2012

A Generalized Effectiveness-NTU Based Variable Geometry Microchannel Heat Exchanger Model

Long Huang
long730@umd.edu

Vikrant Aute

Reinhard Radermacher

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

http://docs.lib.purdue.edu/iracc/1206

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at https://engineering.purdue.edu/Herrick/Events/orderlit.html
A Generalized Effectiveness-NTU Based Variable Geometry Microchannel Heat Exchanger Model

Long HUANG\textsuperscript{1,*}, Vikrant AUTE\textsuperscript{2}, Reinhard RADERMACHER\textsuperscript{3}

Center for Environmental Energy Engineering
Department of Mechanical Engineering, University of Maryland
College Park, MD 20742 USA

\textsuperscript{1,2}Tel: 301-405-8726, \textsuperscript{3}Tel: 301-405-5286, Fax: 301-405-2025
Email: \textsuperscript{1}long730@umd.edu, \textsuperscript{2}vikrant@umd.edu, \textsuperscript{3}raderm@umd.edu

*Corresponding Author

ABSTRACT

Microchannel heat exchangers (MCHXs) are now widely used in the HVAC&R industry. Similar to tube-fin heat exchangers used mainly in the indoor units, the innovative variable geometry microchannel heat exchanger will further reduce its compact size and enhance its performance. Variable geometry refers to a heat exchanger with different tube and fin surfaces within the same core with one or more tube banks. A comprehensive literature review reveals that there is no modeling approach in the literature that can handle such arbitrary geometry. This paper introduces a new microchannel heat exchanger model which accounts for variable port, tube and fin geometry, variable tube and fin location and variable number of tubes per bank and variable fin density. This model adopts a port-by-port calculation approach on air side and the refrigerant side. The model is based on three-stream NTU method to account for variable fin type, height and air flow on top and bottom of a given tube. The heat exchanger location is based on a Cartesian grid which would account for the air propagation through multiple banks with variable geometry and fin location. Since the basic heat exchange calculation is performed at the port-level, the model lends itself very easily to account for port-level refrigerant flow maldistribution. In addition, the decrease in air-side heat transfer coefficient along the depth in the direction of the air flow can also be accounted for. Empirical correlations from literature are used for local heat transfer coefficient, pressure drop and void fraction calculations. The model is validated against experimental data for standard geometry microchannel condenser.

1. INTRODUCTION

Air-to-refrigerant MCHXs have wide range of applications in refrigeration and air-conditioning industry. Compared to the traditional time-consuming heat exchanger (HX) development approach of designing and testing prototypes, HX simulation models have been used extensively to predict the MCHX performance and cost within a reasonable accuracy. Several models can be found in the literatures that are dedicated to simulating the conventional uniform geometry MCHXs. Uniform geometry refers to a MCHX having the same geometry parameters such as tubes per bank, tube vertical spacing, fins per inch (FPI), port diameter etc. In recent development of next generation HX design, MCHXs with variable geometry are introduced in order to further enhance the heat transfer performance, decrease the refrigerant pressure drop and reduce the material cost. The variable geometry concept are not only focusing on the reduction of airside heat transfer resistance which dominates the thermal resistance of the HX, but also the refrigerant side flow channel design in order to achieve a better flow distribution. Such an innovative MCHX design can have variable fin types, FPI, fin heights, tube heights, tube widths, port diameters, tubes per bank, tube vertical spacing, tube horizontal spacing etc. In additional to the variable physical parameters, the position of the fin, tube and port can also be different. The current MCHX simulation tools do not have the capability to account for these variable parameters. This paper presents a model with the capability of simulating the MCHXs with most flexible design with variable geometry parameters which would allow the engineers to further push the MCHX technology envelope.

