneous void fraction \( \beta \) are obtained from the computation.
3.2.2.2 Periodic Boundary Conditions

As mentioned in Chapter 2, Taylor flow is a steady, periodic flow in a frame of reference moving with the bubble velocity. Therefore in theory periodic boundary conditions need to be defined for the velocity, the volume fraction and a scaled temperature field. As shown in Fig. 3.4, for the unit cell configuration used here the boundaries of the domain are chosen such that the liquid phase only is always present at the boundaries (and the liquid/gas volume fraction at both the boundaries is same), avoiding the need to force a periodic boundary condition for the volume fraction.

\[
\begin{align*}
  u_{x,r} &= u_{x+L_Uc,r} \\
  v_{x,r} &= v_{x+L_Uc,r}
\end{align*}
\]

Figure 3.4: Schematic of the geometry and boundary conditions used in the dynamic mesh method.

Therefore, in the hydrodynamic solution the velocity field at the upstream boundary \(x\) was wrapped from the downstream boundary \(x+L_Uc\), as shown in Fig. 3.4. As shown in equation (3.15), the term ‘wrapping’ here refers to specifying the upstream value of a variable, such as a velocity component or scaled temperature to be equal to the downstream value at the corresponding radial location. (In the Fluent setup a velocity inlet was used at the left hand boundary and a constant pressure of 0 Pa was set at the outlet.) A user-defined function (UDF) is used to achieve this in ANSYS Fluent, as given in Appendix D.
After one iteration the velocity profile was initialised everywhere and wrapping was commenced. This wrapping makes the velocity periodic but as the velocity gradients are not wrapped a truly periodic flow is not achieved. In Rosaguti et al. (2005) work involving periodic single-phase laminar flow, three unit cells were modelled instead of one, so that the inner cell was closer to being fully-developed. This approach could possibly be extended to multiphase flows but it has not been investigated in this work.

To obtain a periodic temperature field, a modified temperature profile is wrapped from the computational faces having flow out of the domain to the corresponding faces where flow enters the domain. In order to determine the direction in which temperatures are wrapped we must look at the problem in the domain moving with the bubble. As shown in Fig. 3.5, there is now inflow and outflow at each boundary due to the existence of a recirculation zone in the liquid slug when viewed in the bubble frame of reference. It is straightforward to identify the cells that must be treated as being an "inlet" and therefore have a modified temperature mapped to them from the corresponding "outlet" cell. In ANSYS Fluent the option of setting a backflow temperature at an outlet was used to implement this boundary condition. At the inlet, temperature is wrapped at all cells but the values are only used where there is flow into the domain.

![Diagram](image)

Figure 3.5: Velocity profiles with respect to the dynamic mesh at the periodic interfaces. The direction of temperature wrapping is shown.
3. Approaches to CFD Modelling of Taylor Flow

3.2.2.3 Thermal Boundary Condition

A constant wall heat flux (H2) boundary condition is used for studying heat transfer. An approach very similar to that used in the work of Rosaguti et al. (2005) for single-phase periodic heat transfer is used here. For a constant wall heat flux the temperature profile at the inlet is the same as that at the outlet, but shifted by an amount that depends on the heat addition at the wall. Equation (3.16) is obtained from an overall energy balance over a unit cell and is used to determine the inlet temperature from the corresponding inflow temperature, as shown in Fig. 3.5. A detailed derivation of equation (3.16) is presented in Appendix E.

\[ T_{x,r} |_{x+UC} - T_{x,r} |_{x} = \Delta T_{UC} = \frac{q_{w}A_{wUC}}{(1 - \beta)\rho_{L}c_{pL} + \beta\rho_{G}c_{pG}]U_{TP}A_{XS}} \]  

(3.16)

The above equation can be rewritten as

\[ T_{x,r} |_{x+UC} - T_{x,r} |_{x} = \Delta T_{UC} = \frac{q_{w}A_{wUC}}{[(U_{TP} - \varepsilon_{G}U_{B})\rho_{L}c_{pL} + \varepsilon_{G}U_{B}\rho_{G}c_{pG}]A_{XS}} \]  

(3.17)

for which all of the parameters except the bubble velocity \((U_{B})\) are specified at the start of the simulation. The bubble velocity \((U_{B})\), calculated at each time step using equation (3.14), is used in equation (3.17).

3.3 Numerical Methods

The fractional step co-located scheme, in which pressure and velocity are stored at the cell centres, is used for the pressure-velocity coupling and a body-force-weighted interpolation scheme is chosen to compute the face pressure. ANSYS Fluent provides an optional “implicit body force” treatment that takes into account the partial equilibrium of the pressure gradient and body forces and this is used in these simulations. An explicit geometric reconstruction scheme, which is based on the piecewise linear interface calculation (PLIC) method (Youngs, 1982), is applied to keep the gas-liquid interface sharp. A maximum Courant number of 0.25 is used for the volume fraction equation. A
first order non-iterative time-marching scheme having a variable time step based upon a Courant number of 0.25 is used for the integration of the transient term. The convective terms in the momentum and energy equations are discretised using the QUICK scheme. These choices reflect the best practices presented in Gupta et al. (2009).

3.4 Verification Results and Discussion

Two-dimensional, axisymmetric, transient simulations for Taylor flow and heat transfer in a circular channel of diameter 2 mm were carried out using both the stationary and the moving domain methods. The length of the domain was 100\(d\) for the stationary domain approach and was set to the desired unit cell length of 6.75\(d\) in the moving domain method. Nitrogen and water were chosen as the working fluids. The properties of the fluids used, assumed constant as required for a periodic solution, are given in Table 3.1. In the stationary domain case the gas and liquid inlet temperatures were set to 300 K. A heat flux of 32 kW m\(^{-2}\) was applied at the channel wall for the heat transfer simulations.

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Density (kg m(^{-3}))</th>
<th>Dynamic viscosity (Pa s)</th>
<th>Thermal conductivity (W m(^{-1})K(^{-1}))</th>
<th>Specific heat capacity (J kg(^{-1})K(^{-1}))</th>
<th>Surface tension (N m(^{-1}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water @ 25°C</td>
<td>997.0</td>
<td>89\times10^{-5}</td>
<td>0.6000</td>
<td>4182</td>
<td>0.072</td>
</tr>
<tr>
<td>Nitrogen @ 25°C</td>
<td>1.145</td>
<td>2.1\times10^{-5}</td>
<td>0.0242</td>
<td>1040</td>
<td></td>
</tr>
</tbody>
</table>

3.4.1 Comparison of the Stationary Domain and Moving Domain Results

In the following sections, the results obtained for water and nitrogen having a mixture velocity \((U_{TP})\) of 0.32 m s\(^{-1}\) and void fraction \((\epsilon_G)\) of 0.366 using the stationary and moving domain methodologies are presented and compared. These conditions correspond
to a two-phase Reynolds number $Re = 713$ and Capillary number $Ca = 0.004$, based on the liquid properties and mixture velocity.

### 3.4.1.1 Computational Mesh

As explained in the work of Gupta et al. (2009), the computational mesh is maintained square in the near axis region where most of the interface lies so that the direction of the surface tension force is calculated correctly and the spurious velocities caused by discretisation error in the implementation of the surface tension force are minimised. In the near wall region the mesh is refined in the radial direction to capture the thin liquid film and to accurately resolve the temperature gradients in the thermal boundary layer in the film. The liquid film surrounding the bubble had 17 mesh elements in the radial direction and the radial size of the elements reduced from the interface to the wall. Following a mesh-independence study, a mesh having 650,000 cells was used for the stationary domain simulations, compared with one having 92,000 cells in the moving domain case.

In Fig. 3.6, the gas-liquid interfaces obtained from the moving domain simulations using different grid sizes are shown. It can be seen that the bubble shapes and the liquid film thicknesses remain unchanged with increasing the grid numbers more than $93\times1000$. The simulated bubble velocities for the cases 2 and 3 are also very close taking the values of 0.348 and 0.347, respectively. In addition, the average Nusselt numbers ($Nu_{av}$) for these two cases (2 and 3) were calculated and the values obtained were 10.8 and 10.9 which have less than 1% difference. Therefore, the grid resolution of $93\times1000$ was good enough to simulate the flow and heat transfer for this work.
3. Approaches to CFD Modelling of Taylor Flow

3.4.1.2 Computational Time

The simulations using the stationary domain approach were run on a 64 bit HP xw 8600 workstation having an Intel Xeon X5472 CPU, 16 GB RAM. Using 4 processors for the computations it took about 1,460 hours of wall clock time to simulate the injection of 14 bubbles. Some bubbles travelled far enough for the flow to be hydrodynamically fully-developed but the tube was not long enough for thermally-developed flow to occur. The moving domain method simulations were run on a similar 64 bit HP Workstation using only a single processor, as the UDF used to implement the wrapping was not readily parallelised. It took about 29 hours to obtain a steady solution for the hydrodynamics alone. Once the steady solution was obtained, the flow field (continuity, momentum and VOF equations) was ‘frozen’ and only the energy equation was solved, starting with an initial temperature of 300 K, until a steady non-dimensional temperature field was obtained. Solving the energy equation alone also relaxed the requirement of a very low time step governed by a Courant number of 0.25 for the flow field (of around 1 μs) and a time step of typically 0.001 s was used for time-marching of the energy equation. This...
resulted in enormous saving in the computational time and the temperature field reached a steady-state in just 9 hours.

3.4.1.3 Bubble Shape

In the stationary domain case, the simulations were run until there were 14 bubbles in the domain. The second to thirteenth bubbles had the same shape and the velocity field was found to be periodic for these unit cells. Thus the flow was hydrodynamically fully-developed from the second to the thirteenth unit cell. The bubble shape from the sixth unit cell along the channel is compared with that obtained via the moving domain method in Fig. 3.7. The bubble shapes are very similar, each giving a 41 µm film thickness in the constant film region. The lengths of the gas bubbles and liquid slugs are $3.05d$ and $3.70d$, respectively, for both the stationary and moving domains. In the stationary domain, the value of the gas hold-up, $(\varepsilon_g)$, was calculated from the integrated gas volume fraction in a unit cell, which gave a value of 0.366, which is identical to the value set in the periodic domain solution.
3. Approaches to CFD Modelling of Taylor Flow

3.4.1.4 Flow Field

The bubble velocity in the moving and stationary domain computations is calculated using equation (3.14). The values obtained are 0.348 m s\(^{-1}\) and 0.347 m s\(^{-1}\) for the moving and stationary domains, respectively.

Figure 3.8 compares the fully-developed, normalised velocity profiles halfway along the length of the liquid slugs obtained from each of the simulation methods. Also shown for illustration purposes is the parabolic velocity profile for the liquid-only, fully-developed laminar flow. It can be seen from Fig. 3.7 that, for either method, the velocity profile in the liquid slugs is close to parabolic, with the centreline velocity being 1.93 and 1.88 times the average velocity \((U_{TP})\) for the moving and stationary domains, respectively. This degree of approach to a fully-developed pipe flow profile in the liquid slug would
require a channel length of \(~40d\) in a single-phase hydrodynamically developing flow; correspondingly a developing flow pipe length of \(1.85d\) (corresponding to the slug half-length) would produce a centreline velocity of just \(1.3U_T\) (Hornbeck, 1964). Clearly the flow development in the slug cannot be described simply in terms of developing pipe flow – much shorter development lengths in the slug arise because the velocity profile is not flat at the nose of the bubble (start of the slug) and more importantly, the velocity gradients in the near-wall region are well developed at the start of the slug.

![Normalised velocity profile in the middle of the liquid slug for the moving (dashed line) and stationary domain (solid line) approaches. The parabolic velocity profile for the fully-developed laminar flow is also shown by a dotted line.](image)

Figure 3.8: Normalised velocity profile in the middle of the liquid slug for the moving (dashed line) and stationary domain (solid line) approaches. The parabolic velocity profile for the fully-developed laminar flow is also shown by a dotted line.
Figure 3.9: Local wall shear stress in the axial direction. In part (b), the scale has been changed to show the small differences in the calculated wall shear stress in the moving and fixed domain methods. The solid lines are for the stationary domain while the dashed lines are for the moving domain. The location of the bubble is also shown by a dotted line.
Figure 3.9(a) shows that the wall shear stresses obtained from the two methods are very similar. A closer examination reveals that there are slight differences in the bubble film region, as well as in the liquid slug region. Both the shear stresses at the boundaries of the unit cell and the shear force over a unit cell differ by ~2 % in the two cases. It is evident from Figure 3.9(b) that the shear stress is not perfectly periodic, as expected given the velocity gradients were not wrapped. The small differences in the velocity profile shown in Fig. 3.8 were attributed to be a consequence of the small differences in the wall shear stress.

3.4.1.5 Heat Transfer for a Constant Wall Heat Flux (H2) Boundary Condition

Figure 3.10 shows the axial variation of the local wall and bulk temperatures obtained for the stationary domain method. The local bulk temperature at each axial location is calculated using equation (3.18).

\[
T_{b,x} = \frac{\int_0^R \rho |u_x| c_p T(2\pi r) dr}{\int_0^R \rho |u_x| c_p (2\pi r) dr}
\]  

(3.18)

The peak value for the wall temperature occurs at the bubble tail and it decreases in the slug. The high heat capacity rate \((\hat{m}c_p)\) of the liquid in comparison with that of the gas causes the low and high bulk temperatures in the liquid slug and gas bubble regions, respectively. The straight line shown represents the bulk temperature increment for a homogeneously mixed two-phase flow. As discussed in Appendix E, when the temperature field is fully-developed, the increment in bulk temperature over a unit cell is exactly that for a homogeneous flow.

From a comparison of the bulk temperature increment over a unit cell with that obtained for a thermally fully-developed unit cell (Appendix E) only a few unit cells in the middle (fourth to seventh) appear to have reached a thermally fully-developed state. The irregular temperature pattern in the last few unit cells is because of the developing temperature field near the entrance region. While the bulk and wall temperature pattern have developed well in the first few unit cells, the temperature increment (or decrement...
in some cases) is different from that of a fully-developed unit cell. The liquid ahead of the first bubble provides a larger heat sink than the slug between two consecutive bubbles, resulting in lower bulk temperatures in the region of the first few bubbles.

Figure 3.10: Variation of the bulk and wall temperatures for the stationary domain. The dashed straight line shows the temperature increment for a homogeneously mixed two-phase flow. The location of the bubbles is also shown by dotted lines.

In Figure 3.11, the issue of thermal development is investigated further. Figure 3.11(a) shows the volume fraction plot and boundaries (shown by black lines) of the fourth to seventh unit cells. In order to quantify the extent of development of the temperature field the normalised temperature difference was calculated as follows:

\[ T_{\text{dif}} = \frac{(T_{x+\lambda_{UC},r} - T_{x,r})}{\Delta T_{UC}} \]  \hspace{1cm} (3.19)

For perfect thermal development of a unit cell \( T_{\text{dif}} \) would take a value of unity at all radial locations. Temperature data for the unit cells shown in Figure 3.11(a) are shown in
Figure 3.11(b). It shows that the temperature profile of the sixth unit cell is closest to complete development, but it is still not completely periodic having temperature differences of 20% of the bulk temperature rise across the cell. It is evident that the stationary domain method needs even more unit cells to reach a thermally fully-developed state, further highlighting the computational impracticality of this approach.

