Two Phase Flow Distribution and Heat Transfer in Plate Heat Exchanger

March 2017

Department of Science and Advanced Technology
Graduate School of Science and Engineering
Saga University

Mohammad Sultan Mahmud
Two Phase Flow Distribution and Heat Transfer in Plate Heat Exchanger

By

Mohammad Sultan Mahmud

A dissertation submitted in partial fulfillment of the requirements for the degree of

Doctor of Philosophy (Ph. D)

in

Mechanical Engineering

Department of Science and Advanced Technology
Graduate School of Science and Engineering
Saga University
Japan

March 2017
ACKNOWLEDGEMENTS

I am indebted to The Almighty Allah, Who is the Most Gracious, Most Merciful, the Cherisher and Sustainer of the world for giving me the strength and the speed to complete this thesis work.

I would like to express my sincere gratitude to my supervisor, Professor Akio Miyara for his encouragement, guidance, and support during my study and research from the initial to the final step. I am grateful to him for being a great advisor, an excellent motivator, an intellectual magnetic personality and never a conservative. I am especially thankful for his valuable insights, and all his endurance in trying to make me a better researcher. I attribute the level of my Ph. D. degree to his encouragement and effort and without him this thesis would not have been completed or written.

I also would like to extend my sincere thanks to the members of my dissertation committee, Professor Yuichi Mitsutake, Professor Yoichi Kinoue and Dr. Keishi Kariya for their time, comments and valuable ideas. I am grateful for their generous and suggestions that helped improve this work significantly.

I would like to thank ‘Khulna University of Engineering and Technology’ of Bangladesh for granting me study leave for this study. I would like to acknowledge the Japanese Government (MONBUKAGAKUSHO) for furnished me with the scholarship for this study.

I would like to thank all of my lab mates for their support and encouragement. I thank my friends and their families for making my stay at Japan enjoyable and memorable.

Lastly, it is impossible to describe in words the love, encouragement and support that I received from my family.
ABSTRACT

Plate heat exchangers (PHEs) are a type of compact heat exchanger widely used for industrial applications, such as refrigeration, heating, cooling, chemical processing, etc. They provide a large heat transfer surface area per volume, which makes them particularly suited for installation in confined spaces. Consequently, they have a reduced refrigerant charge and require lighter structural supports. Generally, PHEs consist of thin, rectangular, pressed steel plates (most often stainless steel) that are stacked together, such that hot and cold fluid streams alternate through the inter-plate passages. The plates are stamped with corrugated patterns that not only provide a larger effective heat transfer surface area but also modify the flow field in order to promote enhanced thermal–hydraulic performance. The use of compact heat exchangers with reduced channel hydraulic diameter and flow channel length has increased over the last years. To keep the pressure loss at acceptable levels, an implication of the diameter down-scaling is an increase in the number of parallel flow channels through the heat exchanger. The growing number of parallel flow channels increases the challenge of distributing two-phase flow equally among the channels. Due to the complexity of the two-phase flow process, a rigorous theoretical analysis is not feasible. The fundamental understanding of the flow distribution and other mechanisms in this type of channel is rather limited.

In this thesis, two-phase hydraulic performances of a prototype chevron plate heat exchanger have been investigated, modelled and simulated in detail. The hydrodynamic characteristics and flow distribution in two cross-corrugated channels have been investigated numerically. The main advantages of the CFD model of the PHE are the detailed void fraction and velocity distributions obtained and the fact that it is not necessary to collect extensive experimental data to adjust the model parameters. The 3D velocity fields have been obtained through numerical simulation. It is found that the flow around the contact points is separated in two streams and a considerable mixing occurs. Resorting to the results of velocities field it was possible to conclude that a laminar flow occurs in the present operation conditions; the existence of corrugations in the plates confers to velocity a sinusoidal behaviour in the main flow direction.

The local condensation heat transfer characteristics of R1234ze(E) in the plate heat exchanger were investigated experimentally at different mass flux conditions. The experiments were conducted by varying the mass fluxes. The local condensation heat transfer coefficient decreased with increasing of wetness with different values in horizontal direction. At low refrigerant mass flux (G = 10, 20 kg/m²s), the heat transfer coefficients are not dependent on
mass flux and probably condensation is controlled by gravity. For higher refrigerant mass flux
(Gr = 50 kg/m²s) the heat transfer coefficients depend on mass flux and forced convection
condensation occurs. The condensation heat flux decreases with increase of distance along the
direction from inlet to outlet of plate heat exchanger which is the direction of condensation
progress. The local wall temperature distribution decreases with increase of distance along the
downstream.

More advanced development in computer technology will be helpful for further investigations
by CFD. It is very good method to obtain some detail information about two-phase flow in
flow field which cannot be achieved by experiment only. It is possible to obtain details of two
phase flow inside plate heat exchanger by numerical simulations. Some evidence has not been
revealed yet and become a good challenge in future research.
## CONTENTS

Acknowledgements i  
Abstract ii  
Table of contents iv  
List of figures vii  
List of tables x  
Nomenclature xi  

### Chapter 1: Introduction 1  
1.1 Background 1  
1.2 Objective of this study 3  
1.3 Outline of the thesis 3  
References 4  

### Chapter 2: Fundamentals of Plate heat exchanger and Multiphase flow 5  
2.1 Overview of plate heat exchangers 5  
2.1.1 Some advantages of the plate heat exchanger are discussed below 5  
2.1.2 Geometry specification 6  
2.1.3 Complex flow geometry 9  
2.2 Multi-Phase Flow Concepts 9  
2.2.1 Introduction 9  
2.2.2 Types of multi-phase flows 9  
2.2.3 Modeling concepts of multi-phase flows 11  
2.3 Modeling dispersed multi-phase flows 12  
2.3.1 Eulerian-Eulerian approach 13  
2.3.2 Eulerian-Lagrangian approach 14  
2.3.3 Volume of fluid (VOF) approach 16  
References 19  

### Chapter 3: Numerical Simulation of Two Phase Flow in Plate Heat Exchanger 21
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1 Flow Patterns</td>
<td>21</td>
</tr>
<tr>
<td>3.2 Flow Distribution in Plate Heat Exchanger</td>
<td>23</td>
</tr>
<tr>
<td>3.3 Literature review</td>
<td>25</td>
</tr>
<tr>
<td>3.4 Simulation of Plate heat exchanger</td>
<td>27</td>
</tr>
<tr>
<td>3.5 The ANSYS fluent model</td>
<td>27</td>
</tr>
<tr>
<td>3.5.1 Transport equations in ANSYS Fluent</td>
<td>28</td>
</tr>
<tr>
<td>3.5.2 Conservation of mass</td>
<td>28</td>
</tr>
<tr>
<td>3.5.3 Conservation of momentum</td>
<td>28</td>
</tr>
<tr>
<td>3.5.4 Conservation of energy</td>
<td>29</td>
</tr>
<tr>
<td>3.5.5 Volume fraction equation</td>
<td>29</td>
</tr>
<tr>
<td>3.6 Geometry generation</td>
<td>29</td>
</tr>
<tr>
<td>3.7 Meshing of Fluid Domain</td>
<td>30</td>
</tr>
<tr>
<td>3.8 Boundary and operating conditions</td>
<td>30</td>
</tr>
<tr>
<td>3.9 CFD solution procedure</td>
<td>31</td>
</tr>
<tr>
<td>3.10 Results and Discussion</td>
<td>32</td>
</tr>
<tr>
<td>3.11 Conclusions</td>
<td>37</td>
</tr>
<tr>
<td>References</td>
<td>38</td>
</tr>
</tbody>
</table>

**Chapter 4: Local Condensation Heat Transfer of Refrigerant R1234ze(E) in a Plate Heat Exchanger**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1 Introduction</td>
<td>41</td>
</tr>
<tr>
<td>4.2 Literature Review for Local Condensation Heat Transfer</td>
<td>41</td>
</tr>
<tr>
<td>4.3 Data reduction for Condensation heat transfer</td>
<td>43</td>
</tr>
<tr>
<td>4.4 Experimental apparatus and procedures</td>
<td>45</td>
</tr>
<tr>
<td>4.4.1 Refrigerant flow loop</td>
<td>46</td>
</tr>
<tr>
<td>4.4.2 Plate heat exchanger: the test section</td>
<td>47</td>
</tr>
<tr>
<td>4.4.3 Water loop for pre-heater</td>
<td>49</td>
</tr>
<tr>
<td>4.4.4 Water-brine loop for test section</td>
<td>50</td>
</tr>
<tr>
<td>4.4.5 Water-brine loop for after condenser</td>
<td>50</td>
</tr>
<tr>
<td>4.4.6 Data acquisition</td>
<td>50</td>
</tr>
<tr>
<td>4.4.7 Experimental procedures</td>
<td>51</td>
</tr>
<tr>
<td>4.5 Results and discussions</td>
<td>51</td>
</tr>
<tr>
<td>4.6 Conclusions</td>
<td>60</td>
</tr>
</tbody>
</table>
Chapter 5: Local Evaporation Heat Transfer of Refrigerant R1234ze(E) and R32 in Plate Heat Exchanger

5.1 Introduction 63
5.2 Literature Review for Local Evaporation Heat Transfer 63
5.3 Data Reduction 65
5.4 Experimental Setup 67
5.5 Experimental procedures 68
5.6 Result and Discussions for Refrigerant R1234ze (E) 69
5.7 Result and Discussions for Refrigerant R32 74
5.8 Conclusions 79
References 79

Chapter 6: Conclusions and recommendations 81

6.1 Conclusions 81
6.2 Recommendations 82
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure No.</th>
<th>Figure captions</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Plate heat exchanger geometry</td>
<td>09</td>
</tr>
<tr>
<td>3.1</td>
<td>Flow patterns with associated heat transfer regions in uniformly heated tube with a low heat flux</td>
<td>22</td>
</tr>
<tr>
<td>3.2</td>
<td>Manifold pressure and channel flow distribution</td>
<td>24</td>
</tr>
<tr>
<td>3.3</td>
<td>Computational domain for CFD modelling</td>
<td>31</td>
</tr>
<tr>
<td>3.4</td>
<td>Computational mesh for CFD modelling</td>
<td>31</td>
</tr>
<tr>
<td>3.5</td>
<td>Unstructured tetrahedral mesh</td>
<td>32</td>
</tr>
<tr>
<td>3.6</td>
<td>Mid-plane void fraction at time 0.39s, 0.48s and 0.60s</td>
<td>33</td>
</tr>
<tr>
<td>3.7</td>
<td>Mid-plane void fraction at time 0.72s, 0.75s and 0.78s</td>
<td>33</td>
</tr>
<tr>
<td>3.8</td>
<td>Mid-plane velocity distribution at time 0.78s. “●” are contact points</td>
<td>34</td>
</tr>
<tr>
<td>3.9</td>
<td>Velocity distribution at horizontal mid-section of the channel</td>
<td>35</td>
</tr>
<tr>
<td>3.10</td>
<td>Velocity distribution for vertical mid-section of inlet port.</td>
<td>36</td>
</tr>
<tr>
<td>3.11</td>
<td>Velocity distribution for outlet port.</td>
<td>36</td>
</tr>
<tr>
<td>3.12</td>
<td>Velocity distribution for part of vertical mid-section.</td>
<td>36</td>
</tr>
<tr>
<td>3.13</td>
<td>Velocity distribution around the contact points. “●” are contact points</td>
<td>37</td>
</tr>
<tr>
<td>4.1</td>
<td>Schematic diagram of experimental setup</td>
<td>46</td>
</tr>
<tr>
<td>4.2</td>
<td>The test section</td>
<td>48</td>
</tr>
<tr>
<td>4.3</td>
<td>Location of thermocouples in test section</td>
<td>48</td>
</tr>
<tr>
<td>4.3a</td>
<td>Details of thermocouple position in test section (enlarged view of area A shown in Fig. 4.3).</td>
<td>49</td>
</tr>
<tr>
<td>4.4</td>
<td>Variation of local condensation heat transfer coefficient with wetness at mass flux G = 10 kg/m²s</td>
<td>52</td>
</tr>
<tr>
<td>4.5</td>
<td>Variation of heat flux as a function of distance from inlet of the test section at mass flux G = 10 kg/m²s</td>
<td>53</td>
</tr>
<tr>
<td>4.6</td>
<td>Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux G = 10 kg/m²s</td>
<td>53</td>
</tr>
</tbody>
</table>
4.7 Variation of local condensation heat transfer coefficient with wetness at mass flux $G = 20 \text{ kg/m}^2\text{s}$

4.8 Variation of heat flux as a function of distance from inlet of the test section at mass flux $G = 20 \text{ kg/m}^2\text{s}$

4.9 Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 20 \text{ kg/m}^2\text{s}$

4.10a Variation of local condensation heat transfer coefficient on wetness, [0.12 to 0.50] at mass flux $G = 50 \text{ kg/m}^2\text{s}$

4.10b Variation of local condensation heat transfer coefficient on wetness, [0.56 to 0.91] at mass flux $G = 50 \text{ kg/m}^2\text{s}$

4.11a Variation of heat flux as a function of distance from inlet of the test section for wetness from 0.12 to 0.50 at mass flux $G = 50 \text{ kg/m}^2\text{s}$

4.11b Variation of heat flux as a function of distance from inlet of the test section for wetness from 0.56 to 0.91 at mass flux $G = 50 \text{ kg/m}^2\text{s}$

4.12a Variation of wall temperature distribution as a function of distance from inlet of the test section for vapor quality from 0.12 to 0.50 at mass flux $G = 50 \text{ kg/m}^2\text{s}$

4.12b Variation of wall temperature distribution as a function of distance from inlet of the test section for vapor quality from 0.56 to 0.91 at mass flux $G = 50 \text{ kg/m}^2\text{s}$

5.1 Variation of local boiling heat transfer coefficient with vapor quality at mass flux $G = 10 \text{ kg/m}^2\text{s}$

5.2 Variation of heat flux as a function of distance from inlet of the test section at mass flux $G = 10 \text{ kg/m}^2\text{s}$

5.3 Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 10 \text{ kg/m}^2\text{s}$

5.4 Variation of local boiling heat transfer coefficient with vapor quality at mass flux $G = 20 \text{ kg/m}^2\text{s}$

5.5 Variation of heat flux as a function of distance from inlet of the test section at mass flux $G = 20 \text{ kg/m}^2\text{s}$