For fin-and-tube heat exchanger (TFHX), researchers have presented several models to account for variable fin spacing (Jiang et al., 2006, Yang et al., 2006ab). Such capability has been used to optimize the fin spacing for a fin-
and-tube evaporator to reduce the defrost cycle time (Yang et al., 2006ab). It has been experimentally investigated by Domanski et al., (2007) that the tube-to-tube conduction effect of TFHX would result in performance degradation. To study such an effect and explore the optimized fin sheet design, Singh et al. (2009) proposed a TFHX model with arbitrary fin sheet. The Singh et al. model is capable of predicting the performance of TFHX with variable tube diameters, tube locations, tube pitches, variable number of tubes per bank as well as the locations of fin-cut and has been validated against experimental data.

Several segmented MCHX models can be found in the literatures based on the Effectiveness-NTU approach (Tuo et al., 2012, Fronk & Garimella 2010, Schwentker et al., 2005) or an energy balance approach (Jin et al., 2011, Brix 2010, Yun et al., 2007, Kim & Bullard 2001). These model are developed for various applications, however, none of them has the ability to simulate a MCHX with variable geometries. Abdelaziz & Radermacher (2010) proposed a MCHX model to account for the manufacturing uncertainty of tube vertical spacing. This model assumed that the water flow in the channel can be separated into two parts when calculating the heat transfer between different air streams on top and bottom of the tube due to the variation of tube vertical spacing.

This paper introduces a generalized heat exchanger model for MCHX with variable geometry parameters. This model is then validated against experimental data from the test of a conventional CO₂ gas cooler.

2. MODEL DETAILS

2.1 Model Description

Most of current MCHXs have a standard fin and tube configuration as shown in Fig.1. This conventional design of MCHX would facilitate the design and manufacturing process in general. However, the development of MCHX is restricted by such uniformity. The optimum boundary of MCHX design can be extended further with flexible and adaptive geometry to achieve the best HX performance or the most suitable design for a specific application.

![Standard configuration MCHX](image)

Fig. 1 Standard configuration MCHX

The flexibility of proposed MCHX model can be illustrated but not limited by the configuration shown in Fig.2
There are several parameters in the proposed model that can be varied based on the design specifications such as:

1) Port type and size: Extensive experimental studies along with the developed heat transfer and pressure drop correlations (Kandlikar & Balasubramanian 2004, Yun et al., 2005) can result in a better understanding of the port design. This knowledge would lead to a better balance between the maximization of heat transfer, minimization of pressure drop and better flow distribution by varying the number and type of ports in different flow pass of the MCHX.

2) Fin type and fin height: The relative location of fan and the heat exchanger can result in a different level of air-side maldistribution which has a negative effect on the heat exchanger performance. The location and size of the fin can be designed such that the air side heat transfer resistance and maldistribution effects can be minimized while the pressure drop is within an acceptable limit.

3) Tubes per bank and the tube location: The traditional multi-bank MCHX always has the same number of tubes per bank and the MCHX banks are lined uniformly. The proposed model locates all the tubes, fins on a Cartesian grid. The tubes and fins can be located anywhere within the MCHX envelope and the air-side propagation will be conducted based on its location relative to other tubes.

2.2 Modeling Approach

A segment-by-segment microchannel model was developed based on the general approach by Jiang et al. (2006), and the following assumptions were made:

- Steady state model
- Tube shares a half of top fin and a half of bottom fin
- Segment can be sub-divided to track the exact flow pattern change point
- Thermally and hydrodynamically fully developed flow
- The thermophysical properties and heat transfer coefficients are evaluated based on the inlet of each segment/sub-divided segment
- Refrigerant is well-mixed in the intermediate header
- Horizontal air flow

The refrigerant side calculation is processed based on the flow direction. The headers are categorized into inlet header, intermediate header and outlet header. The calculation starts from all the downstream tubes of the inlet headers. Once all the downstream tubes of an inlet header are calculated, the solver would proceed to their intermediate headers. Intermediate headers’ downstream tubes would only be calculated once their upstream tubes’
calculations are done. Finally there will be a check for all outlet header upstream tubes, if all the outlet headers’ upstream tubes are solved, the solver would proceed to the outlet header pressure drop calculation. This completes one iteration of the refrigerant side calculation.