Figure 3.11: Heat transfer development analysis for the fixed domain approach and a constant wall heat flux boundary condition. (a) Volume fraction plot for the fourth to seventh cells, and (b) Radial distribution of the normalised temperature increment across the most thermally-developed unit cells.

Figure 3.12 compares the axial variation of the local wall and bulk temperatures as obtained from the moving domain and the sixth unit cell of the stationary domain method.
In the moving domain simulations the temperature is not fixed at any location and so the level varies during the simulation. In order to allow a direct comparison of the two temperature fields the temperatures from the moving domain calculation were shifted to have the same value at the inlet wall as in the fixed domain case. It can be seen that the wall temperatures calculated by the two methods are very close to each other, but the bulk temperature in the moving domain case is \(~1\) K lower than that in the stationary domain case in the liquid slug region.

![Graph showing temperature variation](image)

**Figure 3.12:** Variation of the bulk and wall temperatures for the unit cell of the moving domain and the stationary domain. The moving domain temperatures are shown by dashed lines. A constant heat flux thermal boundary condition is applied at the walls.

The variation of the local Nusselt number ($N_u$) defined by equation (3.20) (Gupta et al., 2010a), is plotted for the two methods in Fig. 3.13(a).
\[ Nu_x = \frac{q_{wx}}{(T_w - T_{bx}) k_L} \]  

(3.20)

As can be seen in Fig. 3.13(a), the local Nusselt numbers obtained using the two methods have some differences in the gas bubble region, which may be the result of the small difference in the liquid film thickness. In Fig. 3.13(b), the scale of the local Nusselt number axis is changed to show the difference in the liquid slug. The local Nusselt number in the liquid slug region in the moving domain case is approximately 10% lower than that in the stationary domain case.

\[ Nu_{av} = \frac{Q_{W av}}{(T_{W av} - T_{b av}) k_L} \]  

(3.21)

The parameters of \( q_{W av} \), \( T_{W av} \) and \( T_{b av} \) are defined by the following equation:

\[
T_{b av} = \frac{\int_x^{x+L} \int_0^R \rho |u_x| c_p T(2\pi r) dr dx'}{\int_x^{x+L} \int_0^R \rho |u_x| c_p (2\pi r) dr dx'} ; \quad T_{W av} = \frac{1}{L} \int_x^{x+L} T_w dx' \quad \text{and} \quad q_{W av} = \frac{1}{L} \int_x^{x+L} q_{wx} dx' \]  

(3.22)

The average Nusselt number obtained for the stationary and moving domains are 11.4 and 10.8, respectively. It can be seen that the values of the average Nusselt numbers are very close to the local Nusselt number values of the liquid slug regions for each of the methods, consistent with the observation that the contribution of the gas phase to heat transfer is negligible. The fact that the average Nusselt number for the fixed domain exceeds that for the moving domain is consistent with the observation (Fig. 3.11) that the unit cells in the fixed domain are not fully developed thermally.
Figure 3.13: Variation of the local Nusselt number for the unit cell of the moving domain and the stationary domain (sixth unit cell). The moving domain results are shown with a dashed line. A constant wall heat flux is set at the wall. In part (b), the scale of the local Nusselt number is restricted to focus on the slug region. The location of the bubble is also shown by a dotted line.
In addition to the local Nusselt number ($Nu_x$) comparison, the average Nusselt number ($Nu_{av}$) for a unit cell, which is a more meaningful definition for comparison with experimental data, is calculated using equation (3.21) (Gupta et al., 2010a) for both methods.

It is evident that the moving domain methodology is vastly better than the stationary domain methodology, as the simulation time is reduced significantly and fully-developed hydrodynamics and heat transfer results are obtained. In the stationary domain case, even after using a computational domain that incorporated fourteen unit cells, it has proved infeasible to obtain heat transfer data for fully-developed conditions.

3.5 Summary

Two different CFD methodologies have been developed to enable Taylor bubbles with controlled bubble and slug lengths to be simulated. The first, and simplest, is to simulate a bubble train in the laboratory frame of reference. In this approach, the bubbles and slugs of desired length were created by employing a transient inlet boundary condition, which allowed a known amount of gas and liquid to enter the computational domain alternately. By having a sufficiently long solution domain fully-developed values can be obtained. The second methodology was to simulate the behaviour of a unit cell using a flow domain that moved with the bubble velocity.

Simulations for a two-phase Reynolds number of 713, Capillary number of 0.004 and void fraction of 0.366 for water/nitrogen flow are presented to compare the two techniques. In the fixed domain approach thermally fully-developed flow could not be obtained even after generation of 14 bubbles. The computational time required for the moving domain approach was much lower (wall clock time of 38 hours on a single processor) compared with 1460 hours using four processors for the stationary domain approach. The hydrodynamic and heat transfer results obtained from the two approaches were found to be very similar, giving confidence in the implementation of both methods.
In the next Chapter the moving domain method is used to make a study of the hydrodynamics of fully-developed Taylor flow.
Chapter 4

Validation of Simulation Results for Hydrodynamics

In this chapter, the moving domain method, which was described and compared with the stationary domain method in Chapter 3, is used to compare simulation results for the hydrodynamics of Taylor flow with the experimental data of Leung et al. (2010, 2012).

4.1 Input Parameters

Leung et al. (2010, 2012) studied experimentally the hydrodynamics and heat transfer of Taylor flow in vertical upward flow in a 2 mm diameter circular channel. In the experimental study, nitrogen was used as the gas phase and two liquids, water and ethylene-glycol, were investigated. These cover a two phase Reynolds number range of 200-1100 (water/nitrogen system), 10-60 (ethylene-glycol/nitrogen system) and a Capillary number range of 0.001-0.007 (water/nitrogen system), 0.034-0.162 (ethylene-glycol/nitrogen system). The properties of water and ethylene-glycol for temperatures ranging from 20°C to 40°C were determined from steam tables (IAPWS-IF97) and Sun and Teja (2002), respectively.

For validation of the simulation results with the experimental data of Leung et al. (2010, 2012), two-dimensional, axisymmetric, transient simulations in a circular channel of diameter 2 mm were carried out. As gravity acts along the axial direction in vertical upward flow, it is possible to compare the two-dimensional, axisymmetric simulation results with the experimental data including the effect of gravity. The two systems, that used water/nitrogen and ethylene-glycol/nitrogen, were studied in the simulations and the experiments. The physical properties of the working fluids used in the simulations were specified as being constant to allow fully-developed flow and, subsequently, heat transfer simulations to be performed. The values used are given in Table 4.1. It is noteworthy that
4. Validation of Simulation Results for Hydrodynamics

the experimental hydrodynamics results were collected at a room temperature of 21°C, while the temperature changes during the experimental study of heat transfer. Therefore, the constant properties used for the modelling of hydrodynamics were specified at a temperature of 21°C for the ethylene-glycol/nitrogen system. As the effect of the temperature on the water properties was not significant, the temperature used for the constant properties for the water/nitrogen system was selected approximately in the middle range of the experimental temperatures for the heat transfer cases (25°C). However, some of the simulations for the water/nitrogen system were repeated for the hydrodynamics using the properties at 21°C temperature to show the assumption used is acceptable. The results obtained at 21°C for the water/nitrogen system are denoted by a (*) superscript.

Table 4.1: Properties of the gas and the liquids used in the simulations.

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Density (kg m(^{-3}))</th>
<th>Dynamic viscosity (kg m(^{-1}) s(^{-1}))</th>
<th>Thermal conductivity (W m(^{-1}) K(^{-1}))</th>
<th>Specific heat capacity (J kg(^{-1}) K(^{-1}))</th>
<th>Surface tension (N m(^{-1}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water @ 25°C</td>
<td>1.145</td>
<td>2.1×10(^{-5})</td>
<td>0.242</td>
<td>1040</td>
<td>0.048</td>
</tr>
<tr>
<td>Ethylene Glycol @ 21°C</td>
<td>1.114</td>
<td>0.0204</td>
<td>0.2510</td>
<td>2627</td>
<td></td>
</tr>
<tr>
<td>Nitrogen @ 25°C</td>
<td>1.145</td>
<td>2.1×10(^{-5})</td>
<td>0.242</td>
<td>1040</td>
<td></td>
</tr>
<tr>
<td>Nitrogen @ 21°C</td>
<td>1.161</td>
<td>2.08×10(^{-5})</td>
<td>0.242</td>
<td>1040</td>
<td></td>
</tr>
</tbody>
</table>

As the moving domain method was used for the simulations, the length of the domain was set equal to the experimentally determined unit cell length in each case. The boundary and initial conditions used in the simulations and their relationship with measured quantities are described in the following subsections.
4. Validation of Simulation Results for Hydrodynamics

4.1.1 Mixture Velocity ($U_{TP}$)

In the moving domain approach the mixture velocity ($U_{TP}$) is one of the parameters used to define the system. A fully-developed single phase velocity profile (parabolic profile) with the average velocity corresponding to the experimental mixture velocity ($U_{TP}$) is set in the entire domain for initialising the flow. The wrapping process maintains the average velocity the same as the initial value of $U_{TP}$ but allows the shape of the velocity profile to change.

4.1.2 Gas Hold-up ($\varepsilon_G$)

As explained in the moving domain methodology, the gas bubble is stationary inside the domain so the initial gas void fraction ($\varepsilon_G$) must be set to the desired value for the simulation, and in such a manner that bubble and slug lengths similar to the experimental results are obtained as computationally efficiently as possible. The gas hold-up is specified in two different ways depending on the liquid phase material (water or ethylene-glycol). First, the unit cell volume is calculated based on the experimental data, including the chosen unit cell lengths and the diameter of the microchannel ($d$).

For the ethylene-glycol/nitrogen system, an initial cylindrical bubble shape is set in the middle of the domain with the length equal to the experimental bubble length and the radius equal to the tube radius ($R$) minus the measured film thickness ($d_f$). This assumption gives a good estimate of the bubble volume for the low mixture velocity cases. However, the bubble head becomes sharper when the mixture velocity increases (Leung et al., 2010), so in this case the bubble length is modified to give the same bubble volume as that determined from the experimental images by accounting for the elliptical shape of the nose.

In the experimental work using water as the liquid phase, the direct measurement of the film thickness ($d_f$) was not possible. Therefore, the value of the gas hold-up ($\varepsilon_G$) is obtained by using equation (4.1) which is derived from a simple volume balance.
4. Validation of Simulation Results for Hydrodynamics

\[ \frac{\varepsilon_G}{\beta} = \frac{U_{TP}}{U_B} \]  

(4.1)

The experimental values of the bubble velocity \((U_b)\), mixture velocity \((U_{TP})\) and homogenous void fraction \((\beta)\) are used in the equation. Then, the volume of the cylindrical bubble shape is obtained by multiplying the gas void fraction \((\varepsilon_G)\) by the unit cell volume.

4.2 Hydrodynamics Results

As mentioned in Section 4.1, the two-dimensional, axisymmetric simulations were used for the modelling of the experimental data of Leung et al. (2010, 2012) in this study. Leung et al. (2010, 2012) showed that unsteady ripples were observed at the bubble tail at high mixture velocities of 0.42 m s\(^{-1}\) and 0.53 m s\(^{-1}\) for the water/nitrogen system. Therefore, it can be found from Fig. 4.1 taken from Leung et al. (2012) that the accuracy of the two-dimensional, axisymmetric assumption for the CFD modelling becomes worse with increasing \(U_{TP}\) for \(U_{TP}\) values above 0.32 m s\(^{-1}\). However, the effect of this assumption on the simulation results was investigated and the findings are reported below. It is important to mention that the unsteady behaviour was not found in the simulations for the water/nitrogen system until \(U_{TP} = 0.53\) m s\(^{-1}\). In the following sections, the steady and unsteady cases for the experiments and simulations are defined in the various Tables for the water/nitrogen system results.
4. Validation of Simulation Results for Hydrodynamics

Figure 4.1: Experimental images of bubble heads and tails with $\beta = 0.5$ for the water/nitrogen system at (a) different mixture velocities. Taken from Leung et al. (2012). In parts (b) and (c), the transient bubble tails are shown for $U_{TP} = 0.42 \text{ m s}^{-1}$ and $U_{TP} = 0.53 \text{ m s}^{-1}$, respectively.

It is worth noting here that unsteady bubble tail shapes were not observed for the ethylene-glycol/nitrogen system in either the simulations or the experiments because of the low Reynolds numbers ($22 < Re < 40$) studied.
4.2.1 Bubble Velocity ($U_B$)

The bubble velocity is an important parameter which can be used for validating the computational results. In Figures 4.2 and 4.3 the variation of the bubble velocity ($U_B$) with the mixture velocity ($U_{TP}$) is shown for the water/nitrogen and ethylene-glycol/nitrogen systems, respectively. Liu et al. (2005) obtained a bubble velocity correlation for upward flow in vertical channels, given in equation (4.2). This correlation is valid for $Ca$ from 0.0002 to 0.39.

$$\frac{U_B}{U_{TP}} = \frac{1}{1 - 0.61Ca^{0.33}}$$  \hspace{1cm} (4.2)

Figure 4.2: The effect of the mixture velocity ($U_{TP}$) on the bubble velocity ($U_B$) obtained from CFD simulations (○) and the experiments (□) for the water/nitrogen system. The correlation of Liu et al. (2005) is shown by a dashed line.
4. Validation of Simulation Results for Hydrodynamics

Figure 4.3: The effect of the mixture velocity ($U_{TP}$) on the bubble velocity ($U_B$) obtained from CFD simulations (○) and the experiments (□) for the ethylene-glycol/nitrogen system. The correlation of Liu et al. (2005) is shown by a dashed line.

It can be concluded from these plots that there is very good agreement between the bubble velocities obtained from the simulations and experiments for both the water/nitrogen and the ethylene-glycol/nitrogen systems, especially given that the standard deviations for the experimental values are less than 3.8% and 2.6% for the water/nitrogen and ethylene-glycol/nitrogen systems, respectively. In addition, the variation of the simulated bubble velocity for the unsteady cases with $U_{TP} = 0.53$ m s$^{-1}$ in the water/nitrogen system was around 0.3% which is negligible. It can also be seen that the predictions from the correlation of Liu et al. (2005) are in close agreement with these data especially for the water/nitrogen system, as shown in Fig. 4.2.

