CFD Simulation of R134a and R410A Two-Phase Flow in the Vertical Header of Microchannel Heat Exchanger

Yang Zou
University of Illinois at Urbana-Champaign, yangzou1@illinois.edu

Pega Hrnjak
pega@illinois.edu

Follow this and additional works at: http://docs.lib.purdue.edu/iracc

http://docs.lib.purdue.edu/iracc/1719
CFD Simulation of R134a and R410A Two-Phase Flow in the Vertical Header of Microchannel Heat Exchanger

Yang ZOU¹, Pega HRNJAK*¹,²

¹University of Illinois at Urbana-Champaign, Department of Mechanical Science and Engineering, Urbana, IL, USA

²Creative Thermal Solutions Urbana, IL, USA

Contact Information (1-217-244-3677, yangzou1@illinois.edu, pega@illinois.edu)

* Corresponding Author

ABSTRACT

This paper studies refrigerant maldistribution in the vertical header of microchannel heat exchanger through both experiment and CFD simulation. In the experiment, the two-phase R134a or R410A is circulated into the transparent vertical header through multi-parallel microchannel tubes in the bottom pass and exits through multi-parallel microchannel tubes in the top pass representing the flow in the heat pump mode of a reversible system. The experimental results are compared with CFD simulation. The Eulerian-Eulerian model in the commercial software Fluent is used to conduct simulation. Qualitative agreement between experiment and CFD is obtained. Both experiment and CFD show that the distribution is worse with respect to inlet quality due to the flow pattern in the header. With CFD simulation, pressure drop and void fraction information in the vertical header is obtained.

1. INTRODUCTION

Microchannel heat exchangers (MCHX) have come to the frontier of automotive, residential, and commercial air conditioning applications for its advantages in compactness, higher heat transfer, and possible charge reduction. However, refrigerant maldistribution in the header of MCHX creates unwanted superheated region, where the heat transfer is much lower than the two-phase region due to the lower heat transfer coefficient of superheated vapor and less temperature difference between refrigerant and air, so it may reduce MCHX capacity by up to 30%, e.g. as in Byun and Kim (2011) and Zou et al. (2014).

Most studies on refrigerant maldistribution were investigated experimentally. Fei and Hrnjak (2002), Vist (2003), Bowers et al. (2006), and Jin (2007) studied the two-phase flow in the horizontal headers, which usually appeared in the indoor MCHX. Watanabe et al. (1995), Cho and Cho (2004), Lee (2009), Byun and Kim (2011), and Zou and Hrnjak (2013a, 2013b, 2014a, 2014b, 2014c) investigated the refrigerant distribution in the vertical headers, which were commonly used in the outdoor MCHX. Among these studies, some derived empirical distribution functions based on experimental results to simulate refrigerant distribution. Vist (2003) applied the results of T-junction studies to develop a quality distribution function at the round tube junction in the horizontal cylindrical header. Jin (2006) proposed a distribution function in the horizontal header (upward flow in the microchannel tubes) by relating the branch tube quality with the ratio of vapor mass flux in the header immediately upstream to total inlet vapor mass flux. Lee (2009) considered the cylindrical vertical header as a series of T-junctions, and predicted the liquid distribution among flat tubes based on the studies of two-phase flow at T-junction. Watanabe et al. (1995) defined the liquid take-off ratio as the ratio of liquid mass flow rate in the branch tube to liquid mass flow rate in the vertical header immediately upstream. In annular flow, the liquid take-off ratio was constant. In froth or slug flow, the liquid take-off ratio was a function of vapor phase Reynolds number and liquid phase Weber number in the header immediately upstream. Vapor was considered as equally distributed among the tubes based on the measurement. Byun
and Kim (2011) applied the approach of Watanabe et al. (1995). They related both vapor and liquid take-off ratio with vapor phase Reynolds number in the vertical header immediately upstream. With the wider range of test conditions and more fluids, Zou and Hrnjak (2013a, 2013b, 2015) found the inlet quality and liquid phase Froude number were also important parameters to liquid take-off ratio. Zou and Hrnjak (2015) generalized R134a and R410A distribution by relating the liquid take-off with the header inlet quality as well as the vapor phase Reynolds number and liquid phase Froude number in the header immediately upstream.

