2016

Single Phase Pressure Drop and Flow Distribution in Brazed Plate Heat Exchangers

Wenzhe Li
Air-Conditioning and Refrigeration Center (ACRC), University of Illinois Urbana-Champaign, United States of America,
liwz310@illinois.edu

Pega Hrnjak
pega@illinois.edu

Single Phase Pressure Drop and Flow Distribution in Brazed Plate Heat Exchangers

Wenzhe Li[1], Pega Hrnjak[1, 2]*

[1] Air-Conditioning and Refrigeration Center (ACRC),
University of Illinois Urbana-Champaign,
Urbana, IL, USA
liwz310@illinois.edu

[2] Creative Thermal Solutions In. (CTS),
Urbana, IL, USA
pega@illinois.edu

* Corresponding Author

ABSTRACT

Brazed plate heat exchangers (BPHE) have been widely used in the heating, ventilating, air conditioning, and refrigeration industry, but refrigerant distribution among parallel plate channels is still one of the main issues. Maldistribution of refrigerant among different plate channels is greatly affected by the pressure changes along the inlet and outlet header, and it would generally decrease the performance of BPHE by causing higher pressure drop and poor utilization of heat transfer area.

In this paper, the experimental and simulative methods are used to study the single-phase pressure drop and flow distribution in a U-type BPHE. Experiments are conducted to measure the pressure changes along the inlet and outlet header, as well as the pressure difference through each plate channel. And the CFD tool is used to simulate the flow details in the inlet and out header of the BPHE.

1. INTRODUCTION

Heat exchangers with parallel channel structure like microchannel heat exchangers and plate heat exchangers suffer from the problem of refrigerant maldistribution. According to Tuo and Hrnjak (2013), one of the main reasons for refrigerant maldistribution is the header induced pressure drop. Single-phase flow in header is quite common in the applications of HVAC&R. For example, in the condensers, single-phase flow presents in both inlet and outlet headers. And in the evaporator, the vapor flow happens in the outlet header and if the flash gas is bypassed before the evaporator, the pure liquid will be supplied into the inlet header. Therefore, the study on the features of the single-phase pressure drop in the headers is in a great significance for distribution problem. And certainly, header with vapor flow typically has a greater pressure drop and affects distribution more than that with liquid flow.

Several researchers have studied on the single-phase flow distribution in the plate heat exchangers. Bassiouny and Martin (1984a, 1984b) developed a one-dimensional mathematical model to analyze the single-phase flow distribution and pressure drop for the U-type and Z-type plate heat exchangers respectively. They defined a general characteristic parameter which determines the flow behavior in all heat exchangers. It was also found in their work that the ratio of the cross-sectional areas of the intake and exhaust conduits would affect the uniformity of the flow distribution.

To obtain more detailed flow distribution information, CFD (Computational Fluid Dynamics) tool was applied by Wang et al. (2011). They numerically and experimentally investigated the characteristic of single-phase flow distribution in compact parallel flow heat exchangers and found that the jet flow induced by the sudden expansion of the inlet header could reduce the flow rate in the first several tubes.

An experimental study on the pressure drop in U-type plate heat exchangers with different numbers of plate has been conducted by Bobbili et al. (2006). This work demonstrated that the maldistribution of the flow rate in channels would be more severe for the heat exchangers with more plates.

The objective of this paper is to explore the characteristics of the single-phase pressure drop and flow distribution in a U-type BPHE. There are many methods to measure refrigerant distribution indirectly, such as from infrared image (Li and Hrnjak, 2014) or frost pattern (Song et al., 2002). In this study the channel pressure differences were measured as an indication of refrigerant distribution. The static pressure changes along the inlet and outlet header were also measured. Meanwhile, CFD models of the inlet and outlet header are developed in ANSYS Fluent to reveal the flow details.

2. Experiment

Since the flow distribution is highly depending on the header induced pressure drop, experiments have been conducted to measure the pressure changes along the inlet and outlet header and the pressure difference through each plate channel in a U-type BPHE.

