Applied Thermal Engineering 65 (2014) 447–457

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

A computational fluid dynamics and effectiveness-NTU based co-simulation approach for flow mal-distribution analysis in microchannel heat exchanger headers

Long Huang*, Moon Soo Lee, Khaled Saleh, Vikrant Aute, Reinhard Radermacher

Center for Environmental Energy Engineering, Department of Mechanical Engineering, University of Maryland, 4164 Glenn L. Martin Hall Bldg., College Park, MD 20742, USA

HIGHLIGHTS

- MCHX header flow distribution is analyzed by a co-simulation approach.
- The proposed method is capable of simulating both single-phase and two-phase flow.
- An actual header geometry is reproduced in the CFD header model.
- The modeling work is experimentally validated with good accuracy.
- Gravity effect and air side mal-distribution are accounted for.

ARTICLE INFO

Article history:
Received 18 September 2013
Accepted 22 January 2014
Available online 31 January 2014

Keywords:
Microchannel heat exchanger
Flow distribution
Computational fluid dynamic
Header simulation
Effectiveness-NTU based model

ABSTRACT

Refrigerant flow mal-distribution is a practical challenge in most microchannel heat exchangers (MCHXs) applications. Geometry design, uneven heat transfer and pressure drop in the different microchannel tubes are three main reasons leading to the flow mal-distribution. To efficiently and accurately account for these three effects, a new MCHX co-simulation approach is proposed in this paper. The proposed approach combines a detailed header simulation based on computational fluid dynamics (CFD) and a robust effectiveness-based finite volume tube-side heat transfer and refrigerant flow modeling tool. The co-simulation concept is demonstrated on a ten-tube MCHX case study. Gravity effect and uneven airflow effect were numerically analyzed using both water and condensing R134a as the working fluids. The approach was validated against experimental data for an automotive R134a condenser. The inlet header was cut open after the experimental data had been collected. The detailed header geometry was reproduced using the proposed CFD header model. Good prediction accuracy was achieved compared to the experimental data. The presented co-simulation approach is capable of predicting detailed refrigerant flow behavior while accurately predicts the overall heat exchanger performance.

1. Introduction

MCHXs are gaining grounds in the Heating, Ventilation, Air-Conditioning & Refrigeration (HVAC&R) industry due to their high material utilization and enhanced heat transfer performance. However, the uneven flow distribution is one of the challenges in improving MCHX's performance. MCHXs use headers to mix and distribute the working fluid into individual microchannels. The mal-distribution of the working fluids causes reduction in the heat exchanger capacity. Muller and Chiou [1] concluded that there are various types of flow mal-distribution and the causes are: 1) mechanical design of headers and the inlet ducts; 2) self-induced mal-distribution caused by the heat transfer process; 3) gas–liquid flow phase separation in headers; 4) fouling and corrosion.

The objective of this research is to develop a co-simulation approach that provides accurate prediction of flow distribution in header along with the capability of modeling the microchannel tubes in a fast and reliable manner. The mal-distribution effects were investigated for both single-phase and two-phase working fluids within the heat exchanger, under both zero gravity and standard gravity, facing uniform airflow and uneven airflow. The modeling studies focuses on heat exchanger with single-phase flow in inlet headers. Finally, the co-simulation model was validated against experimental data. It should be noted that the proposed

* Corresponding author. Tel.: +1 301 405 7314; fax: +1 301 405 2025.
E-mail address: long730@umd.edu (L. Huang).

1359-4311/$ – see front matter © 2014 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.applthermaleng.2014.01.046

Nomenclatures

A	heat transfer area [m^2]
CFD	computational fluid dynamics
HX	heat exchanger
MCHX	microchannel heat exchanger
NL	non-linear
NTU	number of transfer units
PPCFD	parallel parameterized CFD
STD	standard deviation
U	overall heat transfer coefficient [W m^{-2} K^{-1}]

approach is not limited to single-phase header simulation. Coupling with a proper two-phase CFD solver would allow such approach to be applied to MCHXs with two-phase flow in headers.

2. Literature review

The co-simulation modeling approach presented in the paper addresses three crucial aspects of MCHX analyses: 1) refrigerant flow behavior in headers; 2) refrigerant flow distribution; 3) energy and hydraulic calculation in the main core of the MCHX. Thus, it is important to understand the previous experimental work and modeling effort in these areas. The literature review summarizes the experimental investigations on flow mal-distribution, modeling effort using finite volume MCHX models as well as CFD studies focusing on MCHX simulations, especially header CFD simulation works.