In other studies, numerical methods were applied to study refrigerant distribution. Moura (1995) numerically simulated air-water distribution in a two-pass MCHX with vertical headers based on the two-fluid model. Only qualitative agreement with experiment results was obtained. Tompkins et al. (2002) discretized a header into several control volumes and applied modified separated flow model to simulate distribution. Fei and Hrnjak (2004) conducted CFD simulation of R134a flow in horizontal headers using Eulerian-Eulerian model in the commercial software Fluent. Comparing with experiment data and visualization images, reasonable simulation results were obtained. Ablanque et al. (2010) presented a numerical model, using the results of T-junction studies to simulate the splitting flow phenomenon in the header. The accuracy of this model strongly depended on the selection of the T-junction model. Stevanovic (2012) developed a computational multi-fluid dynamic (CMFD) code based on numerical solving of the mass and momentum balance equations for the flow of each phase and the corresponding closure laws for the calculation of interface transfer of balanced parameters. Huang et al. (2014) proposed a co-simulation approach by combining CFD simulation of the inlet vertical header with a ε-NTU based segmented heat exchanger model. The model is validated against the experimental results. In this study, it is attempted to improve the understanding of two-phase flow in the header through CFD simulation based on experimental results. The upward flow in the intermediate vertical header of an outdoor reversible MCHX is simulated, mimicking the case when the outdoor reversible MCHX functions as evaporator in the heat pump mode.

2. EXPERIMENTAL METHOD

The test loop was constructed to study R410A or R134a distribution in the microchannel heat exchanger, as shown Figure 1. The subcooled liquid refrigerant was pumped into the inlet header. It was assumed that the single-phase subcooled liquid was distributed evenly into the microchannel tubes in the bottom pass, where the refrigerant was heated to desired quality. The two-phase fluid entered into the test header and turned 90° to flow upward in the bottom part. In the upper part of the header, due to maldistribution, different amounts of liquid exited through the microchannel tubes in the top pass. In each exit tube, the refrigerant was heated again to provide equal superheat at the exit. The single phase superheated vapor was then brought to the condenser. Through the receiver and the subcooler, the subcooled liquid was returned to the pump.

![System schematics](image)

**Figure 1:** System schematics
The liquid mass flow rate in each exit tube was obtained by Equation 1.

\[ m_{i,\text{out},i} = m_{\text{out},i} (1 - x_{\text{out},i}) \quad \text{where} \quad x_{\text{out},i} = f(P_{\text{header}}, h_{\text{out},i}) \]  

(1)

The pressure in the header \( P_{\text{header}} \) was estimated as the average of the measured heat exchange inlet and outlet pressures and the outlet enthalpy from the header (i.e., inlet to each exit tube) \( h_{\text{out},i} \) was calculated as in Equation 2.

\[ h_{\text{out},i} = h_{\text{sup},i} - \frac{Q_{\text{out},i}}{m_{\text{out},i}} \quad \text{where} \quad h_{\text{sup},i} = f(P_{\text{sup},i}, T_{\text{sup},i}) \]  

(2)

The liquid mass flow rate were generalized and liquid fraction in Equation 4. Uniform distribution was described as \( LF_i = 0.2 \). An uncertainty propagation analysis carried out in EES (2012). The uncertainty of liquid fraction is usually within 5%.

\[ LF_i = \frac{\sum m_{i,\text{out},i}}{n} \]  

(3)

A high speed camera, Phantom v4.2, was used for visualizing the flow in the transparent header. The exposure time of the camera was 80 μsec. The framing rate was at 2000 frames per second. The resolution was 256x512 pixels. The transparent circular header, made of the PVC tube, had five inlet and five exit microchannel tubes protruded into the ½ depth of header’s inner diameter. The geometries of the transparent header and aluminum microchannel tube are listed in Table 1. The test conditions are shown in Table 2. The inlet mass flux \( G_{in} \), presented in Table 2, is defined by the smallest cross-section area in the header where tube protrusion is presented.