4.2.2 Homogenous Void Fraction ($\beta$)

As noted in Chapter 3, the value of the homogenous void fraction is obtained from the simulation results. In Tables 4.2 and 4.3, the homogenous void fraction for the simulations is calculated using equation (4.1) for the water/nitrogen and ethylene-glycol/nitrogen systems, respectively.
Table 4.2: The homogenous void fraction obtained from the simulations for the water/nitrogen system. The cases denoted by a star were simulated using constant properties at a temperature of 21°C.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>$Re_{@25^\circ C}$</th>
<th>$Ca_{@25^\circ C}$</th>
<th>$U_{TP}$ (m s$^{-1}$)</th>
<th>Exp. $\beta$</th>
<th>Sim. input $\varepsilon_G$</th>
<th>Sim. $U_b$ (m s$^{-1}$)</th>
<th>$\beta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steady cases for the experiments &amp; simulations</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.40</td>
<td>0.36</td>
<td>0.223</td>
<td>0.38</td>
</tr>
<tr>
<td>1*</td>
<td>429</td>
<td>0.0029</td>
<td>0.21</td>
<td>0.40</td>
<td>0.36</td>
<td>0.224</td>
<td>0.38</td>
</tr>
<tr>
<td>2</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.50</td>
<td>0.44</td>
<td>0.223</td>
<td>0.47</td>
</tr>
<tr>
<td>3</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.60</td>
<td>0.53</td>
<td>0.223</td>
<td>0.56</td>
</tr>
<tr>
<td>4</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.70</td>
<td>0.62</td>
<td>0.224</td>
<td>0.66</td>
</tr>
<tr>
<td>5</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.20</td>
<td>0.18</td>
<td>0.348</td>
<td>0.19</td>
</tr>
<tr>
<td>6</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.30</td>
<td>0.27</td>
<td>0.348</td>
<td>0.29</td>
</tr>
<tr>
<td>7</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.40</td>
<td>0.36</td>
<td>0.348</td>
<td>0.39</td>
</tr>
<tr>
<td>7*</td>
<td>653</td>
<td>0.0043</td>
<td>0.32</td>
<td>0.40</td>
<td>0.36</td>
<td>0.350</td>
<td>0.39</td>
</tr>
<tr>
<td>8</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.50</td>
<td>0.45</td>
<td>0.348</td>
<td>0.49</td>
</tr>
<tr>
<td>9</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.60</td>
<td>0.53</td>
<td>0.348</td>
<td>0.57</td>
</tr>
<tr>
<td>Unsteady cases for the experiments, but steady for the simulations</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.30</td>
<td>0.26</td>
<td>0.468</td>
<td>0.29</td>
</tr>
<tr>
<td>11</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.40</td>
<td>0.35</td>
<td>0.467</td>
<td>0.39</td>
</tr>
<tr>
<td>11*</td>
<td>857</td>
<td>0.0057</td>
<td>0.42</td>
<td>0.40</td>
<td>0.35</td>
<td>0.470</td>
<td>0.40</td>
</tr>
<tr>
<td>12</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.50</td>
<td>0.44</td>
<td>0.467</td>
<td>0.49</td>
</tr>
<tr>
<td>13</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.60</td>
<td>0.53</td>
<td>0.467</td>
<td>0.59</td>
</tr>
<tr>
<td>14</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.70</td>
<td>0.61</td>
<td>0.467</td>
<td>0.68</td>
</tr>
<tr>
<td>Unsteady cases for the experiments &amp; simulations</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1189</td>
<td>0.0066</td>
<td>0.53</td>
<td>0.40</td>
<td>0.34</td>
<td>0.607</td>
<td>0.39</td>
</tr>
<tr>
<td>15*</td>
<td>1082</td>
<td>0.0072</td>
<td>0.53</td>
<td>0.40</td>
<td>0.34</td>
<td>0.607</td>
<td>0.39</td>
</tr>
<tr>
<td>16</td>
<td>1189</td>
<td>0.0066</td>
<td>0.53</td>
<td>0.50</td>
<td>0.43</td>
<td>0.606</td>
<td>0.49</td>
</tr>
</tbody>
</table>
Table 4.3: The homogenous void fraction obtained from the simulations for the ethylene-glycol/nitrogen system.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>Re @ 21°C</th>
<th>Ca @ 21°C</th>
<th>$U_{TP}$ (m s$^{-1}$)</th>
<th>Exp. $\beta$</th>
<th>Sim. input $\varepsilon_g$</th>
<th>Sim. $U_B$ (m s$^{-1}$)</th>
<th>Sim. $\beta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.17</td>
<td>0.11</td>
<td>0.300</td>
<td>0.17</td>
</tr>
<tr>
<td>2</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.20</td>
<td>0.18</td>
<td>0.299</td>
<td>0.27</td>
</tr>
<tr>
<td>3</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.30</td>
<td>0.26</td>
<td>0.299</td>
<td>0.39</td>
</tr>
<tr>
<td>4</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.40</td>
<td>0.31</td>
<td>0.299</td>
<td>0.47</td>
</tr>
<tr>
<td>5</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.50</td>
<td>0.39</td>
<td>0.299</td>
<td>0.58</td>
</tr>
<tr>
<td>6</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.60</td>
<td>0.46</td>
<td>0.299</td>
<td>0.68</td>
</tr>
<tr>
<td>7</td>
<td>32</td>
<td>0.1233</td>
<td>0.29</td>
<td>0.30</td>
<td>0.27</td>
<td>0.452</td>
<td>0.38</td>
</tr>
<tr>
<td>8</td>
<td>32</td>
<td>0.1233</td>
<td>0.29</td>
<td>0.40</td>
<td>0.30</td>
<td>0.452</td>
<td>0.48</td>
</tr>
<tr>
<td>9</td>
<td>31</td>
<td>0.1190</td>
<td>0.28</td>
<td>0.50</td>
<td>0.32</td>
<td>0.435</td>
<td>0.56</td>
</tr>
<tr>
<td>10</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.60</td>
<td>0.33</td>
<td>0.593</td>
<td>0.62</td>
</tr>
<tr>
<td>11</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.40</td>
<td>0.25</td>
<td>0.593</td>
<td>0.40</td>
</tr>
<tr>
<td>12</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.50</td>
<td>0.34</td>
<td>0.593</td>
<td>0.54</td>
</tr>
<tr>
<td>13</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.60</td>
<td>0.39</td>
<td>0.593</td>
<td>0.62</td>
</tr>
</tbody>
</table>

It can be seen from Tables 4.2 and 4.3 that the simulated values for the homogenous void fraction are in good agreement with the experimental input data, with the average differences being around 4% for two systems, water/nitrogen and ethylene-glycol/nitrogen. It can be observed that the variations of the homogenous void fractions obtained from equation (4.1) for the unsteady simulated cases ($U_{TP} = 0.53$ m s$^{-1}$) are negligible, as the only simulated result used in this equation is the bubble velocity which showed a negligible change as presented in Section 4.2.1. In addition, it can be found from Table 4.2 for the water/nitrogen system that the bubble velocities and homogenous void fractions obtained from the simulations at 21°C temperature are very close to those of the simulations performed at a temperature of 25°C.
4.2.3 Bubble and Slug Lengths

Although the experimental bubble length has been used to initialise the starting bubble shape in the simulation, the bubble evolves to a steady shape over time. Tables 4.4 and 4.5 show the comparison between the experimental and modelling results for the non-dimensional bubble \( (L_B/d) \) and slug \( (L_S/d) \) lengths in the water/nitrogen and ethylene-glycol/nitrogen systems, respectively. It is necessary to mention that the experimental lengths used for the modelling are the mean values of approximately 100 gas bubble and liquid slug lengths, with the standard deviation being less than 10% for most cases simulated for the water/nitrogen and ethylene-glycol/nitrogen system.
Table 4.4: Comparison of the experimental and simulated gas bubble and liquid slug lengths for the water/nitrogen system. The cases denoted by a star were simulated using constant properties evaluated at a temperature of 21°C.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>Re @ 25°C</th>
<th>Ca @ 25°C</th>
<th>U_{TP} (m s(^{-1}))</th>
<th>Exp. (\beta)</th>
<th>Sim. (\beta)</th>
<th>(L_b/d)</th>
<th>(L_s/d)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Exp.</td>
<td>Sim.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Exp.</td>
<td>Sim.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Exp.</td>
<td>Sim.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Exp.</td>
<td>Sim.</td>
</tr>
<tr>
<td><strong>Steady cases for the experiments &amp; simulations</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.4</td>
<td>0.38</td>
<td>3.40</td>
<td>3.27</td>
</tr>
<tr>
<td>1*</td>
<td>429</td>
<td>0.0029</td>
<td>0.21</td>
<td>0.4</td>
<td>0.38</td>
<td>3.40</td>
<td>3.28</td>
</tr>
<tr>
<td>2</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.5</td>
<td>0.47</td>
<td>4.25</td>
<td>4.09</td>
</tr>
<tr>
<td>3</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.6</td>
<td>0.56</td>
<td>5.34</td>
<td>5.22</td>
</tr>
<tr>
<td>4</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.7</td>
<td>0.66</td>
<td>7.34</td>
<td>7.12</td>
</tr>
<tr>
<td>5</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.2</td>
<td>0.19</td>
<td>1.96</td>
<td>1.96</td>
</tr>
<tr>
<td>6</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.3</td>
<td>0.29</td>
<td>2.41</td>
<td>2.38</td>
</tr>
<tr>
<td>7</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.4</td>
<td>0.39</td>
<td>3.04</td>
<td>3.04</td>
</tr>
<tr>
<td>7*</td>
<td>653</td>
<td>0.0043</td>
<td>0.32</td>
<td>0.4</td>
<td>0.39</td>
<td>3.04</td>
<td>3.05</td>
</tr>
<tr>
<td>8</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.5</td>
<td>0.49</td>
<td>3.93</td>
<td>3.90</td>
</tr>
<tr>
<td>9</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.6</td>
<td>0.57</td>
<td>5.07</td>
<td>5.28</td>
</tr>
<tr>
<td><strong>Unsteady cases for the experiments, but steady for the simulations</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.3</td>
<td>0.29</td>
<td>2.79</td>
<td>2.84</td>
</tr>
<tr>
<td>11</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.4</td>
<td>0.39</td>
<td>3.06</td>
<td>3.06</td>
</tr>
<tr>
<td>11*</td>
<td>857</td>
<td>0.0057</td>
<td>0.42</td>
<td>0.4</td>
<td>0.40</td>
<td>3.06</td>
<td>3.07</td>
</tr>
<tr>
<td>12</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.5</td>
<td>0.49</td>
<td>3.76</td>
<td>3.83</td>
</tr>
<tr>
<td>13</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.6</td>
<td>0.59</td>
<td>4.63</td>
<td>4.72</td>
</tr>
<tr>
<td>14</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.7</td>
<td>0.67</td>
<td>6.06</td>
<td>6.07</td>
</tr>
<tr>
<td><strong>Unsteady cases for the experiments &amp; simulations</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1189</td>
<td>0.0066</td>
<td>0.53</td>
<td>0.4</td>
<td>0.39</td>
<td>4.06</td>
<td>4.03 - 4.05</td>
</tr>
<tr>
<td>15*</td>
<td>1082</td>
<td>0.0072</td>
<td>0.53</td>
<td>0.4</td>
<td>0.39</td>
<td>4.06</td>
<td>4.04 - 4.07</td>
</tr>
<tr>
<td>16</td>
<td>1189</td>
<td>0.0066</td>
<td>0.53</td>
<td>0.5</td>
<td>0.49</td>
<td>4.21</td>
<td>4.18 - 4.20</td>
</tr>
</tbody>
</table>
Table 4.5: Comparison of the experimental and simulated gas bubble and liquid slug lengths for the ethylene-glycol/nitrogen system.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>Re @ 21°C</th>
<th>Ca @ 21°C</th>
<th>U_TP (m s⁻¹)</th>
<th>Exp. β</th>
<th>Sim. β</th>
<th>L₉/d</th>
<th>Lₛ/d</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Exp.</td>
<td>Sim.</td>
</tr>
<tr>
<td>1</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.17</td>
<td>0.17</td>
<td>2.40</td>
<td>2.44</td>
</tr>
<tr>
<td>2</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.26</td>
<td>0.27</td>
<td>2.93</td>
<td>2.95</td>
</tr>
<tr>
<td>3</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.36</td>
<td>0.39</td>
<td>3.48</td>
<td>3.74</td>
</tr>
<tr>
<td>4</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.46</td>
<td>0.47</td>
<td>4.26</td>
<td>4.29</td>
</tr>
<tr>
<td>5</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.57</td>
<td>0.58</td>
<td>5.21</td>
<td>5.23</td>
</tr>
<tr>
<td>6</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.68</td>
<td>0.68</td>
<td>6.71</td>
<td>6.77</td>
</tr>
<tr>
<td>7</td>
<td>32</td>
<td>0.1233</td>
<td>0.29</td>
<td>0.34</td>
<td>0.38</td>
<td>2.98</td>
<td>3.22</td>
</tr>
<tr>
<td>8</td>
<td>32</td>
<td>0.1233</td>
<td>0.29</td>
<td>0.45</td>
<td>0.48</td>
<td>3.69</td>
<td>3.92</td>
</tr>
<tr>
<td>9</td>
<td>31</td>
<td>0.1190</td>
<td>0.28</td>
<td>0.55</td>
<td>0.56</td>
<td>4.63</td>
<td>4.66</td>
</tr>
<tr>
<td>10</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.32</td>
<td>0.31</td>
<td>2.20</td>
<td>2.19</td>
</tr>
<tr>
<td>11</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.42</td>
<td>0.40</td>
<td>2.64</td>
<td>2.70</td>
</tr>
<tr>
<td>12</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.54</td>
<td>0.54</td>
<td>3.79</td>
<td>3.81</td>
</tr>
<tr>
<td>13</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.66</td>
<td>0.62</td>
<td>5.06</td>
<td>4.87</td>
</tr>
</tbody>
</table>
The Tables above show that the results obtained from the modelling are in good agreement with the experimental data for the bubble and slug lengths with the average difference being 2.1% and the largest being 8.6%. It can be also seen that the difference of the bubble and slug lengths obtained for the water/nitrogen system at 21°C and 25°C temperatures are very small as expected. Moreover, the ripples observed at the bubble tails of the unsteady simulated cases for the water/nitrogen system cause a maximum variation of 0.9% for the bubble and slug lengths. In Figs. 4.4 and 4.5 the variations of the simulated non-dimensional bubble, slug and unit cell lengths versus the homogenous void fraction ($\beta$) are plotted for the water/nitrogen system for $U_{TP} = 0.32$ m s$^{-1}$ and for the ethylene-glycol/nitrogen system for $U_{TP} = 0.20$ m s$^{-1}$. Figures 4.4 and 4.5 show that the bubble length becomes longer whilst the slug length becomes shorter as the homogenous void fraction ($\beta$) is increased, which corresponds with increasing gas-liquid volume flow rate ratio. For the water/nitrogen system, the unit cell length ($L_{UC}$) decreases with increasing $\beta$, for values below $\sim 0.3$, and then increases slowly with increasing $\beta$, as can be seen in Fig. 4.4. In Fig. 4.5, a similar trend can be seen for the ethylene-glycol/nitrogen system, where the unit cell length ($L_{UC}$) decreases for $\beta$ below 0.47, then it remains constant until $\beta = 0.58$ which it starts to increase.

![Graph](image.png)

Figure 4.4: The effect of the homogenous void fraction ($\beta$) on the normalized bubble, slug and unit cell lengths ($L/d$) for the water/nitrogen system with $U_{TP} = 0.32$ m s$^{-1}$. 

