AMERICAN UNIVERSITY OF BEIRUT

CFD Simulation of Corrugated Plate Heat Exchangers: Single phase and Boiling Flows

by

Ali I. Al Abed

A thesis
submitted in partial fulfillment of the requirements
for the degree of Master of Engineering
to the Department of Mechanical Engineering
of the Faculty of Engineering and Architecture
at the American University of Beirut

Beirut, Lebanon
October 2020
CFD Simulation of Corrugated Plate Heat Exchangers: Singlephase and Boiling Flows

by

Ali I. Al Abed

Approved by:

Dr. Fadl Moukalled, Professor
Mechanical Engineering

Advisor

Dr. Aristides M. Bonanos, Assistant Professor
Mechanical Engineering, The Cyprus Institute
Co-advisor

Dr. Marwan Darwish, Professor
Mechanical Engineering
Member of Committee

Date of thesis defense: October 15, 2020
AMERICAN UNIVERSITY OF BEIRUT

THESIS, DISSERTATION, PROJECT RELEASE FORM

Student Name: Al Abed Ali Ibrahim

Last First Middle

☒ Master’s Thesis ☐ Master’s Project ☐ Doctoral Dissertation

☒ I authorize the American University of Beirut to: (a) reproduce hard or electronic copies of my thesis, dissertation, or project; (b) include such copies in the archives and digital repositories of the University; and (c) make freely available such copies to third parties for research or educational purposes.

☐ I authorize the American University of Beirut, to: (a) reproduce hard or electronic copies of it; (b) include such copies in the archives and digital repositories of the University; and (c) make freely available such copies to third parties for research or educational purposes after: One ___ year from the date of submission of my thesis, dissertation or project.
Two ___ years from the date of submission of my thesis, dissertation or project.
Three ___ years from the date of submission of my thesis, dissertation or project.

Signature: Ali

Date: Oct 23, 2020

This form is signed when submitting the thesis, dissertation, or project to the University Libraries.
Acknowledgements

The present project has been carried out at the Computational Fluid Dynamics Laboratory at the American University of Beirut (AUB) and at the Cyprus Institute under the supervision of Prof. Fadl Moukalled (AUB) and Prof. Aristides Bonanos (Cyprus Institute). The access to the HPC units and the financial support have been provided by Maroun Semaan faculty of engineering and architecture at AUB and they are gratefully acknowledged.

I express my sincere gratitude to Prof. Fadl Moukalled and Prof. Aristides Bonanos for the opportunity to work on such an interesting and challenging engineering research project. Since the beginning of the project they always provided constructive advice, knowledge and experience till the end of the project. I always appreciated their full confidence in my research skills, decision making and leadership abilities. Also, I would like to thank again Prof. Bonanos for performing the single phase and flow boiling experimental runs at the Cyprus Institute.

Furthermore, I would like to thank all my CFD lab members for the great time and the strong bonds that we had which generated an all time unique social and scientific environment. Much appreciation for my dearest friend Leonie Schmid who endlessly listened and supported me through all of the project details and milestones. I like to thank also my best friends from my neighborhood who always felt proud of my work and kept the bond of friendship between us for such long time.

My warmest thankfulness is for my parents: Ibrahim Al Abed and Jihan Khalaf for their sacrifices, endurance, encouragement, believe, love and the smiles throughout all my life moments.
An Abstract of the Thesis of

Ali Ibrahim Al Abed     for     Master of Engineering
Major: Mechanical Engineering - Applied Energy

Title: CFD Simulation of Corrugated Plate Heat Exchanger: Single phase and Boiling Flows

Corrugated plate heat exchangers are characterized by enhanced hydraulic and thermal performance due to highly turbulent flow generated by the complex geometry and the large compacted heat transfer area. In part 1 of the present work, CFD simulations are performed to study the single phase flow inside a full corrugated channel. The thermal and hydraulic characteristics as well as the low and high Reynolds turbulence modeling were analyzed and discussed. In part 2, the two phase subcooled and saturated boiling flow heat transfer and pressure drop for water were simulated at low mass flux and vapor quality. The Semi Mechanistic Wall boiling model based on Chen’s method is used. The results shows that the two phase heat transfer coefficient increased with the increasing mass flux and degree of subcooling and it decreased with increasing the wall temperatures. Whereas the pressure drop increased with the increasing mass flux, wall temperature and vapor quality. The dominant boiling mechanism is found to be nucleate boiling. Lastly, the flow dynamic and vapor formation were processed and analyzed.
# Contents

Acknowledgements v  
Abstract vi

1 Introduction 1  
1.1 Overview of Plate Heat Exchangers 1  
1.2 Thesis Objective 1  
1.3 Thesis Structure 3  

2 Literature Review 5  
2.1 Single Phase Flow Simulation Studies 5  
2.2 Introduction to Flow Boiling Heat Transfer and Pressure Drop 7  
2.3 Flow Boiling Experimental Studies 10  

3 Corrugated Plate Geometry Modeling 14  

4 Experimental Setup 18  

5 Single Phase Simulation 20  
5.1 Governing Equations 20  
5.2 Turbulence Modeling 21  
5.3 PHE CFD Single Phase Model 22  
5.3.1 Numerical Setup 22  
5.3.2 Boundary conditions 22  
5.3.3 Materials Properties 23  
5.3.4 Data Reduction 24  
5.3.5 Grid Independence Study 25  
5.3.6 Model Validation and Assessment 29  
5.4 Results and Discussion 31  
5.5 Conclusion 36  

6 Flow Boiling Simulation 37  
6.1 Semi Mechanistic Wall Boiling Model 37  
6.2 Mixture Governing Equations 40
6.3 Turbulence Modeling ............................................. 41
6.4 Mass Transfer Model ............................................. 42
6.5 PHE CFD Two Phase Flow Model ................................. 42
  6.5.1 Numerical Setup ............................................. 42
  6.5.2 Boundary Conditions ......................................... 44
  6.5.3 Material Properties ............................................ 44
  6.5.4 Grid Generation and Independence Study .................. 45
  6.5.5 Data Reduction ................................................ 46
  6.5.6 Model Validation .............................................. 48
6.6 Results and Discussion ........................................... 49
  6.6.1 Assessment of heat transfer coefficient and pressure drop
        literature correlations ........................................ 49
  6.6.2 Evaluation of mass flux effect and flow inlet temperature . 51
  6.6.3 Evaluation of the heating wall temperature ................ 53
  6.6.4 Two phase flow analysis ..................................... 56
  6.6.5 Dominant boiling regime ................................... 68
6.7 Conclusion ....................................................... 70

7 Conclusion ......................................................... 71
  7.1 Main Findings of the Present Study ............................. 71
  7.2 Recommendations for Future Work .............................. 72

A Nomenclature ..................................................... 73
# List of Figures

1.1 Flow arrangement inside PHE. Courtesy of Alfa Laval .......................... 2

2.1 Development of the two phase flow in smooth pipe under uniform heat flux (Left) and variation of heat transfer coefficient with vapor quality (Right)[1] ......................................................... 8

3.1 Corrugated plate and zones description ............................................. 15
3.2 Plate Geometry (left) Plane view (right) cross sectional view normal to the corrugation [2] ................................................................. 16
3.3 Plan view and cross sections of the computational fluid domain ....... 17

4.1 Schematic diagram of the experimental test unit ................................. 18
4.2 Channels flow diagram ..................................................................... 19

5.1 Applied boundary conditions for the single phase simulations .......... 23
5.2 3D and sectional view of the Coarse mesh for high Re modeling ..... 27
5.3 3D and sectional view of the fine mesh for low Re modeling .......... 28
5.4 The variation of Nusselt number with the channel Reynolds number based on the predictions of CFD and Correlation and experimental results ................................................................. 30
5.5 The variation of Fanning friction factor with the channel Reynolds number based on the predictions of CFD and Correlation .......... 30
5.6 Variation of temperature with respect to the axial distance at different Reynolds number ............................................................... 31
5.7 Variation of temperature with respect to the axial distance at different Re .............................................................. 32
5.8 Pressure contours on the plate wall ..................................................... 33
5.9 Velocity streams for $Re = 530$ (Left) and $Re = 1500$ (Right) ...... 34
5.10 variation of inner wall temperature predictions according to $Re =$ 530 (Left) and $Re = 1500$ (Right) ........................................... 35

6.1 Applied boundary conditions for the flow boiling simulations .......... 44
6.2 Polyhedral grid for the flow boiling simulations: 3D side and cross sectional views (for section A-A’ and B-B’ locations refer to 3.3) . 47
6.3 Comparison between the CFD model predictions and experimental results for Surface heat flux at varying inlet mass flux. Inlet and wall temperatures are set to $T_{in} = 30 \, ^\circ C$ and $T_w = 110 \, ^\circ C$ respectively ........................................ 49

6.4 Comparison between the CFD model predictions and experimental results for the outlet mass quality at varying inlet mass flux and wall temperatures. Inlet temperature was fixed at $T_{in} = 30 \, ^\circ C$ .... 50

6.5 Comparison between the CFD and correlations predictions for the the heat transfer coefficient and pressure drop at different mass flux. The wall and inlet temperatures are set to $T_w = 110 \, ^\circ C$ and $T_{in} = 30 \, ^\circ C$ respectively ........................................ 52

6.6 Variation of heat transfer coefficient, pressure drop, and outlet vapor mass quality with mass flux for subcooled and saturated liquid inlet flows. Wall temperature is set to $T_w = 110 \, ^\circ C$ ........ 54

6.7 Comparison of heat flux at different wall temperature and mass flux. $T_{in}$ and $T_{sat} = 100^\circ C$ .................................................. 55

6.8 Variation of the two phase heat transfer coefficient (A), Pressure drop (B) and outlet vapor mass quality (C) as function of wall temperature at different mass fluxes. In all cases the inlet is subcooled $T_{in} = 30 \, ^\circ C$ .................................................. 57

6.9 Variation of the two phase heat transfer coefficient (A), Pressure drop (B) and outlet vapor mass quality (C) as function of wall temperature at different mass fluxes. In all cases the inlet is saturated $T_{in} = 100 \, ^\circ C$ .................................................. 58

6.10 Vapor volume fraction distribution at the heating wall. $G = 30$ $Kg/m^2s$, $T_w = 104 \, ^\circ C$, and $T_{in} = 30 \, ^\circ C$ .................................................. 59

6.11 Average bulk fluid temperature variation with the downward axial distance for the case reported in fig. 6.10. Note that the point center zero is the center of the geometry. Refer to chapter 3 for center location .................................................. 60

6.12 Plots of mass transfer rate, vapor fraction, and velocity vectors for area between section 1-1' and 2-2' ........................................ 61

6.13 Mixture temperature (A) and vapor volume fraction (B) contours plot at Section 1-1', 2-2' and 3-3' respectively from top to bottom. Refer to fig. 6.10 for section location ........................................ 62

6.14 Effect of mass flux on vapor distribution. From left to right $G = 20, 40, 60 \, Kg/m^2s$ .................................................. 63

6.15 Mass transfer rate for subcooled (left) and saturated (right) inlet. The mass flux is set to $G = 30 \, Kg/m^2s$ and the wall temperature $T_w = 104 \, ^\circ C$. .................................................. 65

6.16 Velocity vector plot for subcooled inlet (left) and saturated (right). The mass flux is set to $G = 30 \, kg/m^2s$ and the wall temperature $T_w = 104 \, ^\circ C$. .................................................. 66
6.17 Pressure contours for $G = 30 \text{ kg/m}^2\text{s}$ at $T_w = 104 \degree C$ and $T_w = 110 \degree C$ for saturated inlet $T_{in} = 100 \degree C$ ........ 67
6.18 Thonon criteria for determining the dominant boiling. A logarithmic scale is applied on the x-axis and y-axis ........ 69
List of Tables

3.1 Corrugated plate geometrical characteristics and dimensions . . . 16
5.1 Grid A independence study . . . . . . . . . . . . . . . . . . . . . . . . 26
5.2 Grid B independence study . . . . . . . . . . . . . . . . . . . . . . . . 26
6.1 Polyhedral grid independence study . . . . . . . . . . . . . . . . . . . 46
Chapter 1

Introduction

1.1 Overview of Plate Heat Exchangers

Corrugated plate heat exchangers (PHE) have been recognized as efficient and practical heat exchangers for many industries such as in chemicals, food processing, refrigeration and air conditioning etc. This is primarily due to compactness, low fouling rates, flexibility of adding or removing plates to match different heat loads, and high thermal efficiency [3]. The geometrical characteristics of corrugated PHE extends the heat transfer surface area per unit volume and makes the flow in the channels highly turbulent even at low Reynolds number resulting in high heat transfer coefficients and reduced size. Also, swirls and vortex flow generation is expected in addition to distraction and re-attachment of the boundary layers [3].

In general, the PHE consist of a stack of thin plates arranged at 180° to each other resulting in the formation of inlet/outlet ports and two separate group of flow channels for a hot and cold fluid streams flows in an alternating manner across the channels from separate inlet/outlet ports (Figure 1.1). This stacking formation produces a large number of contact points and groovy passages [4]. Several PHE types are available commercially, but the main four are gasketed, brazed, welded and shell and plate [2].