2.1 Experimental Facility

The test system consists of a pressurized tank, a temperature pre-conditioner and a U-type BPHE, as shown in Figure 1. For a U-type BPHE, the fluid enters and exits the heat exchanger at the same side. In the experiment, compressed nitrogen gas is released from the pressurized tank, and passes through the pre-conditioner to equalize its temperature with the ambient temperature. The total mass flow rate is controlled by the regulator and measured by a mass flowmeter. Then the nitrogen gas enters the BPHE from the inlet header at the bottom, flows upward through the plate channels and exits from the outlet header at the top. Two quarter-inch steel tubes with a small hole on the surface at the end are plugged into the inlet and the outlet header as the pressure probes. Two absolute pressure sensors are connected with these two probes to measure the static pressure in the headers. And the pressure difference between the inlet and outlet header or pressure difference through the plate channel is obtained by a pressure transducer. By moving the pressure probes forward and backward, the pressure changes along the inlet and outlet header and the pressure difference through different plate channels are measured.

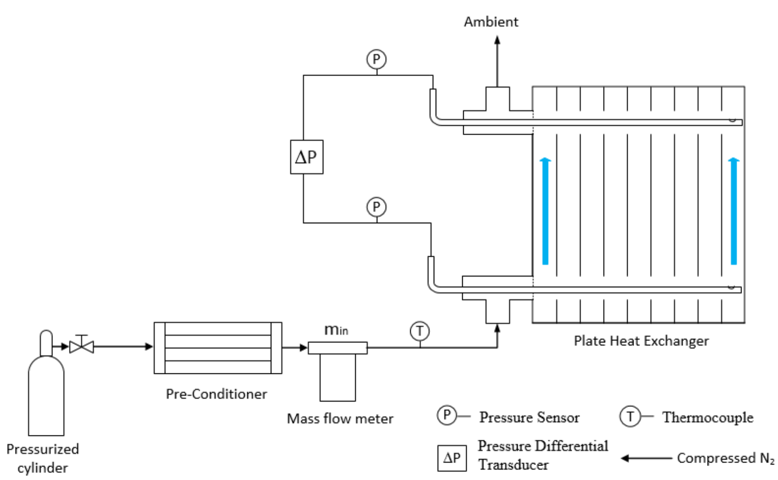
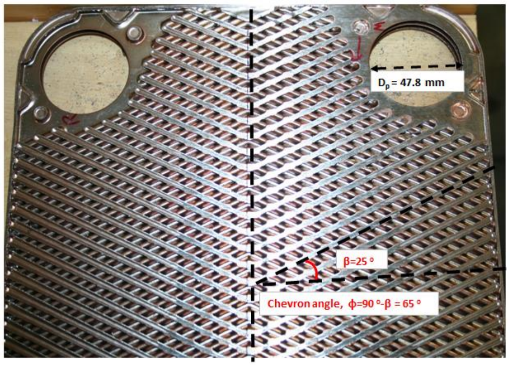


Figure 1: Schematic of experimental facility

The heat exchanger used in this study is a KAORI K205S series BPHE with 54 plate channels for each fluid. A similar model of BPHE (K205 type) was cut to accurately obtain the internal geometric dimensions. Figure 2 and Table 1 show the measured geometric parameters after cutting the heat exchanger.



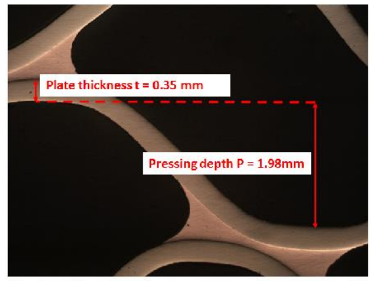


Figure 2: Measured BPHE geometric dimensions (courtesy of CTS)

Table 1: Measured BPHE geometric dimensions (courtesy of CTS)