58
4. Validation of Simulation Results for Hydrodynamics

4.2.4 Film Thickness ($\delta_F$)

In the experimental work for the water/nitrogen system, the film thickness was not measured directly, but it was calculated by Leung et al. (2010) using equation (4.3), which was derived from volume conservation by Suo and Griffith (1964), assuming a stationary, constant thickness film.

$$\frac{\delta_F}{d} = \frac{1}{2} \left( 1 - \frac{U_{TP}}{U_B} \right)$$

(4.3)

In Fig. 4.6, the film thicknesses from the simulations and the calculated values from equation (4.3) evaluated using the experimental data are plotted versus the Capillary number ($Ca$). In this figure, the simulated film thicknesses obtained from the modellings which used constant properties evaluated at 21°C temperature are
presented. Also, the film thickness from the Aussillous and Quéré (2000) correlation (given in equation (4.4)) which is proposed for negligible inertia (low Reynolds numbers) without gravity is plotted.

\[
\frac{\delta_F}{R} = \frac{1.34Ca^{2/3}}{1 + 2.5 \left(1.34Ca^{2/3}\right)}
\]  

(4.4)

Figure 4.6: The effect of Capillary number \((Ca)\) on the non-dimensional film thickness \((\delta_F/R)\) for the water/nitrogen system for an experimental homogenous void fraction \((\beta)\) of 0.4. The symbols (□) and (Δ) are used for film thicknesses calculated from equation (4.3) using the experimental and simulated bubble velocities \((U_B)\), respectively. The symbol (○) is used for the results obtained directly from the simulations. The Aussillous and Quéré (2000) correlation is also shown by a dashed line.

Figure 4.6 shows that the dimensionless film thickness \((\delta_F/R)\) increases with increasing Capillary number. The results derived from equation (4.3) using the experimental data show the liquid film thickness is approximately constant until the Capillary number reaches 0.0043, then it increases. Equation (4.3) is also used to determine the film
4. Validation of Simulation Results for Hydrodynamics

thickness using the simulation results for the bubble velocity \((U_B)\). The calculated film thicknesses, shown by the triangle symbols, are in good agreement with the simulation results, showing an increasing film thickness with increasing Capillary number. The film thickness data calculated using the Suo and Griffith correlation (equation 4.3) together with the experimentally determined bubble velocity (□) are 15-42% higher than those calculated using the simulated bubble velocity. This can be explained by the fact that a small difference between the simulation and experimental bubble velocities gives a large deviation for the film thicknesses obtained using Suo and Griffith correlation. For example, there is only a 4.7% difference between the simulated and experimental bubble velocities which are 0.224 m s\(^{-1}\) and 0.235 m s\(^{-1}\), respectively for the case with \(U_{TP} = 0.21\) m s\(^{-1}\) and \(\beta = 0.4\) and this small difference causes a 42% deviation between the film thicknesses calculated using equation (4.3) as shown in Fig. 4.6.

All the data calculated using equation (4.4) are lower than the simulation results and this difference becomes larger with increasing Capillary number, increasing from 19% to 35% as the Capillary number is increased from 0.0029 to 0.0072. The increase in the Capillary number arises from an increase of the mixture velocity \((U_{TP})\) and it means the inertial effects become more important, so the difference between the simulated film thickness and equation (4.4), which was obtained by neglecting the inertial effects, increases.

de Ryck (2002) studied the effect of inertia on the film thickness by using a regular perturbation method for the front half of the bubble for Reynolds numbers below 1000. He showed that the ratio of the Reynolds and Capillary numbers is important in determining the inertial effect on the liquid film thickness. In Fig. 4.7, taken from his work, it can be seen that the film thickness first decreases with increasing Reynolds number for a specified Capillary number and the slope of the line is greater for higher Capillary numbers. Then, the film thickness increases rapidly with an increase in the Reynolds number. The Capillary numbers for the simulated cases for water/nitrogen are located below the line of \(Ca = 0.02\) and their values for the axial coordinates \((Re/2Ca)\) are around \(7.5\times10^4\), so using Fig. 4.7 the film thickness in these cases is expected to
increases with increasing Reynolds number, as observed in the simulations and in the results calculated from equation (4.3) using the bubble velocity from the simulations.

![Graph](image)

Figure 4.7: The inertial effect on the liquid film thickness. Taken from the work of de Ryck, (2002).

For the water/nitrogen system at the high mixture velocity of 0.53 m s\(^{-1}\), the liquid film thickness remains unchanged even though the bubble tail shape changes. In Fig. 4.8, the gas-liquid interfaces for the case with \(U_{TP} = 0.53 \text{ m s}^{-1}\) and \(\beta = 0.4\) are shown at two different times and it can be seen that the film thicknesses are almost the same. Therefore, a constant value has been used for presenting the non-dimensional film thickness for \(Ca = 0.0072\) in Fig. 4.6.
Figure 4.8: The effect of the variation of the bubble tail shape on the non-dimensional film thickness \((S_f/R)\) for \(U_{TP} = 0.53\) m s\(^{-1}\) and \(\beta = 0.4\) for the water/nitrogen system.

Experimental film thicknesses for the ethylene-glycol/nitrogen system could be measured directly from photographic images. The experimental and simulated film thicknesses are measured at the half length of the bubble, as the film thickness varies significantly along the bubble. The direct comparison of the film thickness obtained from the experiments and simulations shows an average difference of 3.3% and a maximum difference of 7.8%, according to the data given in Table 4.6.
4. Validation of Simulation Results for Hydrodynamics

Table 4.6: Comparison of the liquid film thickness between the simulations and experiments for the ethylene-glycol/nitrogen system.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>$Re$ @ 21°C</th>
<th>$Ca$ @ 21°C</th>
<th>$U_{TP}$ (m s$^{-1}$)</th>
<th>Exp. $\beta$</th>
<th>Sim. $\beta$</th>
<th>$\delta_f/R$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Exp.</td>
<td>Sim.</td>
<td>Diff. (%)</td>
</tr>
<tr>
<td>1</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.17</td>
<td>0.17</td>
<td>0.172</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.183</td>
</tr>
<tr>
<td>2</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.26</td>
<td>0.27</td>
<td>0.170</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.179</td>
</tr>
<tr>
<td>3</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.36</td>
<td>0.39</td>
<td>0.170</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.179</td>
</tr>
<tr>
<td>4</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.46</td>
<td>0.47</td>
<td>0.170</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.179</td>
</tr>
<tr>
<td>5</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.57</td>
<td>0.58</td>
<td>0.166</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.179</td>
</tr>
<tr>
<td>6</td>
<td>22</td>
<td>0.0850</td>
<td>0.20</td>
<td>0.68</td>
<td>0.68</td>
<td>0.174</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.179</td>
</tr>
<tr>
<td>7</td>
<td>32</td>
<td>0.1233</td>
<td>0.29</td>
<td>0.34</td>
<td>0.38</td>
<td>0.196</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.197</td>
</tr>
<tr>
<td>8</td>
<td>32</td>
<td>0.1233</td>
<td>0.29</td>
<td>0.45</td>
<td>0.48</td>
<td>0.191</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.196</td>
</tr>
<tr>
<td>9</td>
<td>31</td>
<td>0.1190</td>
<td>0.28</td>
<td>0.55</td>
<td>0.56</td>
<td>0.189</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.194</td>
</tr>
<tr>
<td>10</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.32</td>
<td>0.31</td>
<td>0.219</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.224</td>
</tr>
<tr>
<td>11</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.42</td>
<td>0.40</td>
<td>0.212</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.216</td>
</tr>
<tr>
<td>12</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.54</td>
<td>0.54</td>
<td>0.208</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.208</td>
</tr>
<tr>
<td>13</td>
<td>40</td>
<td>0.1573</td>
<td>0.37</td>
<td>0.66</td>
<td>0.62</td>
<td>0.208</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.207</td>
</tr>
</tbody>
</table>
The effect of the Capillary number on the film thickness for the ethylene-glycol/nitrogen system, together with the results from the experiments, the simulations and the Aussillous and Quéré (2000) correlation are plotted in Fig. 4.9. It can be seen that the thickness of the liquid film increases with increasing $Ca$ in all cases. The scatter of the simulation and experimental film thickness at a specified Capillary number is due to the different homogenous void fractions used.

![Figure 4.9: The effect of the Capillary number ($Ca$) on the non-dimensional film thickness ($\delta_f/R$) for the ethylene-glycol/nitrogen system. The symbols (□) and (○) are used for the experimental and simulation results, respectively. Values from the Aussillous and Quéré, (2000) correlation are shown by a dashed line.](image)

In Fig. 4.9, the range of the simulation result deviations from the Aussillous and Quéré (2000) correlation, which was developed for negligible inertia, is from 5% to 14%. It can be seen that the difference between the results obtained from the modelling and Aussillous and Quéré (2000) correlation has decreased for the ethylene-glycol/nitrogen system in comparison with the water/nitrogen system, for which the range of the deviation was 19-35%. This can be explained using the results of de Ryck, (2002), shown in Fig. 4.7. The value of $Re/2Ca$ is 128 for the ethylene-glycol/nitrogen system.
which corresponds to conditions where the inertial force does not have a significant effect on the film thickness. Therefore, the film thicknesses obtained from the simulations for the ethylene-glycol/nitrogen system are closer to the results from the Aussillous and Quéré (2000) correlation. As mentioned above, this correlation was established without including the effect of the gravity, so the film thicknesses obtained from this correlation are expected to differ from the simulated and experimental results for the vertical upward flow.

### 4.2.5 Bubble Shape

In this section, the bubble shapes obtained from the computations are compared with the experimental bubble shapes of Leung et al. (2012). The gas-liquid interface in the experimental images was extracted using the DigitizeIt software for the ethylene-glycol/nitrogen system. This was not possible for the water/nitrogen system as the location of the gas and liquid interface could not be differentiated from the wall of the tube over 50% of the tube diameter. In Fig. 4.10 (a), the bubble shapes from the experiments and simulations are compared for the mixture velocity of 0.20 m s\(^{-1}\) at various homogenous void fractions (\(\beta\)). In this figure, the mixture velocities and the experimental homogenous void fractions are also presented. In Fig. 4.10 (b), the bubble shape for case 6 with \(U_{TP} = 0.20\) m s\(^{-1}\) and \(\beta = 0.68\) is shown, to better compare the shape at the nose and tail. The experimental and simulation bubble lengths differ by only 0.9%. It can be seen that the differences in the shapes of the nose, tail and film thickness are very small.
Figure 4.10: Comparison of the bubble shapes obtained from the simulations and experiments for the ethylene-glycol/nitrogen system with $U_{TP} = 0.20 \text{ m s}^{-1}$. In part (a), the effect of the homogenous void fractions ($\beta$) on the bubble shapes is shown. In part (b), the bubble shapes for case 6, with $U_{TP} = 0.20 \text{ m s}^{-1}$ and $\beta = 0.68$, are plotted. The simulation results are shown by solid red lines.
4. Validation of Simulation Results for Hydrodynamics

The effect of the mixture velocity \((U_{tp})\) on the experimental and simulated bubble shapes is shown in Fig. 4.11 for the cases having experimental homogenous void fractions \((\beta)\) in the range 0.54-0.57. Whilst there are small differences between the experimental and simulation results, the increasing sharpness of the bubble nose and flattening of the bubble tail with increasing mixture velocity \((U_{tp})\) is captured very well in the simulations.

![Figure 4.11: The effect of the mixture velocity \((U_{tp})\) on the bubble shape for the experimental homogenous void fractions \((\beta)\) of ~0.55 for the ethylene-glycol/nitrogen system. The simulation results are shown by solid red lines.](image)

4.2.6 Pressure Drop

As there was no experimental data from Leung et al. (2010, 2012) for the pressure drop, the simulated pressure drops were validated using the correlations in literature. In the following figures, the normalized pressure drops from the simulations and correlations have been calculated via
where \( \frac{\Delta P}{L_{\text{LO}}} \) is the pressure drop for the liquid only, fully-developed flow with the average velocity and the liquid length the same as the mixture velocity used for the simulations.

In Figs. 4.12 and 4.13 the variation of the normalized pressure drops with the homogenous void fraction are plotted for the water/nitrogen system for \( U_{tp} = 0.32 \text{ m s}^{-1} \) and for the ethylene-glycol/nitrogen system for \( U_{tp} = 0.20 \text{ m s}^{-1} \), respectively. In these figures, the normalized pressure drops from the Kreutzer et al. (2005) correlation (given in equation (4.6)) which is obtained neglecting the pressure drop in the liquid film around the gas bubble and high Reynolds numbers \( (100 < Re < 1000) \) are also shown.

\[
\left( \frac{\Delta P}{L_{\text{UC}}} \right)^* = \varepsilon_L \left( 1 + a \frac{d}{L_S} \left( \frac{Re}{Ca} \right)^{0.33} \right) \quad \text{and} \quad \varepsilon_L = \frac{L_S}{L_{\text{UC}}} \quad (4.6)
\]

In this equation, the value of the constant \( a = 0.07 \) is used which was found numerically by Kreutzer et al. (2005).
4. Validation of Simulation Results for Hydrodynamics

Figure 4.12: The effect of the homogenous void fraction (\( \beta \)) on the normalized pressure drop (\( \ddot{\Delta}P \)) at \( U_{TP} = 0.32 \text{ m s}^{-1} \) for the water/nitrogen system. The normalized pressure drops calculated from the equation (4.6) are also shown by (□) symbol.

Kreutzer et al. (2005) proposed another correlation for predicting the pressure drop in Taylor flow given by equation (4.7) for the negligible inertial effects. Therefore, the simulated normalized pressure drops are also compared with equation (4.7) for the cases of ethylene-glycol/nitrogen system having low Reynolds numbers (22<Re<40).

\[
\left( \frac{\Delta P}{L_{UC}} \right)^* = \varepsilon_L \left( 1 + \frac{d}{L_s} \frac{7.16(3Ca)^{2/3}}{32Ca} \right) \tag{4.7}
\]

In this equation, the two-phase pressure drop is the sum of the pressure drop over the bubble caps using Bretherton (1961) correlation (equation (4.8)) and the pressure drop in the liquid slug.

\[
\Delta P_{cap} = 7.16(3Ca)^{2/3} \frac{\sigma}{d} \tag{4.8}
\]
4. Validation of Simulation Results for Hydrodynamics

Figure 4.13: The effect of the homogenous void fraction ($\beta$) on the normalized pressure drop ($\circ$) at $U_{tr} = 0.20$ m s$^{-1}$ for the ethylene-glycol/nitrogen system. The normalized pressure drops calculated from equations (4.6) and (4.7) are also shown by the (□) and (△) symbols, respectively.

It can be seen in Figs. 4.12 and 4.13, the normalized pressure drop decreases when the homogenous void fraction increases for both the water/nitrogen and ethylene-glycol/nitrogen systems. When $\beta$ increases, it means the volume fraction of the liquid has decreased in a unit cell, so a decrease in the pressure drop can be expected. It can also be seen that there is good agreement between the normalized pressure drops obtained from the simulations and equation (4.6). The difference between the two increases with increasing $\beta$ and reaches 19% and 17% when $\beta$ is increased to 0.57 and 0.68 for the water/nitrogen and ethylene-glycol/nitrogen systems, respectively. In Fig. 4.13, the maximum difference between the simulated results and equation (4.7) proposed for the negligible inertia is also found for the highest $\beta = 0.68$ taking a value of 40%.

Figures 4.14 and 4.15 show the effect of the mixture velocity on the normalized pressure drop for the water/nitrogen and ethylene-glycol/nitrogen systems, respectively. It is worth noting here that there is a significant variation of the normalized pressure drop for the
water/nitrogen system at $U_{TP} = 0.53\ \text{m}\ \text{s}^{-1}$. For example, there is a wide range of the normalized pressure drop from 0.2 to 2.1 for the case with $U_{TP} = 0.53\ \text{m}\ \text{s}^{-1}$ and $\beta = 0.49$. This can be explained by the fact that the flow is unsteady as mentioned in Section 4.2 at this mixture velocity which corresponds to a high Reynolds number ($Re = 1189$). Therefore, the simulated pressure drop for $U_{TP} = 0.53\ \text{m}\ \text{s}^{-1}$ has not been included in Fig. 4.14. Given the current simulations are two-dimensional the unsteady oscillations are not necessarily physical and further work is needed to investigate their significance.