1.2 Thesis Objective

Flow boiling is held as an effective heat transfer method where large amount of heat flux can be absorbed/released in the heat exchange process [5]. This is due to to having large temperature difference and boiling flow simultaneously in the case of subcooled flow or only boiling flow in the case of a saturated flow. It is of high importance for many applications such as in nuclear reactors, high performance electronic devices (e.g. electric cars battery cooling) etc.
The demand for higher efficiency and increased range and operational conditions particularly in water desalination makes the PHE a suitable candidate. PHE are able to operate at low mass flux, the heat can be supplied from low temperature heat sources such as waste heat or solar energy, as well as they are known to be have high thermal efficiency and mass transfer compared to other types of heat exchangers. In this research, the focus will be on developing and applying Computational Fluid Dynamics (CFD) simulations to better understand the hydrodynamics and heat transfer of PHE with water as the working fluid. So far the simulations of single phase flow in gasket and frame plate heat exchangers have been covered extensively in the literature for a wide range of topic. However, most of the simulations on full channel geometry are based on modeling the wall effects with wall function due to the complexity of forming a very fine mesh in narrow channels with contact regions. On the other hand, CFD simulations of flow boiling dynamics and phase change in PHE has not been explored properly yet. The selected PHE for the study is a gasketed frame and plate with chevron corrugation design and both the single phase and subcooled as well as the saturated boiling flow will be simulated and analyzed under different operating conditions.

Objectives of the simulations in part one:

- Generate a full geometrical model for chevron type corrugated channel and two different computational grids. A fine grid for low Reynolds turbulence modeling and a coarse grid for high Reynolds number modeling
• Develop the single phase simulation setup and validate the model based on experiment and literature correlations

• Predict and analyze the flow hydrodynamics and heat transfer characteristics

Objectives of the simulations in part two:

• Develop a boiling CFD model for predicting the two phase flow in PHE

• Compare the model results with the experiment and correlations of the literature

• Analyze the relation between the two phase heat transfer coefficient and pressure drop with the mass flux, inlet temperature conditions, wall temperatures and vapor quality.

• Analyze the evaporation, pressure, velocity and vapor fraction formation and distribution using contour and vector plots

1.3 Thesis Structure

The thesis consist of 6 chapters. Chapter 1 represented a brief introduction to PHE concept and operation. In chapter 2, the main literature on simulating the single phase flow, general flow boiling and pressure drop formulation, and the experimental work on two phase boiling flows and pressure drop in PHE are listed and discussed. Chapter 3 describes the geometrical and fluid domain modeling and characteristics. In Chapter 4, the experimental setup for the model validation is described in addition to how its operating. Chapter 5, contains the details of the single phase simulation including the governing equation, numerical setup in Ansys Fluent, boundary conditions applied, a data reduction section on how the Nusselt number and fanning friction factors are evaluated, generation of a fine and a coarse grid with their sensitivity studies, model validation against well cited literature correlations and experimental results. Finally, this is followed by the single phase simulation results where the low and high turbulence approaches are compared, as well as the velocity, temperature and pressure distribution are discussed. Chapter 6, contains the flow boiling simulation. It starts with introducing the concept of the Semi Mechanistic Wall boiling model, the mixture conservation equations and $K – \epsilon$ turbulence model solved in Ansys Fluent, the Lee mass transfer model, boundary conditions, polyhedral grid generation and data reduction section that includes how the heat transfer coefficient and pressure drops are evaluated. The last section in Chapter 6 express the results of the subcooled and saturated flow boiling in details along with contour plots for pressure, mass evaporation, volume fractions and velocity vectors to analyze the
vapor formation and behavior. The last chapter numbered 7 concludes the thesis and highlights the main findings in addition to a proposition of several recommendations and opportunities for future work.
Chapter 2

Literature Review

The objective of this literature survey is to give a general overview and fundamental knowledge about some important concepts related to this thesis work. The first section starts by summarizing the main studies published on the simulation of single phase flow inside PHE. The second section will introduce the main mechanisms of flow boiling and pressure drop in smooth tubes. Finally, the third section will cover only the experimental studies published on the flow boiling heat transfer and pressure drop in PHE because no major work is published in the open literature on the simulation of flow boiling in PHE.

2.1 Single Phase Flow Simulation Studies

An extensive body of literature is available for single phase flow simulations in PHE. Several topic were discussed and worked on including the hydrodynamics and heat transfer, parametric analysis for the geometrical parameters, geometrical and operational optimization, fluid flow pattern, and effect of fouling etc...

The majority of the simulations had a coarse grid and a high Reynolds number turbulence model with a wall function. Jain et al. [6] studied the heat transfer, pressure drop and flow distribution in chevron plate heat exchanger with one full cold channel and two half hot channels on each sides with periodic boundary conditions. However, they excluded the port and flow distribution channel parts and they used the $K - \varepsilon$ turbulence model with the non equilibrium wall function (NEWF). Another 3D CFD model for a brazed type corrugated heat exchanger, also without considering the inlet/outlet channel distribution zones, were developed by Sarraf et al. [7] and they studied the effect of helical and cross flow types formed by geometrical modifications. The work by Li et al. [8] investigated, numerically and experimentally, the particulate fouling on different corrugated plate geometries. Han et al. [9] applied CFD simulations to study and optimize the PHE thermal and hydraulic performances and concluded that the enlargement factor and the chevron angle are the most influential parameters.
Gherasim et al. [10, 11] were one of the first to simulate laminar and turbulent flows in corrugated channels. They compared the two turbulence models $k - \omega$ and $K - \varepsilon$ with different formulations and wall functions and deduced that the $K - \varepsilon$ with NEWF provided the closest prediction to the experimental results. A passive modification to the basic design of the corrugated channel consisting of adding a separator in the mid of the channel to improve flow distribution was proposed by Al Zahrani et al. [12]. They reported an enhancement in Nusselt number up to 75%. According to their study, the scalable wall function provided the most accurate results. Tsai et al. [13] studied the flow maldistribution and local characteristic around contact region for a full two channel model with inlet and outlet ports.

Other researchers used a fine grid with inflation layers to fully resolve the boundary layer however they did not consider the full geometry of the plate. Ettem and Sunden [14] investigated several low and high Reynolds number turbulence models for a unitary cell of cross-corrugated channel to study the thermal performance. Their results indicate that the heat transfer is enhanced due to the interaction between the upper and lower flow inside the unit which creates a shear force and rotational flow motion. The work by Lee et al. [15, 16] consisted of unsteady numerical analysis with large eddy simulations on a unitary cell of chevron type PHE. They concluded that the flow in PHE branches into the streamwise and the furrow direction and this resulted in more turbulence even at low Reynolds number. Additionally, they produced a correlation, based on the CFD data, for the friction factor and the Colburn factor with considering the effect of the geometrical parameters and fluid properties on the correlation predictions [16]. Freund and Kabelac [17] generated experimental visualizations for heat transfer coefficients and simulated a unitary cell of PHE channel using the Shear Stress Transport (SST) and Reynolds Stress models (RSM) with low average $Y^+ = 1.3$. The found that the CFD results underpredicts the experimental by 50% and 25% for the SST and RSM turbulence models.

The CFD simulation developed by Gullapalli and Sunden [4] for brazed type PHE approximated the braze junctions as variable radius fillets and included the full size of fluid channel. They generated a fine grid with inflation layers to study the performance of different turbulence models, boundary conditions, and local velocity profiles. The reported results showed that the CFD tool under predicted the heat transfer by 15 – 25% and the pressure drop by 15 – 30 % and whether using using a constant wall temperature, external wall heat transfer with ambient temperature or conjugate heat transfer boundary condition the prediction accuracy for heat transfer is similar.

Too many research using the CFD simulation have been reported in literature on a wide range of topic in PHE of different types however most of the models used a coarse grid with a high Reynolds turbulence model formulation or they did not consider the full channel geometry and analyzed a unitary cell or a section.
2.2 Introduction to Flow Boiling Heat Transfer and Pressure Drop

In contrary to the single phase heat transfer, the two phase boiling heat transfer is more complicated since it depends on many factors such as the vapor quality, saturation temperature, mass flux, heat flux, local flow regimes, surface material, and geometry [2, 18]. Boiling heat transfer model consist of two important mechanisms: macro-convective mechanism, also known as convective and it is dependent on the fluid flow, and a micro-convective mechanism, also known as nucleate and its associated with bubble formation and growth [19].

By considering evaporation in a tube under forced convection, we can describe several hydrodynamics conditions encountered in other channel types. A well described analysis is the subcooled flow entering a vertical pipe under relatively low uniform heat flux as can be seen in fig. 2.1. Initially, the liquid is subcooled and the temperature is below the saturation temperature and thus only single phase heat transfer to the liquid exists. As more heat is transferred to the flow, the liquid temperature next to the wall increases above saturation and vapor nucleation takes place only in the vicinity of the wall. A superheat thermal boundary layer forms at the wall and the vapor condenses as it drifts from the wall into the core of the pipe (Known as subcooled partial boiling). When the bulk fluid temperature reaches the saturation temperature the flow becomes saturated and it different regimes can be identified such as churn, bubbly, or annular based on the vapor mass quality.

At high qualities and mass flow rates the flow regime is usually considered annular flow and convective boiling is the dominate mechanism. On the other hand, at low mass flow rates and sufficient wall superheat, bubble nucleation at the wall occurs within the liquid film and the nucleate boiling is the dominate mechanism. As the flow velocity increases and convection augments, the wall may be cooled down below the minimum onset nucleate boiling temperature ($T_{ONB}$) and nucleate boiling will be suppressed resulting in mainly convective boiling only heat transfer [20].

In flow boiling, the nucleate boiling is similar to pool boiling except that there is an effect of the flow velocity on the bubble growth and departure. Convective boiling refers to the convective process between the liquid phase and the heated wall. The general form of prediction methods for flow boiling heat transfer coefficient in pipes follows the power law format:

$$h_{tp} = [h_{cb}F_{cb})^n + (h_{nb}S_{nb})^n]^\frac{1}{n}$$  \hspace{1cm} (2.1)

The two terms $h_{cb}$ and $h_{nb}$ represents the convective and nucleate boiling heat transfer coefficients respectively. In general, convective heat transfer coefficient is usually linked to the only liquid heat transfer coefficient $h_l$ (Assuming that the
Figure 2.1: Development of the two phase flow in smooth pipe under uniform heat flux (Left) and variation of heat transfer coefficient with vapor quality (Right)[1]

liquid phase occupies the whole pipe cross section) and it is determined using single phase flow correlations such as the Dittus-Boelter correlation. The nucleate heat transfer coefficient is determined using a pool boiling correlation such as Cooper or Forster and Zuber correlations. The factor $F_{nb}$ accounts for the enhancement by the increased velocity and turbulence due to bubble formation and factor $S_{nb}$ accounts for the suppression of nucleate boiling sites by the convection of the flow [18]. Most boiling prediction models can be classified based on the value specified for $n$. Setting $n = 1$ will result in simple additive model known as Chen type model [19]. Steiner and Tabrouk [21] proposed a comprehensive evaporation model known as the asymptotic model based on $n = 3$. Setting $n = \infty$ yields the larger of the two boiling terms such as in Shah model [22].

The total pressure drop for two phase flow is calculated as the summation of static, acceleration, inlet and outlet, and frictional pressure drops [18]:

$$
\Delta p_{total, tp} = \Delta p_{st} + \Delta p_{acc} + \Delta p_{ports} + \Delta p_{fr}
$$

(2.2)
The static and acceleration pressure drops are evaluated based on the homogeneous model while for the ports pressure drop a correlation is usually used assuming also homogeneous flow. The readers are referred to [23] for details on pressure drop calculations models summary. For the prediction of frictional pressure drop there are three main models that are used:

- **Lockhart and Martinelli model**

  \[
  \Delta p_{tp} = (\phi_i)^2 \cdot \Delta p_{fri,l}
  \]  
  \(\text{(2.3)}\)

  \[
  X_{tt} = \frac{\Delta p_{fri,l}}{\Delta p_{fri,v}}
  \]  
  \(\text{(2.4)}\)

  \[
  \phi_i^2 = 1 + \frac{C}{X_{tt}} \cdot \frac{1}{X_{tt}^2}
  \]  
  \(\text{(2.5)}\)

  Where \(X_{tt}\) is the Lockhart-Martinelli parameter, \(\phi_i^2\) is a two phase friction multiplier, and \(C\) is defined as Chisholm parameter and is determined from experimental data. \(\Delta p_{fri,l}\) and \(\Delta p_{fri,v}\) are the liquid and vapor frictional pressure drop respectively and they are calculated assuming that the liquid or vapor would be occupying the whole pipe volume.

- **Kinetic energy model**

  \[
  \Delta p_{fri,tp} = \zeta \cdot \frac{Ke}{volume} = \zeta \cdot \frac{G}{2 \rho_m}
  \]  
  \(\text{(2.6)}\)

  \[
  \frac{1}{\rho_m} = \frac{X_m}{\rho_v} + \frac{1 - X_m}{\rho_l}
  \]  
  \(\text{(2.7)}\)

  Where \(\rho_m\) and \(X_m\) represents the average mixture density and the average vapor quality between inlet and outlet.

- **Two phase fanning friction**

  \[
  \Delta p_{fri,tp} = 2 \cdot f_{tp} \cdot \frac{L_p \cdot G^2}{d_h \cdot \rho_m}
  \]  
  \(\text{(2.8)}\)

  where the Two phase fanning friction correlated in terms of equivalent Reynolds number.
2.3 Flow Boiling Experimental Studies

The flow boiling details in cross corrugated narrow plates can be quite different than for smooth pipes. Very often the concepts and prediction correlations for smooth tubes are employed for PHE after modifications of the constants and correction factors [18]. In PHE most published work shows that the boiling heat transfer is governed by the two mechanisms namely, convective and nucleate boiling. Nucleate boiling is dependent on the heat flux while the convective boiling is dependent on the mass flux and the vapor quality [24].