### Table 1: Vertical header and microchannel tube geometries

<table>
<thead>
<tr>
<th>Item</th>
<th>Data</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Header geometry</strong></td>
<td></td>
</tr>
<tr>
<td>Inner diameter</td>
<td>15.44 mm</td>
</tr>
<tr>
<td>Header length</td>
<td>170 mm for 5+5 header; 300mm for 10+10 header</td>
</tr>
<tr>
<td>Tube pitch</td>
<td>13 mm</td>
</tr>
<tr>
<td>Tube protrusion</td>
<td>½ depth and ¾ depth of inner diameter</td>
</tr>
<tr>
<td><strong>Microchannel geometry</strong></td>
<td></td>
</tr>
<tr>
<td>Shape</td>
<td>Rectangular</td>
</tr>
<tr>
<td>Number of ports</td>
<td>17</td>
</tr>
<tr>
<td>Length</td>
<td>0.54 mm</td>
</tr>
<tr>
<td>Width</td>
<td>0.5 mm</td>
</tr>
<tr>
<td>Hydraulic diameter</td>
<td>0.5 mm</td>
</tr>
</tbody>
</table>

### Table 2: Test conditions

<table>
<thead>
<tr>
<th>Item</th>
<th>Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Saturation temperature</td>
<td>5 °C for R410A; 10 °C for R134a</td>
</tr>
<tr>
<td>Inlet quality</td>
<td>0.2 – 0.8</td>
</tr>
<tr>
<td>Inlet mass flow rate</td>
<td>2 – 6 g/s for 5+5 header</td>
</tr>
<tr>
<td>Inlet mass flux</td>
<td>21.80 – 129.00 kg/m²-s</td>
</tr>
</tbody>
</table>

### 3. CFD Model Description
The commercial software Fluent was used to model the two-phase flow in the intermediate vertical header. The model was based on the work of Fei and Hrnjak (2004) which modeled the two-phase flow in the horizontal header with downward round tubes. As suggested by Fei and Hrnjak (2004), due to the complexity of two-phase flow and irregular geometry, the 3-D domain of the round header was modeled in this study, as shown in Figure 2. The Hex/Wedge mesh as shown in Figure 2 was generated, whose size was kept small enough to have more than 2 meshes along the height (tube minor) of the microchannel tube.

Figure 2: System schematics

The steady simulation was conducted with integrated solver and implicit scheme. The Eulerian method in the Eulerian-Eulerian multiphase model was used, which treated both vapor and liquid phases as continuous phases. (Other methods in the Eulerian-Eulerian multiphase model are the Volume of Fluid method and the Mixture method.) Fei and Hrnjak (2004) showed that the Eulerian method would give the best results in simulating the two-phase flow in the header, so it is chosen in this study. The standard k-ε turbulence model was used for each phase because the turbulence transfer among the phases played a dominant role. The vapor flow was considered as the primary continuous phase, while the liquid droplets flow was considered as the secondary phase. Fei and Hrnjak (2004) determined the uniform droplet diameter based on Phase Doppler Particle Anemometry measurement. In this study, the droplet diameter was adjusted so that the simulated distribution results agreed best with the experimental distribution results, i.e. 25μm for R134a and R410A. Between the two phases, the drag force was modeled with symmetric drag coefficient. The Phase Coupled SIMPLE (PC-SIMPLE) algorithm was used for the pressure-velocity coupling. The continuity residual history of Fluent in this study was $10^{-3}$. The stable convergence was observed after 2000 iterations.

4. RESULTS AND DISCUSSION

Figure 3 compares the simulated distribution results with the experimental results for R134a and R410A at $m_{in} = 6.25$ g/s. The darkness of bar color represents different branch tubes, the pale being the lowest exit branch and the dark being the highest exit branch. For both fluids, CFD results show similar trend as the experiment. The best distribution is at $x_{in} = 0.2$. As quality increases, the distribution is worse because the bottom tubes receive less liquid than the top tubes. The main difference between CFD and experiment is the top tube. The liquid fraction of the top tube decreases with respect to inlet quality in experiment while the liquid fraction of the top tube is higher with respect to inlet quality in CFD. However, for the other 4 tubes, the distribution profiles are very similar between experiment and CFD.
Figure 3: Comparison between experiment and CFD distribution profiles of R134a and R410A