![Diagram](image)

**Figure 4.14:** The effect of the mixture velocity ($U_{TP}$) on the normalized pressure drop ($\bigcirc$) for the homogenous void fraction of $\sim0.5$ for the water/nitrogen system. The normalized pressure drops calculated from the equation (4.6) are also shown by the ($\Box$) symbol.
4. Validation of Simulation Results for Hydrodynamics

Figure 4.15: The effect of the mixture velocity ($U_{TP}$) on the normalized pressure drop ($\circ$) for the homogenous void fraction of $\sim$0.56 for the ethylene-glycol/nitrogen system. The normalized pressure drops calculated from equation (4.6) and (4.7) are also shown by the (□) and (Δ) symbols, respectively.

The normalized pressure drop obtained from the CFD modellings increases with increasing mixture velocity for both the liquid phases used and this trend is more significant for the water/nitrogen system increasing from 0.79 to 1.14 than for the ethylene-glycol/nitrogen system where the range is 0.44 – 0.52. Although the simulated normalized pressure drop depends on the mixture velocity, it can be understood from equation (4.6) that the normalized pressure drop is independent of the mixture velocity, and the very small changes seen in Figs. 4.14 and 4.15 are created by a tiny variation in the slug and unit cell lengths upon increasing the mixture velocity. The normalized pressure drop independency of the mixture velocity is also true for equation (4.7) established for the low Reynolds numbers for which it remains almost constant ($\sim$0.52) with increasing mixture velocity shown in Fig. 4.15.
4. Validation of Simulation Results for Hydrodynamics

4.4 Summary

Based on the comparison of the simulation results using the moving domain method with the experimental data of Leung et al. (2010, 2012) for the hydrodynamics of Taylor flow, it can be concluded that:

1. The bubble velocities obtained from the simulations are close to the experimental results for both the water/nitrogen and ethylene-glycol/nitrogen systems with the average difference being 3.4%.

2. The values of the homogenous void fractions for the simulations obtained using equation (4.1) are very similar to the experimental input values for water/nitrogen and ethylene-glycol/nitrogen systems.

3. The bubble and slug lengths obtained from the simulations are in good agreement with the experiments for both the water/nitrogen and ethylene-glycol/nitrogen systems.

4. For the ethylene-glycol/nitrogen system, for which the direct measurement of the experimental film thickness was possible, the simulations captured the liquid film thicknesses with a maximum deviation of 7.8%.

5. The simulated bubble shapes for the ethylene-glycol/nitrogen system obtained are also very similar to those obtained experimentally for different mixture velocities and homogenous void fractions.

6. The normalized pressure drops from the simulations are compared with the Kreutzer et al. (2005) correlation given in equation (4.6) for both the liquid phases studied, water and ethylene-glycol with the average differences being 16% and 6% for all of the cases simulated.

In the next chapter a comparison of the experimental and numerical heat transfer behaviour is presented.
Chapter 5

Validation of Modelling Results for Heat Transfer

In this chapter, the heat transfer results obtained from the CFD modelling of fully-developed Taylor flow, for the cases which were described in Chapter 4, are presented and compared with the experimental heat transfer data of Leung et al. (2010, 2012). In Section 5.1 the experimental and simulation heat transfer conditions are presented. The Nusselt numbers obtained from the CFD modelling and experiments are presented for both the water/nitrogen and the ethylene-glycol/nitrogen systems in Section 5.2. The effects of the mixture velocity and the homogeneous void are presented in Sections 5.3 and 5.4, respectively. Section 5.5 draws some conclusions.

5.1 Heat Transfer Conditions

Only a brief explanation of the experimental facility used to determine the mean heat transfer rates is presented here, details can be found in Leung et al. (2010, 2012). Figure 5.1 shows the experimental facility for the water-nitrogen system taken from Leung et al. (2010). The experimental heating section consisted of ten cylindrical copper blocks 25 mm in length to each of which a separate resistance band heater was attached. There was a 2 mm thick insulated gap between each block and the entire test section was isolated from the environment. The wall temperature increases from one block to another and it is approximately constant in each block. The average wall heat flux is nearly the same for all of the blocks. However, it was discussed in Leung et al. (2010, 2012) that the experimental arrangement does not correspond strictly to any standard thermal boundary condition for laminar flow. It was shown that the pseudo-local heat transfer coefficients for single-phase flows agreed closely with theoretical results for a constant heat flux thermal boundary condition.

The experimental setup of the heating section for the ethylene-glycol/nitrogen system was modified from the water/nitrogen system by including a closed-loop for ethylene-glycol.
The properties of ethylene-glycol changes with water content, so the closed-loop was required to prevent the effect of the air moisture on the ethylene-glycol properties.

Figure 5.1: Schematic of the experimental setup for the water/nitrogen system. Taken from the work of Leung et al. (2010). $T_n$ ($n = 1, 2, \ldots, 10$) shows the thermocouples used for measuring the temperature of the copper blocks.

For comparison of the simulation results with the experimental data of Leung et al. (2010, 2012), a constant wall heat flux ($H_2$) boundary condition was also applied for the modelling, with this being set to 32 kW m$^{-2}$. In addition, as mentioned in Chapter 4, the temperature changes during the experimental study of heat transfer, so the constant properties were specified at the temperatures of 25°C and 30°C for the water/nitrogen and ethylene-glycol/nitrogen systems, respectively. These temperatures were selected as lying
approximately in the middle of the range of experimental temperatures for most heat transfer cases studied.

5.1.1 Heat Transfer Mechanism

Leung et al. (2012) discussed the possible heat transfer mechanisms in Taylor flow. According to Fig. 5.2, taken from the work of Leung et al. (2012), the heat transfer process in Taylor flow for the constant wall heat flux boundary condition can be explained via three pathways: (1) transport of the heat from the wall to the liquid film above the bubble where the heat is stored prior to transfer to the next liquid slug; (2) transfer of the heat collected in the liquid film to the passing liquid slug and (3) heat transfer from the wall to the liquid slug in the recirculation zone.

![Figure 5.2: Schematic of the heat transfer mechanisms in Taylor flow. Taken from the work of Leung et al. (2012).](image)

These different pathways are captured in the CFD simulations which consider a unit cell and have a mesh sufficiently fine to resolve the liquid film.
5.2 Nusselt Number Results

The average Nusselt number ($N_{uav}$) is a meaningful definition for the comparison of the simulation results with the experiments. In the experimental work for both of the systems, water/nitrogen and ethylene-glycol/nitrogen, the time-averaged Nusselt number at the centre of each block ($N_{uj}$) was calculated using equation (5.1) (Leung et al., 2010, 2012). Then, the average Nusselt number ($N_{uav}$) was obtained by averaging the local Nusselt numbers ($N_{uj}$) over blocks 4 to 9 for the ethylene-glycol/nitrogen system and over blocks 2 to 9 for the water/nitrogen system to remove entrance, development and end-block effects on the average value. In Leung et al. (2012) it is noted that the long development length observed in the ethylene-glycol/nitrogen system was not seen in the water/nitrogen system and heat transfer was thermally fully-developed after the first heating block.

\[
N_{uj} = \frac{q_j}{\frac{T_{b,j,\text{out}} - T_{b,j,\text{in}}}{k_L}} \int \frac{d}{\ln \left( \frac{T_{w,j} - T_{b,j,\text{in}}}{T_{w,j} - T_{b,j,\text{out}}} \right)}
\]  

(5.1)

In equation (5.1), $q_j$ and $T_{w,j}$ are the wall heat flux and wall temperature, respectively; the inlet mean fluid temperature is $T_{b,j,\text{in}}$ and the outlet mean fluid temperature is given by $T_{b,j,\text{out}}$.

In the simulations, the average Nusselt number ($N_{uav}$) for a unit cell is calculated using equation (3.21).

A normalized Nusselt number is defined via

\[
N_{u}^* = \frac{N_{uav}}{N_{uLO}}
\]

(5.2)

where $N_{uLO}$ is the Nusselt number for the liquid only, fully-developed flow for a constant wall heat flux boundary condition and takes a value of 4.364 (Shah and London, 1978).
5. Validation of Modelling Results for Heat Transfer

5.2.1 Water/Nitrogen System

The normalized Nusselt numbers obtained from the simulations and experiments are presented for the water/nitrogen system in Table 5.1. It can be seen that the differences between the simulated and experimental $Nu^*$ are small for the mixture velocities of 0.21 m s$^{-1}$ and 0.32 m s$^{-1}$ with the largest difference being 8.9%, but the deviation increases as the mixture velocity becomes larger for a specified homogenous void fraction ($\beta$). For a mixture velocity of 0.53 m s$^{-1}$, having a Reynolds number of 1189, the differences of 32% and 29% values are significant at $\beta \sim 0.4$ and 0.5, respectively.

Table 5.1: Comparison of the experimental and simulated normalized average Nusselt numbers for the water/nitrogen system.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>$Re$ @ 25°C</th>
<th>$Ca$ @ 25°C</th>
<th>$U_{TP}$ (m s$^{-1}$)</th>
<th>Exp. $\beta$</th>
<th>Sim. $\beta$</th>
<th>$Nu^*$ Exp.</th>
<th>$Nu^*$ Sim.</th>
<th>Diff. (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.40</td>
<td>0.38</td>
<td>2.57</td>
<td>2.45</td>
<td>4.7</td>
</tr>
<tr>
<td>2</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.50</td>
<td>0.47</td>
<td>2.15</td>
<td>2.09</td>
<td>2.8</td>
</tr>
<tr>
<td>3</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.60</td>
<td>0.56</td>
<td>1.70</td>
<td>1.72</td>
<td>-1.2</td>
</tr>
<tr>
<td>4</td>
<td>475</td>
<td>0.0026</td>
<td>0.21</td>
<td>0.70</td>
<td>0.66</td>
<td>1.24</td>
<td>1.35</td>
<td>-8.9</td>
</tr>
<tr>
<td>5</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.20</td>
<td>0.19</td>
<td>3.16</td>
<td>3.07</td>
<td>2.8</td>
</tr>
<tr>
<td>6</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.30</td>
<td>0.29</td>
<td>2.70</td>
<td>2.82</td>
<td>-4.4</td>
</tr>
<tr>
<td>7</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.40</td>
<td>0.39</td>
<td>2.61</td>
<td>2.50</td>
<td>4.2</td>
</tr>
<tr>
<td>8</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.50</td>
<td>0.49</td>
<td>2.27</td>
<td>2.18</td>
<td>4.0</td>
</tr>
<tr>
<td>9</td>
<td>713</td>
<td>0.0040</td>
<td>0.32</td>
<td>0.60</td>
<td>0.57</td>
<td>1.76</td>
<td>1.72</td>
<td>2.3</td>
</tr>
<tr>
<td>10</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.30</td>
<td>0.29</td>
<td>2.75</td>
<td>2.59</td>
<td>5.8</td>
</tr>
<tr>
<td>11</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.40</td>
<td>0.39</td>
<td>2.96</td>
<td>2.36</td>
<td>20.3</td>
</tr>
<tr>
<td>12</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.50</td>
<td>0.49</td>
<td>2.52</td>
<td>2.09</td>
<td>17.1</td>
</tr>
<tr>
<td>13</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.60</td>
<td>0.59</td>
<td>2.04</td>
<td>1.79</td>
<td>12.3</td>
</tr>
<tr>
<td>14</td>
<td>951</td>
<td>0.0052</td>
<td>0.42</td>
<td>0.70</td>
<td>0.68</td>
<td>1.76</td>
<td>1.44</td>
<td>18.2</td>
</tr>
</tbody>
</table>

Unsteady cases for the experiments & simulations

<table>
<thead>
<tr>
<th>Case No.</th>
<th>$Re$ @ 25°C</th>
<th>$Ca$ @ 25°C</th>
<th>$U_{TP}$ (m s$^{-1}$)</th>
<th>Exp. $\beta$</th>
<th>Sim. $\beta$</th>
<th>$Nu^*$ Exp.</th>
<th>$Nu^*$ Sim.</th>
<th>Diff. (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15</td>
<td>1189</td>
<td>0.0066</td>
<td>0.53</td>
<td>0.40</td>
<td>0.39</td>
<td>3.16</td>
<td>2.15</td>
<td>32.0</td>
</tr>
<tr>
<td>16</td>
<td>1189</td>
<td>0.0066</td>
<td>0.53</td>
<td>0.50</td>
<td>0.49</td>
<td>2.77</td>
<td>1.97</td>
<td>29.0</td>
</tr>
</tbody>
</table>
In Fig. 5.3, the simulated and experimental normalized average Nusselt numbers are shown for different mixture velocities. The error bars show two standard deviations calculated from the scatter between blocks of the experimental Nusselt numbers for each case. As mentioned in Chapter 4, the simulated homogenous void fraction is a result of the simulation and it cannot be set as an input parameter, so there are small differences between the simulated and experimental homogenous void fractions in every case. It can be seen in Figs. 5.3a and 5.3b that the simulated values of $\ Nu^\star \ $ all lie within the uncertainty range for $\ U_{TP} = 0.21 \text{ m s}^{-1} \text{ and } 0.32 \text{ m s}^{-1}$. However, the differences between the simulated and experimental mean values of $\ Nu^\star \ $ are not explained by experimental uncertainty at high mixture velocities, especially $\ U_{TP} = 0.53 \text{ m s}^{-1}$. 
Figure 5.3: Normalized average Nusselt number ($Nu^*$) versus the homogenous void fraction ($\beta$) for the water/nitrogen system for different mixture velocities of (a) 0.21 m s$^{-1}$, (b) 0.32 m s$^{-1}$ (c) 0.42 m s$^{-1}$ ($\triangle$) and 0.53 m s$^{-1}$ ($\bigcirc$). The experimental and simulation results are shown by solid and open symbols, respectively.
A detailed study of the case with $\beta = 0.4$, $U_{TP} = 0.53 \text{ m s}^{-1}$ (case 15) of the water/nitrogen system is presented below as it is the case that shows the largest difference between the simulated and experimental results for $Nu^*$ (Table 5.1).

5.2.1.1 Fully-Developed Heat Transfer

It should be mentioned here that the strategy of freezing the flow field when solving the energy equation described in Chapter 3 was not used at the mixture velocity of 0.53 m s$^{-1}$ as the unsteady behaviour of the flow field was significant at this value of $U_{TP}$, so the flow and energy equations were solved simultaneously. Figure 5.4 shows the normalized Nusselt number versus time for the simulated case with $U_{TP} = 0.53 \text{ m s}^{-1}$ and $\beta = 0.39$. It can be seen $Nu^*$ reaches a near constant value of 2.15 as the variation of the bubble tail shape causes only very small fluctuations in $Nu^*$ from 2.145 to 2.154 which are negligible.