Ayub [2] suggested that Small hydraulic diameter passages of PHE makes the majority of the heat transfer to be convective boiling instead of the nucleate boiling as in other types of evaporators (e.g. flooded shell and tube evaporators where the bulk heat transfer is based on pool boiling). The flow in PHE is usually vertical and in opposite direction to gravity which induces less phase separation even at low mass fluxes compared to the shell and tube evaporators. Also, it is possible that the contact points, formed between the two plates with opposite orientation, leads to local nucleate boiling enhancement around them and the contact points can create void or cavity effects for the flow field [2]. Yan and Lin [25] showed that higher evaporation heat transfer coefficients are achieved in corrugated plates compared to circular tubes particularly at high vapor quality when the dominated regime is the convective boiling regime. They found that the two phase heat transfer coefficient and pressure drop increased with the increase in mass flux and vapor quality, whereas heat flux had insignificant effect on the overall heat transfer. Han et al. [26] performed an experimental parametric analysis on brazed type PHE to investigate the relation between the chevron angle and the corrugation pitches with the heat transfer coefficient and pressure drop under varying operating conditions (i.e. mass flux, evaporation temperature, vapor quality and heat flux). The heat transfer coefficient increased slightly with the increase in heat flux due to the strong turbulence in PHE which led to a highly convective flows even at low vapor quality. Moreover, the heat transfer coefficient and pressure drop increased with increasing mass flux and vapor quality but they decreased at higher chevron angles.

Hsieh and Lin [27] performed analysis on the evaporation in PHE where they applied heating from one side of the a corrugated channel. Their results indicate that the heat transfer coefficient and friction factor increased with the increase in mass flux and vapor quality. However, the heat transfer coefficient was slightly affected by the increase in mass flux at low vapor quality. Also the increase in friction factor with the vapor quality is more evident than the rise in heat transfer coefficient. They also concluded that at low vapor quality the nucleate boiling regime dominated and the Nusselt number was independent of the mass flux and vapor quality while at higher vapor qualities the convective boiling regime dominated. Lastly, they recommended the use of Ungor and Winterton correlation
[28] for predicting the heat transfer coefficient in PHE. In the work of Longo et al. [29] the heat transfer coefficient showed more sensitivity towards heat flux, outlet conditions and fluid properties and and they concluded that the dominate heat transfer mechanism is the nucleate boiling mechanism Similar to the conclusion made by Palm and Claesson [30].

A comparative study by Wang et al. [31] showed that the heat transfer coefficient for R410A was by 1 – 20 % higher than that for R22 and pressure drop of R410A was about 30 – 40 % lower than that of R22. They attributed the difference to having higher latent heat pf vaporization, thermal conductivity and specific heat and lower liquid viscosity for R410A in comparison to R22. Ebisu et al. [32] performed a similar study and got similar results and they attributed the difference to lower vapor density for R410A.

The work done by Lee et al. [33] is one of the few published research on PHE having water as the working fluid. They focused on low mass flux conditions of saturated water flow (i.e. mass flux in the range of 14.5 to 36 kg/m²s). Their results exhibit a different relation than the literature at which the flow boiling heat transfer coefficient decreased with increasing mean vapor quality. on the other hand, the heat transfer coefficient increased with the mass flux but it remained constant when different heat fluxes were applied. Moreover, the two phase frictional pressure drop increased with both the mass flux and the vapor quality yet it remained constant with respect to the heat flux. Based on their experimental results they developed a pre and post partial dry out condition correlation for the heat transfer coefficient and pressure drop that is based on evaluating the equivalent Reynolds number. Additionally, they concluded that the flow is in the convective boiling regime even at low mean vapor quality (i.e. vapor quality in the range of 0.09 to 0.6) as per the thonon et al. criteria [34]. The authors linked it to the fact that water achieve lower Boiling number due to the large latent heat compared to refrigerants.

The Thonon et al. criteria [34] determines the transition line between nucleate and convective boiling and can be defined using the product of the boiling number (Bo) and the Lockhart-Martinelli parameter (Xu). The criteria states that for Xu · Bo > 0.00015 the nucleate boiling is dominate, while for Xu · Bo < 0.00015 the convective boiling is dominant unless the mass flux and vapor quality are low then the dominate is the nucleate boiling [18]. It should be noted that this criteria does not include the effect of geometrical parameters yet its a useful as an indicative tool.

Another well cited correlation was developed by Donowski and Kandlikar [35] for R – 134 refrigerants. The correlation assumes the convective boiling contribution is dominate since in PHE the flow is turbulent even at very low Reynolds number. Ta’boas et al. [24] proposed a correlation with a criteria based on the superficial velocities of liquid and vapor phases in PHE to determine whether the flow is nucleate boiling or a competition between nucleate and convective boiling.
The correlation predicted their database of mixture ammonia/water flow by 98 % with an error of 20 %. Also they found that Donowski and Kandlikar [35] correlation had the best fit of the their experimental results compared to other correlations.

Amalfi et al. [36] formulated a generalized prediction methods for the thermal-hydraulic performance of PHE by considering the operating conditions, fluid types and plate geometries. Using their correlation they concluded that the convective boiling regime is dominate even at low vapor qualities and they were able to predict 92.3 % of the total 1903 data points in literature within ±50 %. However, when comparing their correlation with literature data for subcooled flow they could only predicate the data trend qualitatively since their correlation did not include any subcooling parameters. A newer version of this correlation is published by Li et al. [37] where they modified the correlation constants and proposed a new criterion for the transition from macro to micro scale boiling. The new improved correlation gave more accurate results especially in the micro-convection region.

In a comprehensive survey of the research published on the evaporation heat transfer and frictional pressure gradients in PHE, Amalfi et al. [23] concluded that a large number of parameters involved in the mechanism of heat transfer process such as plate geometry, flow distribution and transition between different flow regimes and working conditions. The survey shows that the two phase pressure drop increases when the vapor quality and mass flux increases and most correlation for frictional pressure drop has been developed based on either the homogeneous flow model (i.e. Fanning friction factor is correlated to an equivalent Reynolds number), separated flow model (i.e. Based on the Lockhart-Martinelli parameter) or a kinetic energy of the flow model. As for the heat transfer coefficient, its affected by the vapor quality, heat flux and mass flow rate, however, their relation to the heat transfer coefficient is contradicting across the published studies. These contradictions originates from not knowing where is the transition from nucleate boiling dominated flows to convective boiling dominated flows. Also, they show that many authors reported that at high mass flux the convective heat transfer is dominate while and that the heat transfer coefficient is strongly dependent on the vapor quality and less dependent on the applied heat flux. Lastly Chen type boiling model (i.e. setting n equal to 1 in eq. (2.1)), which is also the basis for the newly introduced semi mechanistic wall boiling model in fluent [38], is the most frequently used model for predicting thermal performance in PHE.

Most of the experimental research work done on plate evaporators are done on refrigerants as working fluids. Also, most experimental work is done on saturated flows and very few on subcooled boiling flows [23]. The mentioned literature shows that more investigation is required in predicting and understanding the variation of the heat transfer coefficient and pressure drop in particular for water.
evaporation applications. In addition there are no information published in the open literature on the use of CFD simulations in such applications for corrugated channels.
Chapter 3

Corrugated Plate Geometry Modeling

The type of corrugated plates modeled in this study is a gasketed plate and frame manufactured by Alfa Laval. Each plate has a sinusoidal shape with chevron style corrugations which represents the main heat transfer surface area in addition to inlet and outlet distribution zones with special corrugations to direct the flow inside the channel. A gasketed sealing is used to prevent leakage and separate the hot and cold fluid channels (as in fig. 1.1) When the plates are installed at angle of 180 ° to each other (fig. 3.1)

The main important geometrical parameters for the PHE are as follows [2, 3]:

- Chevron angle: represents the angle between the corrugations axis and the horizontal axis. The Chevron angle has huge effect on the thermal hydraulic performance.

- Mean flow channel gap: defined as the actual cross section gap and its calculated by subtracting the total corrugation gap from the thickness

\[ b_p = b_{co} - t_p \]

- Channel cross sectional area: calculated by multiplying the mean channel gap by the channel width

\[ A_{ch} = b_p \cdot L_w \]

- Channel hydraulic diameter: represents the channel equivalent diameter and since \( b_p \ll L_w \) it is calculated as:

\[ D_h = 2b_p/\phi \]

The geometrical characteristics definitions and values of the plate and the formed channel are described in fig. 3.2 and table 3.1 respectively.
Figure 3.1: Corrugated plate and zones description
Figure 3.2: Plate Geometry (left) Plane view (right) cross sectional view normal to the corrugation [2]

Table 3.1: Corrugated plate geometrical characteristics and dimensions

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chevron angle</td>
<td>$\beta$</td>
<td>60</td>
</tr>
<tr>
<td>Plate thickness</td>
<td>$t_p$</td>
<td>0.6 mm</td>
</tr>
<tr>
<td>Plate width</td>
<td>$L_w$</td>
<td>100 mm</td>
</tr>
<tr>
<td>Plate height</td>
<td>$L_p$</td>
<td>357 mm</td>
</tr>
<tr>
<td>Corrugation pitch</td>
<td>$P_{co}$</td>
<td>10 mm</td>
</tr>
<tr>
<td>Enhancement factor</td>
<td>$\phi$</td>
<td>1.156</td>
</tr>
<tr>
<td>Mean flow channel gap</td>
<td>$b_p$</td>
<td>2.4 mm</td>
</tr>
<tr>
<td>Hydraulic diameter</td>
<td>$D_h$</td>
<td>4.2 mm</td>
</tr>
<tr>
<td>Channel flow area</td>
<td>$A_{ch}$</td>
<td>$2.4 \times 10^{-4} m^2$</td>
</tr>
</tbody>
</table>
In the present work, the computational domain is represented by the channel fluid volume formed between two corrugated plates inversely positioned with each other. First, sinusoidal corrugated plates were created based on a widely used chevron corrugation geometrical equation eq. (3.1) [9], then the inlet and outlet port distribution zones were created from scratch following the dimensions of the physical plate sample. The formed shape of the plates were then edited to replicate the exact physical plate design. Lastly, the fluid volume was extracted from within the two full plates in contact. Figure 3.3 illustrates the final computational fluid domain along with the contact regions formed at plate intersections (i.e. white areas) as well as the inlet and outlet distribution fluid zones. In total, the top and the bottom surfaces represents a heating surface area of $A_s = 0.0659 \text{m}^2$.

$$F(x) = \frac{b_p}{2} \sin \left( \frac{2\pi}{P_{co}} \left( x - \frac{P_{co}}{4} \right) \right) + \frac{b_p}{2}$$ (3.1)

Figure 3.3: Plan view and cross sections of the computational fluid domain
Chapter 4

Experimental Setup

An experimental setup was built at the Cyprus Institute to validate the single phase and flow boiling CFD model results. The setup main components are: steam generator, plate heat exchanger (Test section), water reservoirs, separation vessels, magnetic flow meters, and K-type thermocouples. A schematic diagram for the experimental setup is shown in fig. 4.1. Finally, a LabView control system was installed to record the measurements of steam, water and vapor flow rates and temperatures.

Four corrugated plates identical to one in fig. 3.2 were combined to form a three flow channel PHE. Two channels are used for the pressurized steam (i.e. hot fluid) and one channel for the cold water (i.e. cold fluid). The generated steam was condensed in the hot channel at a specified saturation pressure to provide the required heat flux for single phase heating or evaporation of the cold water following in the cold channel in a counter flow arrangement as shown in fig. 4.2. The same setup is used to validate the results of part 1 of the thesis simulations.

Figure 4.1: Schematic diagram of the experimental test unit
which includes only heating water single phase simulations (chapter 5) and for the second part of the thesis simulations which consist of flow boiling two phase simulation (chapter 6). In the case of only single phase heating the outlet flow of the cold channel is completely water phase and was directed to a collection tank while for the case of boiling flow, the outlet flow of the cold channel consist of saturated water and water vapor thus a separation tank was used to separate the two phases. In the latter case, the outlet vapor mass flow rate of the cold channel was calculated based on the mass flow rate difference between the inlet and the remaining brine mass flow rate.

Figure 4.2: Channels flow diagram
Chapter 5

Single Phase Simulation

In this chapter, the development of a simulation model for the single phase flow in PHE is discussed in details and the hydrodynamics and heat are analyzed for a full corrugated channel following two approaches: a wall function for bridging the boundary layer and a fine grid for resolving the boundary layer.

5.1 Governing Equations

The set steady state mass, momentum and energy conservation equations can be written as follows [9]:

- Mass conservation

\[ \nabla \cdot (\rho \mathbf{V}) = 0 \]  \hspace{1cm} (5.1)

- Momentum conservation

\[ \nabla \cdot (\rho \mathbf{V} \mathbf{V}) = -\nabla P + \nabla \cdot \left[ \mu (\nabla \mathbf{V} + (\nabla \mathbf{V})^T) - \frac{2}{3} \nabla \mathbf{V} \right] + \rho \mathbf{g} \]  \hspace{1cm} (5.2)

where \( P \) is the static pressure and \( \rho \mathbf{g} \) represents the gravitational body force.