Parameter	Units	Value
Chevron angle, $\varphi=90^o-\beta$	^{o}	65
Pressing depth, P	mm	1.98
Corrugation pitch, P_c	mm	7.4
Plate thickness, t	mm	0.35
Diameter of the header, D_p	mm	47.8
Diameter of the connection conduit, D_c	mm	26.5
Port length, L_p	mm	456
Total length, L_v	mm	528
Port width, L_h	mm	174
Total width, L_w	mm	246
Heat transfer area/plate, A_p	m^2	0.1099
Number of channels/fluid, N_c	-	54

2.2 Results and Discussions

In the experiments, the total mass flow rate is varying from 10 g/s to 30 g/s and the data is taken at the location of the 1st, 14th, 29th, 42nd and 54th plate channel. Figure 3(a) ~ 3(c) give the static pressure changes along the inlet and outlet header with different total mass flow rates.

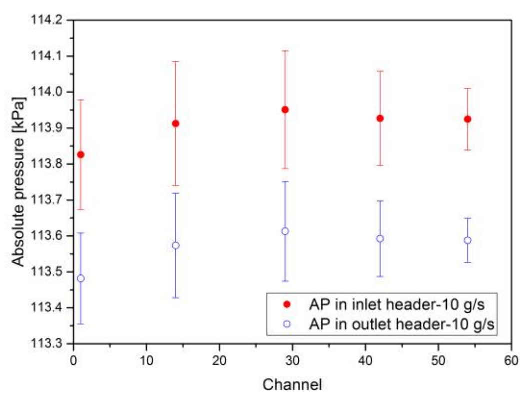


Figure 3(a): Absolute pressure in the headers-10 g/s

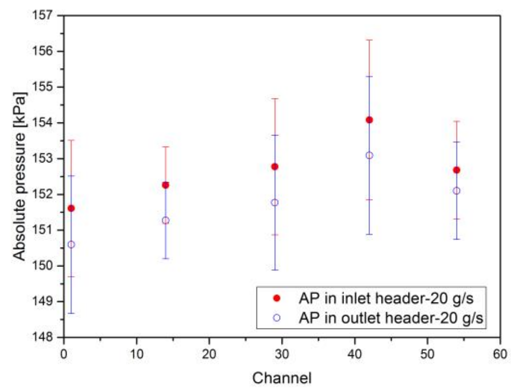


Figure 3(b): Absolute pressure in the headers-20 g/s

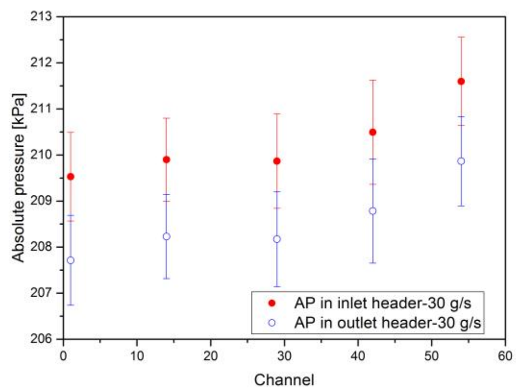


Figure 3(c): Absolute pressure in the headers-30 g/s

As Figure 3(a) ~ 3(c) showed above, for all three total mass flow rates, the absolute pressure generally increases along the flow direction in the inlet header, and decreases along the flow direction in the outlet header. This can be explained by the conversion between the static pressure and the dynamic pressure along the headers. In the inlet header, the working fluid is branching out through the plate channels which leads to the reduction of the mass flow rate of the header flow. With a decreasing mass flow rate, the flow decelerates in the inlet header. Therefore, the dynamic pressure is converted to the static pressure and that gives a pressure rise along the inlet header. The physical process in the outlet header is opposite. With fluid continuously joining in the header flow, the flow actually accelerates in the flowing direction. And, that converts the static pressure to the dynamic pressure and causes pressure drop along the outlet header.

Following the analysis above, ideally without any pressure loss, the increase of the static pressure in the inlet header would be equal to the decrease of the static pressure in the outlet header which means the pressure curves of the inlet and outlet header would be parallel and the pressure difference for each plate channel is identical. In this case, the flow distribution will be uniform.

The measured pressure differences between the inlet and outlet header for several plate channels with different total mass flow rates are shown in Figure 4.