Figure 4 and Figure 5 compare CFD liquid contours with the experiment flow visualization. The churn and semi-annular regimes are identified from the visualization for both R410A and R134a. At low inlet quality in Figure 4, it is observed in experiment that the flow pattern is churn flow. Most of the header is occupied by liquid refrigerant with bubbles, but at the top it is almost vapor only. Bubbles stir the liquid though the mean velocity of liquid is upward. It is hard to distinguish the interface of liquid and vapor. They are mixed almost homogeneously. As illustrated in Figure 4(a), the liquid contour of CFD shows similar churn flow pattern and vapor-only region at the top of the header. Both experiment visualization and CFD velocity contour in Figure 4(b) show that there is local vortex between the neighboring 2 microchannel tubes, which helps to mix vapor and liquid uniformly. Therefore, the opportunity of liquid supply to each branch tube is similar, except for the top tube close to the vapor-only region. The distribution is good at low quality.

(a) Liquid volume fraction (compared with experiment visualization)
At high inlet quality in Figure 5, the flow pattern observed in experiment is semi-annular flow. The semi-annular flow is like annular flow, but due to the tubes protrusion, the annulus is not complete. Most volume of the header is taken by vapor, but liquid is present in the form of liquid film along the inner wall of the header. In the top exiting region, vapor with lighter density is much easier to turn and branches out, but liquid with larger density and higher momentum tends to run through the header and bypassed the first few microchannel tubes. As some fluid branches out, the velocity in the header is reduced, and the liquid film starts to separate from the wall at certain height. The flow pattern becomes locally churn flow. Some liquid flows horizontally and leaves through the outlet microchannel tubes. Other liquid falls down through the gap between microchannel tube and round header, so that creates a large vortex in the header. At the top of the header, the momentum is further reduced due to the two-phase fluid branching out. It results in liquid cannot reach the top and the tubes there get very little if any liquid. Thus, the tubes in this small local churn flow region have higher opportunities to receive liquid resulting in bad distribution. CFD liquid contour in Figure 5(a) also illustrates high void fraction in the header and the liquid is present as liquid film. However, unlike the experiment visualization, the liquid film flows all the way to the top header, turns and comes down from the other side of the header, as also shown in the velocity contour in Figure 5(b). It also creates a large vortex in the header, similar to the experiment. However, it results in the liquid exits through the top tube first and then the bottom tubes, so the top tube has highest liquid fraction and it causes different distribution profiles from experiment at high quality as in Figure 3.
With the help of CFD, more information regarding two-phase flow in the vertical header such as pressure drop and void fraction can be obtained, which may be difficult to measure during experiment. Figure 6 presents the locations and notations of pressure drop and void fraction in the following analysis. The pressure or void fraction of each plane is the average of the cross-section area. Figure 7 shows the pressure drop of R134a and R410A in the top exiting region of the vertical header. Zou and Hrnjak (2014b) measured the two-phase pressure drop of R134a in this vertical header, and the experimental results are compared with CFD results in Figure 7(a). The trend of pressure drop along the header is similar between CFD and experiment. Besides, both experiment and CFD show that at low inlet qualities the pressure drop is positive while at high inlet qualities the pressure drop is negative, i.e. it is pressure gain instead of pressure drop at high qualities. Zou and Hrnjak (2014b) showed that such negative overall pressure drop at high qualities was because that the negative momentum pressure drop (due to losing mass and flow decelerating) was more dominant than the friction and gravity pressure drops. Such negative pressure drop in the top exiting region at high inlet qualities is also illustrated in the pressure contour from CFD in Figure 8.
Figure 7: Pressure drop of R134a and R410A in the vertical header

![Pressure drop diagram](image.png)