![Figure 5.4: The simulated $Nu^*$ versus time for case 15 of the water/nitrogen system with $U_{TP} = 0.53 \text{ m s}^{-1}$ and $\beta = 0.39$.](image)
In Fig. 5.5, the experimental $N u_j^*$ at the centre of each block from 2 to 9 are shown for case 15 with $U_{TP} = 0.53 \text{ m s}^{-1}$ and $\beta = 0.4$ for the water/nitrogen system together with the normalized average Nusselt number calculated by averaging over these blocks shown by a dashed line. It can be seen from this figure that the flow has reached a thermally fully-developed through the heating section used for the experimental work although the fluctuations were observed at the bubble tail shapes (Leung et al., 2010, 2012).

![Figure 5.5](image)

Figure 5.5: The experimental results for the normalized local Nusselt number at the centre of each block from 2 to 9 are shown by ($\Delta$) symbol for the water/nitrogen system with $U_{TP} = 0.53 \text{ m s}^{-1}$ and $\beta = 0.4$. The mean experimental $N u^*$ is also shown by a dashed line.

Whilst Figs. 5.4 and 5.5 show that the heat transfer has reached a thermally fully-developed state in both the CFD modelling and the experiments, the simulated results for the mixture velocity of $0.53 \text{ m s}^{-1}$ which are unsteady cases have not been included in the following sections as the variation of the bubble tail shape leads to unphysical pressure drops reported in Chapter 4.
5.2.2 Ethylene-Glycol/Nitrogen System

Table 5.2 shows the non-dimensional Nusselt numbers from the simulations and experiments for the ethylene-glycol/nitrogen system. As seen in this table the difference between the simulated and experimental results lies in the range of 1.4-21.7% with the average difference being 8%.

Table 5.2: Comparison of the experimental and simulated normalized average Nusselt numbers for the ethylene-glycol/nitrogen system.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>Re @ 30°C</th>
<th>Ca @ 30°C</th>
<th>U TP (m s⁻¹)</th>
<th>Exp. β</th>
<th>Sim. β</th>
<th>Nu* Exp.</th>
<th>Nu* Sim.</th>
<th>Diff. (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>32</td>
<td>0.0583</td>
<td>0.20</td>
<td>0.17</td>
<td>0.17</td>
<td>1.44</td>
<td>1.51</td>
<td>-4.9</td>
</tr>
<tr>
<td>2</td>
<td>32</td>
<td>0.0583</td>
<td>0.20</td>
<td>0.26</td>
<td>0.27</td>
<td>1.37</td>
<td>1.49</td>
<td>-8.8</td>
</tr>
<tr>
<td>3</td>
<td>32</td>
<td>0.0583</td>
<td>0.20</td>
<td>0.36</td>
<td>0.37</td>
<td>1.35</td>
<td>1.47</td>
<td>-8.9</td>
</tr>
<tr>
<td>4</td>
<td>32</td>
<td>0.0583</td>
<td>0.20</td>
<td>0.46</td>
<td>0.46</td>
<td>1.26</td>
<td>1.40</td>
<td>-11.1</td>
</tr>
<tr>
<td>5</td>
<td>32</td>
<td>0.0583</td>
<td>0.20</td>
<td>0.57</td>
<td>0.56</td>
<td>1.12</td>
<td>1.28</td>
<td>-14.3</td>
</tr>
<tr>
<td>6</td>
<td>32</td>
<td>0.0583</td>
<td>0.20</td>
<td>0.68</td>
<td>0.67</td>
<td>0.92</td>
<td>1.12</td>
<td>-21.7</td>
</tr>
<tr>
<td>7</td>
<td>46</td>
<td>0.0846</td>
<td>0.29</td>
<td>0.34</td>
<td>0.35</td>
<td>-</td>
<td>1.47</td>
<td>-</td>
</tr>
<tr>
<td>8</td>
<td>46</td>
<td>0.0846</td>
<td>0.29</td>
<td>0.45</td>
<td>0.45</td>
<td>1.40</td>
<td>1.42</td>
<td>-1.4</td>
</tr>
<tr>
<td>9</td>
<td>44</td>
<td>0.0817</td>
<td>0.28</td>
<td>0.55</td>
<td>0.54</td>
<td>-</td>
<td>1.35</td>
<td>-</td>
</tr>
<tr>
<td>10</td>
<td>59</td>
<td>0.1079</td>
<td>0.37</td>
<td>0.32</td>
<td>0.29</td>
<td>1.54</td>
<td>1.49</td>
<td>3.2</td>
</tr>
<tr>
<td>11</td>
<td>59</td>
<td>0.1079</td>
<td>0.37</td>
<td>0.42</td>
<td>0.37</td>
<td>1.54</td>
<td>1.49</td>
<td>3.2</td>
</tr>
<tr>
<td>12</td>
<td>59</td>
<td>0.1079</td>
<td>0.37</td>
<td>0.54</td>
<td>0.53</td>
<td>1.49</td>
<td>1.42</td>
<td>4.7</td>
</tr>
<tr>
<td>13</td>
<td>59</td>
<td>0.1079</td>
<td>0.37</td>
<td>0.66</td>
<td>0.65</td>
<td>1.35</td>
<td>1.31</td>
<td>3.0</td>
</tr>
</tbody>
</table>

The normalized average Nusselt numbers obtained from the simulations and experiments are shown in Fig. 5.6 for different mixture velocities for the ethylene-glycol/nitrogen system together with error bars covering two standard deviations calculated from the scatter of the experimental Nusselt numbers for each case. It can be seen that most of the simulated $Nu^*$ fall in the range of experimental uncertainty for the different mixture velocities and homogenous void fractions.
5. Validation of Modelling Results for Heat Transfer

Figure 5.6: Normalized average Nusselt number ($N_{u}^{*}$) versus the homogenous void fraction ($\beta$) for the ethylene-glycol/nitrogen system for different mixture velocities of (a) 0.20 m s$^{-1}$ ($\circ$), (b) 0.29 m s$^{-1}$ ($\Box$) and 0.37 m s$^{-1}$ ($\Delta$). The experimental and simulated results are shown by solid and open symbols, respectively.

In the following section, more investigations have been performed for the case with $U_{TP} = 0.20$ m s$^{-1}$ and $\beta = 0.67$ (case 6) of the ethylene-glycol/nitrogen system as it has the
largest difference between the simulated and experimental results for $Nu^*$ (Table 5.2) and the maximum experimental standard deviation (Fig. 5.6a).

5.2.2.1 Fully-Developed Heat Transfer

The higher value of the simulated $Nu^* = 1.12$ in comparison with that of the experimental $Nu^* = 0.92$ raises the possibility that the simulated result has not reached fully-developed heat transfer. In Fig. 5.7, the simulated normalized Nusselt number versus time is shown and it can be found that the simulated $Nu^*$ has reached a steady value of 1.12. Thus, developing heat transfer does not explain the difference for the case with $U_T = 0.20$ m s$^{-1}$ and $\beta = 0.67$ which has a larger simulated $Nu^*$ than the experiments.

![Figure 5.7: The simulated $Nu^*$ versus time for case 6 of the ethylene-glycol/nitrogen system with $U_T = 0.20$ m s$^{-1}$ and $\beta = 0.67$ at 30°C temperature.](image-url)
5.2.2.2 Effect of Temperature on the Physical Properties

As mentioned for the ethylene-glycol/nitrogen system, the simulations were performed using constant properties at a temperature of 30°C which was approximately in the centre of the experimental range of temperatures for most cases studied. The simulation for case 6 was repeated solving for both the hydrodynamics and heat transfer using the constant properties at the first experimental heating block temperature (23.6°C) for this case to investigate the effect of the temperature change on the simulated heat transfer results. It is worth noting here that the maximum temperature reached was 31°C through the experimental heating section for this case, so the simulated normalized Nusselt number at 30°C temperature was used as the result for the maximum temperature. The simulated normalized Nusselt numbers at $T = 23.6°C$ and $T = 30°C$ were 1.08 and 1.12, respectively. This shows that the simulated $Nu^*$ is still 18% larger than the experimental mean value of $Nu^* = 0.92$ even using the properties at $T = 23.6°C$ (those for the temperature of the first heating block). Therefore, it can be concluded that the effect of the variations of the properties with temperature on the simulated Nusselt number is small and is not the explanation for the difference observed between the mean value of the experimental and simulated Nusselt numbers for case 6 of ethylene-glycol/nitrogen system. However, it can be seen that the simulated results obtained for both the temperatures ($T = 23.6°C$ and $30°C$) are located in the range of the experimental uncertainty, as shown in Fig. 5.8.
Figure 5.8: The simulated $Nu^*$ are shown for case 6 of the ethylene-glycol/nitrogen system with $U_{TP} = 0.20$ m s$^{-1}$ and $\beta = 0.67$ at 23.6°C and 30°C temperatures. Error bars representing two standard deviations in the variation of the mean Nusselt number are also shown.

5.3 Effect of the Mixture Velocity

As shown in Fig. 5.9 for the water/nitrogen system, the simulated $Nu^*$ remains approximately unchanged taking a value of 2.12 for $\beta \sim 0.5$ and $L_s/d \sim 3.4$ for the mixture velocities ranging from 0.21 m s$^{-1}$ to 0.42 m s$^{-1}$. This behaviour is similar to that observed in the computational work of Gupta et al. (2010) for the water/nitrogen system, in which a normalized Nusselt number with a value of $\sim 2.5$ was found independent of the mixture velocity over the range 0.3-1.0 m s$^{-1}$ for $\beta = 0.51$ and $L_s/d \sim 1$. It is interesting to mention here that Gupta et al. (2010) applied the stationary domain method to model Taylor flow which is different from the moving domain method used in this study, so the independence of the simulated Nusselt number on the mixture velocity is not related to the method used. However, the experimental $Nu^*$ increases slowly with the mixture velocity until 0.32 m s$^{-1}$ where the influence of $U_{TP}$ becomes greater, which causes an increase in the difference between the simulated and experimental results.
5. Validation of Modelling Results for Heat Transfer

Figure 5.9: The effect of the mixture velocity \( (U_{TP}) \) on the normalized Nusselt number for the water/nitrogen system. The experimental and simulated results are shown by solid and open symbols, respectively. The mean value of the simulated \( Nu^* \) over different mixture velocities is also shown by a dashed line.

In Fig. 5.10, the experimental \( Nu_j^* \) at the centre of blocks 2 to 9 is shown for the case with \( U_{TP} = 0.42 \text{ m s}^{-1} \) and \( \beta = 0.5 \). It can be seen that the flow is thermally fully-developed, thus the high value of the experimental \( Nu^* \) is not created by the effect of developing heat transfer.
5. Validation of Modelling Results for Heat Transfer

Figure 5.10: The experimental results for the normalized local Nusselt number at the centre of blocks 2 to 9 are shown by (Δ) symbol for the water/nitrogen system with \( U_{TP} = 0.42 \text{ m s}^{-1} \) and \( \beta = 0.5 \). The mean experimental \( Nu^* \) is also shown by a dashed line.

In Fig. 5.11 the effect of the mixture velocity on the normalized Nusselt number obtained from the simulations and experiments is shown for the ethylene-glycol/nitrogen system for \( \beta \sim 0.55 \). As shown in this figure, the value of \( L_g/d \) decreases from 2.6 to 3.5 when the mixture velocity changes in the range of 0.20-0.37 m s\(^{-1}\). It can be seen that although the value of \( Nu^* \) increases with increasing mixture velocity for the simulations and experiments, the increase in the simulated \( Nu^* \) (1.28-1.42) is still less than that of the experiments (1.12-1.49). However, for understanding the effect of higher mixture velocity on the simulated Nusselt number, case 12 of the ethylene-glycol/nitrogen system was repeated by changing only the mixture velocity from 0.37 m s\(^{-1}\) to 0.53 m s\(^{-1}\). In this case the initial value of gas hold-up (\( \epsilon_G \)) was also set similar to that of case 12 to keep the value of the liquid slug length almost unchanged. However, it can be seen that there is still a small change of around 9% in \( L_g/d \). Therefore, the increase of the simulated \( Nu^* \) from 1.40 to 1.49 by increasing the mixture velocity from 0.37 m s\(^{-1}\) to 0.53 m s\(^{-1}\) may still be affected the slug length decrease although it is small from 2.6 to 2.4.
5. Validation of Modelling Results for Heat Transfer

Figure 5.11: The effect of the mixture velocity ($U_{TP}$) on the normalized Nusselt number for $\beta \sim 0.55$ for the ethylene-glycol/nitrogen system. The experimental and simulation results are shown by solid and open symbols, respectively. The symbols (○) and (△) are used for $L_s/d$ and $Nu^*$, respectively. The results obtained for a case with $U_{TP} = 0.53 \text{ m s}^{-1}$ and $\beta = 0.56$ are also shown in red.

Overall, further investigation is needed to understand the reason of different sensitivity observed in the simulated and experimental Nusselt Numbers against mixture velocity for the two systems studied, especially for the water/nitrogen system at high mixture velocities.

5.4 Effect of the Homogenous Void Fraction

The variation of the normalized Nusselt numbers with the homogenous void fraction for different mixture velocities is shown in Figs. 5.12 and 5.13 for the water/nitrogen and the ethylene-glycol/nitrogen systems, respectively.

In Fig. 5.12, the simulated $Nu^*$ starts from the highest value, decreases with an increase in $\beta$ for both the presented mixture velocities of $0.21 \text{ m s}^{-1}$ and $0.42 \text{ m s}^{-1}$ for the water/nitrogen system. It can be seen from this figure that although the maximum value
of the experimental $Nu^*$ occurs at $\beta = 0.4$ which is different from the simulation results for $U_{TP} = 0.42$ m s$^{-1}$, the trend of the experimental and simulation data are similar. The experimental $Nu^*$ increases with increasing $\beta$ until a maximum value is reached, beyond which it decreases with increasing $\beta$.

![Graph](image.png)

Figure 5.12: The effect of the homogenous void fraction ($\beta$) on the normalized Nusselt number for the water/nitrogen system. The experimental and simulated results are shown by solid and open symbols, respectively. The symbols (○) and (△) are used for the mixture velocities of 0.21 m s$^{-1}$ and 0.42 m s$^{-1}$, respectively.

For the ethylene-glycol/nitrogen system, the simulated $Nu^*$ and the experimental results show the same trend for increasing $\beta$ for both the mixture velocities (0.20 m s$^{-1}$ and 0.37 m s$^{-1}$) in Fig. 5.13. The simulated and experimental normalized Nusselt numbers decrease as $\beta$ increases in the range of 0.17-0.68 for the mixture velocity of 0.20 m s$^{-1}$. Moreover, it can be seen for $U_{TP} = 0.37$ m s$^{-1}$ firstly the values of $Nu^*$ remain constant until $\beta = 0.42$ and 0.37 for the experiments and simulations, respectively. Then both the simulated and experimental $Nu^*$ decrease with increasing $\beta$ in this range studied.
5. Validation of Modelling Results for Heat Transfer

Figure 5.13: The effect of the homogenous void fraction ($\beta$) on the normalized Nusselt number for the ethylene-glycol/nitrogen system. The experimental and simulated results are shown by solid and open symbols, respectively. The symbols (○) and (△) are used for the mixture velocities of 0.20 m s$^{-1}$ and 0.37 m s$^{-1}$, respectively. The result obtained for a case with $U_{TP} = 0.20$ m s$^{-1}$ and $\beta = 0.87$ is shown in red.