The air side propagation between each tube is conducted iteratively. Initially all the air side state for all tubes is assumed to be the MCHX’s air inlet state. As the schematic in Fig. 3 shows, the corner of the fin and tube has a coordinate on the Cartesian grid. Tube 3’s inlet air state would be the mixed air from tubes 1 and 2.

The top level methodology to solve the problem is presented in Fig. 4. The air-side heat transfer coefficient (HTC) and pressure drop (DP) would be updated after air-side propagation.

![Fig. 3 Air propagation explanation](image)

![Fig. 4 Solution methodology for proposed model](image)
Compared to the previous models (Tuo et al., 2012, Fronk & Garimella 2010, Schwentker et al., 2005, Jin et al., 2011, Brix 2010, Yun et al., 2007, Kim & Bullard 2001) which have a per tube control volume shown in Fig. 5(a), the proposed model has per port based control volume as shown in Fig. 5(b). Having such a finite control volume would allow the model to account for the refrigerant per-port maldistribution as well as the variation in air side temperature and heat transfer coefficient in the direction of air flow.

The ports are divided into segments in the refrigerant flow direction. Since the fins on top and bottom could be different, there are two air streams (1 and 3) and one refrigerant stream (2) as presented in Fig. 6. In the evaporator case, if there is condensation on one of the outer surface, the simultaneous heat and mass transfer is calculated based on enthalpy potential method (McQuiston & Parker, 1994). The flow regime of the segment outlet would be compared to the inlet condition. If there is a phase transition in the segment, segment would be further sub-divided to locate the flow regime transition point iteratively based on the sub-divided segment model as described in Jiang et al. (2006). Within one tube, the air side and refrigerant side outlet condition would be propagated to the next port and segment respectively once the heat transfer and pressure drop in the port/segment is calculated.

For the segments/sub-divided segments with single phase refrigerant flow inside the channel, the heat transfer between two air streams and the refrigerant is calculated based on the 3-stream NTU approach developed by Baclic et al. (1982). The heat capacity ratios on top side and bottom side are calculated as:

\[ C_{i,2} = \frac{(mC_p)_1}{(mC_p)_2} \quad \text{for } i = 1,3 \]  
\[ (1) \]

The number of transfer unit (NTU) on both sides is defined as:

\[ NTU_i = \frac{(UA)_{i,2}}{(mC_p)_i} \quad \text{for } i = 1,3 \]  
\[ (2) \]
Once the heat capacities, NTU and inlet conditions of the air streams and refrigerant streams are determined, outlet temperatures of these three streams can be calculated based on the analytical equation sets in the 3-stream NTU approach. The equations are lengthy and are not presented in this paper for brevity. For the details of the equations, the reader can refer to Bałic et al. (1982).

For the segments/sub-divided segment with two-phase refrigerant flow inside the channel, since the minimum heat capacity one both sides would always be the heat capacity of the air stream, heat transfer between refrigerant and air streams is solved based on the effectiveness-NTU approach (Kays & London, 1984) under a dry surface condition. After the NTU is obtained from equation (2), the effectiveness on both sides can be calculated based on the following equation:

\[ \varepsilon_i = 1 - e^{NTU_i} \quad for \ i = 1,3 \] (3)

Air-side outlet temperature

\[ Tout_i = Tin_i - \varepsilon_i \times (Tin_i - Tin_2) \quad for \ i = 1,3 \] (4)

Heat load

\[ Q = (mC_p)_1 \times (Tin_1 - Tout_1) + (mC_p)_3 \times (Tin_3 - Tout_3) \] (5)

3. MODEL VALIDATION AND RESULTS

The proposed model was built and validated against the experimental data of a CO₂ gas cooler (Zhao et al., 2001). For the detailed geometries and test conditions, the reader can refer to Zhao et al. (2001). The thermophysical properties of CO₂ were calculated using the NIST REFPROP 8.0 (Lemmon et al., 2007). In this validation, several empirical correlations are selected for the heat transfer and pressure drop calculation on both air side and refrigerant side. The correlations are summarized and tabulated in Table 1.