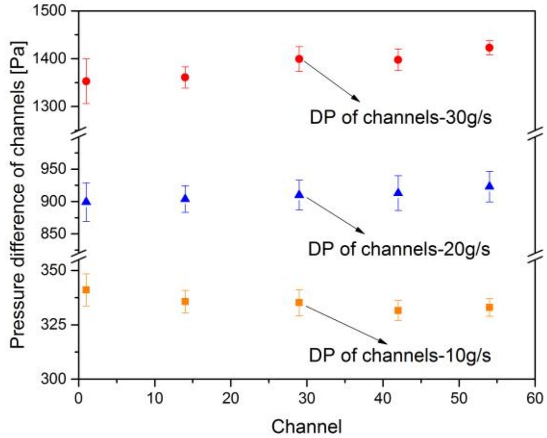


Figure 4: Pressure difference between inlet and outlet for a few channels

Data in Figure 4 indicate that the pressure difference of each plate channel is not identical and consequently the mass flow rate distribution is not uniform.

One influencing factor is the local pressure loss (the friction loss might be negligible for the header flow). The local pressure losses always decrease the static pressure along the flowing direction. Thus, in the inlet header, the local pressure losses would decrease the pressure rise along the header. But in the outlet header, such pressure losses will increase the pressure drop along the flowing direction. Therefore, the pressure rise in the inlet header would be smaller than the pressure drop in the outlet header. This makes the pressure difference of the first several plate channels is greater than that of the last several plate channels which means channel mass flow rate is decreasing from the channels near the inlet and outlet ports to the channels away from the inlet and outlet ports.

However, in Figure 4, only the flow with a relatively low total mass flow rate (10 g/s) shows the decreasing trend of the channel pressure difference as discussed above. For the flow with relatively higher total mass flow rates (20, 30 g/s), the pressure difference of channels increases from the channels near the inlet and outlet ports to the channels away from the inlet and outlet ports. To explain this phenomenon, CFD tool is applied.

3. CFD MODELING

CFD models have been developed for the inlet and outlet header to explore the flow details.

3.1 Establishment of CFD Models

Because the flow in the header is essential to the mass flow distribution, and to avoid the massive computation of the flow in the plate channels, the CFD models are only developed for the inlet and outlet header. In order to accurately simulate the local pressure losses, the small round edge part of each plate, shown in Figure 5, is also included in the header geometry.

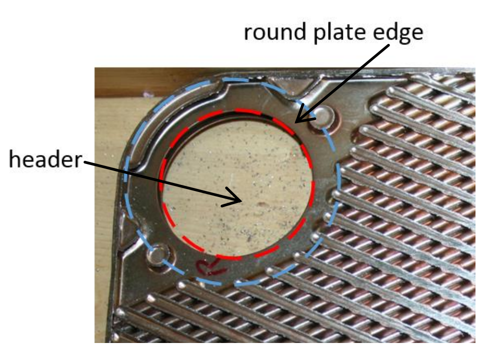


Figure 5: Header-plate geometry used for CFD modeling

The computational mesh for CFD modeling, shown in Figure 6, is generated in the preprocessor software Gambit. This is a structured mesh constituted by hexahedra elements and the grid contains about 2.3 million elements.

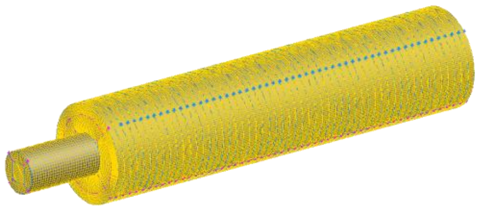


Figure 6: Computational mesh for CFD modeling of the headers

In the test system, there is a tee structure in front of the inlet and outlet header (Figure 7). This tee structure induces the changes of the flow direction and cross area which might affect the flow condition in the headers and then the mass flow distribution. Therefore, in order to accurately model the flow field in the headers and meanwhile explore the impact of this instrumental factor, another header CFD model with such a tee structure is also developed (Figure 8).