(a) R134a  
(b) R410A

---

**Figure 8:** CFD pressure contours of R134a at $m_{in}=6.25g/s$ and $x_{in}=0.6$

---

Figure 9 presents the void fraction of R134a and R410A in the vertical header from CFD. Even though the flow visualization is taken during experiment, it is very difficult to quantify void fraction because of the complex flow patterns. This information is added with the help of CFD. It is shown in Figure 9 that even at low inlet quality, at least 80-90% volume of the vertical header is occupied by the vapor. Thus, it may be very difficult to mix vapor and liquid uniformly in the header. To achieve good distribution, venting some vapor out of the header may be a more effective solution, as presented in Tuo and Hrnjak (2011) with the method called flash gas bypass.
This study investigates the two-phase flow of R134a and R410A in the vertical header of microchannel heat exchanger. The CFD simulation is carried out in the commercial software Fluent and the simulation results are compared with the experimental results. The distribution profiles from CFD are very similar to those from experiment except for the top tube. Both CFD and experiment show that as inlet quality increases more liquid exits through the top tubes, and the distribution becomes worse. This is due to the flow pattern in the header. At low inlet qualities, the flow pattern is churn, and the mixing of vapor and liquid is uniform. However, the flow pattern in the header is semi-annular at high inlet qualities. The high speed liquid film would bypass the bottom exit tubes and flow to a higher location, then the liquid is only available for a few tubes at the top. These flow patterns are observed from both CFD and experiment. Besides, the CFD simulation shows that the pressure drop in the top exiting region (top half part) of the header is positive at low quality but negative at high inlet quality. The negative pressure drop in the header may seem counterintuitive, but this is because that the flow decelerates as the two-phase fluid branches out and the negative momentum pressure is dominant at high qualities. Based on the void fraction from CFD simulation results, there is at least 80-90% vapor in the header. It might be very difficult to mix vapor and liquid uniformly, especially at high qualities. Probably some other method (e.g. flash gas bypass method) should be applied to vent out some vapor for improving refrigerant distribution.

5. CONCLUSIONS

This study investigates the two-phase flow of R134a and R410A in the vertical header of microchannel heat exchanger. The CFD simulation is carried out in the commercial software Fluent and the simulation results are compared with the experimental results. The distribution profiles from CFD are very similar to those from experiment except for the top tube. Both CFD and experiment show that as inlet quality increases more liquid exits through the top tubes, and the distribution becomes worse. This is due to the flow pattern in the header. At low inlet qualities, the flow pattern is churn, and the mixing of vapor and liquid is uniform. However, the flow pattern in the header is semi-annular at high inlet qualities. The high speed liquid film would bypass the bottom exit tubes and flow to a higher location, then the liquid is only available for a few tubes at the top. These flow patterns are observed from both CFD and experiment. Besides, the CFD simulation shows that the pressure drop in the top exiting region (top half part) of the header is positive at low quality but negative at high inlet quality. The negative pressure drop in the header may seem counterintuitive, but this is because that the flow decelerates as the two-phase fluid branches out and the negative momentum pressure is dominant at high qualities. Based on the void fraction from CFD simulation results, there is at least 80-90% vapor in the header. It might be very difficult to mix vapor and liquid uniformly, especially at high qualities. Probably some other method (e.g. flash gas bypass method) should be applied to vent out some vapor for improving refrigerant distribution.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$G$</td>
<td>Mass flux</td>
<td>(kg/m²·s⁻¹)</td>
</tr>
<tr>
<td>$i$</td>
<td>Enthalpy</td>
<td>(kJ/kg)</td>
</tr>
<tr>
<td>$LF$</td>
<td>Liquid fraction</td>
<td>(-)</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass flow rate</td>
<td>(g/s)</td>
</tr>
<tr>
<td>$n$</td>
<td>Number of the outlet tubes</td>
<td>(-)</td>
</tr>
<tr>
<td>$P$</td>
<td>Pressure</td>
<td>(kPa)</td>
</tr>
<tr>
<td>$Q$</td>
<td>Power of the heaters</td>
<td>(kW)</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
<td>(K)</td>
</tr>
<tr>
<td>$x$</td>
<td>Quality</td>
<td>(-)</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Void fraction</td>
<td>(-)</td>
</tr>
</tbody>
</table>

Subscripts

- i: Branch number
- in: At the smallest
- out: Out of the header
- l: Liquid
- sup: Superheated
- sub: Subcooled
- v: vapor

REFERENCES


ACKNOWLEDGEMENT

This paper is part of the Ph.D. work of the first author at University of Illinois Urbana-Champaign. The authors would like to thank the support from the sponsors of Air Conditioning and Refrigeration Center at the University of Illinois Urbana-Champaign.