Numerical Optimization of Baffle Design in A Parallel Heat Exchanger Header Using CFD and Genetic Algorithm

Nguyen Ba Chien†*, Nguyen Dinh Vinh†, Vu Huy Khue† & Jong-Taek Oh‡

†Department of Refrigeration and Air-conditioning Engineering, Hanoi University of Science and Technology, 1 Dai Co Viet, Hanoi, Vietnam, 10000
‡Department of Refrigeration and Air-conditioning Engineering, Chonnam National University, 50 Daehak-ro, Yeosu-si, 59626, South Korea
*E-mail: chien.nguyenba@hust.edu.vn

ABSTRACT: Maldistribution of refrigerant flow in parallel heat exchanger is one of the main reasons which reduces the thermal performance. This study demonstrates a numerical method to automatically evaluate the baffles height in order to optimize the flow distribution of two-phase flow refrigerant inside a heat exchanger header. This method couples the computational fluid dynamics (CFD) model and a genetic algorithm to generate the optimal dimensions of multi baffles simultaneously. The numerical results show that the maldistribution of flow could be reduced up to 9 times compare to original design. Furthermore, this framework also could expand the design space, thus it could be a useful tool to boost up the design of headers using in parallel heat exchanger.

KEYWORDS: refrigerant; computational fluid dynamics (CFD); parallel heat exchanger

INTRODUCTION

The flow maldistribution inside inlet header is one of the most important concern in designing air-cooled heat exchanger. The performance of the heat exchanger could be significantly reduced by the maldistribution of flow [1-2]. Thus, the behavior of flow inside the inlet header is necessarily analyzed as well as optimized to improve the efficiency of parallel flow heat exchanger.

In practical design, flow maldistribution in parallel inlet headers could be induced by both of the geometry and operating conditions [3]. Jiao et al. [4] reported that the flow distribution could be significantly improved by an optimal design of header configuration. The experimental results in this study also stated flow distribution in plate fin heat exchanger was affected by various geometry parameters including the inlet pipe diameter, the first header’s diameter, and the second header’s diameter.

A serious flow maldistribution problem in the heat exchanger using a conventional header is reported by Wen and Li [5]. To improve the uniformity of flow, they recommended to install a baffle with small-sized holes inside the header. Wang et al. [6] presented additional solutions to prevent flow maldistribution. Five modified inlet headers were proposed in their study included trapezoidal type, step types and baffle types.

Beside of using experiment, various studies have employed numerical method to evaluate the flow distribution. Using CFD model, Tonomura et al. [7] showed that the shape of manifolds, length and location of inside fins, and inlet flow rate strongly affect to the flow uniformity. The authors also proposed a CFD-based optimization method to find the optimal design of plate-fine microdevices. Wang et al. [8] investigated a different computational approach. The authors presented a Lattice Boltzmann method for shape optimization of the distributor. Recently, Facao [9] developed a correlation model to improve the uniformity of flow in a solar collector. The model was then validated by the CFD simulation.

In this study, we demonstrated a further step to enhance the uniformity of flow inside the header by deploying an automated design process. The procedure is established by coupling the CFD simulation and a surrogated based optimization method [10]. The approach presented in this study is used to optimize the baffle height of inside inlet header to maintain the uniformity of flow. On the other hand, by using this method, we could expand the design space effectively. Therefore, the uniformity of flow which depends on various parameters and has no general form for the specific applications, could be tackled.
NUMERICAL MODEL

Mathematical model

In present study, two-phase flow of refrigerant is simulated using the OpenFOAM software. The working fluid is R-134a. Since the experimental test was taken under the adiabatic condition, the simulation will only solve the continuity and momentum equations. VOF method, therefore, has been implemented. In this work, VOF solver in OpenFOAM calculates one momentum and one continuity equation for both liquid and gas phases of R134a. The physical properties of one phase are determined based on the volume fraction, α, of two phases in a cell. The continuity equation is defined as follow:

\[ \nabla \cdot \mathbf{U} = 0 \]  

(1)

The momentum equation is determined as follow:

\[ \frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) - \nabla p + \rho \mathbf{g} - \nabla \cdot \mathbf{F}_s = 0 \]  

(2)

where \( \mathbf{F}_s \) is the surface tension force at free surfaces and is defined as:

\[ \mathbf{F}_s = \sigma \kappa(x) \mathbf{n} \]  

(3)

where \( \kappa \) is the curvature of the interface and \( \mathbf{n} \) is a unit vector normal to the interface. \( \kappa \) and \( \mathbf{n} \) are calculated as follow:

\[ \kappa(x) = \nabla \cdot \mathbf{n} \]  