- Conservation of energy in fluid domain

\[ \nabla \left[ \rho \mathbf{V} (H + \frac{V^2}{2}) \right] = \nabla \cdot (K_{eff} \nabla T) \]  \hspace{1cm} (5.3)

where \( H \) is the enthalpy, \( K_{eff} \) is the effective thermal conductivity and its equal to the thermal conductivity in addition to the turbulent thermal conductivity \((K + K_{turb})\)
• Conservation of energy in solid domain

\[ \nabla^2 T = 0 \]  
\[ (5.4) \]

## 5.2 Turbulence Modeling

The two equation Reynolds Average Navier Stokes (RANS) Realizable $K - \varepsilon$ turbulence model is considered for the single phase simulations in PHE. As was discussed in the literature section the $K - \varepsilon$ is recommended for simulating flows in PHE. It is based on a transport equation for the turbulence kinetic energy ($K$) and the turbulence dissipation rate ($\varepsilon$). The wall effects can be modeled through low Re models or wall functions. The wall functions consist of empirical formulas linking the dependent variable at the near wall cells to the parameters at the wall. Also, The wall functions are based on mean velocity and temperatures as well as near wall turbulent quantities. It should be noted that as the Reynolds number increases their accuracy also increases however if in the case where low Reynolds near wall effects are important a low Re Turbulence models should be used to have a more accurate results. On the other hand, the low Re turbulence models require more CPU time [39]. In this work, the Realizable $K - \varepsilon$ model with enhanced wall treatment (EWT) and the Realizable $K - \varepsilon$ model with Non equilibrium wall function (NEWF) are used. The enhanced wall treatment option allows for resolving the near the wall boundary and thus its considered a low Reynolds model while the NEWF bridges the boundary with a wall function and its a high Reynolds model. According to [40] the NEWF is recommended for the use in complex flows involving separation, reattachment, and impingement. In such flows improvements can be obtained especially in predicting the skin friction and heat transfer (Nusselt and Stanton numbers). More information about the formulations can be found in [40].

\[ \nabla \cdot (\rho \nabla K) = \nabla \cdot \left( \frac{\mu}{\sigma_k} \nabla k \right) + G_k - C_b - \rho \varepsilon - Y_M \]  
\[ (5.5) \]

\[ \nabla \cdot (\rho \nabla \varepsilon) = \nabla \cdot \left( \frac{\mu}{\sigma_\varepsilon} \nabla \varepsilon \right) + C_1 \frac{\varepsilon}{k} (G_k + C_2 G_b) - C_2 \rho \frac{\varepsilon^2}{k} \varepsilon \]  
\[ (5.6) \]

$G_K$ and $G_b$ represent the kinetic energy generation due to mean velocity gradients and due to buoyancy respectively. $C_1$, $C_2$ and $C_3$ constant values are left as default as per [40]; The $\sigma_k$ and $\sigma_\varepsilon$ are the Prandtl number for $K$ and $\varepsilon$. The turbulent viscosity is defined as:

\[ \mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \]  
\[ (5.7) \]
5.3 PHE CFD Single Phase Model

5.3.1 Numerical Setup

The simulations for the single phase flow in PHE were performed using the software ANSYS Fluent CFD code [40]. The coupled algorithm was used for the velocity-pressure coupling and it showed significant reduction in computational time compared to the SIMPLE algorithm, the body force weighted average scheme was used for pressure equation, the 2nd order scheme for the momentum, energy, and the $K - \varepsilon$ turbulence equations. Lastly, the gradients were calculated using the cell based least square method. The pseudo transient formulation was also used since the simulation residuals showed more stability when used and the time scale factors were set to 0.5 and the solid scale factor 1. The explicit under relaxation factors for the pressure, momentum, turbulence, energy were set to 0.5, 0.5, 0.75, 0.75 respectively.

To ensure convergence of the solution three main factors were analyzed. The normalized residuals in all cases were in order of $10^{-4}$ for all equations except for the energy in the order of $10^{-7}$. The mass imbalance representing the difference between the total inlet and outlet flow were kept in the order of $10^{-6}$ while the energy imbalance, which is the difference between the energy passing through the plate walls and the energy the fluid receives, was kept less than 1% of the total system energy. Additionally, the pressure, temperature, and heat flux were monitored and the solution is assumed to be converged when they became constant with the increasing number of iterations. Astonishingly each simulation took only around 100 to 120 iterations to converge.

5.3.2 Boundary conditions

As shown in fig. 5.1 at the top and bottom walls of the channel a fixed temperature boundary condition is applied over a virtual solid domain of 0.6 mm. The virtual solid domain allows for the simulation of heat transfer in the solid plate thickness without the need to mesh it [40]. The side of the channel is surrounded by a gasket and thus a wall with zero flux boundary condition is applied. At the inlet of the channel the velocity, and the static temperature. It was assumed that the velocity is uniformly distributed and that the direction of the inlet velocity is normal to the two ports inlet curves. While at the outlet of the channel the flow is open to the atmosphere and thus a 0 pa is specified. For turbulence modeling, the turbulence length scale $l_t$ and the turbulence intensity $T_u_{in}$ are calculated using the equations eq. (5.9) and eq. (5.8). The length scale of turbulence is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows and in fully developed pipe flows the turbulence length scale is restricted by the size of the pipe since the eddies can not be bigger than that size [40]. For wall bounded flows in which the inlets involve a turbulent boundary
layer this combination of boundary conditions is the most suitable option.

\[ T u_{in} = 0.16 \cdot Re^{-1/8} \]  \hspace{1cm} (5.8)

\[ l_t = \frac{0.07 \cdot D_h}{C_{\mu}^{0.75}} \]  \hspace{1cm} (5.9)

where \( C_{\mu} \) is a constant equal to 0.09 for \( K - \varepsilon \) and \( K - \omega \) models. Under the study \( Re \) number the \( T u_{in} \) is in the range of 5 to 6 % and the \( l_t \) is set to 2.64 \( \times 10^{-3} \).

Figure 5.1: Applied boundary conditions for the single phase simulations

5.3.3 Materials Properties

All water properties were calculated as function of temperature through the relations published by Nayar et al. and Sharqawy et al. [41, 42, 43], except for the specific heat it was specified constant \( C_{pv} = 4180 \ j/kg/k \) since its change with the temperature range in this study is negligible. The below relations for water density, dynamics viscosity, thermal conductivity were complied in the fluent solver as user defined functions. The temperature is set in \(^\circ\)C.

\[ \rho_t = a_0 + a_1 T + a_2 T^2 + a_3 T^3 + a_4 T^4 \]
where, \( a_0 = 9.99922 \times 10^2 \); \( a_1 = 2.0341 \times 10^{-2} \); \( a_2 = -6.1624 \times 10^{-3} \); \( a_3 = 2.2614 \times 10^{-5} \); \( a_4 = -4.657 \times 10^{-8} \)

\[
\mu_i = 4.2844 \times 10^{-5} + \left( 0.157 + (T + 64.993)^2 - 91.296 \right)^{-1}
\]

\[
\log_{10}(K_i) = \log_{10}(240) + 0.434 \left( 2.3 - \frac{343.5}{T} \right) \left( 1 - \frac{T}{647} \right)^{0.33}
\]

The plate thickness material (i.e. virtual shell thickness) is defined as titanium with constant properties as follows: \( \rho_s = 4850 \) kg/m\(^3\); \( C_{ps} = 544.25 \) J/kg.k; \( K_s = 20 \) W/m \cdot k.

### 5.3.4 Data Reduction

To find the Nusselt number \( Nu \) and the fanning friction factor \( f \) of the cold channel the following equations in this section are evaluated [9]. All water properties were evaluated at the average temperature of the channel. The Nusselt number is a function of the outlet temperature and the total heat transfer are required. In the case of the simulation the total heat transfer is calculated as the area weighted average over all the surface while for experimental work the total heat transfer is calculated based on the following:

\[
Q_{exp} = \dot{m} \cdot C_p \cdot (T_{C_{out}} - T_{C_{in}})
\]  \( (5.10) \)

- Calculation of Nusselt number First, the heat transfer coefficient is calculated using the method:

\[
h = \frac{Q}{A_s \cdot \Delta LMTD}
\]  \( (5.11) \)

Where, \( A_s \) is the total surface area and the \( \Delta LMTD \) is Logarithmic Mean Temperature Difference and calculated as follows:

\[
\Delta T_{LMTD} = \frac{\Delta T_1 - \Delta T_2}{\ln \frac{\Delta T_1}{\Delta T_2}}
\]  \( (5.12) \)

\[
\Delta T_1 = T_w - T_{c_{in}}
\]  \( (5.13) \)

\[
\Delta T_2 = T_w - T_{c_{out}}
\]  \( (5.14) \)
• Calculation of Fanning friction factor $f$ [4] The fanning friction factor is based on the friction factor that can be calculated using Darcy’s equation.

$$f = \frac{\zeta}{4}$$  \hspace{1cm} (5.15)

$$\zeta = \frac{0.5 \cdot \Delta P \cdot D_h}{\rho \cdot L_p \cdot U_{ch}^2}$$  \hspace{1cm} (5.16)

The pressure drop is calculated as the area weighted average pressure difference between the inlet and outlet sections. $L_p$ represents the total length of the plate (Refer to chapter 3). The channel velocity is calculated as:

$$U_{ch} = \frac{\dot{m}}{A_{ch} \cdot \rho}$$  \hspace{1cm} (5.17)

### 5.3.5 Grid Independence Study

To correctly utilize the use of the two turbulence models considered. Two different grids were introduced. For the simulation $K - \varepsilon$ with EWT a sufficiently fine grid ($y^+ < 5$) was generated. For having such a grid in narrow channels with contact regions while preserving the element shapes, the area around the contact regions were meshed first with high very fine tetrahedron elements with no inflation layers while outside that area 10 layers of inflation were applied. This preserved the quality of the elements at the small space where the two plates meet. Also, different levels of refinement were applied at the locations of high gradients and flow separation separation. This resulted in a large size grid however the total time of the simulation and convergence were improved. For the simulation with the $K - \varepsilon$ with NEWF a coarser grid with a single body element size is generated without inflation or refinement. The latter type of grid is the most reported in use in the literature. For simplicity In what follows the coarse and fine grid will be referred to as Grid A and Grid B respectively. To determine the optimal grid size a grid sensitivity analysis was performed and the results are indicated in table 5.1 and table 5.2 for Grid A and Grid B respectively. The flow inlet velocity and temperature were set to 0.4 $m/s$ and $17.5 \ ^\circ C$ respectively and the wall temperature was set to $T_w = 101 \ ^\circ C$. As seen With the increasing number of iterations the prediction of the outlet temperature and the pressure drop decreases. Based on the mentioned study Grid 3 is selected as an optimum size for Grid A and Grid B since the change in prediction of the temperature and pressure drop became almost not changing and they will be used in the following comparison between the two turbulence models formulations. A 3D and sectional views of Grid A and B are shown in fig. 5.2 and fig. 5.3 respectively.
### Table 5.1: Grid A independence study

<table>
<thead>
<tr>
<th>Case</th>
<th>Number of elements</th>
<th>Outlet Temperature (k)</th>
<th>Pressure drop (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid 1</td>
<td>2,155,773</td>
<td>93.21</td>
<td>859</td>
</tr>
<tr>
<td>Grid 2</td>
<td>2,662,170</td>
<td>93.85</td>
<td>866.3</td>
</tr>
<tr>
<td>Grid 3</td>
<td>3,335,904</td>
<td>94.55</td>
<td>880.6</td>
</tr>
<tr>
<td>Grid 4</td>
<td>5,542,499</td>
<td>94.85</td>
<td>881.9</td>
</tr>
</tbody>
</table>

### Table 5.2: Grid B independence study

<table>
<thead>
<tr>
<th>Case</th>
<th>Number of elements</th>
<th>Outlet Temperature (k)</th>
<th>Pressure drop (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid 1</td>
<td>2,893,685</td>
<td>94.85</td>
<td>922</td>
</tr>
<tr>
<td>Grid 2</td>
<td>7,165,439</td>
<td>94</td>
<td>925.7</td>
</tr>
<tr>
<td>Grid 3</td>
<td>14,611,698</td>
<td>94.09</td>
<td>931.2</td>
</tr>
<tr>
<td>Grid 4</td>
<td>16,992,502</td>
<td>94.1</td>
<td>933.08</td>
</tr>
</tbody>
</table>
Figure 5.2: 3D and sectional view of the Coarse mesh for high Re modeling
Figure 5.3: 3D and sectional view of the fine mesh for low Re modeling
5.3.6 Model Validation and Assessment

In this section, simulations using the EWT and NEWF turbulence models are compared against the experimental results and the predictions of four main correlations reported in the literature. The Nusselt number and fanning friction factor correlations reported by Thonon [44], Wanniarachchi et al. [45], Martin [46], and Lee et al. [15] were used to further assist the validation process. These correlations were selected since they match to a good extent the geometrical characteristic in the current study.

Figure 5.4 shows a comparison among the CFD predictions, correlations predictions and experimental results as function of Re ranged from $Re = 361$ to 2019. The inlet temperature $T_m = 17.5 \,^\circ C$ and the wall temperature is $T_w = 101.5 \,^\circ C$. The results show better predictions using the EWT model than the NEWF. The highest difference in Nu between the experimental and the EWT model is 3.518 where as the highest difference between the experimental and the NEWF model is 8.49 at $Re = 518$. Moreover, the EWT model predictions follows the exact increase in the correlations of Nu with respect to Re. It can be highlighted that the correlations by thonon [44] and Wanniarachchi et al. [45] are the closest to the CFD EWT and experimental results.

In conclusion, there is a very good agreement among the CFD results, experimental data and the correlations where the EWT showed an improved predictions in comparison to the NEWF wall function model.

Figure 5.5 shows a comparison of NEWF and EWT models predictions along with the correlations for the fanning friction factor $f$. Both CFD models had close prediction for $f$ across the full range of Re and they had the same decreasing trend in the prediction of $f$ with the increasing Re. The predictions by EWT model are higher than the predictions by NEWF at high Re. Furthermore, both models predictions were in between the correlation by Martin [46] and wanniarachchi et al. [45]. In conclusion the prediction of $f$ by the NEWF and EWT is close and both lies within the ranges of the correlation selected. For the rest of the analysis the EWT model is used only and the final grid and its specification is shown in fig. 5.3.
Figure 5.4: The variation of Nusselt number with the channel Reynolds number based on the predictions of CFD and Correlation and experimental results.

Figure 5.5: The variation of Fanning friction factor with the channel Reynolds number based on the predictions of CFD and Correlation.
5.4 Results and Discussion

For the results analysis the $K - \varepsilon$ with EWT model is chosen since it exhibited more stability and faster convergence in addition to better Nu predictions as were shown in the previous section. In the all the simulations provided the inlet temperature fixed at $T_{in} = 17.5 \, ^\circ C$ and wall temperature $T_w = 101.5 \, ^\circ C$. Starting with Figure 5.6 the variation of local temperature prediction at different Re is plotted along the axial distance of the channel and at $y = 0$ and $Z = 0$ (Refer to fig. 3.3 for details on the geometry center). The plot shows an increase in the temperature prediction as the flow progresses in the channel for all the three Re numbers considered additionally the temperature predictions increases with the decrease of Re which is expected since the flow velocity is reduced.