5.6 Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 20 \text{ kg/m}^2\text{s}$
5.7 Variation of local boiling heat transfer coefficient with vapor quality at mass flux $G = 30 \text{ kg/m}^2\text{s}$
5.8 Variation of heat flux as a function of distance from inlet of the test section at mass flux $G = 30 \text{ kg/m}^2\text{s}$
5.9 Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 30 \text{ kg/m}^2\text{s}$
5.10 Variation of local boiling heat transfer coefficient of R32 with vapor quality at mass flux $G = 10 \text{ kg/m}^2\text{s}$
5.11 Variation of heat flux of R32 as a function of distance from inlet of the test section at mass flux $G = 10 \text{ kg/m}^2\text{s}$
5.12 Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 10 \text{ kg/m}^2\text{s}$
5.13 Variation of local boiling heat transfer coefficient of R32 with wetness at mass flux $G = 20 \text{ kg/m}^2\text{s}$
5.14 Variation of heat flux of R32 as a function of distance from inlet of the test section at mass flux $G = 20 \text{ kg/m}^2\text{s}$
5.15 Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 20 \text{ kg/m}^2\text{s}$
5.16 Variation of local boiling heat transfer coefficient of R32 with vapor quality at mass flux $G = 40 \text{ kg/m}^2\text{s}$
5.17 Variation of heat flux of R32 as a function of distance from inlet of the test section at mass flux $G = 40 \text{ kg/m}^2\text{s}$
5.18 Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 40 \text{ kg/m}^2\text{s}$
LIST OF TABLES

<table>
<thead>
<tr>
<th>Table No.</th>
<th>Table captions</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1</td>
<td>Geometrical characteristics of the PHE</td>
<td>50</td>
</tr>
<tr>
<td>4.2</td>
<td>Experimental conditions for condensation of R1234ze(E)</td>
<td>51</td>
</tr>
<tr>
<td>5.1</td>
<td>The research has been done with carbon dioxide, as reported in open literature</td>
<td>64</td>
</tr>
<tr>
<td>5.2</td>
<td>Geometrical characteristics of the PHE</td>
<td>68</td>
</tr>
<tr>
<td>5.3</td>
<td>Experimental conditions for evaporation of R1234ze(E)</td>
<td>68</td>
</tr>
</tbody>
</table>
NOMENCLATURE

\(a\)  Respective amplitude (depth of corrugation)
\(b\)  Pitch between two plates
\(d\)  Diameter
\(f\)  Force
\(h\)  Enthalpy
\(K\)  Thermal conductivity
\(l\)  Distance
\(P\)  Pressure
\(Z\)  Dimensionless corrugation parameter
\(M\)  Interface momentum
\(Q\)  Heat transfer
\(r\)  Radius
\(S\)  Momentum source term
\(v\)  Velocity
\(g\)  Gravity
\(m\)  Mass
\(q\)  Local heat flux
\(T\)  Temperature
\(x\)  Local vapor quality

Subscripts

\(B\)  Basset
$D$ Drag

$E$ External body

$eq$ Equivalent

$G$ Gravity

$h$ Hydraulic

$hp$ Hydrostatic pressure

$q$ The $q^{th}$ phase

$l$ The $l^{th}$ phase

$loss$ Heat loss

$L$ Lift

$p$ pressure

$P$ Particle

$pre$ Preheater

$ref$ Refrigerant

$sat$ Saturation

$v$ vapour

$V$ Virtual mass

$wall$ Wall of working fluid side

**Greek Symbols**

$\alpha$ Volume fraction

$\alpha_s$ Local heat transfer coefficient

$\beta$ Chevron angle

$\Lambda$ Wavelength (pitch)
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>γ</td>
<td>Plate corrugation aspect ratio</td>
</tr>
<tr>
<td>λ</td>
<td>Thermal conductivity of stainless steel</td>
</tr>
<tr>
<td>φ</td>
<td>Enlargement factor</td>
</tr>
<tr>
<td>ρ</td>
<td>Density</td>
</tr>
<tr>
<td>σ</td>
<td>Viscous stress tensor</td>
</tr>
<tr>
<td>Γ</td>
<td>Source term of mass</td>
</tr>
</tbody>
</table>
CHAPTER ONE

Introduction

1.1 Background

In today’s highly industrialized world, energy is one of the major concerns. With rapid consumption of fossil fuels, saving energy has become an attractive topic for the researchers. Recently, a marked improvement in heat exchanger technology has been observed. With improved technology vivid energy savings are now possible with efficient natural refrigerants in compact heat exchangers, used in various applications (Khan et al., 2014).

Plate heat exchangers (PHE’s) were first commercially introduced in the 1920’s to meet the hygienic demands of the dairy industry, while some patents existed as early as in the 1870’s in Germany (Clark, 1974). Design of this type of exchanger reached maturity in the 1960’s with the development of more effective plate geometries, assemblies, and improved gasket material, and the range of possible applications has widened considerably (Kakac S. and Liu H., 2002). PHE’s are nowadays widely used in a broad range of heating and cooling applications in food processing, chemical reaction processes, petroleum, pulp and paper, as well as in many water-chilling applications. Some basic features of PHE’s include high efficiency and compactness (i.e., high heat transfer capacity per unit volume compared to conventional, shell-and-tube heat exchangers), high flexibility for desired load and pressure drop, easy cleaning, and cost competitiveness.

While PHE’s became popular for liquid-to-liquid heat transfer duties, their use in phase-changing applications was not common initially. Before the 1990’s such applications were mostly in the fields of concentrating liquid food and drying of chemicals. Applications in refrigeration systems were rare, mainly because of concerns over refrigerant leakage, and also because of the pressure limits required, especially on condensation applications. In the last two decades, with the introduction of semi-welded and brazed PHE’s, this type of exchanger has been increasingly used in refrigeration systems, from domestic heat pumps to large ammonia installations for water-chilling duties.
Plate heat exchangers (PHEs) are a type of compact heat exchanger widely used for industrial applications, such as refrigeration, heating, cooling, chemical processing, etc. They provide a large heat transfer surface area per volume, which makes them particularly suited for installation in confined spaces. Consequently, they have a reduced refrigerant charge and require lighter structural supports. Generally, PHEs consist of thin, rectangular, pressed steel plates (most often stainless steel) that are stacked together, such that hot and cold fluid streams alternate through the inter-plate passages. The plates are stamped with corrugated patterns that not only provide a larger effective heat transfer surface area but also modify the flow field in order to promote enhanced thermal–hydraulic performance. Their most important feature is their larger heat transfer surface area per unit volume compared to a shell and-tube unit and thus PHEs have increasingly become the heat exchanger of choice in many industrial and domestic applications in the small to medium size range. They are compact, flexible to alter the thermal size for accommodating varying heat load capacities by adding/removing the plates as well as changing their geometry, cleanable (e.g. the gasketed-PHE) and attractive to enhance heat transfer characteristics (Kakaç and Liu H., 2002). The improved convective heat transfer encouraged by the corrugated plates and the consequent complex interpolate channels is primarily linked to the effective heat transfer area enlargement, smaller flow cross section or hydraulic diameter, flow disruption and reattachment of boundary layers and the generation of swirl, vortex and helical secondary flows.

Heat exchangers, including PHE’s and other types, are designed and employed according to two criteria: heat transfer and pressure drop. Characteristics of the heat exchanger’s thermo-hydraulic performance are the primary interest of most investigations. The corrugated channels of PHE’s have probably the most complicated geometry of all flow ducts. As the number of applications has increased, single-phase flows in PHE’s have been well investigated in the last few decades. Although a general theory about the influence of some geometric parameters is lacking, due to the complex geometry of the flow channel, a large number of correlations for heat transfer and pressure drop are available in the open literature. The number of published investigations on two-phase heat transfer and pressure drop in PHE’s is increasing but is not yet comprehensive.

Due to the complexity of the two-phase flow process, a rigorous theoretical analysis is not feasible. The fundamental understanding of the flow distribution and other mechanisms in this type of channel is rather limited.
1.2 Objective of this study

The purpose of the present study is to investigate the performance of plate heat exchangers using refrigerant used as condenser and evaporators. The objective is to extend the present knowledge of the thermo-hydraulic performance of PHE evaporators with consideration of flow conditions (mass flux, heat flux, vapour fraction) and channel geometries (chevron angles). This study aims to obtain a generally applicable model, which is able to provide design and operating guidelines for this type of evaporator, and to be used for the simulation of complete refrigeration systems. The specific objectives of this thesis are:

1. To carry out a thorough literature review on plate heat exchangers, their terminology, working principles, two phase performance characteristics and most importantly the state of the art of evaluating the condensation and evaporation heat transfer and simulation of two-phase flow in plate heat exchanger channels.

2. To obtain two-phase air-water flow distribution phenomena for a corrugated plate heat exchanger. This serves to provide a quantitative understanding of the performance of plate heat exchanger in two-phase applications, and to provide the information for flow behavior inside the plate heat exchanger.

3. To obtain condensation heat transfer characteristics of refrigerant R1234ze(E) and flow conditions (mass flow rate, heat flux, vapour quality).

4. To obtain evaporation heat transfer characteristics of refrigerant R1234ze(E) and R32 and flow conditions (mass flow rate, heat flux, vapour quality).

As the final and principal aim, to extend the present knowledge of the thermo-hydraulic performance of plate heat exchanger. There are a few types of PHE’s available, but this study was limited to the chevron (herringbone) corrugation type.

1.3 Outline of the thesis

This thesis contains six chapters.
Chapter 1 provides an introduction to plate heat exchanger applications in refrigeration, followed by the objectives, scope, motivation of the current study.

Chapter 2 provides a critical review of the literature. This chapter includes a thorough theories and methods for Computational Fluid Dynamics in general, an in-depth review of plate heat exchanger theory, terminology, working principles.

Chapter 3 provides detailed description of CFD methodology, construction of computational domain and simulation of two phase flow with appropriate boundary conditions of plate heat exchanger.

Chapter 4 provides a detailed description of the experimental facility, including the plate heat exchangers, and the experiments of condensation heat transfer of refrigerant R1234ze(E) in plate heat exchanger.

Chapter 5 provides the experiments of evaporation heat transfer of refrigerant R1234ze(E) in plate heat exchanger and present results of the evaporator performance tests using R32.

Chapter 6 summarizes the overall results of the simulations and heat transfer characteristics and finalize the conclusions and gives suggestion for future work.

References:


CHAPTER TWO

Fundamentals of Plate Heat Exchanger and Multiphase flow

2.1 Overview of plate heat exchangers

In this thesis, two-phase hydraulic performances of a prototype chevron plate heat exchanger have been investigated, modelled and simulated in detail. In fact, plate heat exchangers are an important type of compact heat exchanger widely used for many applications: refrigeration, heating, cooling, chemical processing, etc. Generally, plate heat exchangers consist of thin, rectangular, pressed steel plates (most often stainless steel) that are stacked together, such that hot and cold fluid streams alternate through the inter-plate passages. The plates are stamped with corrugated patterns, providing a larger effective heat transfer surface area and additionally modify the flow field in order to promote enhanced thermal-hydraulic performance. Their most important feature is the larger heat transfer surface area per unit volume compared to shell-and-tube and tube-in-tube units and thus plate heat exchangers have increasingly become the preferred solution in many industrial and domestic applications in the small to medium size range. Several types of plate heat exchangers are utilized, depending on the application. Mainly, there are four types of plate heat exchangers: Gasketed, brazed, welded and semi-welded and shell and plate.

The plate heat exchanger is designed with either single-pass or multi-pass flow, depending on the application. For most applications, the single-pass solution is suitable and preferred because it keeps all connections on the stationary frame part and thus makes disassembly easier. However, for applications with low rates or close approach temperatures, the multi-pass solution is required.

2.1.1 Some advantages of the plate heat exchanger

*Expandable:* the expandability of the plate heat exchanger is one of its significant features. Increasing the heat transfer requirement simply means adding plates instead of buying a new heat exchanger, thus saving time and money.

*High efficiency:* due to the plate patterns and relatively narrow gaps, very high turbulence is achieved at relatively low fluid velocity. This combined with counter directional flow results in very high heat transfer coefficient.
**Compact size:** consequent to it’s high efficiency, a much smaller heat exchanger than would be needed for the same duty using other heat exchangers is obtained. Typically, a plate heat exchanger requires between 20-40% of the space requirement for a tube and shell heat exchanger.

**Multiple duties in a single unite:** the plate heat exchanger can be bolt in sections separated with simple divider plates or more complicated divider frames with additional connections. This makes it possible to heat, regenerate and cool a fluid in one heat exchanger or heat or cool multiple fluids with same cooling or heating source.

**Less fouling:** the very high turbulence achieved as a result of the plate pattern and narrow gap, combined with the smooth plate surface reduces fouling considerably in comparison to other heat exchanger types.

Easy to remove and clean: this is done by simply removing the tie bolts and sliding the movable frame part back. The plates can then be inspected, pressure cleaned or removed for refurbishment if required.

To better understand the mechanisms of plate heat exchangers, their flow patterns, swirl flows and effects on heat transfer and pressure drop and to translate this into the development of new products, the numerical simulation of various design parameters is essential.

In the numerical simulation of plate heat exchangers, some assumptions are made, most of which do not correspond to real plate heat exchanger conditions. For example, the constant wall temperature/heat flux thermal boundary condition commonly specified does not correspond to real situation as the temperature and heat flux varies along the plate due to complex 3D flow caused by the shape of the corrugation pattern. For large plate heat exchangers, a fully developed flow approximation in the central region may be assumed, which is invalid for small plate heat exchangers because at the central region of the fully developed regions.

To obtain an approximation of the actual results, the smallest geometry that simulates the heat exchanger would be analyzed.

### 2.1.2 Geometry specification

An approximately sinusoidal chevron corrugation geometry is the main pattern normally adopted to fabricate the plates for different commercial applications. The angle between the
corrugation and the vertical axis is typically defined as the chevron angle, $\beta$, in the literature (see Fig. 2.1). In some publications, however, an inclination angle has been used, i.e. the angle between the furrows and the horizontal axis ($90^\circ - \beta$), in place of the chevron angle. The first definition (chevron angle, $\beta$) is used here for consistency with most of the literature (see Fig. 2.1). Plates with different chevron angles, such as low and high chevron angles, can be stacked together in either a symmetric or mixed arrangements. In a mixed arrangement, plates of two different chevron values are employed alternately. It should be noted that conventionally the mixed-plate arrangement is usually designated by the average chevron angle of the two plates.

The multiple metal-to-metal inter-plate contact points lend to their increased rigidity and mechanical support of the stack. A plate with a larger chevron angle provides higher thermal performance compared to that with lower corrugation angle, however, accompanied by a higher pressure drop penalty. The lateral and transversal cross sectional profiles of the channels are shown in Figure 1.2b for plates with chevron angles of $\beta = 0^\circ$ and $\beta = 90^\circ$, respectively.