(4)

and

\[ \mathbf{n} = \nabla \alpha \]  

(5)

Volume fraction, \( \alpha \), in a cell is 0 if it is completely filled by vapor phase and, on the contrary, volume fraction equals 1 if cell is completely filled by fluid phase. Volume fraction is calculated by a transport equation as following:

<table>
<thead>
<tr>
<th>Table 1. Properties of R134a</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sat. Tem. p. (°C)</td>
</tr>
<tr>
<td>-----------------------------</td>
</tr>
<tr>
<td>24.47</td>
</tr>
</tbody>
</table>
The compression of the surface in OpenFOAM is defined as follow:

\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{U}) = 0
\] (6)

The compression of the surface in OpenFOAM is defined as follow:

\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{U}) + \nabla \cdot (\alpha (1 - \alpha) \mathbf{U}_r) = 0
\] (7)

where \( \mathbf{U}_r \) is a suitable velocity field to compress the interface.

Geometry and grid generation

In this study, the original dimensions of the header is obtained from a report of Fei and Hrnjak [11].

In the first step, we set up a CFD model to validate the experimental data. A three-dimensional geometry was then created. The grid was generated with fully hexahedral elements to reduce the computational cost as well as to enhance the accuracy and stability of simulation. In addition, to validate the independence of grid, three types of

![Figure 1. Geometry and grid](image)

<table>
<thead>
<tr>
<th>Table 2. Dimension of header</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet length (mm)</td>
</tr>
<tr>
<td>---------------------</td>
</tr>
<tr>
<td>120</td>
</tr>
</tbody>
</table>
mesh were also made. The detail dimension of manifold is illustrated in Table. 1. The geometry and sample of a grid are shown in Fig. 1. And the properties of two-phase flow of R134a obtained in REFPROP 8 [12] are listed in Table 2.

In the second step, we applied the validated numerical model to the same dimension geometry but inserting the inside baffles between each outlet channel. Since the baffles divide the header to separated subsections, the geometry now can be expressed in 2D model. The geometry and grid are then generated using script. The structure of proposed header design is shown in Figure 2.

Optimization procedure

In present work, a surrogate-based optimization model and single objective genetic algorithm (SOGA) were implemented as the optimizers. The purpose of algorithm is to minimize the standard deviation of the flow rate at the outlets. The optimization block automatically generates baffle dimensions and other continues design space. CFD code, driven by a script, then runs and exports the results to sample-dataset. Total 120 samples were generated. The optimizer then runs evaluation processes bases on the data set and proposed the best results. In SOGA block, the fitness type was set as domination count. The cross over is 80%, the mutation scale is 10%, and the number of evaluations in this study was set as 150 [13-16].

RESULTS AND DISCUSSIONS

![Figure 3. History of mass fraction and liquid mass flow rate](image1)

![Figure 4. History of evaluation](image2)

**Figure 3.** History of mass fraction and liquid mass flow rate

**Figure 4.** History of evaluation

Fig. 3 shows the history of volume fraction liquid phase and the mass flow rate of five outlet channels. In this case, the total mass flow rate at the inlet of header is 25g/s and the vapor quality e 0. As seen in the figure, after few first seconds, the liquid flow rate is stable and most of liquid in concentrated at fourth and fifth outlets are
stable and mostly equal. The interference of the forward flow from inlet of header and the backward flow bouncing from the last wall is believed to be the reason for this phenomenon. This case also addresses both the non-uniformity and fluctuation of flow could occur inside a generic header. Thus, optimizing of structure of inlet is needed.

The height of baffle and standard deviation of mass flow rate during the evaluation were illustrated in Fig. 4. In this work, total 120 samples were generated from CFD solver. The baffle height is ranged from 0.1 to 7.9 mm and 150 evaluations were created. The optimal point is successfully found at evaluation 148th with the baffle height of 2.44 mm. The optimal results show the optimal design could reduce the maldistribution up to 9 times.

CONCLUSIONS

In current work, the CFD simulation and a novel optimization approach of two-phase flow distribution of R134a in an inlet header of heat exchanger have been investigated. The simulations were established using VOF model. An optimization procedure which used SBO-GA coupled with CFD simulation has been developed. The results from this study show that the optimal design could reduce the maldistribution of flow up to 9 times compared to the original one. In addition, this procure could be expanded its application to other design parameters that help designers to saving time in deploying new structure of inlet header used in heat exchanger.

ACKNOWLEDGEMENT

This research is funded by the Hanoi University of Science and Technology (HUST) under project number T2018-PC-084 (Vietnam) and Basic Science Research Program through Ministry of Education, Science, and Technology (NRF-2016R1D1A1A09919697) (South Korea).

REFERENCES


