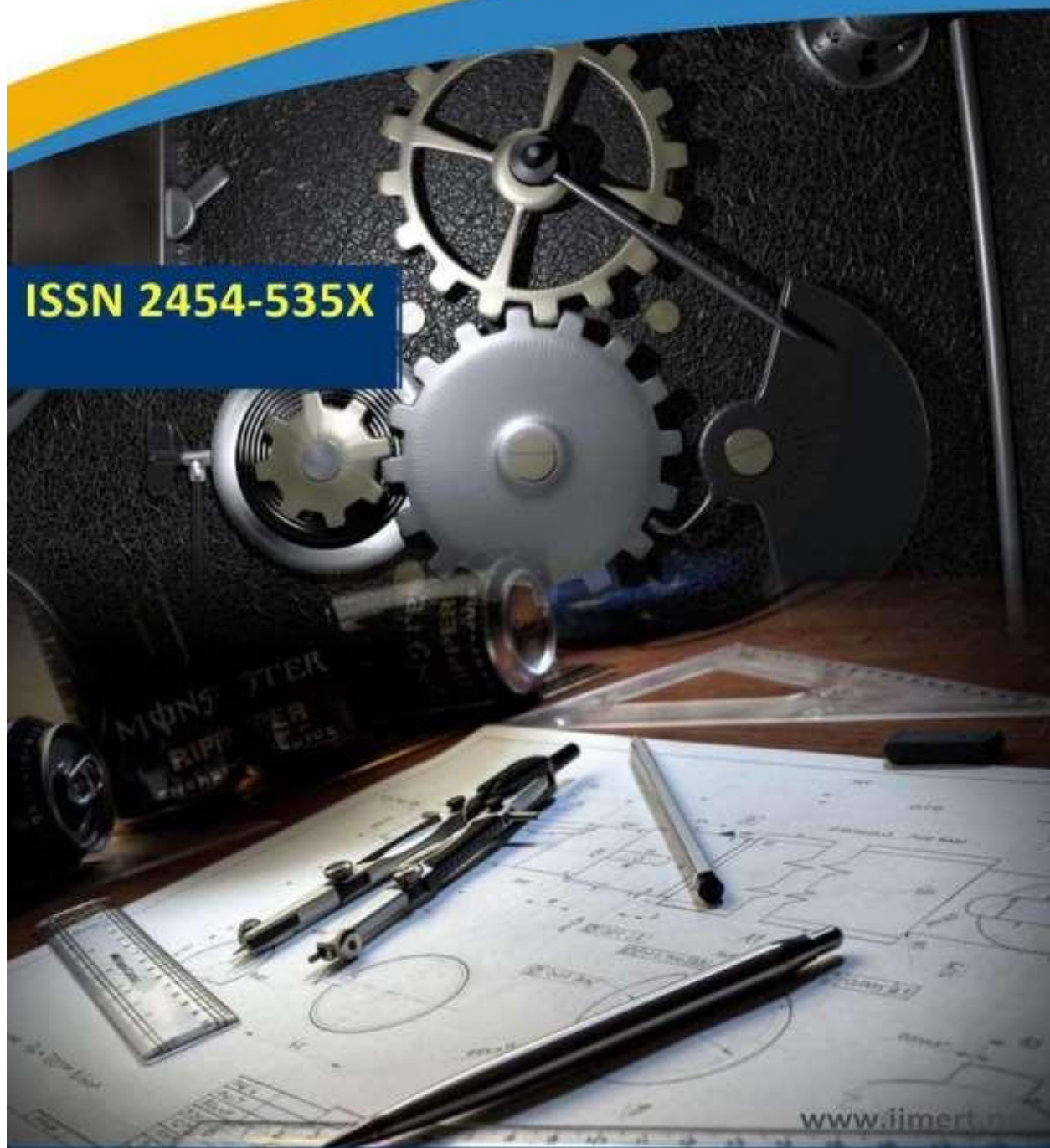




International Journal of Mechanical Engineering Research and Technology

ISSN 2454-535X



Email ID: info.ijmert@gmail.com or editor@ijmert.net



Using CFD and a genetic algorithm, the baffle design of a parallel heat exchanger header is numerically optimized.