For studying the behaviour of the simulated Nusselt number when the homogenous void fraction increases further, a case was modelled with a similar unit cell length ($L_{UC} = 18.98$ mm) and mixture velocity ($U_{TP} = 0.20$ m s$^{-1}$) to that of case 6 of the ethylene-glycol/nitrogen system, but the input value of the gas hold-up ($\varepsilon_G$) was increased from 0.48 to 0.62. It can be seen in Fig. 5.13 that the simulated normalized Nusselt number declines steadily towards a gas-only value ($Nu^* \ll 1$) when $\beta \to 1$ and it has reached 0.76 for the case with $\beta = 0.87$.

Overall, the observed trend of $Nu^*$ versus $\beta$ shown in Figs. 5.12 and 5.13 can be explained based on the works of Thulasidas et al. (1997) and Leung et al. (2010, 2012). Thulasidas et al. (1997) showed theoretically and experimentally in circular tubes for
Capillary numbers in the range of $10^{-4} < Ca < 10^{-2}$, the required time for a fluid parcel to move from one side of the liquid slug to the other side is equal to the time that the liquid slug moves a distance of twice its length, so it can be concluded that shorter slugs have more complete recirculation cycles as compared with longer slugs which have the same gas and liquid flow rates. Moreover, it was reported in the work of Leung et al. (2010, 2012) that the heat transfer enhancement would increase with increasing $\beta$ as the length of the liquid slug decreases and the recirculation number increases. However, an increase in $\beta$ means the volume fraction of the liquid has decreased in a unit cell, so the heat transfer performance decreases as the heat capacity rate of the gas is negligible compared with that of the liquid (Gupta et al., 2010). Therefore, first the effect of the shorter slug is to enhance $Nu^*$ by increasing $\beta$ until it reaches the maximum value of $Nu^*$, then this effect is offset by the liquid flow rate reduction effect which $Nu^*$ decreases by increasing $\beta$ further.

5.5 Summary

Fully-developed heat transfer results from the CFD modelling was compared with the experimental results of Leung et al. (2010, 2012) for the constant wall heat flux boundary condition for the water/nitrogen and ethylene-glycol/nitrogen systems. Based on the comparison of the simulated and experimental normalized average Nusselt numbers, it can be concluded that there is good agreement between the results obtained for both of the liquid phases used, water and ethylene-glycol. It is worth noting that most of the cases studied for the ethylene-glycol/nitrogen system and all of the cases for the water/nitrogen system with $U_{TP} = 0.21$ m s$^{-1}$ and 0.32 m s$^{-1}$ lie within the experimental uncertainty range. The only significant difference between the simulated and experimental $Nu^*$ for the water/nitrogen system was found for the unsteady cases with high mixture velocity of 0.53 m s$^{-1}$ for which the variation of the bubble tail shape appeared.

The largest difference between the simulated and experimental Nusselt numbers for the ethylene-glycol/nitrogen system was observed for the case with the high homogenous void fraction ($\beta \sim 0.7$) and $U_{TP} = 0.20$ m s$^{-1}$. Thus this case was chosen for further study. The effect of the temperature used for evaluation of the constant properties on the
simulated Nusselt numbers was found to be negligible over the range of temperatures occurring in the experimental heating section studied. However, it should be stressed that the simulated Nusselt number of this case with the largest difference was located in the range of the experimental uncertainty.

Moreover, the simulated average Nusselt number remains almost unchanged by increasing the mixture velocity for the water/nitrogen system and it is different from the experiments which $Nu^*$ increases with increasing mixture velocity. Although for the ethylene-glycol/nitrogen system the trend of the simulated average Nusselt number with increasing mixture velocity is similar to the experiments, the enhancement of the simulated $Nu^*$ with increasing $U_{TP}$ is still less than that of the experiments. The effect of the homogenous void fraction on the average Nusselt number was also studied. It was observed that the trend of the normalized average Nusselt number from the simulations was similar to the experiments. First $Nu^*$ increases with increasing homogenous void fraction from $\beta = 0$, to a critical value changing for different mixture velocities, then it decreases on further increasing the homogenous void fraction for both of the systems studied. This is explained in terms of having more complete recirculation cycles in shorter liquid slugs which enhances the heat transfer, then increasing $\beta$ further leads to a stronger effect of the liquid flow rate reduction than the shorter liquid slug effect.
Chapter 6

Conclusions and Recommendations

This work was aimed at improving the knowledge of the flow field and heat transfer behaviour in gas-liquid Taylor flow in microchannels by applying CFD modelling. Two methodologies have been developed in this thesis for the computational modelling of Taylor flow. In the two approaches, bubble and slug lengths can be simulated with prescribed values. Therefore, the CFD simulation results have been compared hydrodynamically and thermally with the experimental data having very close bubble and slug lengths. These investigations improved computational techniques for understanding the complex behavior of the flow and heat transfer in Taylor flow.

The aim of this thesis was designed according to the important gaps of this field revealed from the literature review in Chapter 2. The following sections summarize the major findings of this work and these are followed by suggestions for further investigations in this area of gas-liquid Taylor flow in microchannels.

6.1 Approaches to Modeling Taylor Flow

Two approaches were developed to simulate gas-liquid Taylor flow implemented in ANSYS-Fluent. The first, and conceptually simplest, (named stationary domain method) is to generate bubbles and slugs in a long tube by using a time-dependent boundary condition. In the second method (called moving domain method), the flow and heat transfer in a single unit cell, consisting of a bubble surrounded by liquid slugs, is solved in a frame of reference moving with the bubble velocity. Both methods used the volume of fluid (VOF) method to capture the gas-liquid interface.

Simulations for a two-phase (liquid-only) Reynolds number of 713, Capillary number of 0.004 and void fraction of 0.366 for water/nitrogen flow were performed to compare the two techniques. The bubble shapes were very similar, each giving a 41 µm film thickness in the constant film region for both the stationary and moving domains. The wall shear
stresses obtained from the two methods were also very close with small differences in the bubble film region and the liquid slug region which can be explained by the velocity gradients not being wrapped in the moving domain method.

In addition, there was a very large difference between the required computational mesh sizes and times for the two methods, with a wall clock time of 38 hours on a single processor for the moving domain simulation compared with 1460 hours using four processors for the stationary domain approach. For a constant wall heat flux boundary condition, the average Nusselt number obtained for the stationary domain was larger than that of the moving domain taking the values of 11.4 and 10.8, respectively. The difference observed was explained by the fact that even with 14 bubbles present in a long tube thermal development was not achieved. Overall, the hydrodynamic and heat transfer results obtained from the two approaches were found to be very similar, giving confidence in the implementation of both methods.

6.2 Validation of the Simulated Hydrodynamics Results

Fully-developed flow results obtained from the simulations using the moving domain method were compared with the experimental data of Leung et al. (2010, 2012) for two systems of water/nitrogen and ethylene-glycol/nitrogen system. The cases simulated covered a wide Reynolds number range of 475-1189 (water/nitrogen), 22-40 (ethylene-glycol/nitrogen) and a Capillary number range of 0.0026-0.0066 (water/nitrogen), 0.0850-0.1573 (ethylene-glycol/nitrogen).

The homogenous void fractions ($\beta$) obtained from the simulations were very close to the experimental data using the void fraction ($\epsilon_0$) and the mixture velocity ($U_{TP}$) as the input parameters for two systems, water/nitrogen and ethylene-glycol/nitrogen. After reaching the final bubble shape over time, the lengths of the gas bubbles and liquid slugs were also in good agreement with the experimental results.

For the ethylene-glycol/nitrogen system, the liquid film thicknesses obtained from the simulations were compared with those measured directly from experimental photographic
images with the average difference being 3.1%. Also, the bubble shapes obtained for this system from CFD simulations were very similar to those obtained from experiments for different mixture velocities and homogenous void fractions. While for the water/nitrogen system as the location of the gas and liquid interface could not be differentiated from the wall of the tube over 50% of the tube diameter, the film thickness from the simulations were compared with those from the Suo and Griffith (1964) correlation (established for a stationary and constant thickness film) using the experimental bubble velocity and mixture velocity and the values calculated were 15-42% higher than the simulations.

The pressure drops from the simulations were validated using the Kreutzer et al. (2005) correlation (developed for long slugs and thin liquid films), as there was no experimental data from Leung et al. (2010, 2012) for the pressure drop. The average differences between the results obtained from the modelling and correlation were 16% and 6% for the water/nitrogen and ethylene-glycol/nitrogen, respectively.

Overall, the simulated hydrodynamics results were very similar to the experimental results for both of the liquid phases studied, water and ethylene-glycol. Another interesting finding was the observation of the oscillations at the bubble tail shape of the simulations for the water/nitrogen system at high mixture velocity ($U_{TP} = 0.53 \text{ m s}^{-1}$) corresponds to $Re = 1189$. This unsteady behaviour also appeared at the bubble tail for the experiments from the mixture velocity of 0.42 m s$^{-1}$. The good agreement between the simulations and experiments was found for the bubble and slug lengths, the liquid film thickness and the bubble velocity at $U_{TP} = 0.53 \text{ m s}^{-1}$ although unstable flow was created at the end of the gas bubble. The time-dependence of the bubble tail shape was found to affect only the simulated normalized pressure drop significantly for example for a case with $U_{TP} = 0.53 \text{ m s}^{-1}$ and $\beta = 0.49$, the simulated normalized pressure drop changed from 0.2 to 2.1.

### 6.3 Validation of the Simulated Heat Transfer Results

Fully-developed heat transfer results obtained from the CFD modellings for the cases described in Section 6.2 for the water/nitrogen and ethylene-glycol/nitrogen systems were
compared with the experimental heat transfer data of Leung et al. (2010, 2012) for the constant wall heat flux boundary condition.

For the water/nitrogen system, the ratio of the average Nusselt number to that of the liquid-only flow (normalized average Nusselt number) from the simulations were close to the experiments with the average difference being 4% for the mixture velocities of 0.21 m s$^{-1}$ and 0.32 m s$^{-1}$. The simulated normalized average Nusselt numbers ($Nu^*$) all lie within the experimental uncertainty range which is calculated from the scatter between heating section blocks of the experimental Nusselt numbers for each case. The maximum deviation value of around 30% was found for a mixture velocity of 0.53 m s$^{-1}$, having a Reynolds number of 1189 and these cases were defined as unsteady cases due to observing the fluctuations at the bubble tail shape versus time for the modelling and experiments.

For the ethylene-glycol/nitrogen system, the normalized average Nusselt number ($Nu^*$) from the simulations and experiments were in good agreement having an average difference of 8%. All of the simulated $Nu^*$ fell in the range of the experimental uncertainty even the case with the largest difference of 22% between the simulation and experiments for $U_{tp} = \text{0.20 m s}^{-1}$ and $\beta = 0.67$.

The simulated $Nu^*$ for the water/nitrogen system remained approximately constant for similar homogenous void fractions for the mixture velocities ranging from 0.21 m s$^{-1}$ to 0.42 m s$^{-1}$, whereas the experimental $Nu^*$ increased by increasing the mixture velocity. For the ethylene-glycol/nitrogen system, both the simulated and experimental normalized Nusselt numbers increased with increasing mixture velocity, but still the effect of the mixture velocity was more significant in the experimental results than the simulations.

For both of the systems, water/nitrogen and ethylene-glycol/nitrogen, the trend of the simulated and experimental $Nu^*$ versus the homogenous void fraction were similar. The normalized Nusselt numbers increased with increasing homogenous void fraction until a maximum value which varied with the mixture velocity was reached and this behaviour
was explained by the fact that there are more complete recirculation cycles in the liquid slug zone for shorter slugs to enhance heat transfer from the wall to the liquid. After reaching the critical value, the $Nu^*$ decreases with increasing homogenous void fraction because the effect of the liquid flow rate reduction becomes stronger than the shorter slug effect.

### 6.4 Recommendations

The following recommendations are given for further research in the CFD modelling of gas-liquid Taylor flow in microchannels which are not covered in this thesis:

- On increasing the mixture velocity to high values of 0.42 m s$^{-1}$ and 0.53 m s$^{-1}$ for the water/nitrogen system corresponding to Reynolds numbers of 951 and 1189, respectively, unsteady ripples at the bubble tail were observed in the experimental work (Leung et al, 2010; 2012). Therefore, the accuracy of the two-dimensional, axisymmetric assumption for the CFD simulations used in this work becomes worse and three-dimensional simulations are required to understand the effect of such unsteady oscillations at the bubble tail shape. It is recognized that three-dimensional modelling even for one unit cell applying the moving domain method needs significant larger meshes and the flow and energy equations should be solved simultaneously at high mixture velocity because of the unsteady behaviour of the flow field, so parallelizing the user defined functions used for the moving domain method would definitely be helpful to decrease the time for running the transient simulations.

- The existence of the thermal capacity of the solid wall was neglected in the present CFD modelling study of heat transfer in a fully-developed Taylor flow. It would be interesting to investigate the effect of the wall on the heat transfer behaviour.

- The moving domain method which is more efficient than the stationary domain method for studying the flow field and heat transfer in a fully-developed Taylor flow has been developed only for the constant wall heat flux boundary condition.
Extending this method for the constant wall temperature boundary condition would make this method suitable to investigate Taylor flow in a wider range of conditions.
References


APPENDIX A

Harmonic Averaging Viscosity

The default average viscosity cannot be changed directly from arithmetic to harmonic averaging in ANSYS Fluent. Therefore, it was done via a UDF (given in Appendix B) in which the liquid viscosity is given a constructed value, such that when the viscosities are averaged arithmetically the result is equivalent to harmonic averaging.

\[
\mu_{L2} = \frac{1}{\alpha_L} \left[ \left( \frac{(1 - \alpha_L)}{\mu_G} + \frac{\alpha_L}{\mu_L} \right)^{-1} - (1 - \alpha_L)\mu_G \right]
\]

(A1)

The liquid viscosity was used in the construction as it avoided the calculation of negatives values for the artificial viscosity.
APPENDIX B

UDF for Calculating the Harmonic Averaged Viscosity

#include "udf.h"

DEFINEPROPERTY(liquid_viscosity,c,t)
{
    real mul2, mug, mul,vfl;

    mug = 0.000021; /*The gas viscosity*/
    mul = 0.014; /*The liquid viscosity*/

    vfl = C_VOF(c,t); /*The volume fraction of the liquid*/

    if (vfl > le-8) /*If liquid is present*/
    {
        mul2 = (l/vfl)*((l/(((l-vfl)/mug)+((vfl)/mul)))-(l-vfl)*mug); /*The constructed value for the liquid viscosity using equation (A1)*/
        return mul2;
    }
    else
    {
        mul2 = mul;
        return mul2;
    }
}
APPENDIX C

UDF for Transient Inlet B.C. for Stationary Domain Method

```c
#include "udf.h"

DEFINE_PROFILE(length_control,t,i)
{
    face_t f;
    int ti;
    real tUC, tG, t_nG, t_n, R, R_in, r, r_in, y;

    real xc[ND_ND]; /*This will hold the position vector*/
    real flow_time=RP_Get_Real("flow-time");

    tUC = 0.0388;  /*Time for creating one unit cell*/
    tG  = 0.0165;  /*Time for creating one bubble*/
    t_nG  = tG/tUC; /*Non-dimensional bubble creation time*/
    t_n = flow_time/tUC; /*Non-dimensional flow time*/
    ti = floor(flow_time/tUC); /*Integer part of non-dimensional flow time*/

    R = 0.001;  /*Radius of tube in m*/
    R_in = 0.00097;  /*Inlet radius of gas entrance in m*/
    r_in = R_in/R; /*Non-dimensional inlet radius*/

    if ((t_n-ti) < t_nG ) /*If non-dimensional time is less than non-dimensional time to create a bubble, the gas should enter from the inside part of the inlet, otherwise liquid should enter from whole of the inlet*/
```