The severity of sinusoidal surface waviness can be essentially defined by two dimensionless parameters: the plate corrugation aspect ratio, $\gamma$, and the enlargement factor, $\phi$, defined below:

$$
\gamma = \frac{4a}{\Lambda} = \frac{2b}{\Lambda}
$$

(2.1)

$$
\phi = \frac{\text{Effective area}}{\text{Projected area}} = \frac{\int_{0}^{A} \sqrt{1 + \left(\frac{\gamma \pi}{2}\right)^2 \cos^2 \frac{2\pi}{\Lambda} x} \, dx}{\Lambda}
$$

(2.2)

Where, $a$ and $\Lambda$ are the respective amplitude (depth of corrugation) and wavelength (pitch) of a sinusoidal surface corrugation. The enlargement factor can be approximated for a sinusoidal corrugation from a three-point integration formula, using the dimensionless corrugation parameter $Z$:

$$
Z = \frac{2\pi a}{\Lambda}
$$

(2.3)
\[ \varphi \approx \frac{1}{6} \left( 1 + \sqrt{1 + Z^2} + 4 \sqrt{1 + Z^2/2} \right) \]

(2.4)

In general, both heat transfer coefficients and flow friction losses increase with increasing \( \beta, \varphi, \) and \( \gamma. \) The length scale or the equivalent diameter for the inter-plate spacing that confines the flow channel then becomes:

\[ d_{eq} = 4a = 2b \]

(2.5)

and \( a \) is the amplitude of the corrugation, while \( b \) is the pitch between two plates \( (b = 2a) \).

A modified length scale in the form of hydraulic diameter is defined as follow:

\[ d_h = \frac{d_{eq}}{\varphi} = \frac{4a}{\varphi} = \frac{2b}{\varphi} \]

(2.6)

---

Figure 2.1: Plate heat exchanger geometry (Amalfi et al. 2016).
2.1.3 Complex flow geometry

Due to the angle of the corrugations, $\beta$ in figure 2.1, the flow pattern of the fluid becomes complex. The plate pattern promotes early transition to turbulent flow with a secondary swirl flow. This complex flow results in high heat transfer coefficients and pressure drops. In fact, for a commercial plate heat exchanger in a heat pump installation, the heat transfer coefficient on the two fluid sides are at the same order of magnitude or even with a higher heat transfer coefficient on the secondary refrigerant.

2.2 Multi-phase Flow Concepts

2.2.1 Introduction

Multi-phase flow is a type of fluid flow which consists of more than one phase or component. The phase here means the thermodynamic state of the fluids, i.e., gas, liquid, or solid. Multi-phase flows encompass a wide range of fluid flows such as gas-liquid, liquid-liquid, gas-solid, liquid-solid, or gas-liquid-solid flows. There are a lot of examples of multi-phase flow in industry such as gas-solid flow in cyclones, gas-liquid flow in pipelines, gas-solid or gas-liquid-solid flow in chemical reactors, etc. Multi-phase flow regimes depend on the geometry, the velocities and properties of the flowing fluids. For example, in multi-phase flow reactors such as three-phase packed bed reactors there are four different flow regimes called trickle flow, bubble flow, mist flow and pulsating flow. The main difference between these flow regimes are the velocity of the gas and liquid phases. Multi-phase reactors are used in different industries, e.g., in the production of oil and gas, in the food processing, or in water treatment plants, etc. So, it is important to know more about these reactors. In addition, with understanding more about these reactors we will be able to design more effective and high performance multi-phase reactors and improve the performance of the existing multi-phase reactors in industry.

2.2.2 Types of multi-phase flows

When two or more phases flow together in a heat exchanger, reactor, pipeline or canal a lot of different flow regimes can be seen. It is possible to classify these flow regimes in different ways. The first way is the classification of the flow regime with the thermodynamic state of the phases, for example, gas-liquid, liquid-solid, gas-solid, gas-liquid-solid, etc. In addition, if the properties of two liquids are different and they do not mix very well (immiscible liquids)
we can also have a liquid-liquid two-phase flow. For example, mixtures of refrigerant and water can be classified as two-phase liquid-liquid flow. Broadly, multi-phase flow regimes can be divided in dispersed flows, separated flows and mixed flows. Dispersed flow is a kind of multi-phase flow with all phases dispersed except for one that is continuous. In this sort of flow all dispersed phases flow through the continuous phase. Fluid flow in bubble column reactors, trickle bed reactors and cyclones are examples of dispersed flow regime. Separated flows are flows in which none of the phases is dispersed and all phases are continuous or semi-continuous. The examples of these kind of flow are film flow and annular flow. Mixed flow is a combination of dispersed flow and separated flow. So, in mixed flow we have both dispersed and separated phases. The example of this sort of multi-phase flow can be bubbly annular flow and slug flow. In three-phase reactors such as packed bed reactors we can have both separated and mixed flow regimes. For example, in these reactors in the trickling flow regime the gas phase is the continuous phase and the liquid phase is the dispersed phase. Depending on the superficial velocity of gas and liquid, the continuous and dispersed phases the flow regime can change in trickle bed reactors.

In two-phase gas-solid reactors where the solid phase is not moving (fixed bed reactors) it is possible to consider the solid phase as a porous medium. In a porous medium, gas flows over the surface of the solid phase as well as in/through the pores of the solid phase. In this case, the size of the pores, the properties of solid particles and distribution of the gas phase can have an effect on the flow of gas. There are applications for this kind of reactor in industry such as the Ethylene Oxide reactor. There are other kind of gas-solid reactors in which both gas and solid phases are moving. They are called fluidized bed reactors. In these reactors, the gas phase is the continuous phase and the solid phase is the dispersed phase. It is also possible to have some other sub-regimes in these reactors. The sub-regimes can be the dense bed regime, the turbulent bed or the fast fluidized bed regime depending on the velocity of gas and solid phases.

For gas-liquid-solid reactors, such as packed bed reactors, in the trickling flow regime, the liquid phase is the dispersed phase and the gas phase is the continuous phase. We have discussed the flow regimes of trickle bed reactors in chapter 1. In the three-phase bubble column reactors the gas and solid phases are the dispersed phase and the liquid phase is the continuous phase. In bubble column reactors depending on the operating condition, property of the phases and velocity of the phases, other sub-regimes such as turbulent flow and slug flow are also possible.
2.2.3 Modeling concepts of multi-phase flows

For computational methods, the development of these methods was based on finite difference discretization of the continuity and Navier-Stokes equations. The initial idea was based on using velocity and pressure as the initial variables. Another option for computational methods with introducing the basic principles of the finite volume method. The first commercial CFD code (PHOENICS) for solving multi-phase flow problems was produced by Spalding.

As mentioned before multi-phase flows can be classified as separated flows and dispersed flows. This classification is important from a computational point of view as well as a physical point of view. For the modeling of dispersed flow systems such as most multi-phase flow reactors three main issues need to be addressed:

- Definition of phase, flow regime and required resolution
- Formulation of governing equations
- Solution of governing equations

As mentioned before the first classification of the multi-phase flows is based on the thermodynamic state (gas, liquid or solid). It is also possible to define different dispersed phases based on the particle size. For example, particles with size $A$ as phase $a$ and size $B$ as phase $b$. Both $a$ and $b$ phases can be gas, liquid or solid. In addition, it is also possible to define two different thermodynamic states as one phase. For example, a mixture of liquid and solid as slurry phase in three-phase (gas, liquid and solid) bubble column reactors. So, in three-phase bubble column reactors there will be two phases: gas and slurry (liquid-solid). The first steps in selection of the best multi-phase model are the definition of the phases and flow regime. Depending on the flow regime and properties of phases, different modeling approaches can be used.

There are three different approaches for the modeling of multi-phase flows:

(i) Eulerian-Eulerian approach

(ii) Eulerian-Lagrangian approach

(iii) Volume of fluid (VOF) approach

The Eulerian-Eulerian multi-phase approach is based on the assumption of each phase as interpenetrating continuum. The phases can be solid, liquid or gas and any combination of these
three phases. In this method any number of secondary or dispersed phase is possible. The number of phases is only limited by memory requirements and convergence issues. In this method, a single pressure is shared by all phases. The continuity, momentum, energy and species transfer equations are solved for each phase separately. This method is suitable for multi-phase flow modeling with a range of volume fraction between 0 and 1. As there is no limitation in the volume fraction or number of phases in this method, this method is a suitable approach for the simulation of multi-phase reactors with more than one dispersed phase in the system. For all these methods the number of phases are limited only by the computation time and availability of memory. It is not a priori clear which of these methods is the best for the simulation of multiphase systems. The appropriate method should be chosen based on the complexity of the dispersed phase, the size of the equipment and the parameters which we are interested in. For more understanding of multi-phase flows for some cases it is possible to model a problem with two of these approaches.

The Eulerian–Lagrangian approach is more complicated than the volume of fluid approach. In this method the fluid phase is considered as continuum and Navier-Stokes equations are solved for the continuous phases while the dispersed phases are solved by tracking the particles (bubbles or droplets) through the calculated flow field. The dispersed phases exchange momentum, mass and energy with the continuous phase. This method is suitable for the modeling of multi-phase flow systems with low volume fraction of dispersed phases (less than 10%). This model is an appropriate model for e.g. liquid fuel combustion and spray dryers.

The first method (VOF) is conceptually the simplest method of multi-phase flow modeling. In this method, all phases (two or more) are considered as non-interpenetrating continuum. In this method a single set of momentum equations is solved and the volume of each phase is tracked in the computational domain. The interfaces between the phases are tracked by the solution of a continuity equation for the volume fraction of the phases. This method is suitable for the modeling of multi-phase systems in a small domain. This method is also suitable for the multiphase flow systems for which the behavior of the interface is a point of interest. This method is not suitable for the modeling of large scale systems because it is computationally very expensive.

2.3 Modeling dispersed multi-phase flows

Dispersed multi-phase flow is present in a number of multi-phase flow reactors such as fluidized beds, bubble columns and trickle beds. The modeling of three phase reactors is very
complex due to the quite complex phenomena which occur in three-phase reactors. The reactor engineer has to deal with a lot of phenomena such as flow, species transport, heat transfer, mass transfer, chemical reaction, evaporation, condensation, etc. In addition, the dispersed phase will have an effect on the continuous phase flow specially when the volume fraction of the dispersed phase is increasing.

For selection of the most suitable model for the simulation of multi-phase flow reactors it is very important to have a detailed look at the coupling issues between the continuous and dispersed phases. After this step and considering computation time it will be possible to choose the best approach. Governing equations and more details of three modeling approaches are discussed below.

### 2.3.1 Eulerian-Eulerian approach

In the Eulerian-Eulerian approach all phases (dispersed and continuous) are taken as interpenetrating continuum. This method is the most suitable method for the modeling of multi-phase flow reactors such as fluidized bed reactors, bubble column reactors and trickle bed reactors with high volume fraction of dispersed phases (>10%). The coupling between the phases should be implemented via suitable interphase transport models. It is not easy to model complex phenomena (such as reaction, evaporation, condensation, mass transfer, etc) at the particle level with the Eulerian-Eulerian approach. For single-phase flows, basic transport equations are given in the form of mass, momentum and energy conservation. For multi-phase flows such equations should be solved with averaging. Several different averaging methods can be used for this purpose. In this section, we present a general form of the governing equations for the Eulerian-Eulerian multi-phase flows. With this approach, it is assumed that the sum of volume fraction of phases is equal to 1.0. If there are $n$ phases in total, this gives:

$$
\sum_{q=1}^{n} a_q = 1.0
$$

(2.7)

For the Eulerian-Eulerian multi-phase approach the averaged conservation equations for mass and momentum for each phase are given by:

$$
\frac{\partial (a_q \rho_q)}{\partial t} + \nabla \cdot (a_q \rho_q \vec{v}_q) = \Gamma_q
$$
where $\rho_q$, $\vec{v}_q$, $\alpha_q$ and $\sigma_q$ are the density, velocity, volume fraction and viscous stress tensor of the $q^{th}$ phase respectively. $P$ is the pressure, $\Gamma_q$ is a source term of mass, $M_{q,l}$ is the interface momentum exchange between phase $q$ and phase $l$, $\vec{v}_l$ is the relative velocity and $S_q$ is a momentum source term of phase $q$ due to external forces other than the gravity.

In addition, the Eulerian-Eulerian approach is computationally less expensive in comparison to the Eulerian-Lagrangian approach as an alternative. The main disadvantage of the Eulerian-Eulerian multi-phase approach is the need for closures for the exchange between the phases. Unfortunately, these closure relations are not available for all case of fluid-fluid or fluid-solid systems and they are not very accurate. So, the accuracy of this model is less than the Eulerian-Lagrangian model.

### 2.3.2 Eulerian-Lagrangian approach

In the Eulerian-Lagrangian approach, the particles of the dispersed phase are considered as rigid spheres which do not deform. The motion of a particle is governed by the Lagrangian form of the Newton’s second law:

\[
\frac{d(m_p \vec{v}_p)}{dt} = f_{hp} + f_p + f_E + f_G + f_D + f_V + f_L + f_B
\]

(2.10)

where $m_p = \rho_p V_p$ is the mass of the particle. As mentioned, the dispersed phase is assumed to be rigid spheres, so $m_p$, $\rho_p$ and $V_p$ are constant in time. The forces in the equation 2.8 are surface and body forces acting on a particle. $f_{hp}$ is the force due to the hydrostatic pressure, $f_p$ is the force due to any external pressure gradients, $f_E$ is any external body force except the gravity, $f_G$ is the body force due to the gravity, $f_D$ is the steady drag force, $f_V$ is the virtual mass force, $f_L$ is the lift force and $f_B$ is the Basset force.

The particle trajectory is calculated from the definition of the translational velocity of the center of mass of the particle:
\[
\frac{dr_p(t)}{dt} = v_p(t, r_p(t))
\]  

(2.11)

In a one way coupled system, any effect of the dispersed phase on the continuous phase is neglected. So, the local velocity of the dispersed phase has no effect on the continuous phase but the local velocity of the continuous phase has an impact on the dispersed phase. This is only true for systems with small volume fraction of the dispersed phase. For a system with higher volume fraction the effects of the dispersed phase on the continuous phase cannot be neglected. So, for denser systems, it is necessary to consider particle-particle interaction and also its effects on the continuous phase. Hence, four-way coupling is recommended.

The dispersed phase volume fraction and the number of particles of dispersed phase are the main issues in the coupling between the phases in the Eulerian-Lagrangian method. For very dilute systems \((\alpha_p < 10^{-6})\) a simple one-way coupling between dispersed and continuous phase is sufficient for considering the interaction between these two phases. For denser dispersed phases \((10^{-6} < \alpha_p < 10^{-3})\) two-way coupling is considered enough and reasonable. Finally, for the phases which \(\alpha_p > 10^{-3}\) four-way coupling is recommended. In the Eulerian-Lagrangian simulations the computational time should be taken into account. For the flows with a number of not more than 106 particles it is possible to solve Lagrangian equations for each element. But, if the number of particles are more than 104 particles a statistical approach is more useful and practical.