The refrigerant distribution in parallel flow MCHX has been experimentally studied and several refrigerants and different header geometries have been tested by researchers. Most of the previous researches are focused on flow distribution improvement with different header shapes and inlet arrangements. Cho et al. [2] identified that the header orientation was the most important factor affecting flow distribution in MCHXs. A study on phase separation and distribution in headers has been published by Vist & Pattersen [3]. The authors proposed using a short inlet tube to enhance the vapor and liquid distribution in the headers. Hrnjak [4] presented three methods for the study of mal-distribution: infrared imaging, frost accumulation and exit air temperature profile. To reduce the mal-distribution, he suggested placing the inlet and outlet header on the opposite side of the heat exchanger with single-phase fluid and creating a misty flow with extremely small droplet sizes in the headers with two-phase fluid. Poggi et al. [5] studied single-phase flow behavior in a vertical header. The experimental test showed that the pressure drop in the flow channels is greater than the header pressure drop by a ratio of 10. Severe flow maldistribution was not found in this test for single-phase HydroFluoroEther 7100 and water. Ren and Hrnjak [6] correlated the single-phase pressure drop in D shaped headers of both heat exchanger inlet and outlet. The correlation is capable of predicting the pressure difference between each channels. Jin [7] observed that the quality mal-distribution was more severe than the mass flow mal-distribution. Hwang et al. [8] recommended placing the inlet connecting tube in the center of the horizontal header to improve the liquid distribution. Byun and Kim [9] tested a parallel flow microchannel evaporator with vertical headers with R410A. Significant improvement of flow distribution was obtained while placing the inlet at the top instead of using a middle inlet. Also, they concluded that top and bottom outlets are better compared to a middle outlet. Zou and Hrnjak [10,11] presented flow visualization experiments of R134a and R410A distribution in vertical header. The best flow distribution in Zou and Hrnjak's test was at high mass flow rate and low quality where the flow is in the churn and separated regimes.

Extensive simulation efforts on microchannel heat exchanger modeling using the effectiveness-NTU method and the energy balance method can be found in the open literature. Most of these finite volume (also known as segmented) models assume equal distribution of refrigerant in all tubes [12–23]. The reported deviations of heat load in these models are typically within 10 percent. These models use empirical correlations for the calculation of header and tube pressure drop. It is often a challenge to match the predicted and experimentally measured pressure drop due to the limitation on the applicability of the correlations and the high uncertainty in experimental pressure measurement. It should be noted that there is no correlation specifically developed for pressure drop in MCHX headers.

Flow mal-distribution in finned tube heat exchangers has been modeled by Domanski [24]. He developed a simulation model with the capability of solving flow mal-distribution in different circuits. One-dimensional air mal-distribution can be accounted for in this model. Kim et al. [25] studied the effect of non-uniform airflow distribution based on a cycle simulation model of a residential R410A heat pump system. It was concluded that the control of individual refrigerant circuit flow rates would benefit the evaporator performance significantly when the airflow mal-distribution is severe. Brix et al. [26] developed a two-tube MCHX model and then conducted a parametric study of refrigerant flow distribution profile to evaluate its impact on the MCHX's performance. Brix et al. investigated the effect of the non-uniform airflow distribution on the local UA value and studied the coupled influence of refrigerant-side and air-side mal-distribution. Brix et al. proposed that the air mal-distribution effect can be compensated by a suitable phase distribution in the header. Huang et al. [27] presented a steady-state MCHX model that accounts for both refrigerant flow distribution and air side flow distribution. In the Huang et al. [27] model, the header pressure drop is calculated using empirical round tube correlations while the complicated flow behavior within the actual header geometry cannot be accounted for. Thus, in this model, the mal-distribution of refrigerant is induced by the uneven heat transfer of the microchannel tubes but not affected by the header geometry. Tuo et al. [28] presented an evaporator model that considered both refrigerant flow distribution and detailed header pressure drop. In Tuo's header pressure drop formulation, contraction loss, expansion loss, pressure loss due to tube protrusion, frictional loss, momentum loss as well as gravity effect are taken into account. The simulated pressure drop between inlet and outlet header is validated against experimental data within 12 percent deviation. The iterative flow distribution calculation method and correlation based header pressure drop prediction approach were also adopted by Ren et al. [29] in a port-by-port MCHX model.

Zhang and Li [30] conducted a CFD study on flow distribution in a plate-fin heat exchanger by simulating the entire heat exchanger domain. Based on the modeling effort, the authors concluded that applying modified headers with a two-stage-distributing structure could reduce the mal-distribution effect. Lalot et al. [31] reported 7–25% performance degradation for different types of heat exchangers based on the numerical CFD simulation results. Fei and Hrnjak [32] modeled the two-phase flow in a header using the Eulerian model. Deviations of the experimental data and simulation results were presented. The actual droplet diameter, void fraction, the effect of flow expansion and re-circulation cannot be well captured by the presented model. Habib et al. [33] correlated flow mal-distribution parameter of a HX using air as tube-side fluid.