111
{
begin_f_loop(f,t) /*Loop over all inlet faces*/
{
    F_CENTROID(xc,f,t);
    y=xc[1]; /*Radial coordinate*/
    r=y/R; /*Non-dimensional radial coordinate*/

    if (r<r_{in}) /*If the radial coordinate is less than the inlet radius, the gas
    volume fraction should be one for that face otherwise it should be zero*/
    {
        F_PROFILE(f,t,i)= 1;
    }

    else
    {
        F_PROFILE(f,t,i)= 0;
    }

} 
end_f_loop(f,t)

}

else
{
begin_f_loop(f,t)
{
    F_PROFILE(f,t,i)=0.0;
}
end_f_loop(f,t)

}

}
APPENDIX D

UDFs for the Moving Domain Method

#include "udf.h"

double UB; /* Bubble Velocity, a global variable */

double yo[65]; /*Outlet radial coordinate, a global variable*/
double uo[65]; /*Outlet axial velocity, a global variable*/
double vo[65]; /*Outlet radial velocity, a global variable*/
double to[65]; /*Outlet temperature, a global variable*/

double yi[65]; /*Inlet radial coordinate, a global variable*/
double ti[65]; /*Inlet temperature, a global variable*/

UDF 1: Initialization of the Bubble Shape

DEFINE_INIT(initial_bubble_shape,domain)
{
    Thread *t;
    Thread **pt;
    Thread **st;
    cell_t c;
    Domain *pDomain=DOMAIN_SUB_DOMAIN(domain,P_PHASE);
    Domain *sDomain=DOMAIN_SUB_DOMAIN(domain,S_PHASE);
    real R, Ri, r, ri, y, x, b;
    real xc[ND_ND]; /*This will hold the position vector*/
R = 0.001; /*The radius of the tube in m*/
Ri = 0.000902; /*The radius of initial bubble shape in m*/
ri = Ri/R; /*The non-dimensional initial radius*/

/*Specify the initial cylindrical shape of the bubble by setting the volume fraction of
primary phase, which is the liquid, here*/

mp_thread_loop_c(t, domain, pt)
{
  Thread *tp=pt[P PHASE];
  begin_c_loop(c,t)
  {
    C_CENTROID(xc,c,t);
    x=xc[0];
    y=xc[1];
    r=y/R; /*non-dimensional radial coordinate*/

    if ((x>0.00386)&&(x<0.00994))
    {
      if (r<ri)
        C_VOF(c,tp)=0;
      else
        C_VOF(c,tp)=1;
    }
    else
    {
      C_VOF(c,tp)=1;
    }
  }
  end_c_loop(c,t)
}
/*Specify the initial cylindrical shape of the bubble by setting the volume fraction of secondary phase, which is gas, here*/

mp_thread_loop_c(t, domain, st)
{
    Thread *sp=st[S_PHASE];
    begin_c_loop(c,t)
    {
        C_CENTROID(xc,c,t);
        x=xc[0];
        y=xc[1];
        r=y/R; /*non-dimensional radial coordinate*/

        if ((x>0.00386)&&(x<0.00994))
        {
            if (r<ri)
                C_VOF(c,sp)=1;
            else
                C_VOF(c,sp)=0;
        }
        else
        {
            C_VOF(c,sp)=0;
        }
    }
    end_c_loop(c,t)
}
}
UDF 2: Initialization of the Velocity Profile

DEFINE_INIT(init_velocity, domain)
{
    Thread *t;
    cell_t c;

    real R, r, y, x, UTP;
    real xcl[ND_ND]; /*This will hold the position vector*/

    R = 0.001; /*The radius of the tube in m*/
    UTP = 0.32; /*The mixture velocity in m/s*/

    thread_loop_c(t, domain)
    {
        begin_c_loop_all(c, t) /*Loop over all cells*/
        {
            C_CENTROID(xc1, c, t);
            x = xc1[0];
            y = xc1[1];
            r = y/R; /*Non-dimensional radial coordinate*/

            C_U(c, t) = 2*UTP*(1-(r*r)); /*Specify the initial parabolic velocity profile for whole of the domain*/
        }
        end_c_loop_all(c, t)
    }
}
UDF 3: Storing the Outlet Data

DEFINE_ADJUST(outlet_store,domain)
{
    real FC[2];
    face_t f;

    int IDo=7;/*Zone ID for outlet zone from Boundary Conditions task page*/
    int n=0;

    Thread *thread=Lookup_Thread(domain,IDO);

    /*Getting the radial position, velocities and temperature of the outlet zone, IDo=7*/

    begin_f_loop(f,thread)
    {
        F_CENTROID(FC,f,thread);

        yo[n]=FC[1];
        uo[n]=F_U(f,thread);
        vo[n]=F_V(f,thread);
        to[n]=F_T(f,thread);

        n+=1;
    }
    end_f_loop(f,thread)
}
UDF 4: Storing the Inlet Data

DEFINE_ADJUST(inlet_store, domain)
{
    real FC[2];
    face_t f;

    int IDi=6;/*Zone ID for inlet zone from Boundary Conditions task page*/
    int n=0;

    Thread *thread1=Lookup_Thread(domain, IDi);

    /*Getting the radial position and temperature of the inlet zone, IDi=6*/

    begin_f_loop(f, thread1)
    {
        F_CENTROID(FC, f, thread1);

        yi[n]=FC[1];
        ti[n]=F_T(f, thread1);

        n+=1;
    }
    end_f_loop(f, thread1)
UDF 5: Wrapping the Axial Velocity from the Outlet to the Inlet

```cpp
DEFINE_PROFILE(u_velocity,t,i)
{
    face_t f;
    real r, R, y, UTP;
    real xc[ND_ND]; /*This will hold the position vector*/
    real time_step=RP_Get_Integer("time-step");
    int n=0;

    R = 0.001; /*The radius of tube in m*/
    UTP = 0.32; /* Mixture velocity in m/s*/

    /*For the first time step, use the parabolic velocity profile at the inlet*/

    if ((time_step)<2)
    {
        begin_f_loop(f,t)
        {
            F_CENTROID(xc,f,t);
            y=xc[1];
            r=y/R; /*Non-dimensional radial coordinate*/

            F_PROFILE(f,t,i)= 2*UTP*(1-(r*r));
        }
        end_f_loop(f,t)
    }

    /*After first time step, start to wrap the axial velocity from the outlet to the inlet*/
```
else
{
    begin_f_loop (f,t)
    {
        F_CENTROID(xc,f,t);
        for (n=0; n<65; n++)
        {
            if ((fabs(xc[1] - yo[n])) < 0.000000001)
            {
                F_PROFILE (f,t,i) = uo[n];
            }
        }
    }
    end_f_loop (f,t)
}
UDF 6: Wrapping the Radial Velocity from the Outlet to the Inlet

DEFINE_PROFILE(v_velocity,t,i)
{
    face_t f;
    int n=0;

    real xc[ND_ND];/*This will hold the position vector*/
    real time_step=RP_Get_Integer("time-step");

    /*For the first time step, set the radial velocity to zero at the inlet*/

    if ((time_step)<2)
    {
        begin_f_loop(f,t)
        {
            F_PROFILE(f,t,i)= 0;
        }
        end_f_loop(f,t)
    }

    /*After first time step, start to wrap the radial velocity from the outlet to the inlet*/

    else
    {
        begin_f_loop (f, t)
        {
            F_CENTROID(xc,f,t);
        }
    }
for (n=0;n<65;n++)
{
    if ((fabs(xc[1]-yo[n]))<0.000000001)
    {
        F_PROFILE (f,t,i) =vo[n];
    }
}
end_f_loop (f, t)
UDF 7: Calculating the Bubble Velocity at Every Time Step

DEFINE_ADJUST(bubble_velocity, domain)
{

FILE *f1;

Thread *t;
Thread *st;
cell_t c;

real alphau, alpha;
real xc[ND_ND]; /* This will hold the position vector*/

f1 = fopen("bubble_velocity.txt","a"); /* A file to record bubble velocity which is used for the dynamic mesh velocity*/

/* Calculating the bubble velocity*/

thread_loop_c(t, domain)
{

st=THREAD_SUB_THREAD(t,1);

begin_c_loop(c, t)
{

C_CENTROID(xc,c,t);

alphau+=C_VOF(c,st)*C_U(c,t)*C_VOLUME(c,t); /* Sum all the axial velocities of the bubble cells*/
alpha+=C_VOF(c,st)*C_VOLUME(c,t);
}

end_c_loop(c,t)
UB = alphau/alpha;
fprintf(f1,"UB = %g\n", UB);
}
fclose(f1);
}

UDF 8: Moving the Dynamic Mesh with an Axial Velocity equal to the Bubble Velocity

DEFINE_CG_MOTION(mesh_velocity, dt, vel, omega, time, dtime)
{
    vel[0] = UB;
}
UDF 9: Wrapping the Modified Temperature from the Inlet to the Outlet for the Back Flow Part

DEFINE_PROFILE(outlet_temp, thread, i)
{
    real xc[ND_ND];
    real deltaT;
    face_t f;
    int n=0;

    deltaT = 1.103; /* The difference between the inlet and outlet temperature is calculated based on the energy balance over a unit cell for H2 B.C.*/

    begin f loop (f, thread);
    {
        F_CENTROID(xc,f,thread);

        for (n=0; n<65; n++)
        {
            if ((fabs(xc[1]-yi[n]))<0.0000000001) /*Specify the outlet faces corresponding to the inlet faces*/
            {
                F_PROFILE (f,thread,i) = ti[n] + deltaT; /*Wrap the modified inlet temperatures to the corresponded outlet*/
            }
        }
    }
    end_f_loop (f, thread)
}
UDF 10: Wrapping the Modified Temperature from the Outlet to the Inlet for the Forward Flow Part

```
DEFINE_PROFILE(inlet6_temp,thread, i)
{
    real xc[ND_ND];
    real deltaT;
    face_t f;
    int n=0;

    deltaT = 1.103;

    begin_f_loop (f, thread);
    {
        F_CENTROID(xc,f,thread);

        for (n=0;n<65;n++)
        {

            if ((fabs(xc[l]-yo[n]))<0.0000000001) /*Specify the inlet faces corresponding to the outlet faces*/
            {

                F_PROFILE (f,thread, i) = to[n]-deltaT; /*Wrap the modified outlet temperatures to the corresponded inlet*/
            }
        }
    }
    end_f_loop (f, thread)
}
```
APPENDIX E

Energy Balance in Taylor Flow

The conservation of energy for a multiphase system can be written as in equation (E1) below:

\[
\frac{\partial}{\partial t} \left( \sum \alpha_i \rho_i h_i \right) + \nabla \cdot \left( \sum \alpha_i \rho_i h_i \mathbf{v} \right) = -\nabla \cdot \mathbf{q} \quad (E1)
\]

Integrating equation (C1) over a control volume gives

\[
\int \frac{\partial}{\partial t} \left( \sum \alpha_i \rho_i h_i \right) \, dV + \int \nabla \cdot \left( \sum \alpha_i \rho_i h_i \mathbf{v} \right) \, dV = \int -\nabla \cdot \mathbf{q} \, dV \quad (E2)
\]

The volume integrals in the second and third terms of equation (E2) can be written as surface integrals using Gauss theorem. In the first term, the integration is performed over a fixed control volume so the limits are independent of time and the time derivative can be taken out of the integrand.

\[
\frac{\partial}{\partial t} \int \sum \alpha_i \rho_i h_i \, dV + \int \sum \alpha_i \rho_i h_i \mathbf{v} \cdot dS = \int -\mathbf{q} \cdot dS \quad (E3)
\]

In a laboratory frame of reference, the flow is not steady but is periodic with a time period of \( t_B \). \( t_B \) is the time in which a unit cell passes through a given cross-section. Integrating equation (E3) with respect to time from time \( t \) to \( t_B \), we get

\[
\int_t^{t+t_B} \frac{\partial}{\partial t} \int \sum \alpha_i \rho_i c_{pi} T \, dV \, dt = \left[ \int \alpha_i \rho_i c_{pi} T \, dV \right]_t^{t+t_B}
\]

\[
= \int_t^{t+t_B} \int -\mathbf{q} \cdot dS \, dt - \int_t^{t+t_B} \int \sum \alpha_i \rho_i c_{pi} T \mathbf{v} \cdot dS \, dt \quad (E4)
\]
As the flow is periodic, the first term (on the left) in the equation (E4) will be zero because the accumulation term will be the same at times \( t \) and \( t + t_B \). The heat conduction from the inlet and outlet boundaries would be cancelled out as the flow is periodic. So the equation (E4) can be rewritten as

\[
\int_t^{t+t_B} \int_{\text{in}} \alpha_i \rho_i c_{pi} T v \, dS \, dt - \int_t^{t+t_B} \int_{\text{out}} \alpha_i \rho_i c_{pi} T v \, dS \, dt + \int_{\text{wall}} q \, dS \, t_B = 0 \quad (E5)
\]

Equation (E5) is valid for constant wall heat flux, as well as constant wall temperature boundary conditions.

Combining the first two terms, we can write

\[
\int_t^{t+t_B} \int_{X_S} \alpha_i \rho_i c_{pi} (T_{\text{in}} - T_{\text{out}}) v \, dS \, dt + \int_{\text{wall}} q \, dS \, t_B = 0 \quad (E6)
\]

For a constant wall heat flux boundary condition, the difference between outlet and inlet temperature is a constant, say \( \Delta T \).

\[
\int_t^{t+t_B} \int_{X_S} \alpha_i \rho_i c_{pi} \Delta T \, v \, dS \, dt = q_w A_w t_B \quad (E7)
\]

Now \( dt \) can be written as \( dx / U_B \) and \( t_B \) as \( (L_{UC} / U_B) \) so the equation (E7) can be modified as

\[
\int_x^{x+L_{UC}} \int_{X_S} \alpha_i \rho_i c_{pi} \Delta T \, v \, dS \, dx' = q_w A_w L_{UC} \quad (E8)
\]

At first instance, the choice of \( U_B \) appears to be arbitrary and therefore is important to note here if any velocity other than \( U_B \) is used to convert \( dt \) to \( dx \), the result would be different as the velocity will not cancel out.
Expanding above equation in terms of average quantities gives

$$\Delta T \left( \rho_G c_{pg} \int_{cv} \alpha_G \, v \, dV + \rho_L c_{pl} \int_{cv} \alpha_L \, v \, dV \right) = q_w A_w L_{UC} \quad (E9)$$

$$\Delta T A_{XS U TP} (\beta \rho_G c_{pg} + (1 - \beta) \rho_L c_{pl}) = q_w A_w \quad (E10)$$

$\Delta T$ obtained from the equation (E10) gives the increment in the bulk temperature over a unit cell thermally fully-developed Taylor flow for constant wall heat flux boundary condition.