<table>
<thead>
<tr>
<th>Heat transfer coefficient</th>
<th>Frictional pressure drop</th>
<th>Contraction pressure drop</th>
<th>Expansion pressure drop</th>
</tr>
</thead>
<tbody>
<tr>
<td>Supercritical region</td>
<td>Liao &amp; Zhao (2002)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Two-phase region</td>
<td>Shah (1979)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Liquid-phase region</td>
<td>Gnielinski (1976)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Single-phase region</td>
<td>Churchill (1977)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The validation of the CO₂ gas cooler simulation was conducted against 11 test points. The comparison between experimental data and simulated results is shown in Fig. 4, 5 and 6. The maximum deviation between simulated heat load and experimental data is ±4.4%. Most of the pressure drop deviations were within ±30% of test data. However, there are several test points with around 40% deviation. There are several factors may influence the pressure drop calculation. One of them is that the refrigerant-side maldistribution is not considered in this validation. As the control volume is per-port based, the refrigerant distribution in different ports would be an important factor for both heat transfer and pressure drop calculation which is due to the fact that the air-side condition is propagating from the first port to the end port. It is worthwhile to note that there are limited correlations available to calculate the supercritical CO₂ pressure drop. Single phase frictional pressure drop correlation developed by Churchill (1977) is selected in this validation. Thus, a suitable pressure drop correlation for supercritical CO₂ in the microchannel is
essential. Most of the deviations of CO$_2$ outlet temperature are within ±2.5 K. No correction factor is applied in this validation. Application of correction factors to correlations can significantly improve the validation results.

Fig. 7 Comparison between experimental heat load and simulated results

Fig. 8 Comparison between experimental refrigerant pressure drop and simulated results
4. CONCLUSIONS

A new heat exchanger model for variable geometry microchannel air-to-refrigerant heat exchangers was developed. This model has the capability of accounting for variable geometry parameters, such as fin types, fin dimensions, tube geometries, port shapes and the location of the tubes and fins. The model adopted a segment-by-segment approach along with sub-divided model that can locate the refrigerant flow regime change point. The three-stream NTU method is implemented in order to account for the variable fins on top and bottom of the tube. The air-side propagation is conducted based on the location of the tubes and fins on a Cartesian grid. This allows one to account for the change in air side heat transfer coefficient in the direction of air flow. The developed model was validated against experimental data for a conventional microchannel heat exchanger. The heat load prediction was within ±4.4% of experimental data. The refrigerant side pressure drop is found to be within ±30% of the tested pressure drop value for most test points. The validation was carried out without applying any correction (performance adjustment) factors to the model.

The current model’s heat transfer coefficient and pressure drop calculations are based on empirical correlations. It should be noted that most of the correlations developed are based on experimental data for conventional uniform geometry MCHX. While simulating the variable geometry MCHX, computational fluid dynamics (CFD) techniques can be applied to predict the complex air distribution within the core. In addition, refrigerant side flow distribution should be addressed in the study of variable geometry MCHX. The ports within the tubes need to be designed in order to achieve better flow distribution. Furthermore, this generalized model can be incorporated with heat exchanger optimizer (Aute et al., 2004) to design the parameters such as tube/fin/port dimensions, tube/fin locations etc. for heat transfer performance enhancement, pressure drop minimization and material cost reduction.

**NOMENCLATURE**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Subscripts</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>area</td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>heat capacity</td>
<td></td>
</tr>
<tr>
<td>C_p</td>
<td>specific heat</td>
<td></td>
</tr>
<tr>
<td>Q</td>
<td>heat load</td>
<td>i, stream</td>
</tr>
<tr>
<td>T</td>
<td>temperature</td>
<td>in, Inlet</td>
</tr>
<tr>
<td>U</td>
<td>Overall heat transfer coefficient</td>
<td>out, Outlet</td>
</tr>
<tr>
<td>m</td>
<td>mass flow rate</td>
<td></td>
</tr>
<tr>
<td>ε</td>
<td>effectiveness</td>
<td></td>
</tr>
</tbody>
</table>

Fig. 9 Comparison between experimental refrigerant outlet temperature and simulated results
REFERENCES