For the same location of monitoring points a plot for the local pressure variation with respect to Re and axial distance is provided in fig. 5.7. The plot show a decrease in pressure with the axial distance for all three Re numbers however at low Re $Re = 530$ the change in pressure was much smaller. Also at higher Re a significant increase in pressure distribution is predicted.

![Figure 5.6: Variation of temperature with respect to the axial distance at different Reynolds number](image)

Contour maps and velocity streamlines for the corrugated channel are shown in this section. For all cases the flow enters from top and leaves from below. The pressure contours on the surface wall of the PHE is plotted for $Re = 1500$ in fig. 5.8. The figure shows a large stagnation pressure at the inlet section due to the
Figure 5.7: Variation of temperature with respect to the axial distance at different Re

geometrical blockage in front of the inlet port and a more uniform distribution across the corrugation area. Further investigation can help in optimizing the shape design of the inlet distribution zone. At the outlet section a pressure negative zones is formed due to separation of the boundary layer at curved outlet section walls. A comparison of the velocity stream lines between $Re = 530$ and $Re = 1500$ is performed to study the flow distribution (fig. 5.9). The plot shows a higher velocity magnitude for the higher Re and similar flow distribution in both cases. The velocity magnitude at the inlet and outlet distribution is zone is the highest and then it gets slower while passing through the chevron corrugation zones. A curve flow is formed due to the contact regions and the flow flows in the both the streamwise and furrow directions which forms a shear layer that induces more turbulence and improves the heat transfer [16]. Moreover it can be seen that at the higher Re $Re = 1500$ the velocity in the left section of the plate is higher since the inlet port on the left side. This indicates the presences of flow maldistribution between the left and right sections. The last contour plot fig 5.10;C inner wall temperature $Re=530$ and $Re=1500$ consist of the prediction temperature on the plate walls for $Re = 530$ and $Re = 1500$. A higher wall temperature is predicted for the the $Re = 530$ and the temperature is almost uniform across the plate.
Figure 5.8: Pressure contours on the plate wall
Figure 5.9: Velocity streams for $Re = 530$ (Left) and $Re = 1500$ (Right)
Figure 5.10: variation of inner wall temperature predictions according to $Re = 530$ (Left) and $Re = 1500$ (Right)
5.5 Conclusion

In conclusion a detailed CFD simulation model is presented for simulating single phase flow inside corrugated plate channel. Two different modeling approaches for the wall effects were compared using the turbulence $K - \varepsilon$. Enhanced wall treatment is employed for a fine grid with inflation and refinement and Non equilibrium wall function (i.e. The most used in the literature) for a coarser grid. Better predictions of the Nusselt number were achieved using the EWT. Furthermore, the flow hydrodynamic and heat transfer characteristics were analyzed and reported.
Chapter 6

Flow Boiling Simulation

In this chapter, CFD simulations are performed to gain insight into the complex two phase flow and heat transfer underneath flow boiling in PHE. CFD is capable of capturing developed and undeveloped flows, separation, swirls as well as wakes in the flow domain. The modeling approach followed includes simulation of the two phase flow, model the mass transfer between liquid and vapor phases and calculate the heat transfer at the wall while accounting for effects of enhancement or suppression due to vapor formation.

6.1 Semi Mechanistic Wall Boiling Model

The common CFD modeling approach for boiling flows is the Rensselaer Polytechnic Institute (RPI) wall boiling model [47]. It is a mechanistic model based on wall heat flux partitioning into convective, evaporation and quenching terms and built in eulerian framework with detailed consideration to sub-mechanism of boiling such as near wall bubble dynamics, and frequency of bubble departure [48]. However, test runs with the RPI model for boiling flows on PHE geometries under the boundary conditions reported in this thesis showed difficulties in convergence and numerical instability. It might be attributed to the fact that RPI empirical equations were developed for high pressure applications found in boiling conditions in nuclear reactors [49].

In the present chapter, the newly added Semi-Mechanistic Wall Boiling Model (SMB) to ansys fluent [38] is used to simulate boiling phase change process in PHE. The SMB model is implemented in the mixture multi-phase framework and it combines different models to predicate the saturated boiling flows at low pressure. It is made suitable for the use high Reynolds turbulence models with wall functions (i.e. The boundary layer is bridged with mathematical wall functions instead of resolving it with fine mesh elements) therefore its cheap in terms of computational time and power. Most importantly, its suitable for complex shape passages and suitable for low pressure applications. In SMB, heat transfer aug-
mentation at the wall due to boiling is captured by a modified Chen correlation [19]. The effective heat flux at the wall is partitioned into two components: single phase and nucleate boiling heat flux. Furthermore the evaporation/condensation mechanism in the bulk of the fluid as well as near the wall is captured using the Lee phase change model.

The effective heat flux:

$$q_w = F \times q_{sp} + S \times q_{nb}$$  \hspace{1cm} (6.1)

where $q_{sp}$ and $q_{nb}$ are the single phase and nucleate boiling heat fluxes respectively. The term $F$ represents the forced convective augmentation factors, while $S$ represents the nucleate boiling suppression factor, and they are added under the assumption that vapor formation leads to increase liquid velocity, the convective heat transfer is augmented compared to the single phase liquid flow and at the same time flow convection suppresses the nucleation boiling sites and reduces the effect of nucleate boiling.

The single phase heat flux is given as

$$q_{sp} = h_{sp}(T_w - T_n)$$  \hspace{1cm} (6.2)

The nucleate phase heat flux is given as

$$q_{nb} = h_{nb}(T_w - T_{sat})$$  \hspace{1cm} (6.3)

where, $T_w$, $T_n$ and $T_{sat}$ are the wall, cell next to the wall and saturation temperature respectively. The $h_{nb}$ is the nucleate heat transfer coefficient and is calculated using foster and Zuber pool boiling correlation eq. (6.4). While the $h_{sp}$ is a combination of $h_l$ and $h_v$. According to Lee et al. [33] who performed experimental work on water evaporation in PHE, the partial dry out is predicted to occur beyond 0.3 vapor quality and in another study [50] they showed that partial dry out of the liquid film with ammonia as working fluid occur when the vapor quality was in the range of 0.6 to 0.98. Since in this current study the range of vapor mass quality considered is low (i.e. $X_{out} < 15\%$), the $h_{sp}$ will be set to liquid heat transfer coefficient under the assumption that no dry spots will form in the current operating range.

$$h_{nb} = 0.00122 \frac{k_l^{0.79} \cdot c_{pl}^{0.45} \cdot \rho_l^{0.49}}{\sigma^{0.5} \cdot \mu_l^{0.29} \cdot h_{fg}^{0.24} \cdot \rho_v^{0.24}} (P_{sat,T_w} - P_{sat,T_{sat}})^{0.75} (T_w - T_{sat})^{0.24}$$  \hspace{1cm} (6.4)

where,

- $k_l$ is liquid thermal conductivity
- $c_{pl}$ is the liquid specific heat
- $\rho_l$ is the liquid density

38
\( \rho_v \) is the vapor density

\( \sigma \) is the surface tension coefficient of water liquid and vapor

\( \mu_l \) is the liquid dynamic viscosity

\( h_{fg} \) is the latent heat of vaporization

\( P_{sat,T_w} \) and \( P_{sat,T_{sat}} \) are the saturation pressure at the \( T_w \) and \( T_{sat} \) respectively

The augmentation factor \( F \) is based Martinelli parameter as follows:

\[
F = 2.35 \left( \frac{1}{X_{tt}} + 0.213 \right)^{0.736}
\]  

(6.5)

\[
X_{tt} = \left( \frac{1 - x}{x} \right)^{0.9} \left( \frac{\rho_v}{\rho_l} \right)^{0.5} \left( \frac{\mu_l}{\mu_g} \right)
\]  

(6.6)

The suppression factor \( S \) is composed of two components: suppression due to subcooling effects and suppression due to the forced convection.

\[
S = S_{conv} \cdot S_{sub}
\]  

(6.7)

The subcooled suppression is defined as the following where \( T_{ref} \) was set to \( T_{bulk} \) (It's only included if the flow is subcooled)

\[
S_{sub} = \frac{T_w - T_{sat}}{T_w - T_{ref}}
\]  

(6.8)

The forced convection suppression factor is based on the a modified two phase Reynolds number as follows:

\[
S_{conv} = \begin{cases} 
\frac{1}{(1+0.12Re_{tp}^{1.14})} & R_{ctp} < 32.5 \\
\frac{1}{(1+0.42Re_{tp}^{0.78})} & 32.5 < R_{ctp} < 70 \\
0.1 & R_{ctp} > 70
\end{cases}
\]  

(6.9)

The two phase Reynolds number is defined:

\[
R_{ctp} = 10^{-4} \cdot Re_{l,ref} \cdot F^{1.25}
\]  

(6.10)

The reference Reynolds number is based on liquid properties at the bulk temperature, the length is set to the PHE hydraulic diameter, and the reference velocity is calculated from the wall function internally:

\[
R_{e,ref} = \frac{\rho_{l,ref} \cdot U_{ref} \cdot D_h}{\mu_{l,ref}}
\]  

(6.11)
6.2 Mixture Governing Equations

The SMB model is implemented within the simplified mixture multi-phase model. Where the model solves the continuity equation, the momentum equation, the energy equation for the mixture, and the volume fraction equation for the secondary phases (i.e. water vapor), as well as algebraic expressions for the relative velocities of the phases. The conservation equations are described below as per [51]; in what follows the subscripts \(m\), \(l\) and \(v\) denotes mixture, liquid and vapor respectively:

- Conservation of mass

\[
\nabla \cdot (\rho_m \mathbf{V}_m) = 0 \tag{6.12}
\]

where \(\rho_m\) is the mixture density and \(\mathbf{V}_m\) is the mass average mixture velocity field and defined as [52]:

\[
\rho_m = \alpha_l \rho_l + \alpha_v \rho_v \tag{6.13}
\]

\[
\mathbf{V}_m = \frac{\alpha_l \rho_l \mathbf{V}_l + \alpha_v \rho_v \mathbf{V}_v}{\rho_m} \tag{6.14}
\]

- Conservation of volume fraction

\[
\nabla \cdot (\alpha_v \rho_v \mathbf{V}_m) = \dot{m}_{l\rightarrow v} - \dot{m}_{v\rightarrow l} \tag{6.15}
\]

The volume fraction equation is obtained from the continuity equation for the vapor phase. \(\alpha\) refers to the volume fraction. The right hand side terms represents the rate of mass transfer due to evaporation and condensation and they are calculated as mentioned in section 6.4 chapter 6.

- Conservation of momentum

\[
\nabla \cdot (\rho_m \mathbf{V}_m \mathbf{V}_m) = -\nabla p + \nabla \cdot [\mu_m (\nabla \mathbf{V}_m + (\nabla \mathbf{V}_m)^T)] + \rho_m \mathbf{g} \tag{6.16}
\]

where \(\mu_m\) represents the viscosity of the mixture and defined as:

\[
\mu_m = \sum_{n=l,v} (\alpha_n \mu_n) + \mu_{t,m} \tag{6.17}
\]
• Conservation of energy

\[ \nabla \cdot \sum_{n=l,v} (\alpha_n \mathbf{V}_m (\rho_n H_n + P)) = \nabla \cdot (K_{eff} \nabla T) + S_E \quad (6.18) \]

where \( H_n \) is the phase sensible enthalpy, \( S_E \) is the corresponding heat source/sink term due to the phase change, and lastly \( K_{eff} \) is the the effective thermal conductivity which is function of the phases thermal conductivity and the mixture turbulent conductivity calculated from the turbulence model.

\[ K_{eff} = \sum_{n=l,v} \alpha_n K_n + K_{t,m} \quad (6.19) \]

### 6.3 Turbulence Modeling

In mixture model one set of equations is solved to model the turbulence. The Realizable \( K - \varepsilon \) model, which is a two equation Reynolds average Navier Stokes model (RANS), with non equilibrium wall function is used for all of the two phase simulations. The non equilibrium wall function is recommended for swirly and separation flows [51]. The \( k - \varepsilon \) model equations are formulated as a transport equations of the turbulence kinetic energy \( K \) and dissipation rate \( \varepsilon \).

\[ \nabla \cdot (\rho_m \mathbf{V}_m k) = \nabla \cdot (\mu_m \nabla k) + G_{k,m} - G_{b,m} - \rho_m \varepsilon \quad (6.20) \]

\[ \nabla \cdot (\rho_m \mathbf{V}_m \varepsilon) = \nabla \cdot (\mu_m \nabla \varepsilon) + C_1 \frac{\varepsilon}{K} (G_{k,m} + G_{b,m}) - C_2 \rho_m \frac{\varepsilon^2}{K} \quad (6.21) \]

\( G_{k,m} \) and \( G_{b,m} \) represent the kinetic energy generation due to mean velocity gradients and due to buoyancy respectively. \( C_1, C_2 \) and \( C_3 \) constant values are left as default as per [40]; for information on the turbulence model refer to [51]. The turbulent viscosity and thermal conductivity defined before in eq. (6.19) and eq. (6.19) are:

\[ \mu_{t,m} = \rho_m C_\mu \frac{k^2}{\varepsilon} \quad (6.22) \]

\[ K_{t,m} = \frac{C_p \mu_{t,m}}{Pr_t} \quad (6.23) \]
6.4 Mass Transfer Model

The phase change near the wall and in the flow domain is modeled using the evaporation/condensation Lee model [51] which is a mechanistic model with physical basis. Based on the cell temperature the mass transfer is calculated as follows:

If $T_m > T_{sat}$

$$\dot{m}_{l\rightarrow v} = f_{evap} \alpha_l \rho_l \frac{T_m - T_{sat}}{T_{sat}}$$  \hspace{1cm} (6.24)

If $T_m < T_{sat}$

$$\dot{m}_{v\rightarrow l} = f_{cond} \alpha_v \rho_v \frac{T_{sat} - T_m}{T_{sat}}$$  \hspace{1cm} (6.25)

In eq. (6.24) and eq. (6.25) the terms $f_{evap}$ and $f_{cond}$ represents the evaporation and condensation frequencies respectively and their values is function of several parameters: accommodation factors, phase volume fraction, fluid properties, and vapor bubble diameter [48] [51]. Both frequencies has to be tuned based on the experimental data. The condensation frequency represents the fraction of vapor molecules that are impinging on the vapor-liquid interface that actually condenses, same concept for evaporation frequency. In other words, if the condensation (evaporation) coefficient is set to unity, then this means that every vapor (liquid) molecule reaching the separation interface will condense (evaporate) [53]. In the current PHE simulations the $f_{evap}$ is set to 30 and $f_{cond}$ is set to 200. Note that since the density ratio between the liquid and vapor phase at atmospheric pressure is 20,000, the condensation frequency should be higher than evaporation frequency [51].