For simulations with two and four way coupling the continuous phase is described by modified single phase momentum equations. The momentum equations are solved considering the interaction terms between particles. These interaction terms are taken into account based on Newton’s third law (action=re-action).

The main advantage of the Eulerian-Lagrangian approach in comparison with the Eulerian-Eulerian approach is its flexibility with respect to the incorporation of the microscopic transport phenomena. The Eulerian-Lagrangian approach is computationally more expensive than the Eulerian-Eulerian approach. This is the main disadvantage of the Eulerian-Lagrangian approach. In general, multi-phase flow reactors are dense systems. So, it is very expensive to track a high number of dispersed particles. Therefore, averaged methods should be used for the modeling. For this purpose, appropriate closure laws are needed for the interfacial transport of
momentum. It is worth mentioning that these closures are not complete and they are still under development.

2.3.3 Volume of fluid (VOF) approach

In this approach, a single set of conservation equations is shared between the fluids. The governing equations can be written as follows:

\[
\frac{\partial (\rho \mathbf{v})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = \sum_{q=1}^{n} (S_q)
\]

(2.12)

\[
\frac{\partial (\rho \mathbf{v})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla P + \rho \mathbf{g} + \mathbf{F}
\]

(2.13)

Where, \( \rho \) is the density, \( \mathbf{v} \) is the velocity vector, \( S \) is the mass source, \( P \) is the pressure, \( \mathbf{g} \) is the gravity acceleration vector and \( \mathbf{F} \) is the force vector.

The equations 2.10 and 2.11 are the same equation which we use in single-phase flow problems. For multi-phase flows it is also possible to use these equations with the desired boundary conditions at the interface of different phases. But the important issue is that density, viscosity and other physical properties should be changed at the interface for the calculations. The other important issue is that when the volume fractions are changing the interface is also changing.

In the volume of fluid approach the movement of all phases is simulated and not the motion of the interface. The movement of the interface is inferred indirectly through the movement of different phases separated by an interface. The VOF formulation relies on the fact that two or more fluids (or phases) are not interpenetrating. For each additional phase that added to model, a variable is introduced: the volume fraction of the phase in the computational cell. Motion of the different phases is tracked by solving an advection equation of a phase volume fraction. In each control volume, the volume fractions of all phases sum to unity. The fields for all variables and properties are shared by the phases and represent volume-averaged values, as long as the volume fraction of each of the phases is known at each location. Thus the variables and properties in any given cell are either purely representative of one of the phases, or representative of a mixture of the phases, depending upon the volume fraction values. If the control volume is occupied by one phase, the properties of that phase are used in the
calculations but if a control volume is not entirely occupied by one phase, mixture properties are used while solving governing equations 2.12 and 2.13. In other words, if the $q^{th}$ fluid's volume fraction in the cell is denoted as $\alpha_q$ then the following three conditions are possible:

- $\alpha_q = 0$: the cell is empty (of the $q^{th}$ fluid)
- $\alpha_q = 1$: the cell is full (of the $q^{th}$ fluid)
- $0 < \alpha_q < 1$: the cell contains the interface between the $q^{th}$ fluid and one or more other fluids

Based on the local value of $\alpha_q$, the appropriate properties and variables will be assigned to each control volume within the domain. The properties appearing in equations 2.12 and 2.13 are related to the volume fraction of the $q^{th}$ phase as follows:

\[
\rho = \sum_{q=1}^{n} \alpha_q \rho_q
\]  

(2.14)

\[
\mu = \sum_{q=1}^{n} \frac{\alpha_q \rho_q \mu_q}{\alpha_q \rho_q}
\]  

(2.15)

For two phase flow, equation 2.14 becomes,

\[
\rho = \alpha_2 \rho_2 + (1 - \alpha_2)\rho_1
\]  

(2.16)

The volume fraction of each fluid, $\alpha_q$, is calculated by tracking the interface between different phases throughout the solution domain. Tracking of the interfaces between $n$ different phases is accomplished by solving continuity equations for $n-1$ phases. For the $q^{th}$ phase, this equation has the following form:

\[
\frac{1}{\rho_q} \frac{\partial}{\partial t} \left( \alpha_q \rho_q \right) + \nabla \cdot \left( \alpha_q \rho_q \bar{v}_q \right) = \frac{1}{\rho_q} S \alpha_q + \frac{1}{\rho_q} \sum_{p=1}^{n} (\dot{m}_{pq} - \dot{m}_{qp})
\]  

(2.16)

where $\dot{m}_{pq}$ is the mass transfer from phase $q$ to phase $p$ and $\dot{m}_{qp}$ is the mass transfer from phase $p$ to phase $q$. By default, the source term on the right-hand side of Eq. 2.6, $S \alpha_q$, is zero, but can be specified a constant or user-defined mass source for each phase. The volume fraction equation will not be solved for the primary phase; the primary-phase volume fraction will be computed based on the following constraint:
\[ \sum_{q=1}^{n} \alpha_q = 1 \]  

(2.17)

The volume fraction equation may be solved either through implicit or explicit time discretization.

**The Implicit Scheme**

When the implicit scheme is used for time discretization, the standard finite-difference interpolation schemes, QUICK, Second Order Upwind and First Order Upwind, and the Modified HRIC schemes, are used to obtain the face fluxes for all cells, including those near the interface.

\[
\frac{\alpha_{q,f}^{n+1} \rho_{q,f}^{n+1} - \alpha_{q,f}^{n} \rho_{q,f}^{n}}{\nabla t} V + \sum_f \left( \rho_{q,f}^{n+1} U_f^{n+1} \alpha_{q,f}^{n+1} \right) = \left[ S_{\alpha_q} + \sum_{p=1}^{n} (\dot{m}_{pq} - \dot{m}_{qp}) \right] V
\]

(2.18)

Since this equation requires the volume fraction values at the current time step (rather than at the previous step, as for the explicit scheme), a standard scalar transport equation is solved iteratively for each of the secondary-phase volume fractions at each time step.

The implicit scheme can be used for both time-dependent and steady-state calculations.

**The Explicit Scheme**

In the explicit approach, the standard finite-difference interpolation schemes are applied to the volume fraction values that were computed at the previous time step.

\[
\frac{\alpha_{q,f}^{n+1} \rho_{q,f}^{n+1} - \alpha_{q,f}^{n} \rho_{q,f}^{n}}{\nabla t} V + \sum_f \left( \rho_{q,f}^{n} U_f^{n} \alpha_{q,f}^{n} \right) = \left[ \sum_{p=1}^{n} (\dot{m}_{pq} - \dot{m}_{qp}) + S_{\alpha_q} \right] V
\]

(2.19)

Where;

- \( n+1 \) : index for new (current) time step
- \( n \) : index for previous time step
- \( \alpha_{q,f} \) : face value of the \( q^{th} \) volume fraction, computed from the first-or second-order upwind, QUICK, modified HRIC, or CICSAM scheme
\[ V \]: volume of cell, infinitesimal control volume
\[ U_f \]: volume flux through the face, based on normal velocity

This formulation does not require iterative solution of the transport equation during each time step, as is needed for the implicit scheme. When the explicit scheme is used, a time-dependent solution must be computed.

When the explicit scheme is used for time discretization, the face fluxes can be interpolated either using interface reconstruction or using a finite volume discretization scheme. The reconstruction based schemes available in FLUENT are Geo-Reconstruct and Donor-Acceptor. The discretization schemes available with explicit scheme for VOF are First Order Upwind, Second Order Upwind, CICSAM, Modified HRIC, and QUICK.

The volume of fluid method is computationally very expensive. This is the main disadvantage of this method. So, it is very difficult to use this method for two-phase flows. Therefore, this method is not very effective in the modeling of large scale systems, however, it can be helpful for understanding of the local phenomena of immiscible and dispersed multi-phase flows.

Considering all advantages and disadvantages of the Eulerian-Eulerian, the Eulerian-Lagrangian multi-phase approaches and volume of fluid (VOF) approach such as ability to model two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain and accuracy, computation time, etc for two-phase flow in a heat exchanger the volume of fluid (VOF) method is the best choice for the modeling.

References:

CHAPTER THREE

Numerical Simulation of Two Phase Flow in Plate Heat Exchanger

3.1 Flow Patterns

Various flow structures are observed in flow boiling, and those are defined as two-phase flow patterns with identifying characteristics. Flow patterns of vertical and horizontal flow generally differ, as flow in horizontal pipes is influenced by the effect of gravity, which acts to stratify the liquid to the bottom and the gas/vapour to the top of the channel. Refrigerant evaporators usually have vertically orientated channels, where two-phase liquid and vapour refrigerant flows in a co-current upward manner.

Four flow patterns are usually identified for vertical up-flows in conventional tubes: bubble, slug (plug), churn, and annular (McQuillan and Whalley, 1985). Figure 3.1 gives the flow patterns in an uniformly heated vertical tube with a low heat flux, with associated wall and fluid temperature variations and heat transfer regions. Churn flow is not present in Figure 3.1, but it can be identified as a highly disturbed flow pattern between slug and annular flow. Flow patterns in horizontal flow will be somewhat different from those shown in this figure. Taitel and Dukler (1976) specified five flow regimes in horizontal two-phase flow, those are smooth stratified, wavy stratified, intermittent (slug and plug), annular and dispersed bubble. It should be pointed out that a two-phase flow pattern is a subjective observation, and there is no general assessing method which identifies and describes flow patterns precisely.

One simple method of representing flow pattern transitions that occur at particular local conditions is by the form of a two dimensional flow pattern map. Respective flow patterns are represented as areas on a graph, with the coordinates being most often chosen as superficial phase velocities, \( j_g \) and \( j_f \), or other generalized parameters containing these velocities. Coordinates of flow patterns can be arbitrarily chosen, as pointed out by Taitel et al. (1980). Because there was little theoretical basis for selection of coordinates, their generality and accuracy are limited. Nevertheless, flow pattern maps have long been widely used in the industry, the most recognized ones are probably that proposed by Hewitt and Roberts for vertical flow, and that of Baker for horizontal flow.
The channel geometry of a plate heat exchanger is very different from a straight circular tube; flow patterns identified in conventional tubes may not be entirely applicable. For example, due to the highly three-dimensional flow directions, annular flow is not likely to be present. Investigations on flow patterns in plate heat exchangers have been very few, and limited to adiabatic air-water flows.

Figure 3.1: Flow patterns with associated heat transfer regions in uniformly heated tube with a low heat flux (Collier and Thome, 1994).
3.2 Flow Distribution in Plate Heat Exchanger

The concept of flow distribution, in the case of plate heat exchangers, refers to that between channels and thus applies only to multi-channel units. Although the flow distribution inside an individual channel can be another important technical issue, it is rarely addressed in the open literature. Thermo-hydraulic theories and correlations for PHE’s largely work on the assumption of uniform flow distribution between channels, this is however not the case in practice. It is generally accepted that flow maldistribution can increase the overall pressure drop and decrease the thermal performance. There is, however, no generally accepted quantitative determination method accounting for this effect.

Flow distribution is determined by inlet and exit manifold pressure profiles, for which two factors can have influences: frictional force and momentum change. Frictional force always causes pressure drops, the momentum change may however have two different effects: in the inlet manifold the deceleration of fluid, due to outflow (into channels), results in actually a pressure rise, whereas in the exit manifold the fluid confluence gives pressure drop in the flow direction. The net effect of the two mechanisms depends on many factors, most importantly channel pressure drop characteristics which determines the flow rate of the fluid leaving the manifold and entering the channels, fluid flow rate, and port size which determines the frictional losses in the manifolds. Most industrial PHE units are arranged in single-pass U or Z type flows; effects of friction loss and momentum change on the pressure profile in these two arrangements are schematically illustrated in Figure 3.2. A vertical line between the two pressure profile lines represents the pressure drop. Even distribution of pressure drops means even distribution of flow, since all channels are identical.
The Z-type arrangement is normally expected to give more uniform flow distributions than the U-type (Kakac and Liu, 2002). This was confirmed by Wilkinson (1974) in his test on units of six channels with pressure measurement on each individual channel. The result also indicated that in the U-type arrangement the liquid stream always “favors” channels closer to the inlet nozzle, more so at higher volume flow rates. The author further pointed out that this holds true also for large units.

Bassiouny and Martin (1984 a, 1984 b) carried out a theoretical analysis on the flow distribution and pressure drop in PHE’s, based on the assumption that the friction loss in the manifolds is negligible. The direct result from this assumption is that pressure always rises in the inlet manifold along the flow direction, and consequently the Z-type arrangement always suffers more severe maldistribution than the U-type. This is because, with such an assumption, pressure profiles in the Z-type arrangement always tend to diverge, while in a U-type they tend to remain relatively parallel (see Figure 3.2). While this assumption remains itself highly arguable, Rao and Das (2004) concluded using this same model that maldistribution is more severe in the Z-type compared with the U type. They conducted tests of overall pressure drop.
with the two arrangements, but no obvious difference of the overall pressure drop at the two conditions could be seen.

3.3 Literature review

There have been a number of investigations based on simulations and experimental investigation of two phase flow in plate heat exchangers.

Shiomi et al. (2004) presented the experimental results of liquid single-phase flow and gas-liquid two phase flow in a channel formed by chevron type plates. The effects of the corrugation angle of the plate were considered. The authors found that the stratified flow and dispersed flow were observed in horizontal condition and only dispersed flow was observed in the vertical condition.

Lozano et al. (2008) analyzed the flow distribution inside one channel of a plate heat exchanger by creating a 3D model without considering the heat transfer. They found that the flow was not uniform and preferentially moved along the lateral extremes of the plates.

Dovic et al. (2002) investigated the flow mechanism in cross-corrugated channels with chevron angles 28º and 61º using a number of dye-based visualization tests in the Reynolds number range of 0.5 to 300. Visualization tests revealed the presence of two substreams flowing along the furrows on the opposite plates. These substreams interact, and the channel flow pattern is characterized by how these interactions are influenced by the channel geometry.

Nilpueng and Wongwises (2006) experimentally investigated the flow patterns and pressure drop of upward liquid single-phase flow and air–water two-phase flow in sinusoidal wavy channels. Different phase shifts between the side walls of the wavy channel of 0º, 90º and 180º are investigated. The slug flow pattern was only found in the test sections with phase shifts of 0º and 90º. Recirculating gas bubbles were always found in the troughs of the corrugations. The recirculating was higher when the phase shifts were larger.