Figure 7: The tee structure in front of the headers

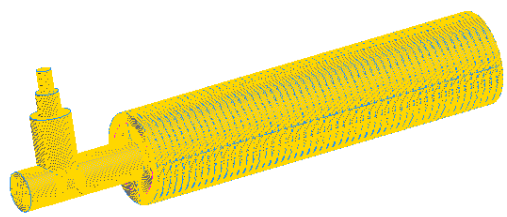


Figure 8: Computational mesh for CFD modeling of the headers with a tee

It can be seen in Figure 4 that, for all three total mass flow rates, the pressure difference for different channels is actually not varying too much and that implies the mass flow rate for each channels doesn't change too much. Hence, it's acceptable to assume a constant friction factor for each channel at a certain total mass flow rate. In addition, the pressure difference for each channel can be obtained by curve-fitting the data provided in Figure 4. With the assumption of the constant friction factor and the pressure difference for each channel, the channel mass flow distribution can be calculated out for a certain total mass flow rate. This mass flow distribution is given to the header CFD model as the boundary condition.

The standard k-epsilon turbulent model with the scalable wall function is used to simulate the flow in the inlet and outlet header.

3.2 Modeling Results and Discussion

The flow in the inlet and outlet header has been simulated for the total mass flow rate of 10 g/s. Modeling results are obtained for two header geometries (with/without a tee). Figure 9 gives a comparison between the simulated centerline static pressure changes in the headers and the experimental data.

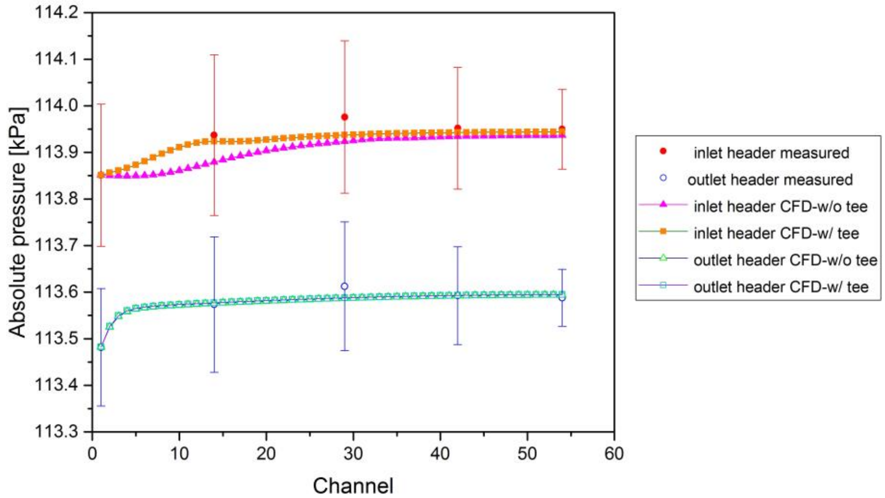


Figure 9: CFD modeling vs. experiments for 10 g/s

Figure 9 shows that, the CFD simulation of the inlet header with a tee gives a higher static pressure rise than that of the inlet header without a tee, and matches the experimental data better. But in the outlet header, the simulated static pressure curves are almost overlapping for the outlet header with and without a tee structure. The comparison between the modeling results and experimental data illustrates that the CFD models can accurately simulate the flow in the headers.

The impact of the tee structure can be seen more clearly in the diagrams of velocity contours. Figure 10(a) and 10(b) gives the axial velocity contours of the vertical center plane of the inlet header with and without a tee respectively.