6.5 PHE CFD Two Phase Flow Model

6.5.1 Numerical Setup

The numerical simulation on the two phase flow boiling were performed under steady state conditions using the commercial CFD software Ansys Fluent 2020 [40]. The coupled algorithm was used for the velocity-pressure coupling and it showed significant reduction in computational time compared to the SIMPLE algorithm, the body force weighted average scheme was used for pressure equation, the 2nd order scheme for the momentum, energy, and the $K - \epsilon$ turbulence equations while for the volume fraction equation the Quick scheme was used. Lastly, for gradient calculations, the node based Green Gauss method was used.

The interfacial area concentration, defined as the interfacial area between the water liquid and vapor phases per unit volume, was calculated using the algebraic
symmetric model \cite{40} assuming that the interface is spherical and its based on phases volume fractions and the vapor/droplet diameter (6.26). This interfacial concentration is used in predicting the mass, momentum and energy through the interface.

\[ A_i = \frac{6\alpha_v\alpha_l}{d_{v,i}} \] (6.26)

where, \(\alpha_v\) and \(\alpha_l\) are the vapor and liquid volume fractions respectively and \(d_{v,i}\) is the characteristic diameter for the dispersed phase whether its the liquid or the vapor.

For momentum interactions between the two phases in the mixture formulation, a Drag function should be specified and its usually a function of Drag coefficient \(C_D\). In this study, the builtin Schiller-Neumann model is used and its formulation is as follows:

\[ f = \frac{C_D Re}{24} \] (6.27)

\[ C_D = \begin{cases} 
\frac{24(1+0.15Re^{0.087})}{Re} & \text{if } Re < 1000 \\
0.44 & \text{if } Re > 1000
\end{cases} \] (6.28)

where \(Re\) is the relative Reynolds number defined as:

\[ Re = \frac{\rho|V_i - V_v|d_v}{\mu_l} \] (6.29)

To ensure convergence of the solution three main factors were analyzed. The normalized residuals in all cases were in order of \(10^{-4}\) for all equations except for the energy and volume fraction equations in the order of \(10^{-6}\). The mass imbalance representing the difference between the total inlet and outlet flow were kept in the order of \(10^{-5}\) and the energy imbalance, which is the difference between the energy passing through the plate walls and the energy the fluid receives, were kept less than 1\% the total energy. Additionally, the pressure, temperature, vapor quality, and mass of evaporation predictions at several points were monitored and the solution is assumed to be converged when they became constant with the increasing number of iterations. On average each simulation took around 2500 to 3000 iterations to converge. It should be noted that for the best solution convergence all the explicit relaxation factors had to be set to low values and a pseudo transient formulation was used.
6.5.2 Boundary Conditions

The full volume of fluid of the corrugated channel described in chapter 3 is considered for the boiling flow simulation. The heat is being supplied through the plate surface to the cold channel from steam condensing at constant temperature in the hot channel and thus the top and bottom surfaces were set to wall boundary condition with a uniform specified temperature which is in all simulation cases is higher than the saturation temperature of the flow. Also, since the side surface of the fluid volume is sealed by a gasket, a wall boundary condition is specified there with zero heat flux assuming heat transfer is resisted by the gasket. For the channel inlet, a velocity inlet condition with constant temperature and volume fraction are specified while for the channel outlet a pressure outlet condition is specified. In all simulations, only water phase is assumed to be entering thus $\alpha_l$ and $\alpha_v$ are set to 1 and 0 at inlet respectively, on the other hand, the outlet is assumed to be open to the atmosphere and thus atmospheric pressure is specified at outlet. fig. 6.1 illustrates the type of the boundary conditioned applied.

![Diagram of applied boundary conditions for the flow boiling simulations](image)

Figure 6.1: Applied boundary conditions for the flow boiling simulations

6.5.3 Material Properties

For the flow boiling simulation the material properties are specified for each of the water liquid and vapor phases in addition to the latent heat of vaporization
and surface tension. The density, dynamics viscosity, specific heat, and thermal conductivity of liquid water were set as a function of cell temperature using the same relations defined in material properties section of chapter 5. While the vapor properties were specified as constants at the saturation temperature \( T_{\text{sat}} = 100 \, ^\circ C \) and exported from library in [40]. The vapor density, dynamic viscosity, thermal conductivity, and specific heat were set respectively as follows:

\[
\rho_v = 0.5542 \, Kg/m^3, \mu_v = 1.34 \times 10^{-5} \, pa \cdot s, \, k_v = 0.0261 \, W/m^2 \cdot K, \, c_{pv} = 2014 \, j/kg \cdot K.
\]

A reference temperature has to be set for both liquid and vapor phases in material properties for evaporation simulations and it is equal to the saturation temperature of the flow \( T_{\text{sat}} = 100 \, ^\circ C \). The latent heat of vaporization is included in Fluent as the standard state of enthalpy in \( j/kg \cdot mol \) which is calculated by the multiplication of the latent heat of vaporization \( h_{fg} = 2257 \times 10^3 \, j/kg \) by the molecular weight of water \( M = 18.01 \, kg/kg \cdot mol \).

Lastly, the average surface tension is specified as constant \( \sigma = 0.06 N/m \) since its variation is very small at current simulation temperature ranges.

### 6.5.4 Grid Generation and Independence Study

Four different grids, made of tetrahedrons elements, were generated for the fluid domain in fig. 3.3 using the ANSYS meshing tool. In each grid, different levels of refinement were applied at the location of high variations (i.e. Edges of contact regions, corrugation face, etc..) and without inflation layers since for the SMB model the \( y+ \) range recommended to be above 12. A two million element cell count difference was kept between each grid where grid 4 represented the most refined grid (table 6.1) under consideration. Then each grid was transformed into a polyhedral element grid to reduce the computational time in the fluent solver.

To quantify the effect of the grid size on the solution accuracy a thorough grid sensitivity analysis was carried out (table 6.1) with a selection criteria of having less than \( 5e^{-3} \) difference in the predictions of the heat flux and the pressure drop parameters between two consecutive meshes. The applied boundary conditions for the independence study are \( G = 40 \, Kg/m^2s \), inlet liquid temperature \( T_{\text{in}} = 100 \, ^\circ C \) and wall temperature \( T_w = 110 \, ^\circ C \).

Based on the study and the criteria mentioned, grid 2 was selected as the final grid. Both the aspect ratio and the orthogonal quality were 19 and 0.16 respectively and in the range recommended by [40]. 3D and cross sectional views of the final selected grid are shown in fig. 6.2.
Table 6.1: Polyhedral grid independence study

<table>
<thead>
<tr>
<th>Case</th>
<th>Cell Count</th>
<th>Node Count</th>
<th>$q$ (W/m$^2$)</th>
<th>$\frac{q^i - q^{i+1}}{q^i}$</th>
<th>$\Delta p$ (Pa)</th>
<th>$\frac{\Delta p^i - \Delta p^{i+1}}{\Delta p^i}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid 1</td>
<td>556,139</td>
<td>2,771,458</td>
<td>28,058</td>
<td>5.55e$^{-3}$</td>
<td>5,881</td>
<td>0.0194</td>
</tr>
<tr>
<td>Grid 2</td>
<td>986,516</td>
<td>5,247,319</td>
<td>27,903</td>
<td>3.95e$^{-3}$</td>
<td>5,769</td>
<td>1e$^{-3}$</td>
</tr>
<tr>
<td>Grid 3</td>
<td>1,406,569</td>
<td>7,652,084</td>
<td>27,793</td>
<td>1.15e$^{-3}$</td>
<td>5,763</td>
<td>0.009</td>
</tr>
<tr>
<td>Grid 4</td>
<td>1,780,899</td>
<td>9,799,350</td>
<td>27,761</td>
<td>–</td>
<td>5,704</td>
<td>–</td>
</tr>
</tbody>
</table>

6.5.5 Data Reduction

In this study, the channel mass flux $G$, total heating surface area $A_s$, inlet velocity and inlets area are required. The inlet area $A_{in}$ and the total heating surface area $A_s$ are calculated from the geometrical model as $74 \times 10^{-6}$ m$^2$ and 0.0659 m$^2$ respectively. The inlet velocity $v_{in}$ and mass flux $G$ are calculated based on the required channel mass flow rate.

$$ v_{in} = \frac{\dot{m}_{in}}{A_{in}\rho_{t}} \quad (6.30) $$

$$ G = \frac{\dot{m}_{in}}{A_{ch}} \quad (6.31) $$

Through out the experimental analysis, the heat flux is calculated as a summation of the single phase and the evaporation heat flux as follows:

$$ q = \dot{m}_{in} \cdot C_{pl} \cdot (T_{sat} - T_{in}) + x_{out} \cdot \dot{m}_{in} \cdot h_{fg} \quad (6.32) $$

Through out the simulations analysis, the frictional pressure drop is calculated as the area weighted average pressure difference (static) between the inlet and outlet sections.

The two phase heat transfer coefficient, in the case of a saturated flow boiling only (i.e. $T_{in} = T_{sat}$), is calculated as the total surface heat flux divide by the super heat temperature difference ($\Delta T_{sat} = T_{w} - T_{sat}$):

$$ h_{tp} = \frac{q}{T_{w} - T_{sat}} \quad (6.33) $$

While in the case of subcooled flow boiling the two phase heat transfer coefficient is calculated as the area and temperature weighted average of the single phase and saturated boiling heat transfer coefficients as listed below:
Figure 6.2: Polyhedral grid for the flow boiling simulations: 3D side and cross sectional views (for section A-A’ and B-B’ locations refer to 3.3)
\[ h_{tp} = \frac{q_{sp} \cdot A_{sp} \cdot \Delta T_{LMTD} + q_{sat} \cdot A_{sat} \cdot \Delta T_{sat}}{A_{sp} \cdot \Delta T_{LMTD} + A_{sat} \cdot \Delta T_{sat}} \]  \hspace{1cm} (6.34)

where \( A_{sp} \) and \( A_{sat} \) are the surface areas required for the single phase and evaporation heat fluxes respectively. The \( A_{sp} \) is assumed to start at inlet of the channel until the section where bulk fluid temperature reaches the saturation and its determined manually for each case in the post processing. \( \Delta T_{LMTD} \) is defined as the logarithmic mean temperature difference and \( \Delta T_{sat} \) defined as super heat temperature difference.

\[ \Delta T_{LMTD} = \frac{\Delta T_1 - \Delta T_2}{\ln \frac{\Delta T_1}{\Delta T_2}} \]  \hspace{1cm} (6.35)

\[ \Delta T_1 = T_{in} - T_w \]  \hspace{1cm} (6.36)

\[ \Delta T_2 = T_{sat} - T_w \]  \hspace{1cm} (6.37)

### 6.5.6 Model Validation

The developed CFD model was validated against experimental tests done using the experimental setup described in chapter 4. In fig. 6.3 the heat flux, at various mass fluxes, predicted by the CFD is compared with the heat flux calculated from the experimental results. The inlet and wall temperatures were fixed at \( T_{in} = 30 \degree C \) and \( T_w = 110 \degree C \) respectively. At higher mass flux, the CFD predictions agrees more with the experimental measurements than at low mass flux where the maximum percentage difference reached 12.8 % at \( G = 20 \ Kg/m^2s \).

Another comparison between the CFD predictions and experimental measurements for the outcome vapor quality at the channel exit is undertaken with varying mass inlet mass flux and heating wall temperatures as shown in fig. 6.4. For the two different wall temperatures \( (T_w = 110 \degree C \text{ and } T_w = 105 \text{ si} \)\degree C\) the predictions of CFD are closer to experimental at high mass flux than that at low mass flux. Yet, a wider difference between the predictions and experimental measurements at the higher wall temperature \( (T_w = 110 \degree C) \) compared to that at lower wall temperature \( (T_w = 105 \degree C) \). The CFD results underpredicts most of the experimental results in terms of vapor quality percentage. The largest difference reached is at \( G = 20 \ Kg/m^2 \cdot s \) and \( T_w = 110 \degree C \) where CFD results under predicted the experimental by 4.6 % while the smallest difference happens at \( G = 38 \ Kg/m^2 \cdot s \) and \( T_w = 105 \degree C \) where CFD results over predicted the experimental by 0.45 %. Based on the above analysis, the results of the validation shows a good agreement between the CFD predictions and experimental results.
Figure 6.3: Comparison between the CFD model predictions and experimental results for Surface heat flux at varying inlet mass flux. Inlet and wall temperatures are set to $T_{in} = 30 \degree C$ and $T_{w} = 110 \degree C$ respectively.