Vlasogiannis et al. (2002) investigated the flow pattern and heat transfer coefficient of air–water two-phase flow in a plate heat exchanger. In this study, they used the Alfa-Laval P-01 heat exchanger as a test section. The plate having a length of 0.43 m and a width of 0.123 m was used in the experiment. The corrugation form of plate was a herringbone pattern with an angle of 60 relative to the direction of flow. The test runs were performed with direction of downward flow and at superficial liquid velocity ranging between 0.01 and 0.25 m/s and superficial gas velocity ranging between 0.3 and 10 m/s. They reported that the flow regime
with a gas continuous phase covering the core of the channel and liquid flowing in the form of rivulets inside the furrows showed particularly favorable heat transfer characteristics.

Nilpueng and Wongwises (2010) experimentally investigated the flow patterns and pressure drop of air–water two-phase flow in a plate heat exchanger. Different flow pattern was revealed and discussed, such as bubbly flow, bubble recirculation flow, and annular-liquid bridge flow for the vertical upward direction, whereas slug flow, annular-liquid bridge flow/air-alone flow, and annular-liquid bridge flow for the vertical downward direction. They also found that the velocity of water and air has a significant effect on the change in pressure drop and the flow pattern shows little effect on the change in pressure drop. The maximum pressure drop was found in the annular-liquid bridge flow pattern.

Kho and Müller-Steinhagen (1999) investigated the effect of flow distribution inside the channel of a plate heat exchanger with flat plates. Different distributors were tested experimentally and numerically for obtaining a more homogeneous flow distribution and thus reducing fouling. CFD numerical simulations were performed using commercial package ANSYS CFX. The geometry of the channel was modeled with grid of 45,000 elements and the k–epsilon turbulence model was used. The plate wall temperature was specified as boundary condition and flow and heat transfer were simulated for incompressible flow under steady state conditions. The CFD predictions were reasonable agreement with the experimental results of flow visualization and fouling, but discrepancies exist on the location of the main flow recirculation zone.

Grijspeerdt et al. (2003) carried out 2D and 3D CFD simulations of flow between two corrugated plates for investigating flow distribution. The grid contained 33,150 points for 2D and 551,265 points for 3D case. The wall temperature was specified, the Baldwin–Lomax turbulent stress model was used and the software FINE-Turbo (Numeca International) was employed. The authors clear the importance of corrugation geometry and point that CFD could be a valuable tool for optimizing the design of plate heat exchangers for milk processing industry with respect to fouling. However, the model would need to simulate the dynamic protein denaturation and wall adhesion. Moreover, the tested geometry differs from that of a
plate heat exchanger channel where the plates have opposing corrugation orientations that provide a series of contact points between plates.

Fernandes et al. (2005) used the CFD tool POLYFLOW of Fluent Inc. to model stirred yogurt flow between two corrugated plates. A geometrical domain with 173634 elements represented half of the corrugated core of a plate heat exchanger chevron channel since symmetry was assumed for simplification. Non-isothermal flow with non-Newtonian behavior (Herschel–Bulkley rheological model) was considered. The geometrical domain was highly complex due to the multiple contractions and expansions along the channel and the mesh was constituted by tetrahedral, hexahedral and pyramidal elements. CFD predictions and experimental data were in very good agreement.

3.4 Simulation of Plate heat exchanger

In recent time, the highly developed computational fluid dynamics have been considered as an effective tool for evaluating the performance of heat exchangers. CFD is a numerical approach, commonly based on the Finite Volume Method (FVM) to solve the Navier Stokes equations. In the present study, a CFD model was used for simulation of air-water two phase flow in a cross-corrugated plate heat exchanger. A real size geometric representation of the plate heat exchanger was generated, including cross corrugated channels and conduits, for creating a virtual prototype of the exchanger that takes into account non-uniform flow distribution inside the channel. The void fraction and flow distribution in a cross-corrugated channel of a plate heat exchanger was investigated numerically.

3.5 The ANSYS fluent model

ANSYS Fluent contains broad physical modeling capabilities for the modeling of flow, heat transfer, mass transfer, reaction and turbulence for industrial applications. These applications can be air flow over an aircraft wing, combustion, multi-phase flow reactors such as bubble column reactors, semiconductors, etc. It is possible to model one phase as well as multi-phase flows with ANSYS Fluent. So, ANSYS Fluent is capable of modeling three-phase trickle bed reactors as multi-phase flow reactors with all phenomena which may occur inside these reactors.
such as chemical reactions, heat transfer, mass transfer, evaporation, condensation, etc. Three
different multi-phase approaches are available in ANSYS Fluent which are: Volume of fluid
(VOF) model, Eulerian-Lagrangian model and Eulerian-Eulerian model. For the modeling of
two phase flow in a plate heat exchanger, we use the volume of fluid approach. More details
about different multi-phase approaches and their applications can be found in chapter two.

3.5.1 Transport equations in ANSYS Fluent

In this part, multi-phase flow equations solved by ANSYS Fluent are discussed. The equations
are discussed for a general case of \( n \) phase flow.

3.5.2 Conservation of mass

The conservation of mass in the Eulerian-Eulerian multi-phase approach for each phase \( q \) is:

\[
\frac{1}{\rho_{rq}} \frac{\partial}{\partial t} (\alpha_q \rho_q \bar{v}_q) + \nabla \cdot (\alpha_q \rho_q \bar{v}_q) = \frac{1}{\rho_{rq}} \sum_{p=1}^{n} (\dot{m}_{pq} - \dot{m}_{qp})
\]

(3.1)

In the equation 3.1, \( \rho_{rq} \) is the phase reference density or the volume averaged density of the \( q^{th} \)
phase in the solution domain, \( \alpha_q \) is the volume fraction of phase \( q \), \( \rho_q \) is the density of phase
\( q \), \( \bar{v}_q \) is the velocity of phase \( q \), \( t \) is the time, \( \dot{m}_{pq} \) is the mass transfer from the \( p^{th} \) to the \( q^{th} \)
phase, \( \dot{m}_{qp} \) is the mass transfer from the \( q^{th} \) to the \( p^{th} \) phase and \( n \) is the number of phases.

3.5.3 Conservation of momentum

The conservation of momentum in the Eulerian-Eulerian multi-phase approach for each phase
\( q \) is:

\[
\frac{\partial}{\partial t} (\alpha_q \rho_q \bar{v}_q) + \nabla \cdot (\alpha_q \rho_q \bar{v}_q \bar{v}_q) = -\alpha_q \nabla P + \nabla \cdot \bar{\tau}_q + \alpha_q \rho_q \ddot{g} + \sum_{p=1}^{n} \left( K_{pq} (\dot{\phi}_{pq} - \dot{\phi}_{pq}) + \dot{m}_{pq} \dot{\phi}_{pq} - \dot{m}_{qp} \dot{\phi}_{qp} + (\ddot{F}_q + \ddot{F}_{lift,q} + \ddot{F}_{vm,q}) \right)
\]

(3.2)

In the equation 3.2, \( P \) is the pressure, \( \bar{\tau}_q \) is the stress strain tensor of phase \( q \), \( \ddot{g} \) is the gravity
vector, \( K_{pq} \) is the momentum exchange coefficient between two fluids or a fluid and solid, \( \bar{\bar{\nu}}_{pq} \)
is the interphase velocity, $\dot{m}_{pq}$ is the mass transfer from the $p^{th}$ to the $q^{th}$ phase, $\dot{m}_{qp}$ is the mass transfer from the $q^{th}$ to the $p^{th}$ phase, $\vec{F}_q^p$ includes all external body forces except gravity, $\vec{F}_{\text{lift} q}$ is the lift force and $\vec{F}_{\text{vm,q}}$ is the virtual mass force.

### 3.5.4 Conservation of energy

The conservation of energy in the Eulerian-Eulerian multi-phase model for the $q^{th}$ phase is:

$$\frac{\partial}{\partial t} (\alpha_q \rho_q \vec{v}_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q h_q) = -\alpha_q \frac{\partial P_q}{\partial t} + \vec{t}_q \cdot \nabla \vec{v}_q - \nabla \cdot \vec{q}_q + S_q + \sum_{p=1}^{n} (Q_{pq} + \dot{m}_{pq} h_{pq} - \dot{m}_{qp} h_{pq})$$

(3.3)

In the equation 3.3, $h_q$ is the specific enthalpy of the $q^{th}$ phase, $\vec{q}_q$ is the heat flux, $S_q$ is a source term which includes sources of enthalpy (for example chemical reaction), $Q_{pq}$ is the heat transfer between the phases $p$ and $q$ and $h_{pq}$ is the interphase enthalpy.

### 3.5.5 Volume fraction equation

As mentioned before, when we consider multi-phase flows as interpenetrating continua (the Eulerian-Eulerian approach) we introduce phase volume fractions, denoted here by $\vec{q}$. Volume fractions represent the space occupied by each phase. The laws of conservation of mass and momentum are satisfied by each phase individually. The volume fraction equation for a $n$ phase flow can be written as:

$$\sum_{q=1}^{n} \alpha_q = 1$$

(3.4)

### 3.6 Geometry generation

The geometry considered consists of the cross-corrugated channel and inlet and outlet ports.

The geometry considered consists of the cross-corrugated channel and inlet and outlet ports. The geometry of plate heat exchanger for this simulation is 64 mm in width, 166 mm in height and a corrugation of 1.5 mm depth at 5.6 mm pitch with inclination angle 60º. Figure 1 shows the computational domain for the single-channel plate heat exchanger with inlet and outlet pipe.
Three-dimensional numerical simulations of the fluid flow in the cross-corrugated channel was carried out by the computer code FLUENT 14.5 based on the finite volume method.

The CFD simulations are performed in two steps: the construction of the computational domain and mesh generation, then the boundary conditions setting. First, a CAD software is used for the construction of the geometry and mesh generation is implemented in the preprocessor software ANSYS ICEM CFD. The main part of the computational domain is the cross-corrugated section which is symmetric with respect to the center plane.

### 3.7. Meshing of Fluid Domain

Meshing of the fluid domain is conducted in ANSYS ICEM CFD. The fluid section or the entire fluid geometries are prepared in popular CAD systems and then converted into ANSYS native Parasolid format. The imported surface geometry is usually grouped into subsets as showing in figure 3.3 for application of desired mesh attributes and to facilitate application of local boundary conditions during the pre-processing stage. The surface connectivity is then redefined once again within ICEM CFD by creating new topology and filtering unnecessary curves and points which otherwise would lead to an unnecessarily large mesh. Depending on the complexity of the plate pattern, each fluid volume can be composed of thousands of complex surfaces.

The grid used for this simulation is an unstructured mesh. Tetrahedral elements are created for computational grid to describe the complexity of the cross corrugated passage as also shown in figure 3.4 and figure 3.5. To reduce the computational expense, three different grid systems have been compared and checked the quality to optimize the mesh size. It was found that the optimum size has a similarity to other reported literatures (Tsai et. al., 2009; Li et. al, 2013.). The used mesh contains approximately 0.7 million elements.

### 3.8 Boundary and operating conditions

The second step was the establishment of the boundary conditions and material properties. The boundary condition at the inlet is the mass flow rate and at the outlet is pressure outlet. All the walls were modeled as no slip wall boundary conditions. The fluid domains were modeled with the properties of air and water as working fluid. The initial condition was air filled computational domain and no water inside the channel.
3.9 CFD solution procedure

The fluid domains were modeled with the properties of air and water as working fluid. The simulation was solved using the laminar flow model and multiphase VOF model with implicit scheme. Simulations were performed with decreasing size grids. The simulations were carried in a Intel® core™ i7 3.40GHz computer with 32GB RAM.

Figure 3.3: Computational domain for CFD modelling.

Fig. 3.4: Computational mesh for CFD modelling.
3.10 Results and Discussion

The detailed results from the CFD model allow the analysis of velocity and flow distribution inside the plate heat exchanger. High velocity regions and stagnation areas could be observed. The results presented here consist of the void fraction and velocity distribution in different time. The effective way to visualize the void fraction and velocity distribution is to consider a mid-plan on plate heat exchanger. The initial boundary condition was the water filled computational domain and no air inside the channel. Figure 3.6 and figure 3.7 show the change of void fraction with respect to time for quality 0.1.
Figure 3.6: Mid-plane void fraction at time 0.39s, 0.48s and 0.60s.

Figure 3.7: Mid-plane void fraction at time 0.72s, 0.75s and 0.78s.
Figure 3.8: Mid-plane velocity distribution at time 0.78s. “●” are contact points.

Numerical results for the velocity vectors around the contact points are shown in figure 3.8. The main flow direction was upward. It can be seen that the velocity is higher at right side portion of channel than left portion. The inclination angle of the chevron is 60° from the main flow direction for present study. The chevron angle is reversed on adjacent plates so that when the plates are brazed together, the corrugations provide numerous contact points. It is obviously that the flow is separated into two streams and passes the contact point. The flow of air-water mixture follows the furrows of the corrugation of upper and lower plates. That is, some velocity vectors are 60° to the right from the main flow direction, but some are in the direction of 60° to the left. The one stream shears with the other stream separated by the neighboring contact points. A considerable mixing occurs and changes the flow direction. Also a lot of secondary flow appears around the contact point. The simulations can already yield useful results, but should be extended to study the fouling phenomenon in more detail.

The typical flow pattern predicted by CFD simulations for the cross-corrugated plate heat exchanger for vapor quality 0.1 is shown in figure 3.9. The simulation proceeds along the flow direction until fully developed situations are achieved. The velocity vectors at the horizontal
mid-section of the channel are shown in figure 3.9. The dark colored vectors are of negligible magnitude, representing the points of contact. Therefore, the dark color vectors are of negligible magnitude, while lighter colored vectors predominantly decide the flow direction. From the figure, it is seen that the velocity vectors describe a zig-zag pattern and fluid flow is mainly aligned along the main flow direction. The flow visualization shows that the high velocity region appears in the upper portion of the channel.

![Velocity Distribution](image)

Fig. 3.9. Velocity distribution at horizontal mid-section of the channel.

The velocity vectors of the inlet port and outlet port at vertical mid-section plane are shown in figure 3.10 and 3.11. It can be seen that fluid enters at low velocity from inlet pipe to the channel and a negligible flow appears in other portion. It is also found that the flow velocity is high at exit of channel to outlet pipe and flow impinged on the outlet pipe wall which led to a significant pressure drop.

As shown in figure 3.12, the fluid near the furrow in the upper plate flowed along the direction of the upper furrow and the fluid near the furrow in the lower plate flowed along the direction of the lower furrow.
Numerical result for the velocity vectors around the contact points are shown in Figure 3.13. The inclination angle of the chevron is 60° from the main flow direction for present study. The chevron angle is reversed on adjacent plates so that when the plates are brazed together, the corrugations provide numerous contact points. It is obviously that the flow is separated into
two streams and passes the contact point. The flow of air-water mixture follows the furrows of the corrugation of upper and lower plates. That is, some velocity vectors are 60° to the right from the main flow direction, but some are in the direction of 60° to the left. The one stream shears with the other stream separated by the neighboring contact points. A considerable mixing occurs and changes the flow direction. In addition, a lot of secondary flow appears around the contact point.