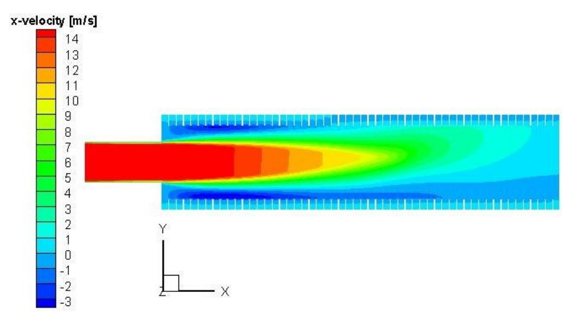


Figure 10(a): x-velocity contour in the inlet header w/o a tee

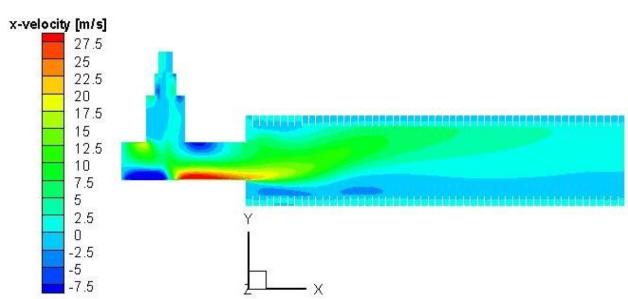


Figure 10(b): x-velocity contour in the inlet header w/ a tee

Comparing the velocity fields in Figure 10(a) and 10(b), it can be observed that the tee structure actually changes the velocity profile at the inlet port of the header. In the Figure 10(a), the axial velocity is almost uniform at the inlet port. But after adding a tee structure, the fluid tends to concentrates at the bottom of the conduit with a much higher axial velocity when entering the header. The different velocity profiles at the inlet port would affect pressure change in the inlet header.

To quantitatively analyze the influence of the tee structure, the calculated dynamic pressure changes are plotted out for these two inlet header geometries in Figure 11.

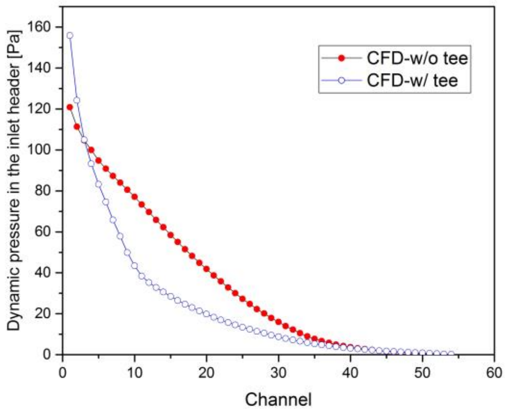


Figure 11: Simulated dynamic pressure in the inlet header

It can be seen in the Figure 11, with the same total mass flow rate (10 g/s), the dynamic pressure near the inlet port (the 1st channel) of the header with a tee is much higher than that of the header without a tee. This demonstrates that the inlet velocity profile is quite uneven for the header with a tee. However, the dynamic pressure is decreasing rapidly in the header with a tee. This might be because the fluid has a relatively large space to expand in the header and convert the dynamic pressure to the static pressure. With a higher dynamic pressure to be converted, the static pressure rise along the inlet header would be larger which is beneficial for the last several channels to receive more mass flow.

The situation is different in the outlet header. Figure 12(a) and 12(b) gives the axial velocity contours of the vertical center plane of two types of outlet headers.