6.6 Results and Discussion

In this chapter, the trends and variations of the heat transfer coefficient and pressure drop as well as the boiling flow dynamics are discussed based on the simulations results. In particular, the effect of varying the mass flux, wall temperature, and inlet temperature are considered for flow boiling in PHE. The fluid enters the channel as either subcooled liquid or saturated liquid (i.e. at inlet $T_{in} = T_{sat} = 100 \degree C$). In what follows the evaluation method for heat transfer coefficient and pressure drop are as listed in the Data reduction section of 6.

6.6.1 Assessment of heat transfer coefficient and pressure drop literature correlations

Many correlations based on experimental studies are reported in the literature for predicting two phase heat transfer coefficients as well as pressure drop. In general, large discrepancies in the predictions are found and no generalized correlation is still agreed upon. This stems from the fact that the heat transfer in flow boiling is a function of many factors such as the mass flux, vapor mass quality, heat flux, saturation pressure, in addition to corrugation geometrical characteristics [37]. In this section, an assessment of some of the most widely used correlations of the heat transfer and pressure drop is performed by comparing with the CFD results.
under the conditions of low mass flux and vapor quality for saturated liquid inlet flow. The mass flux variation ranged from $G = 15 \, Kg/m^2s$ to $G = 90 \, Kg/m^2s$ and the vapor mass quality ranged from $X_{out} = 3.76 \%$ to $X_{out} = 11.25 \%$.

Figure 6.5.1 depicts the two phase heat transfer coefficient ($h_{tp}$) predictions for CFD model and the correlations reported by Amalfi et al. [36], Ta´boas et al. [24], Donowski and Kandlikar [35], Li et al. [37], and Ouazia et al. [54] with respect to varying mass flux. The $h_{tp}$ increases with the increase in the mass flux and its captured by both the CFD model and the correlations. Li et al. [37] and Amalfi et al. [36] correlations have the most similar trend with the CFD model predictions. This can be attributed to the large experimental data bank that they used to formulate their correlations. The CFD results shows very good agreement with newly reported correlation by Li et al. [37] with a maximum deviation equal to 8.4 % at the highest mass flux $G = 90 \, Kg/m^2s$. Ouazia et al. correlation agrees with the CFD predictions at low mass flux while it increasingly overpredicts as the mass flux increase. On the other hand, the correlations by Ta´boas et al. [24] and Donowski and Kandlikar [35] have an identical trend of $h_{tp}$ increase with the mass flux but they under predicts the CFD predictions particularly at low mass flux.
Figure 6.5.2 express the two phase pressure drop ($\Delta P$) predictions for CFD versus the correlations reported by Amalfi et al. [36], Ta’boas et al. [24], Khan et al. [55], and Ouazia et al. [54] with respect to varying mass flux. The results indicate almost linear increase of $\Delta P$ with increasing flux. The predictions by Ta’boas et al. [24] and Ouazia et al. [54] correlations follow the CFD predictions well however Ta’boas correlation predictions are the closest to the CFD predictions with higher agreement as the mass flux is decreased. The highest difference between the CFD predictions and Ta’boas correlation is 15.5 % at $G = 90 Kg/m^2s$.

In conclusion, there is a good agreement between CFD predictions and literature correlations for heat transfer coefficient and pressure drop. The heat transfer coefficient correlation by Li et al. [37] and the pressure drop correlation by Ta’boas et al. [24] achieves the closest fit to the CFD results with the highest agreement at low mass flux region. Hence generally speaking they are recommended for flow hydrodynamics and heat transfer predictions for PHE working under similar operating conditions as reported in this thesis.

### 6.6.2 Evaluation of mass flux effect and flow inlet temperature

In order to evaluate numerically the flow boiling characteristic in PHE a set of simulations with varying channel mass flux for subcooled and saturated liquid flows at inlet were performed. The $h_{tp}$, $\Delta P$ and $X_{out}$ for subcooled liquid flow (i.e. $T_{in} = 30$) and saturated flow(i.e. $T_{in} = 100 ^{\circ}C$) were plotted as shown in fig. 6.6. In all cases, the wall temperature is fixed at $T_w = 110 ^{\circ}C$.

Based on the results, for the subcooled and saturated liquid at inlet, both the $h_{tp}$ and $\Delta P$ increased with the increase in the mass flux (fig. 6.6.1 and 6.6.2). On the other hand, the generated vapor quality increased exponentially as the mass flux is decreased (fig. 6.6.3). The prediction results proves to be inline with almost all experimental observations mentioned in the literature review section.

The $h_{tp}$ in the case of subcooled inlet is much higher than in the case of saturated flow. The $h_{tp}$ for the subcooled inlet, at low mass flux $G = 20 Kg/m^2.s$, is 39 % higher than for saturated inlet whereas at high mass flux $G = 90 Kg/m^2.s$ it reached 47 % since for subcooled flow the heat flux is a summation of single phase heating and evaporation heat fluxes whereas for saturated flow only evaporation heat flux exists and the difference in the evaporation heat flux between the two cases is relatively small which can be seen in the relatively close outlet vapor quality for both cases (fig. 6.6.3). By comparing the heat flux for subcooled and saturated, at $G = 20 Kg/m^2s$ and $G = 90 Kg/m^2s$, the subcooled heat flux is 52 % and 76 % higher than the saturated heat flux respectively. Consequently, since the percentage difference in heat flux between the subcooled and saturated is lower at low mass flux, the two $h_{tp}$ curves approaches to each
Figure 6.5: Comparison between the CFD and correlations predictions for the heat transfer coefficient and pressure drop at different mass flux. The wall and inlet temperatures are set to $T_w = 110 \, ^\circ C$ and $T_{in} = 30 \, ^\circ C$ respectively.
other as the mass flux is reduced. The comparison shows that for subcooled flow boiling, the influence of mass flux on the heat transfer coefficient is more significant and as the degree of subcooling (i.e. $T_{sat} - T_{in}$) is increased $h_{tp}$ will increase. Similar trends were reported in [5] on the comparison of subcooled and saturated flows.

Higher pressure drops were predicted in the case of only saturated flow compared to the subcooled flow at the high mass flux while as the mass flux decreased, the pressure drop became similar in both cases. In the view of the fact that pressure drop is dependent on mass flux as well as on vapor quality ([27, 26]), then at a specific mass flux the difference in $\Delta P$ between saturated and subcooled flow is in accordance with the difference in the vapor quality between saturated and subcooled flow. As can be seen in fig. 6.6.3 the difference in vapor quality between saturated and subcooled flow is reduced as the mass flux is decreased hence the $\Delta P$ difference between saturated and subcooled flow is also reduced.

In the case of saturated flow relatively higher evaporation is achieved compared to subcooled flow. At high mass flux the vapor output quality for the saturated inlet is 1.384% higher than the subcooled flow while at low mass flux it is 0.54% higher owing to the fact that for subcooled flow as the mass flux is reduced the heating area required for single phase heating is smaller and thus more heating surface area available for boiling.

### 6.6.3 Evaluation of the heating wall temperature

To assess the effect of wall temperature on the hydrodynamics and heat transfer in PHE, simulations with different wall temperature were carried out. Figure 6.7 illustrates the variation in area weighted average heat flux at the heating walls with respect to wall temperatures and mass flux. The inlet liquid temperature and saturation temperatures are $T_{in} = 100 \, ^\circ C$ and $T_{sat} = 100 \, ^\circ C$ respectively. The plot shows that as the wall temperature increases, larger amount of heat flux are achieved manifesting the advantage of having flow boiling process. At $G = 20 \, K\!g/m^2s$ an increase in the wall temperature from $T_w = 104 \, ^\circ C$ to $T_w = 115 \, ^\circ C$ resulted considerably in 2.81 times more heat flux. Furthermore, the higher the mass flux, the higher the boiling heat flux though the rate of increase diminishes as the mass flux is increased due to the suppression of boiling sites by the higher velocity convective flow. For instance at $T_w = 110 \, degree C$ the heat flux increased by 18.3% by increasing the mass flux from $G = 20 \, K\!g/m^2s$ to $G = 40 \, K\!g/m^2s$ however the heat flux increased by 7.4%, for an increase in mass flux from $G = 40 \, K\!g/m^2s$ to $G = 60 \, K\!g/m^2s$.

The change in $h_{tp}$, $\Delta P$ and $X_{out}$ with wall temperature for subcooled and saturated inlet flow was captured in fig. 6.8 and fig. 6.9. Three different wall temperatures $T_w = 104 \, ^\circ C$, $T_w = 107 \, ^\circ C$, and $T_w = 110 \, ^\circ C$ at two different inlet
Figure 6.6: Variation of heat transfer coefficient, pressure drop, and outlet vapor mass quality with mass flux for subcooled and saturated liquid inlet flows. Wall temperature is set to $T_w = 110 \, ^\circ C$
Figure 6.7: Comparison of heat flux at different wall temperature and mass flux. 
$T_{in}$ and $T_{sat} = 100^\circ C$

liquid temperatures $T_{in} = 30^\circ C$ and $T_{in} = 100^\circ C$ were considered.

The two plots point out that as the temperature is increased the $h_{tp}$ is decreased at all mass fluxes. On the contrary, the $\Delta P$ and $X_{out}$ increases with the increase in wall temperature.

The reason that the $h_{tp}$ decreases with temperature is because that under the current operating condition as the temperature difference is increased the $h_{tp}$ will decrease if the augmentation of heat flux was not very large enough (Refer to heat flux calculations in data reduction section chapter 6.5.5).

Interestingly, for subcooled inlet at the higher mass flux $G = 90 \, Kg/m^2s$ the effect of temperature on the $h_{tp}$ is insignificant while as the mass flux is reduced the effect is more pronounced (fig. 6.8.1). Moreover, as the temperature decreases, the difference in $\Delta P$ between low and high mass flux becomes less. For example, the difference in $\Delta P$ between $G = 20 \, Kg/m^2s$ and $G = 90 \, Kg/m^2s$ at $T_w = 104^\circ C$, is 3329 Pa while at $T_w = 110^\circ C$ it reached 8109 Pa (fig. 6.8.2). This can be linked to the fact that at lower wall temperatures and higher velocities there are
less vapor content in the channel generated and thus the reduced pressure drop.

For saturated flow inlet as the wall temperature decreases and the mass flux increases, the increase in \( h_{op} \) becomes less (fig. 6.9.1). As for \( \Delta P \), the increase is linear with the mass flux at all three temperatures with closer range of \( \Delta P \) as the mass flux is reduced (fig. 6.9.2)

Furthermore, the analysis of the plots on the outlet vapor quality and pressure drop reveals a relation between the vapor quality and pressure drop. An increase in the outlet vapor mass quality, due to increase in \( T_w \), results in increased pressure drop. The same conclusion can be drawn for subcooled and saturated flows(fig. 6.8.2 and 6.8.3, fig. 6.9.2 and 6.9.3)

6.6.4 Two phase flow analysis

In this section, the contour maps and velocity vectors of the flow boiling simulation are analyzed to highlight the important features about the hydrodynamics and heat transfer characteristics in PHE. Note that for all contour maps the flow enters from the top of the geometry and leaves from below at the outlet section. In fig. 6.10 the distribution of the vapor volume fraction for \( G = 30 \, \text{Kg/m}^2 \cdot \text{s}, \, T_w = 104 \, ^\circ\text{C}, \, \text{and} \, T_{in} = 30 \, ^\circ\text{C} \) is plotted at the wall heating surface. Based on this distribution, two distinct zones can be identified for subcooled flow boiling in PHE. The starting zone A represents the portion of the of the PHE where single phase heating and partial subcooled flow boiling are pertained up to the location of section 2 – 2’ at which the fluid bulk temperature reaches the saturation temperature (fig. 6.11). From that section to the exit of the heat exchanger the flow is saturated and the vapor occupies most of the volume inside the channel.

Figure 6.12 provides top view of the contours plot for the mass transfer rate and vapor fraction distribution as well as velocity vectors at the area between section 1-1’ and 2-2’ of fig. 6.10. The mass transfer rate represents the mass rate of the phase change from liquid to vapor (i.e. evaporation) or from vapor to liquid (i.e. condensation) at each grid volume element, the vapor fraction distribution represents the ratio of vapor volume to the cell volume and lastly the velocity vectors is a representation for the mixture velocity magnitude and direction. The figure indicates that the regions surrounding the contact regions act as the first nucleation sites since the temperature is higher than the bulk fluid due to low proximity between the top and bottom heating surfaces and fluid velocity is low as well due to more friction, stagnation or recirculating swirls. Moreover, more mass transfer on the left side section is a sign on flow and velocity maldistribution (fig. 6.12.1). This partial boiling leads to vapor formation around the contact regions and thus the vapor volume fraction increases and occupies part of the space between the channels (fig. 6.12.3). In addition, the vapor formation around the contact region leads to flow acceleration since the velocity vectors show an increase downwards of the contact regions in particular on the channel left side.
Figure 6.8: Variation of the two phase heat transfer coefficient (A), Pressure drop (B) and outlet vapor mass quality (C) as function of wall temperature at different mass fluxes. In all cases the inlet is subcooled $T_{in} = 30 \, ^\circ C$. 

57
Figure 6.9: Variation of the two phase heat transfer coefficient (A), Pressure drop (B) and outlet vapor mass quality (C) as function of wall temperature at different mass fluxes. In all cases the inlet is sat $T_{in} = 100 \, ^{\circ}C$
Figure 6.10: Vapor volume fraction distribution at the heating wall. $G = 30$ $Kg/m^2s$, $T_w = 104 \, ^\circ C$, and $T_{in} = 30 \, ^\circ C$
Figure 6.11: Average bulk fluid temperature variation with the downward axial distance for the case reported in fig. 6.10. Note that the point center zero is the center of the geometry - Refer to chapter 3 for center location.