![Image of velocity distribution](image)

Fig. 3.13. Velocity distribution around the contact points. “●” are contact points.

### 3.11 Conclusions

The hydrodynamic characteristics and flow distribution in two cross-corrugated channels have been investigated numerically. The main advantages of the CFD model of the PHE are the detailed void fraction and velocity distributions obtained and the fact that it is not necessary to collect extensive experimental data to adjust the model parameters. On the other hand, the computational time required is a limitation for modeling plate heat exchangers with a large number of plates or more complex geometries using CFD. The 3D velocity fields have been obtained through numerical simulation. It is found that the flow around the contact points is separated in two streams and a considerable mixing occurs. Resorting to the results of velocities field it was possible to conclude that a laminar flow occurs in the present operation conditions; the existence of corrugations in the plates confers to velocity a sinusoidal behaviour in the main
flow direction. More advanced development in computer technology will be helpful for further investigations by CFD.

References


CHAPTER FOUR

Local Condensation Heat Transfer of Refrigerant R1234ze(E) in a Plate Heat Exchanger

4.1 Introduction

In today’s highly industrialized world, energy is one of the major concerns. With rapid consumption of fossil fuels, saving energy has become an attractive topic for the researchers. Recently, a marked improvement in heat exchanger technology has been observed. With improved technology vivid energy savings are now possible with efficient natural refrigerants in compact heat exchangers, used in various applications (Khan et al, 2014).

Plate heat exchangers (PHEs) are widely used in a variety of industrial applications. Some of the major applications are in dairy, process, paper/pulp, refrigeration, heating, ventilating, and air-conditioning industries. Several features of PHEs make them more suitable than other types of heat exchangers. Generally, they are characterized by larger heat transfer area to volume ratio, lighter weight, design flexibility, high thermal effectiveness, hence are suitable for energy and space saving. Their design flexibility provides an advantage in varying heat transfer area by easily adding or removing plates without disturbing the piping connections.

Plate heat exchangers have clear advantage over shell and tube type heat exchanger due to their compact size and high thermal effectiveness. Being compact in nature, the PHEs have better heat transfer characteristics, however, may have higher pressure drop. Therefore, for wider engineering applications, experimental data are required for both heat transfer and pressure drop characteristics of the plate heat exchangers (Khan et al, 2012).

4.2 Literature Review for Local Condensation Heat Transfer

In view of the scarcity in the two-phase heat transfer data for PHEs, (Yan et al. 1999 a,b). carried out experiments to investigate the condensation, evaporation heat transfer and pressure drop of refrigerant R134a in a vertical plate heat exchanger. They experimentally measured the condensation heat transfer coefficients and frictional pressure drops for R-134a in a vertical PHE. The authors found that the evaporation heat transfer for R134a flowing in the PHE was much better than that in circular tubes, particularly in high vapor quality convection dominated regime. Both the heat transfer coefficient and pressure drop increased with the imposed heat
flux, refrigerant mass flux and vapor quality. Furthermore, it was noted that at a higher system pressure the heat transfer coefficients were slightly lower. Moreover, the rise in the heat transfer coefficient with the vapor quality was larger than that in the pressure drop.

Pelletier and Palm (1997) investigated and compared R22 and hydrocarbons in a small plate heat exchanger as evaporator. The authors concluded that the three correlations tested for pool boiling may in most cases be useful to estimate boiling heat transfer coefficients while the other correlations over-predict them.

Hsieh and Lin (2002) investigated the saturated and sub-cooled flow boiling heat transfer and associated pressure drop for R410A in the PHE. They concluded that the saturated boiling heat transfer coefficient and pressure drop in the PHE increased almost linearly with the imposed heat flux and the mass flux effects on the heat transfer coefficient were significant at a high imposed heat flux.

Longo and Gasparella (2007 a,b,c) presented experimental heat transfer and pressure drop characteristics of some Hydro-floro-carbons (HFCs) in a brazed plate heat exchanger. They reported that heat transfer and pressure drop to be significantly affected by heat flux, exit vapor quality and refrigerant properties, while effect of saturation temperature was reported to be insignificant.

Vakili-Farahani et al. (2014 a,b) examined the local heat transfer and pressure drop of R245fa within a PHE channel created by two plates that were electrically heated. The plates were pressed together by two PVC plates to stabilize and insulate the test section. Six windows along the length of the PHE were machined into the PVC plates allowing the outer surface temperature and the local HTC to be measured with an IR-camera. The author reported that the two phase flow behavior in PHEs is similar to pipe flow. They also observed that the flow distributions at the PHE inlet and outlet have a significant effect on the overall thermal and hydraulic performance.

From the review of above literatures, it clearly reveals that only few studies of heat transfer characteristics for condensation of different refrigerants have been investigated in plate heat exchanger. However, no studies have been conducted for measuring the local condensation heat transfer characteristics of R1234ze(E) on a chevron type plate heat exchanger. In this study, the characteristics of local heat transfer during condensation for refrigerant R1234ze (E) flowing in a vertical chevron type plate heat exchanger were investigated experimentally.
4.3 Data reduction for Condensation heat transfer

A data reduction analysis is needed in the present measurement to deduce the local heat transfer rate between the refrigerant flow and the water flow in the test section.

The thermocouples, located along the three lines (left, right and center line of the test plate as Fig. 4.3), are used to measure the local heat flux. Assuming that local heat fluxes \( q_x \) can be estimated from one-dimensional, steady-state heat conduction, \( q_x \) can be expressed by Equation (4.1):

\[
q_x = \lambda \frac{T_{ref} - T_{water}}{l_1}
\]  

(4.1)

where \( \lambda \) is the thermal conductivity of stainless steel (SUS303), \( l_1 \) is the distance between each thermocouple of water side and refrigerant side, \( T_{ref} \) and \( T_{water} \) are refrigerant and water side local temperatures respectively.

The wall temperature of the working fluid side \( (T_{wall,x}) \) is calculated by Equation (4.2):

\[
T_{wall,x} = T_{ref} + \frac{q_x l_2}{\lambda}
\]

(4.2)

Where, \( l_2 \) is the distance between the thermocouple and the wall surface.

The local heat transfer coefficient \( (\alpha_x) \) is calculated by Equation (4.3):

\[
\alpha_x = \frac{q_x}{T_{wall,x} - T_{sat}} = \frac{q_x}{\Delta T_{sat}}
\]

(4.3)

Where, \( \Delta T_{sat} \) is the wall superheat and \( T_{sat} \) is the bulk temperature of the working fluid side, derived using the saturation pressure at the inlet of test section. By the way, since the inside of a plate has pressure drop, it along the flow should be considered in deriving the local saturation pressure and local saturation temperature.

The local specific enthalpy \( (h_i) \) on the test plate is derived using the following method. First, the subcooled working fluid is flown into the preheater inlet. Then, the preheater inlet specific
enthalpy \((h_{pre,in})\) is calculated from the fluid temperature and pressure by using the REFPROP (Lemmon, 2013)

\[
h_{pre,in} = f(T_{pre,in}, P_{pre,in})
\]  

(4.4)

The heat transfer rate \(Q_{w,pre}\) at the preheater is calculated by Equation (4.5):

\[
Q_{w,pre} = \dot{m}_w (h_{pre,out} - h_{pre,in})
\]  

(4.5)

Where, \(\dot{m}_w\) is water mass flow rate and \(h_{pre,out}\) is the outlet specific enthalpy.

The test section inlet specific enthalpy \((h_{t,in})\) is obtained from the heat transfer rate \((Q_{w,pre})\), mass flow rate \((\dot{m}_w)\) for the preheater and heat loss \((Q_{loss})\) from the preheater outlet to the test section inlet:

\[
h_{t,in} = h_{pre,in} - \frac{Q_{w,pre}}{\dot{m}} + Q_{loss}
\]  

(4.6)

The heat loss \((Q_{loss})\) is calculated by Equation (4.7):

\[
Q_{loss} = \frac{2\pi L}{\alpha_r r_1} + \sum_{j=1}^{N} K_j \ln \frac{r_j + 1}{r_j} + \frac{1}{\alpha_c r_{N+1}}
\]  

(4.7)

Where, \(L\) is the distance between the preheater outlet and the test section inlet, \(r\) is radius, \(K\) is the thermal conductivity, \(\alpha_r\) is the single-phase heat transfer coefficient of the refrigerant and \(\alpha_c\) is the natural convection heat transfer coefficient of the air.

Here, the increase in stock of specific enthalpy between neighboring thermocouple is calculated using the heat flow \((Q_l)\) between two neighboring thermocouple and the mass flow rate \((\dot{m})\), similar to the calculation shown in Equation (1). Here, \(Q_l\) is obtained from the local heat flux \((q_l)\) and the heat transfer area. The local specific enthalpy is calculated as follows:

\[
h_l = h_{t,in} \pm \frac{Q_l}{\dot{m}}
\]
\[ h_{i+1} = h_i \pm \frac{Q_{i+1}}{m} \]  

(4.8)

The local vapor quality \( \chi_i \) is determined by specific enthalpy as follows:

\[ \chi_i = \frac{h_i - h_l}{h_v - h_l} \]  

(4.9)

Where, \( h_l \) is the specific enthalpy of the saturated liquid at the saturation pressure of the test section inlet and \( h_v \) is the enthalpy of the saturation vapor, which can be calculated using REFPROP.

The experimental conditions are shown in Table 4.2.

4.4 Experimental apparatus and procedures

Fig. 4.1. shows the schematic diagram of experimental setup for the present study. The experimental setup consists of four independent circuits and a data acquisition system to investigate condensation heat transfer of refrigerant R1234ze (E) in a vertical PHE.
It includes a refrigerant loop, one water loop (for pre-heater) and two cold water–brain loops (one for after condenser and another for the test section). Refrigerant R1234ze (E) is circulated in the refrigerant loop. To obtain various test conditions of R1234ze (E) (including the imposed refrigerant mass flux, system pressure and inlet vapor quality) in the test section, it is required to control the temperature and flow rate in the other three loops. The pre-heater circuit provides the desired vapor quality at the inlet of the test section. The heating circuit provides the heating for the test section.

4.4.1 Refrigerant flow loop

The refrigerant loop contains a refrigerant pump, an accumulator, a refrigerant mass flow meter, a pre-heater, a test section "the plate heat exchanger", an after condenser, a sub-cooler, a receiver, a filter/dryer and three sight glasses. The refrigerant pump is driven by a DC motor that is controlled by a variable DC output motor controller. The variation of the liquid R-1234ze(E) flow rate was controlled by a rotational DC motor through the change of the DC current. The refrigerant flow rate was measured by a Coriolis mass flow meter (MASSMAX® 3300) installed between the pump and pre-heater with an accuracy of ±0.1%. 

Fig. 4.1. Schematic diagram of experimental setup.
The pre-heater is used to evaporate the refrigerant to a specified vapor quality at the test section inlet by transferring heat from the hot water to R1234ze (E). Note that the amount of heat transfer from the hot water to the refrigerant in the pre-heater is calculated from the energy balance in the water flow. Meanwhile, an after condenser and sub-cooler were used to condense the refrigerant vapor from the test section by a cold water-brine system to avoid cavitation at the pump inlet. The pressure of the refrigerant loop can be controlled by varying the temperature and flow rate of the water-brine mixture in the after condenser and sub-cooler. After condensed, the liquid refrigerant flows back to the receiver.

Mixing chambers are installed before and after the test section and also before the pre-heater. Absolute pressure and temperature were measured at the mixing chamber. All water and refrigeration temperatures are measured by K-type thermocouples which are calibrated.

4.4.2. Plate heat exchanger: the test section

The test section used in this study, as shown in Fig. 4.2 was formed by eight stainless steel plates. Among two of them were processed chevron plates and a refrigerant channel was formed by the plates (shown in Fig. 3). Other two flat plates beside the chevron plates are set for heat transfer measurements. Other four plates consist on cooling water flow channel.

The main characteristics of the test section are illustrated in Table 4.1. Each of the processed chevron plates is 5 mm thick. The chevron grooves have the corrugation pitch of 5.6 mm, the corrugation depth of 1.5 mm, and the chevron angle of 60°. The total height of the test section is 186 mm and the total width is 84 mm. The distance between the inlet to outlet port centers is 136 mm. Each connection port has a diameter of 18.5 mm. Two processed chevron plate form the refrigerant channel of which height and width are 117.5 mm and 64 mm, respectively.
Fig. 4.2. The test section.

Fig. 4.3. Location of thermocouples in test section.
A pressure transducer is installed at the test section inlet and a differential pressure transducer is used to measure the overall pressure drop. In order to measure local heat transfer characteristics, total 60 thermocouples were set at middle of flow direction and also in the right and left sides of plates in test section. Inside the flat plates beside the chevron plates for heat transfer measurements, five straight grooves of 1.7 mm depth were machined perpendicular to the flow direction with the positions of 23.2, 45.6, 68, 90.4 and 112.8 mm apart from the inlet of refrigerant channel on each side of each flat plate. The thermocouple tubes were inserted in each groove with sheathed three K-type thermocouples in each and soldered to minimum the contact resistance (shown in Fig. 4.3).

Fig. 4.3a. Details of thermocouple position in test section (enlarged view of area A shown in Fig. 4.3).

### 4.4.3. Water loop for pre-heater

A brazed plate heat exchanger was used as the pre-heater. The liquid R1234ze (E) flowing in the inner passage was heated to achieve set condition at test section inlet by the hot water flow in the outer passages. The pre-heater and the connection pipe between the test section and the pre-heater were all thermally insulated. The hot water loop designed for the pre-heater consists of a 34L hot water thermostatic bath with 6.0 kW heaters. Then, a water pump with an inverter is used to drive the hot water at a specified water flow rate to the pre-heater. Similarly, a bypass water valve is also used to adjust the flow rate.
4.4.4. Water-brine loop for test section

The water-brine loop in the system designed for circulating cold water-brine through the test section contains a constant temperature water bath with a 3.6 kW heater and an air cooled refrigeration unit of 1.5 kW cooling capacity intending to accurately control the water-brine temperature. A pump with an inverter is used to drive the cold water-brine to the plate heat exchanger with a specified water-brine flow rate. Another by-pass valve can also be used to adjust the water-brine flow rate.

4.4.5 Water-brine loop for after condenser

The water-brine loop designed for condensing the R1234ze (E) vapor contains another constant temperature bath with a water cooled refrigeration system. The water-brine at a specified flow rate is driven by a pump to the condenser as well as to the sub-cooler. A by-pass valve is also provided to adjust the water-brine flow rate.