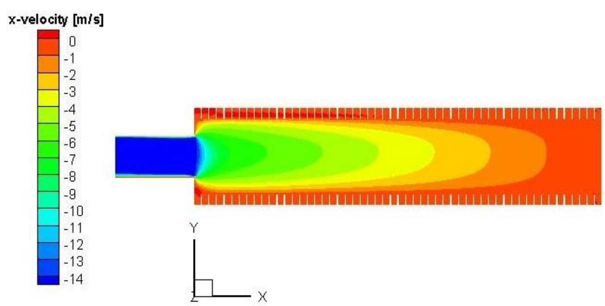


Figure 12(a): x-velocity contour in the outlet header w/o a tee

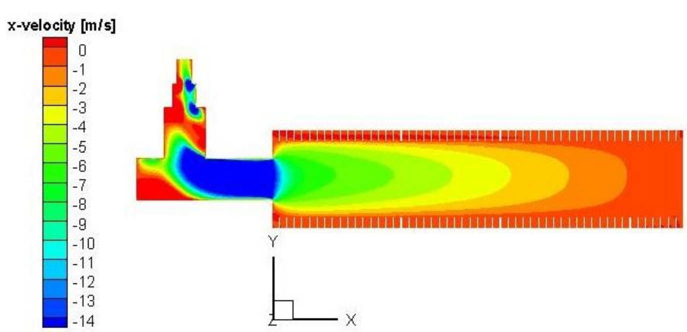


Figure 12(b): x-velocity contour in the outlet header w/ a tee

Comparing the velocity contours in Figure 12(a) and 12(b), one may find that there is no obvious difference in the flow fields of the outlet header with and without a tee. This is because the tee structure is at the downstream of the outlet header, and it does not impact the flow upstream very much.

Based on the analysis above, it is concluded that, a tee structure in front of the inlet header will increase the static pressure rise along the header which increases the channel pressure difference of the last several channels. But the tee structure does not significantly affect the upstream outlet header flow. So far, the CFD modeling has only been conducted for the total mass flow rate of 10 g/s. However, it is rational to infer that the impact of the tee structure on the inlet header flow would be bigger when the total mass flow rate is increased. Therefore, the channel pressure difference for the last several channels will be increased further and even larger than that of the first several channels. That might be the reason why opposite channel pressure difference tend is obtained for different total mass flow rate in Figure 4. The validation of this hypothesis is ongoing.

4. CONCLUSION

The experimental and simulative methods are both used to study the single-phase pressure drop and flow distribution in a U-type BPHE. The pressure changes along the inlet and outlet header are measured, as well as the pressure difference through each plate channel. And CFD models are developed to simulate the flow details in the inlet and out header.

Experimental results show that, due to the conversion between the dynamic pressure and the static pressure, the static pressure increases along the inlet header and decreases along the outlet header. The pressure difference of the first several channels is larger than that of the last several channels at a relatively low total mass flow rate. The opposite trend is obtained when the total mass flow rate is increased.

The CFD simulation of the headers shows that the instrumentation can affect distribution. A tee structure in front of the header would increase the pressure rise along the inlet header, thus increases the pressure difference of the last several channels. But, the flow in the outlet header is not significantly affected by the downstream tee structure. The impact of the tee structure is estimated to be bigger with the higher total mass flow rate and that might explain the opposite trends of the channel pressure difference for different total mass flow rates.

REFERENCES

Bassiouny, M. K., Martin, H. (1984a). Flow distribution and pressure drop in plate heat exchangers-I: U-type arrangement. *Chemical Engineering Science, 39*(4), 693-700.

Bassiouny, M. K., Martin, H. (1984b). Flow distribution and pressure drop in plate heat exchangers-II: Z-type arrangement. *Chemical Engineering Science, 39*(4), 701-704.

Bobbili, P. R., Sunden, B., Das, S. K. (2006). An experimental investigation of the port flow maldistribution in small and large plate package heat exchangers. *Applied Thermal Engineering, 26*, 1919-1926.

Li, H., Hrnjak, P. (2015). Quantification of liquid refrigerant distribution in parallel flow microchannel heat exchanger using infrared thermography. *Applied Thermal Engineering, 78,*410-418.

Song, S., Bullard, C. W., Hrnjak, P. S. (2002). Frost deposition and refrigerant distribution in microchannel heat exchangers. *ASHRAE Trans. 108,* 944-953.

Tuo, H., Hrnjak, P. (2013). Effect of the header pressure drop induced flow maldistribution on the microchannel evaporator performance. *International Journal of Refrigeration, 36*(8), 2176-2186.

Wang, C. C., Yang, K. S., Tsai, J. S., Chen, I. Y. (2011). Characteristics of flow distribution in compact parallel flow heat exchangers, part I: typical inlet header, *Applied Thermal Engineering, 31*, 3226-3234.

ACKNOWLEDGEMENT

The authors thankfully acknowledge the support provided by the Air Conditioning and Refrigeration Center (ACRC) at the University of Illinois at Urbana-Champaign and the instrumentation and other technical support from CTS (Creative Thermal Solution, Inc.).