This velocity increase facilitates the convection of formed vapor across the channel passages. The temperature and vapor volume fractions distribution at section 1-1', 2-2' and 3-3' are illustrated in Figure 6.13. The figure gives insight into the formation and distribution of vapor. At regions with a higher fluid temperatures liquid vapor patches are formed and then they gets convicted and might condenses through the channel. After section section 2-2' the flow reaches saturation temperature $T_{sat} = 100 \, ^\circ C$; the flow becomes saturated and temperature distribution and vapor volume fraction becomes more uniform along the cross section area.

Figure 6.14 display details of the vapor distribution at the top wall for different mass flux $G = 20, 40, 60 \, Kg/m^2 \cdot s$ from left to right. The effect on vapor distribution is significant where with increase in mass flux less vapor content is formed in the channel due to the suppression of boiling sites by the forced convection. Yet in all three cases partial boiling around contact region continued to exist. Also, the evaporation is higher on the left section of the channel due to the flow maldistribution. Since the inlet port is on the right section the velocity is higher and thus more suppression exists at the right section and at which, the maldistribution is increasing with increase in mass flux.
Figure 6.12: Plots of mass transfer rate, vapor fraction, and velocity vectors for area between section 1-1’ and 2-2’
Figure 6.13: Mixture temperature (A) and vapor volume fraction (B) contours plot at Section 1-1', 2-2' and 3-3' respectively from top to bottom. Refer to fig. 6.10 for section location.
Figure 6.14: Effect of mass flux on vapor distribution. From left to right $G = 20, 40, 60 \text{ Kg.m}^2\text{s}$
Figures 6.15 and 6.16 together demonstrates the relation between the mass evaporation rate and the velocity distribution. In fig. 6.15 the mass evaporation rate for the subcooled and saturated cases is shown. In the case of subcooled inlet the mass evaporation rate at inlet section is zero since the fluid temperature is below the saturation then it starts to increase across the channel and it reaches the highest after the bulk fluid temperature is above the saturation saturation temperature. The increased evaporation accelerates the flow and velocity is increased significantly as seen in (fig. 6.16.1). Consequently, with more velocity increase the evaporation is suppressed and thus the mass transfer rate is reduced. On the other hand, for a saturated inlet, the mass transfer rate is the highest at the inlet section and this in turn augments the velocity (fig. 6.16.2). As the flow progresses through the channel the velocity increases and affect the evaporation by more suppression.

The two figures shed light on the complicated relation between heat transfer and flow hydrodynamics. With water evaporation the quality is increased and it occupies most of the volume inside the corrugations and thus the fluid velocity increases. More velocity leads to more shear between the two phases and induces more turbulence which increases the two phase heat transfer coefficient as well as the friction pressure drop.

Figure 6.17 demonstrates the pressure contours for saturated flow at $G = 30$ $Kg/m^2 \cdot s$ for two different wall temperature $T_w = 104\degree C$ and $T_w = 110\degree C$. At higher wall temperatures more heat flux is transferred to the fluid and thus more evaporation happening inside the channel, this in turn leads to higher pressure distribution across the channel and hence the pressure drop increases with the increase in vapor quality. A pressure negative zones forms near the outlet section due to boundary layer separation at some of the curved surfaces.
Figure 6.15: Mass transfer rate for subcooled (left) and saturated (right) inlet. The mass flux is set to $G = 30 \text{ kg/m}^2\text{s}$ and the wall temperature $T_w = 104 \, ^\circ\text{C}$.
Figure 6.16: Velocity vector plot for subcooled inlet (left) and saturated (right). The mass flux is set to $G = 30 \text{ kg/m}^2\text{s}$ and the wall temperature $T_w = 104 \degree C$. 
Figure 6.17: Pressure contours for $G = 30 \text{ kg/m}^2\text{s}$ at $T_w = 104 \text{ °C}$ and $T_w = 110 \text{ °C}$ for saturated inlet $T_{in} = 100 \text{ °C}$.
6.6.5 Dominant boiling regime

The flow boiling heat transfer is considered to be a result of nucleate and forced convective boiling. As described in the literature review section, it is conceived that the $h_{tp}$ is mainly dependent on the heat flux in the case of nucleate boiling whereas it is mainly dependent on vapor quality and mass flux [56]. Thus with the increase in vapor quality the regime changes. In most experimental studies, the nucleate boiling regime is dominant at the low mass flux while convective boiling is associated with high mass flux [37]. Other researchers considered Thonon et al. [34] were one of the first to identify a criteria for PHE to determine the dominate boiling regime. The criteria is based on the boiling number ($Bo$) and the Martinelli parameter ($X_{tt}$) and it is defined as:

$$
\begin{pmatrix}
X_{tt} Bo > 0.00015 & \text{NucleateBoiling} \\
X_{tt} Bo < 0.00015 & \text{ConvectiveBoiling}
\end{pmatrix}
$$

(6.38)

Where the $Bo$ and $X_{tt}$ are calculated using the below equations:

$$
Bo = \frac{q}{Gh_{fg}}
$$

(6.39)

$$
X_{tt} = \left( \frac{1 - x_{mean}}{x_{mean}} \right)^{0.9} \left( \frac{\rho_{l}}{\rho_{g}} \right)^{0.5} \left( \frac{\mu_{l}}{\mu_{g}} \right)
$$

(6.40)

In the SMB model that is used in the current work, the wall heat flux is calculated based on an two phase heat transfer coefficient that takes the contribution of both the convective and nucleate heat transfer coefficients. In this section an investigation for determining the dominate boiling regime for saturated flow was performed in accordance to the literature criterion’s. The Thonon criteria is evaluated based on the CFD model predictions for $Bo$ and $X_{tt}$ parameters for a mass flux range of $G = 15 - 90 \, Kg/m^2.s$ and at two different wall temperatures $T_w = 110 \, ^oC$ and $T_w = 104 \, ^oC$.

According to plot of Thonon et al. criteria [34] in fig. 6.18 the dominate boiling regime is the nucleate boiling under the current study operating range.

As the mass flux is increased and the wall temperature is decreased the points shifts towards the transition line identified by the criteria expressing an increasing dominance of convective flow characteristic.

Another new criterion on the transition from micro to macro scale is introduced by Li et al. [37]. The criteria is based on the bond number and the liquid Reynolds number as described below. The bond number represents the ratio between the buoyancy and the surface tension forces and its used in many studies for the determination of transition between the macro to microscale two phase
boiling flow [36]. The CFD results indicate a $Bd.Re_L^{0.5}$ in the range of 39 to 97 at the operating range of mass flux and wall temperatures mentioned above concluding that the flow boiling in PHE is in the microscale region (another term for nucleate boiling)

$$\begin{align*}
Bd.Re_L^{0.5} &< 200 \quad \text{microscale} \\
Bd.Re_L^{0.5} &> 200 \quad \text{macroscale}
\end{align*}$$  \hspace{1cm} (6.41)

$$Bd = \frac{(\rho_l - \rho_v)gD_h^2}{\sigma}$$  \hspace{1cm} (6.42)

$$Re_l = \frac{GD_h(1 - X)}{\mu_l}$$  \hspace{1cm} (6.43)

![Graph](image)

Figure 6.18: Thonon criteria for determining the dominant boiling. A logarithmic scale is applied on the x-axis and y-axis
6.7 Conclusion

A two phase flow numerical model based on the Semi Mechanistic wall boiling model was developed to simulate the subcooled and saturated boiling flow inside Chevron corrugated plate heat exchangers under the following operating range: low mass flux, low vapor quality, and excess temperature range (i.e. \( G = 15 - 90 \) \( Kg/m^2.s \), \( X_{out} = 1.84 - 11.25 \% \), \( \Delta T_{excess} = 2 - 15 ^\circ C \)). The predictions agrees well with the correlation and experimental data. The two phase heat transfer coefficient increased with increasing the mass flux and decreasing inlet temperature while it decreased with the wall temperature. On the other hand, the two phase pressure drop increased with increasing the mass flux, vapor quality and wall temperature. The main dominant boiling mechanism seems to be the nucleate boiling mechanism. Phase change and vapor fraction were analyzed in details as well as flow distribution.
Chapter 7

Conclusion

7.1 Main Findings of the Present Study

The main findings of the present study are directed towards understanding and analyzing the complicated flow in plate heat exchangers. The research focused on water as the working fluid and the single phase heating as well as the subcooled and saturated boiling flows were considered in details. All the simulations were done using Ansys Fluent CFD package and the numerical setup developed was discussed in details. The major findings can be summarized as follows:

For single phase flow:

- The flow inside the corrugated channel is shown to be a wavy flow and induced more turbulence and mixing. In addition flow and temperature maldistribution exists for current geometrical design

- Resolving the boundary layer with a refined grid and inflation layer showed better Nusselt number predictions

For boiling flow:

- Contact regions surroundings and flow recirculation zones act as the first boiling cavities

- The two phase heat transfer coefficient increased with increasing the mass flux and decreasing inlet temperature while it decreased with the wall temperature.

- The two phase pressure drop increased with increasing the mass flux, vapor quality and wall temperature.

- The main dominant boiling mechanism seems to be the nucleate boiling mechanism based on thonon et al. criteria and Li et al criteria.
• The evaporation process showed to be linked with the velocity magnitude. Higher evaporation increased the velocity of the flow and in turn it suppressed the evaporation downward in the channel as the flow progressed.

7.2 Recommendations for Future Work

For single phase flow many aspects been covered in the literature using CFD simulations however the model with the fine grid can still be extend to investigate other turbulence models and boundary conditions. On the other hand, the main focus of the thesis was on introducing a predictive CFD model for boiling flows since its not covered yet and an extensive research is still required. On that note the CFD simulation on boiling flows can be extended by:

• Improving the model accuracy by investigating different empirical correlations for nucleate boiling heat transfer coefficient

• Simulating the three channels model with conjugate heat transfer between the hot and cold channels

• Validating and simulating evaporation at a wider range of mass flux and vapor quality

• Include and predict the flow of particulates such as salt from sea water evaporation
Appendix A

Nomenclature

Acronyms
PHE  Plate heat exchanger
NEWF  Non equilibrium wall function
EWT  Enhanced wall treatment

Roman Symbols

\( t_p \)  Plate thickness (m)
\( L_w \)  Plate width (m)
\( L_p \)  Plate height (m)
\( b_p \)  Mean flow channel gap (m)
\( P_{ca} \)  Corrugation pitch (m)
\( D_h \)  Hydraulic diameter (m)
\( A \)  area (m\(^2\))
\( h_{fg} \)  Latent heat (J/kg)
\( k \)  Thermal conductivity (W/m\(^2\) \cdot k)
\( C_p \)  Specific heat (J/kg \cdot k)
\( M \)  Molecular mass
\( \mathbf{V} \)  Velocity vector field
\( U \)  Velocity magnitude (m/s)
\( g \)  Gravitational acceleration (m/s\(^2\))
\( P \)  Static pressure (Pa)
\( h \)  Enthalpy
\( T \)  Temperature °C or (k)
\( T_u_{in} \)  Inlet turbulence intensity (%)
\( Re \)  Reynolds number
\( l_t \)  Turbulent length scale
\( Q \)  Heat Transfer (W)
\( \dot{m} \)  Mass flow rate (kg/s)
\( h_{sp} \)  Single phase heat transfer coefficient (W/m\(^2\) \cdot k)
\( f \)  Fanning friction factor
\( y^+ \) Dimensionless logarithmic distance
\( F \) Augmentation factor
\( S \) Suppression factor
\( q \) Heat flux \((W/m^2)\)
\( h_{tp} \) Two phase heat transfer coefficient \((W/m^2 \cdot k)\)
\( h_{nb} \) Nucleate boiling heat transfer coefficient \((W/m^2 \cdot k)\)
\( X_{li} \) Lockhart Martinelli parameter
\( K \) Turbulence Kinetic energy \((J)\)
\( Ke \) Kinetic energy \((J)\)
\( f_{eop} \) Evaporation frequency
\( f_{cond} \) Condensation frequency
\( d \) Phase diameter \((m)\)
\( C_D \) Drag coefficient
\( G \) Mass flux \((kg/m^2 \cdot s)\)
\( x \) Vapor mass quality
\( Bo \) Boiling number
\( Bd \) Bond number

**Greek Symbols**

\( \beta \) Chevron angle \((^\circ)\)
\( \phi \) length Enhancement factor
\( \sigma \) Surface tension between water liquid and vapor \((N/m^2)\)
\( \mu \) Dynamics viscosity \((Pa \cdot s)\)
\( \rho \) Density \((kg/m^3)\)
\( \zeta \) Friction factor based on Darcy’s equation
\( \alpha \) Volume fraction
\( \varepsilon \) Turbulence dissipation rate
\( \Delta T_{LMTD} \) Logarithmic mean temperature difference \((k)\)
\( \Delta T_{sat} \) Excess temperature \((^\circ C)\)
\( \Delta p \) Pressure drop \((Pa)\)
\( \phi^2 \) Two phase multiplier
\( C \) Chisholm parameter

**Subscripts**

s Solid
l Liquid
v Vapor
w Wall
tp Two phase
m Mixture
t Turbulence
ch Channel cross sectional area
s Surface
c Cold
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>sp</td>
<td>Single phase</td>
</tr>
<tr>
<td>nb</td>
<td>Nucleate boiling</td>
</tr>
<tr>
<td>cb</td>
<td>Convective boiling</td>
</tr>
<tr>
<td>sat</td>
<td>Saturation</td>
</tr>
<tr>
<td>conv</td>
<td>Convective</td>
</tr>
<tr>
<td>sub</td>
<td>Subcooled</td>
</tr>
<tr>
<td>st</td>
<td>Static</td>
</tr>
<tr>
<td>acc</td>
<td>Acceleration</td>
</tr>
<tr>
<td>fr</td>
<td>Friction</td>
</tr>
</tbody>
</table>
Bibliography