4.4.6. Data acquisition

The data acquisition unit includes a Keithley 3700 system switch data logger combined with a personal computer. The data logger was used to record the temperature and voltage data. The water flowmeter and differential pressure transducer need a power supply as a driver to output an electric current. The LabVIEW® 2012 interface was used to connect the data logger to computer, allowing all measured data to transmit from the data logger to the computer and then to be analyzed by a spreadsheet.

Table 4.1: Geometrical characteristics of the PHE.

<table>
<thead>
<tr>
<th>Characteristics</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid flow plate length (mm)</td>
<td>117.5</td>
</tr>
<tr>
<td>Plate width (mm)</td>
<td>64</td>
</tr>
<tr>
<td>Area of the plate (m²)</td>
<td>0.0075</td>
</tr>
<tr>
<td>Corrugation type</td>
<td>Herringbone</td>
</tr>
<tr>
<td>Angle of the corrugation (°)</td>
<td>60</td>
</tr>
<tr>
<td>Corrugation pitch (mm)</td>
<td>5.6</td>
</tr>
<tr>
<td>Number of plates</td>
<td>08</td>
</tr>
<tr>
<td>Channels on refrigerant side</td>
<td>1</td>
</tr>
<tr>
<td>Channels on water side</td>
<td>2</td>
</tr>
</tbody>
</table>
4.4.7 Experimental procedures

In each test the system pressure is maintained at a specified level by adjusting the water-brine temperature and its flow rate. The vapor quality of R1234ze (E) at the test section inlet can be kept at the desired value by adjusting the temperature and flow rate of the hot water loop for the pre-heater. Finally, the heat transfer rate between the counter flow channels in the test section can be varied by changing the temperature and flow rate in the water loop for the test section. Any change of the system variables will lead to fluctuations in the temperature and pressure of the flow. It takes about 20-100 minute to reach a statistically steady state condition. Then the data acquisition unit is initiated to scan all the data channels for ten times in 40s. The mean values of the data for each channel are obtained to calculate the heat transfer coefficient. Additionally, the flow rate of water in the test section should be high enough to have turbulent flow in the water side so that the associated heat transfer in it is high enough for balancing the condensation heat transfer in the refrigerant side.

Table 4.2: Experimental conditions for condensation of R1234ze(E).

<table>
<thead>
<tr>
<th>Working fluid</th>
<th>R1234ze(E)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flux, G [kg/m²s]</td>
<td>10 - 50</td>
</tr>
<tr>
<td>Saturation temperature, $T_{sat}$ [ºC]</td>
<td>35 - 40</td>
</tr>
<tr>
<td>Wetness of test section inlet, $x_{in}$ [-]</td>
<td>0 - 0.1</td>
</tr>
<tr>
<td>Wetness of test section outlet, $x_{out}$ [-]</td>
<td>0.9 - 1</td>
</tr>
</tbody>
</table>

4.5 Results and discussions:

The local condensation heat transfer coefficient of R1234ze(E) versus wetness at various mass fluxes were measured.

Fig. 4.4 shows the plots of the measured local condensation heat transfer coefficient versus wetness at mass flux 10 kg/m²s and maximum heat flux of 20 kW/m²s. These data indicate that at a fixed mass flux the condensation local heat transfer coefficient decreases gradually with an increase in wetness of the refrigerant in the plate heat exchanger. This decrease is rather significant. For instance, the condensation heat transfer coefficient at the wetness of 0.14 is higher than that at 0.81.

This obviously results from the simple fact that in the higher vapor quality regime the vapor amounts are much high, which causes to dry out in that region, meanwhile the liquid film is
relatively very thin. This, in turn, reduces the heat transfer from the plate surface to the refrigerant. A close inspection of the data in Fig. 4.4 reveals that heat transfer coefficient of front side of plate heat exchanger is lower than the back side of plate heat exchanger.

Fig. 4.4. Variation of local condensation heat transfer coefficient with wetness at mass flux \( G = 10 \text{ kg/m}^2\text{s} \).

Fig. 4.5 shows variation heat flux with distance from inlet to outlet of plate heat exchanger at mass flux 10 kg/m\(^2\)s and saturation temperature of 36°C. All local condensation heat fluxes tend to decrease in the direction from the inlet to the outlet of the test section. It may also be observed that heat flux follows similar trend with local heat transfer coefficient and wetness at the fixed mass flux of 10 kg/m\(^2\)s.
Fig. 4.5. Variation of heat flux as a function of distance from inlet of the test section at mass flux $G = 10 \text{ kg/m}^2\text{s}$.

Fig. 4.6 shows the variation of the wall temperature of the test section in the direction from the inlet to the outlet of the test section. In this case, the wall temperatures of the plate heat exchanger decrease in the direction from the inlet to the outlet of the test section which indicates the condensation behavior.

Fig. 4.6. Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 10 \text{ kg/m}^2\text{s}$.
Next, results are presented to illustrate the effect of the imposed mass flux 20 kg/m²s on the local condensation heat transfer coefficient. Fig. 4.7 shows the local condensation heat transfer coefficients on the refrigerant side with wetness at mass flux 20 kg/m²s at saturation temperature 38 °C and maximum heat flux 20 kW/m²s.

In the case of the mass flux G = 20 kg/m², the local condensation heat transfer coefficient of R1234ze(E) decreased with an increase of the wetness. It may also be observed from Fig. 4.7 that local heat transfer coefficient decreases moderately with an increase in wetness between 0.29 and 0.53. In addition, the condensation heat transfer coefficients show weak sensitivity at high wetness (> 0.65). Note that the local condensation heat transfer coefficients at mass flux 20 kg/m²s are larger than that at mass flux 10 kg/m²s.

The results in figure 4.8 shows the effect of local heat flux on the condensation of R1234ze(E) inside a plate heat exchanger. It may be observed that the local heat fluxes decrease toward the distance to downstream of condensation process.

![Graph of local condensation heat transfer coefficient with wetness at mass flux G = 20 kg/m²s.](image)

Fig. 4.7. Variation of local condensation heat transfer coefficient with wetness at mass flux G = 20 kg/m²s.

Fig. 4.9 shows the wall temperature distributions with the distance from inlet to outlet of plate heat exchanger. The temperatures decrease gradually along the downstream of test section which is the direction of condensation.
Fig. 4.8. Variation of heat flux as a function of distance from inlet of the test section at mass flux \( G = 20 \text{ kg/m}^2\text{s} \).  

Fig. 4.9. Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux \( G = 20 \text{ kg/m}^2\text{s} \).  

Fig. 4.10a and 4.10b show the local condensation heat transfer coefficient as a function of wetness for mass flux 50 \text{ kg/m}^2\text{s} at saturation temperature 38 °C. The fig. 4.10a presents the
effect of local heat transfer coefficient on wetness from 0.1 to 0.6 at mass flux 50 kg/m²s and fig. 10b shows local heat transfer coefficient as function of wetness from 0.5 to 1.0 for mass flux 50 kg/m²s. The results of local heat transfer coefficient for mass flux 50 kg/m²s wasn’t possible to obtain in one time between wetness range from 0 to 1.0 because of high mass flow rate.

Fig. 4.10a. Variation of local condensation heat transfer coefficient on wetness, [0.12 to 0.50] at mass flux G = 50 kg/m²s.

Results from fig. 4.10a and 4.10b show that the local heat transfer coefficient show weak sensitivity with increase of wetness. The heat transfer coefficient at high wetness (> 0.8).
Fig. 4.10b. Variation of local condensation heat transfer coefficient on wetness, [0.56 to 0.91] at mass flux $G = 50 \text{ kg/m}^2\text{s}$.

Fig. 11a and 11b depict the local heat flux as a function of distance from inlet to outlet of the test section.

Fig. 11a. Variation of heat flux as a function of distance from inlet of the test section for wetness from 0.12 to 0.50 at mass flux $G = 50 \text{ kg/m}^2\text{s}$.
Fig. 4.11b. Variation of heat flux as a function of distance from inlet of the test section for wetness from 0.56 to 0.91 at mass flux $G = 50 \text{ kg/m}^2\text{s}$.

Fig. 4.12a and 4.12b show the wall temperature distributions in test section at mass flux 50 kg/m$^2$s. During condensation, the refrigerant flows downward and cooling water flows upward as counter flow. Results show that the wall temperature decreases linearly with increase of the distance along to direction from the inlet of plate heat exchanger to outlet of the exchanger. Also these data indicate that there a notable temperature differences between the refrigerant side and water side of the test section surface.
G = 50 [kg/m$^2$s]

$q_{\text{max}} = 34$ [kW/m$^2$]

Fig. 4.12a. Variation of wall temperature distribution as a function of distance from inlet of the test section for vapor quality from 0.12 to 0.50 at mass flux $G = 50$ kg/m$^2$s.

G = 50 [kg/m$^2$s]

$q_{\text{max}} = 29$ [kW/m$^2$]

Fig. 4.12b. Variation of wall temperature distribution as a function of distance from inlet of the test section for vapor quality from 0.56 to 0.91 at mass flux $G = 50$ kg/m$^2$s.
4.6 Conclusions

The local condensation heat transfer characteristics of R1234ze(E) in the plate heat exchanger were investigated experimentally at different mass flux conditions. The experiments were conducted by varying the mass fluxes. The local condensation heat transfer coefficient decreased with increasing of wetness with different values in horizontal direction. At low refrigerant mass flux (G =10, 20 kg/m²s), the heat transfer coefficients are not dependent on mass flux and probably condensation is controlled by gravity. For higher refrigerant mass flux (Gr =50 kg/m²s) the heat transfer coefficients depend on mass flux and forced convection condensation occurs. The condensation heat flux decreases with increase of distance along the direction from inlet to outlet of plate heat exchanger which is the direction of condensation progress. The local wall temperature distribution decreases with increase of distance along the downstream.

References


O. Pelletier, B. Palm, (1997), Boiling of Hydrocarbons in Small Plate Heat Exchangers. International Institute of Refrigeration Commission B1, College Park, USA.


CHAPTER FIVE

Local Evaporation Heat Transfer of Refrigerant R1234ze(E) and R32 in Plate Heat Exchanger

5.1 Introduction

The flow boiling heat transfer and pressure drop of refrigerant and hydrocarbon fluid in Plate Heat exchangers have been investigated for various industrial applications. During the evaporation process, the heat transfer mechanisms are strongly dependent on the flow regimes. Saturated refrigerant boiling inside Plate Heat exchanger is governed by two heat transfer mechanisms, one is nucleate boiling and another is convective boiling. Nucleate boiling is mainly dominated by the heat flux, whereas vapour quality and mass flux control convective boiling.

5.2 Literature Review for Local Evaporation Heat Transfer

Han et al. (2003) investigated the evaporative heat transfer in brazed plate heat exchangers with R-410A and R-22 refrigerants. They used plate heat exchangers with different chevron angles of 45º, 35º, and 20º. The authors found that the heat transfer coefficient increases with increasing mass flux and vapour quality and with decreasing evaporation temperature and chevron angle. The authors therefore suggested that convective boiling was the main boiling mechanism.

Djordjevic and Kabelac (2008) studied flow boiling of R-134a and ammonia in a plate heat exchanger. Heat transfer coefficient was reported to be a strong function of vapor quality, which also increased with an increase in heat flux and mass flux. Experimental data were presented, however, a correlation for heat transfer coefficient based on their study was not reported. Nevertheless, ammonia was reported to have better heat transfer characteristics compared to R-134a.

Arima et al. (2010) provided data on evaporation of ammonia in a vertical flat plate heat exchanger and developed a heat transfer correlation. Local heat transfer coefficient is reported to increase with increasing vapor quality. However, a sharp decrease has been reported beyond 0.8 vapor quality, perhaps due to dry-out phenomenon. Furthermore, they investigated effects of saturation pressure, mass and heat flux on the boiling heat transfer characteristics of the heat exchanger and provided flow visualization of ammonia flow within the plate heat exchanger.
It has been shown that for a given saturation pressure and mass flux, the heat transfer coefficient decreased boiling mechanism.

Amalfi et al. (2015a, b) provided a comprehensive literature survey of flow boiling heat transfer and two-phase frictional pressure drops mechanisms within chevron PHEs. In this study, the prediction methods available in the open literature from 1981 until 2014 were detailed and a consolidated experimental database from 13 research studies were culled that included 3601 data points. The authors carried out a sensitivity analysis to investigate the effect of plate geometrical parameters on the thermal and hydraulic performance, and an extensive statistical comparison of all available two-phase prediction methods against the above mentioned databank was also furnished. Using dimensional analysis coupled with the multi-regression technique, new methods for predicting local HTC and local frictional pressure gradient within plate heat exchangers were developed and shown to work better than the available methods.

Table 5.1: The research has been done with carbon dioxide, as reported in open literature:

<table>
<thead>
<tr>
<th>Researcher</th>
<th>Type of heat exchanger</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>Robinson and Groll (1998)</td>
<td>Single straight tubes with outside fins of constant thickness and spacing</td>
<td>CO₂ and R-22 two-phase behavior was compared and CO₂ heat exchanger dimension ratios were proposed.</td>
</tr>
<tr>
<td>Pertersen et al. (1998)</td>
<td>Small diameter mechanically expanded round-tube heat exchanger and brazed micro-channel type heat exchanger</td>
<td>Discussed advantages and disadvantages of how various exchangers may be used for CO₂ refrigeration cycles in automotive and residential air conditioning.</td>
</tr>
<tr>
<td>Kim and Kim (2002)</td>
<td>Counter flow type heat exchanger with concentric dual tubes using R774/134a and R744/290 mixtures</td>
<td>Experimentation and simulation were performed showing that as mass fractions of R744 increased, cooling capacity and compressor power increased but COP decreased.</td>
</tr>
<tr>
<td>Rauch et al. (2005)</td>
<td>Double pipe heat exchangers</td>
<td>Numerical and experimental results of trans-critical CO₂ cycle discovering optimal gas cooler pressure values in the trans-critical CO₂ cycle</td>
</tr>
<tr>
<td>Authors</td>
<td>Type of Heat Exchanger</td>
<td>Description</td>
</tr>
<tr>
<td>------------------</td>
<td>------------------------------------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Rigola et al.</td>
<td>Double pipe counter flow</td>
<td>Numerical analysis was performed</td>
</tr>
<tr>
<td>(2005)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Brown et al.</td>
<td>Evaporator and condenser heat exchanger types</td>
<td>Visual Basic programs were used to simulate single and two-phase CO₂ vapor compression cycles</td>
</tr>
<tr>
<td>(2005)</td>
<td>not specified in simulation program</td>
<td></td>
</tr>
<tr>
<td>Hwang et al.</td>
<td>Fin and tube as well as microchannel heat</td>
<td>Behavior of CO₂ were explored using condensation and evaporation properties. Airside pressure drops were compared according to different air frontal velocities. Various overall heat transfer coefficients were measured.</td>
</tr>
<tr>
<td>(2005)</td>
<td>exchanger</td>
<td></td>
</tr>
<tr>
<td>Hayes et al.</td>
<td>Chevron type three brazed plate heat exchanger</td>
<td>Experimental investigation of carbon dioxide condensation were performed. showing comparisons of dimensionless heat transfer parameters of two phase flow in each plate.</td>
</tr>
<tr>
<td>(2011)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### 5.3 Data Reduction

For calculation purposes, the plate area was divided into several segments along the vertical axis, on the borders of which thermocouples were installed on the cooling liquid side. The energy balance for each of the segments was checked and the amount of heat transferred calculated. Since the heat transfer coefficients were calculated separately for each segment of the plate, they represent local values. For the local heat transfer coefficient, test sections in local heat flux $q_x$, assuming 1-dimensional steady-state heat conduction and calculated from the following equation.

$$ q_x = \lambda \frac{T_{\text{ref}} - T_{\text{water}}}{l_1} $$  

(5.1)

Where $T_{\text{ref}}$ is the refrigerant-side temperature, $T_{\text{water}}$ is water-side temperature, $l_1$ is distance between each thermocouple of water-side and refrigerant-side, $\lambda$ is the thermal conductivity of the plate material (SUS303).
Wall temperature of the working fluid side (refrigerant flow) from outside, $T_{wall,x}$ is calculated by equation (5.2):

$$T_{wall,x} = T_{ref} - \frac{q_x l_2}{\lambda}$$  \hspace{1cm} (5.2)

Where, $l_2$ is the difference between thermo-couple and the wall surface of refrigerant side in evaporation.

Local heat transfer coefficient $\alpha_x$ is defined by using the value obtained by the equations (5.1) and (5.2).

$$\alpha_x = \frac{q_x}{T_{wall,x} - T_{sat}}$$  \hspace{1cm} (5.3)

Where $T_{sat}$ is the saturation temperature of the working fluid obtained from saturation pressure at inlet of the test section.

For evaporation, the local specific enthalpy $h_t$ on the test plate is derived using the following method. First, the subcooled working fluid is flown into the preheater inlet. Then, the preheater inlet specific enthalpy $h_{pre,in}$ is calculated from the fluid temperature and pressure by using the REFPROP.

$$h_{pre,in} = f(T_{pre,in},P_{pre,in})$$  \hspace{1cm} (5.4)

Assume in the preheater, the heat exchange takes place under the fully insulated condition. The heat transfer rate $Q_{w,pre}$ is calculated by Equation (5.5):

$$Q_{w,pre} = \dot{m}_w (h_{w,pre,out} - h_{w,pre,in})$$  \hspace{1cm} (5.5)

Where, $\dot{m}_w$ is water mass flow rate, $h_{w,pre,out}$ is the preheater outlet enthalpy.

The test section inlet specific enthalpy $h_{t,in}$ obtained from the heat transfer rate $Q_{w,pre}$ mass flow rate $\dot{m}_w$ for the preheater and heat loss $Q_{loss}$ from the preheater outlet to the test section inlet:

$$h_{t,in} = h_{pre,in} - \frac{Q_{w,pre}}{\dot{m}} + Q_{loss}$$  \hspace{1cm} (5.6)

The heat loss $Q_{loss}$ is obtained from the equation (5.7):
\[ Q_{loss} = \frac{2\pi L}{\alpha_r r_1^2} + \frac{1}{\alpha_c r_{N+1}} + \sum_{j=1}^{N} \frac{r_{j+1}}{r_j} \ln \frac{r_{j+1}}{r_j} \]  

(5.7)

Where, \( L \) is the distance between the preheater outlet and the test section inlet, \( r \) is radius, \( K \) is the thermal conductivity, \( \alpha_r \) is the thermal conductivity of the refrigerant, and \( \alpha_c \) is the natural convection heat transfer coefficient of the air.

Here, the increase in stock of specific enthalpy between neighboring thermocouple is calculated using the heat flow \( Q_i \) between two neighboring thermocouple and the mass flow rate \( \dot{m} \), similar to the calculation shown in Equation (5.1). Here, \( Q_i \) is obtained from the local heat flux \( q_i \) and the heat transfer area. The local specific enthalpy is calculated as follows:

\[ h_i = h_{i,in} \pm \frac{Q_i}{m} \]  

(5.8)

\[ h_{i+1} = h_i \pm \frac{Q_{i+1}}{m} \]  

(5.9)

\[ x_i = \frac{h_i - \Delta h}{h_v - \Delta h} \]  

(5.10)

The local vapor quality \( x_i \) is calculated from above equation (5.10). \( h_i \) is the specific enthalpy of the saturated liquid at the saturation pressure of the test section inlet and \( h_v \) is the enthalpy of the saturated vapor, which can be calculated using REFPROP.

5.4 Experimental Setup

The schematic diagram of the experimental setup was the same as figure 4.1. It consists of a test section of plate heat exchanger and four independent loops to investigate evaporation heat transfer of refrigerant R1234ze (E). The test section consists of eight stainless steel plates including the test plate heat exchangers. The main geometrical characteristics are illustrated in table (5.2). The detail description of experimental setup including four independent loops and data acquisition system have been discussed in chapter four.
Table 5.2: Geometrical characteristics of the PHE.

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid flow plate length (mm)</td>
<td>117.5</td>
</tr>
<tr>
<td>Plate width (mm)</td>
<td>64</td>
</tr>
<tr>
<td>Area of the plate (m²)</td>
<td>0.0075</td>
</tr>
<tr>
<td>Corrugation type</td>
<td>Herringbone</td>
</tr>
<tr>
<td>Angle of the corrugation (°)</td>
<td>60</td>
</tr>
<tr>
<td>Corrugation pitch (mm)</td>
<td>5.6</td>
</tr>
<tr>
<td>Corrugation height, b (mm)</td>
<td>1.5</td>
</tr>
<tr>
<td>Hydraulic diameter, (d_h = 2b) (mm)</td>
<td>3</td>
</tr>
<tr>
<td>Number of plates</td>
<td>08</td>
</tr>
<tr>
<td>Channels on refrigerant side</td>
<td>1</td>
</tr>
<tr>
<td>Channels on water side</td>
<td>2</td>
</tr>
</tbody>
</table>

5.5 Experimental procedures

In each test the system pressure is maintained at a specified level by adjusting the water-brine temperature and its flow rate. The vapor quality of R1234ze (E) at the test section inlet can be kept at the desired value by adjusting the temperature and flow rate of the hot water loop for the pre-heater. Finally, the heat transfer rate between the counter flow channels in the test section can be varied by changing the temperature and flow rate in the water loop for the test section. Any change of the system variables will lead to fluctuations in the temperature and pressure of the flow. It takes about 20-100 minute to reach a statistically steady state condition. Then the data acquisition unit is initiated to scan all the data channels for ten times in 40s. The mean values of the data for each channel are obtained to calculate the heat transfer coefficient. Additionally, the flow rate of water in the test section should be high enough to have turbulent flow in the water side so that the associated heat transfer in it is high enough for balancing the evaporation heat transfer in the refrigerant side.

Table 5.3: Experimental conditions for evaporation of R1234ze(e).

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>R1234ze(E)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flux, (G) [kg/m²s]</td>
<td>10 - 50</td>
</tr>
<tr>
<td>Saturation temperature, (T_{sat}) [ºC]</td>
<td>5 - 10</td>
</tr>
<tr>
<td>Vapor quality of test section inlet, (x_{in}) [-]</td>
<td>0 - 0.1</td>
</tr>
<tr>
<td>Vapor quality of test section outlet, (x_{out}) [-]</td>
<td>0.9 - 1</td>
</tr>
</tbody>
</table>
5.6 Result and Discussions for refrigerant R1234ze(E):

The local evaporation heat transfer coefficient of R1234ze(E) versus vapor quality at various mass fluxes were measured.

Fig. 5.1 shows experimental local evaporation heat transfer coefficient versus vapor quality at mass flux 10 kg/m²s and maximum heat flux of 21 kW/m². Fig. 5.1 shows the plots of the measured local evaporation heat transfer coefficient versus vapor quality at mass flux 10 kg/m²s and maximum heat flux of 21 kW/m². These data indicate that at a fixed mass flux the local evaporation heat transfer coefficient decreases gradually with an increase in vapor quality of the refrigerant in the plate heat exchanger. This decrease is rather significant. For instance, the evaporation heat transfer coefficient at the vapor quality of 0.25 is lower than that at 0.80. The results indicate that from vapor quality 0.7, the heat transfer coefficient changes very little.

This obviously results from the simple fact that in the higher vapor quality regime the vapor amounts are much high, which causes to dry out in that region, meanwhile the liquid film is relatively very thin. This, in turn, reduces the heat transfer from the plate surface to the refrigerant. A close inspection of the data in Fig. 5.1 reveals that heat transfer coefficient of front side of plate heat exchanger is lower than the back side of plate heat exchanger.

The other results are shown from figure 5.2 to figure 5.9.
Figure: 5.1. Variation of local boiling heat transfer coefficient with vapor quality at mass flux $G = 10 \text{ kg/m}^2\text{s}$.

Figure: 5.2. Variation of heat flux as a function of distance from inlet of the test section at mass flux $G = 10 \text{ kg/m}^2\text{s}$. 
Figure: 5.3. Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 10 \text{ kg/m}^2\text{s}$.

Figure: 5.4. Variation of local boiling heat transfer coefficient with vapor quality at mass flux $G = 20 \text{ kg/m}^2\text{s}$. 
Figure: 5.5. Variation of heat flux as a function of distance from inlet of the test section at mass flux \( G = 20 \text{ kg/m}^2\text{s} \).

Figure: 5.6. Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux \( G = 20 \text{ kg/m}^2\text{s} \).
Figure: 5.7. Variation of local boiling heat transfer coefficient with vapor quality at mass flux $G = 30 \text{ kg/m}^2\text{s}$.

Figure: 5.8. Variation of heat flux as a function of distance from inlet of the test section at mass flux $G = 30 \text{ kg/m}^2\text{s}$.
The experimental results of heat transfer characteristics of R32 shown from figure 5.10 to figure 5.18. Experimental data of refrigerant R32 for local heat transfer coefficient is presented to show the effects of mass flux between 10 and 40 kg/m²s, heat flux between 27 and 40 kg/m² and vapour quality from 0.0 to 1.0. The results show that for the different mass flux conditions, the local heat transfer coefficient is highly dependent on different location of plate heat exchanger.
Figure: 5.10. Variation of local boiling heat transfer coefficient of R32 with vapor quality at mass flux $G = 10 \text{ kg/m}^2\text{s}$.

Figure: 5.11. Variation of heat flux of R32 as a function of distance from inlet of the test section at mass flux $G = 10 \text{ kg/m}^2\text{s}$.
Figure: 5.12. Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 10 \text{ kg/m}^2\text{s}$.

Figure: 5.13. Variation of local boiling heat transfer coefficient of R32 with wetness at mass flux $G = 20 \text{ kg/m}^2\text{s}$. 
Figure: 5.14. Variation of heat flux of R32 as a function of distance from inlet of the test section at mass flux $G = 20 \text{ kg/m}^2\text{s}$.

Figure: 5.15. Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 20 \text{ kg/m}^2\text{s}$. 
Figure: 5.16. Variation of local boiling heat transfer coefficient of R32 with vapor quality at mass flux $G = 40 \text{ kg/m}^2\text{s}$.

Figure: 5.17. Variation of heat flux of R32 as a function of distance from inlet of the test section at mass flux $G = 40 \text{ kg/m}^2\text{s}$. 
Figure: 5.18. Variation of wall temperature distribution as a function of distance from inlet of the test section at mass flux $G = 40 \text{ kg/m}^2\text{s}$.

5.8 Conclusions

The local evaporation heat transfer characteristics of R1234ze(E) and R32 in the plate heat exchanger were investigated experimentally at different mass flux conditions. The experiments were conducted by varying the mass fluxes. The results for R1234ze(E) indicate that at a fixed mass flux the local evaporation heat transfer coefficient decreases gradually with an increase in vapor quality of the refrigerant in the plate heat exchanger. Experimental data of refrigerant R32 for local heat transfer coefficient is presented to show the effects of mass flux between 10 and 40 kg/m$^2$s, heat flux between 27 kg/m$^2$ and 40 kg/m$^2$ and vapour quality from 0.0 to 1.0. The results show that for the different mass flux conditions, the local heat transfer coefficient is highly dependent on different location of plate heat exchanger.

References


CHAPTER SIX

Conclusion and Recommendations

6.1 Conclusions

All forgoing investigation can be summarized in following general conclusions, these are,

**For CFD Simulation:**
The hydrodynamic characteristics and flow distribution in two cross-corrugated channels have been investigated numerically.

- The main advantages of the CFD model of the PHE are the detailed void fraction and velocity distributions obtained and the fact that it is not necessary to collect extensive experimental data to adjust the model parameters.
- The 3D velocity fields have been obtained through numerical simulation. It is found that the flow around the contact points is separated in two streams and a considerable mixing occurs.
- Resorting to the results of velocities field it was possible to conclude that a laminar flow occurs in the present operation conditions; the existence of corrugations in the plates confers to velocity a sinusoidal behaviour in the main flow direction.

**For Condensation and Evaporation heat transfer:**
The local condensation heat transfer characteristics of R1234ze(E) in the plate heat exchanger were investigated experimentally at different mass flux conditions. The experiments were conducted by varying the mass fluxes.

- The local condensation heat transfer coefficient decreased with increasing of wetness with different values in horizontal direction. At low refrigerant mass flux \( G = 10, 20 \text{ kg/m}^2\text{s} \), the heat transfer coefficients are not dependent on mass flux and probably condensation is controlled by gravity.
- For higher refrigerant mass flux \( \text{Gr} = 50 \text{ kg/m}^2\text{s} \) the heat transfer coefficients depend on mass flux and forced convection condensation occurs.
- The condensation heat flux decreases with increase of distance along the direction from inlet to outlet of plate heat exchanger which is the direction of condensation progress.
• The local wall temperature distribution decreases with increase of distance along the downstream.

Experimental results of refrigerant R32 for local heat transfer coefficient show that for the different mass flux conditions, the local heat transfer coefficient is highly dependent on different location of plate heat exchanger.

6.2 Recommendations:

More advanced development in computer technology will be helpful for further investigations by CFD. It is very good method to obtain some detail information about two-phase flow in flow field which cannot be achieved by experiment only. It is possible to obtain details of two phase flow inside plate heat exchanger by numerical simulations. Some evidence has not been revealed yet and become a good challenge in future research.