Naga Babu¹, Chiranjeevi Vara prasad², Pawan Kaylan³

[†]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: pawan@hust.edu.vn

ABSTRACT: One of the primary causes of decreased thermal performance in parallel heat exchangers is improper refrigerant flow distribution. In order to improve the flow distribution of two-phase flow refrigerant inside a heat exchanger header, this work presents a numerical approach to automatically estimate the height of the baffles. This technique uses a genetic algorithm and the computational fluid dynamics (CFD) model to simultaneously produce the best dimensions for several baffles. The numerical results demonstrate that, in comparison to the original design, the flow maldistribution might be reduced by up to 9 times. This framework also has the potential to increase the design area, making it a valuable tool for improving header designs that incorporate parallel heat exchangers.

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

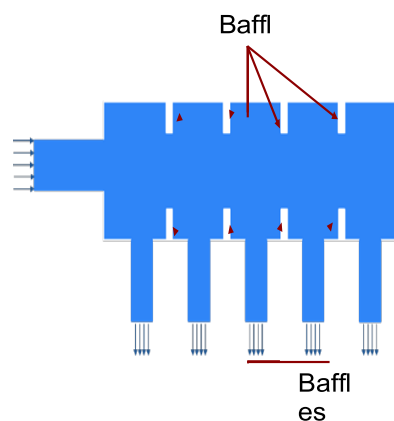
been implemented. In this work, VOF solver in Open FOAM calculates one momentum and one continuity equation for both liquid and gas phases of R134a. The physical properties of one phase are determined based on the volume fraction, α , of two phases in a cell. The continuity equation is defined as follows:

The momentum equation is determined as follows:

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U U) - \nabla \cdot (\mu \nabla U) - \rho g = -\nabla p - F_s \quad (1)$$

where F_s is the surface tension force at free surfaces and is defined as: $F_s = \sigma \kappa(x) n$ (2)

where κ is the curvature of the interface and n is a unit vector normal to the interface. κ and n are calculated as follows: $\kappa(x) = \nabla \cdot n$ (3)



$$\frac{\partial \alpha}{\partial t} + \quad (6)$$



and

Table 1. Properties of R134a

Sat. Tem p. (°C)	P (kPa)	Density of liquid (kg/m ³)	Density of vapor (kg/m ³)	Liquid Vis. (Pa.s)	Vapor Vis. (Pa.s)
24.47	665	1208.7	31.841	1.96E-04	1.17E-05

Volume fraction, α , 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:

The compression of the surface in OpenFOAM is defined as follow: $\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{U}) + \nabla \cdot (\alpha (1-\alpha) \mathbf{U} \partial t) = 0$ (7)

where $\mathbf{U}$ 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

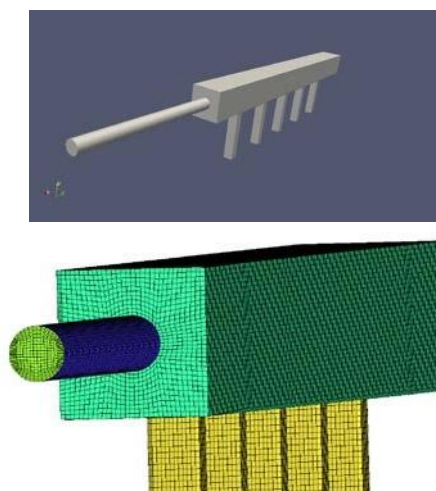


Figure 1. Geometry and grid

Table 2. Dimension of header

Inlet length (mm)	Inlet diameter (mm)	Body Length (mm)	Body Height (mm)	Outlet hydraulic diameter
120	9.5	305	25.4	8.8

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.

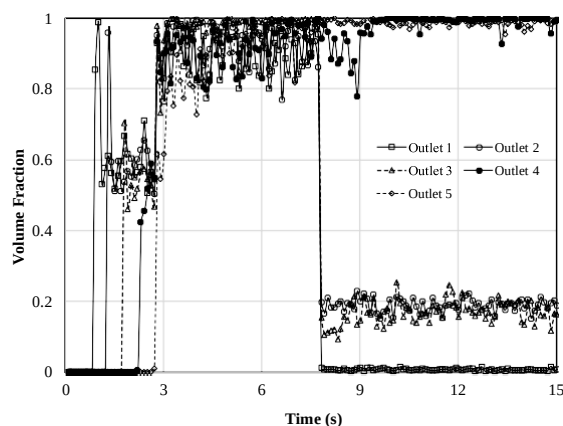
In the second step, we applied the validated numerical model to the same dimension geometry but inserting the inside baffles between each outlet channel. Since the baffles divide the header to separated subsections, the geometry now can be expressed in 2D model. The geometry and grid are then generated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

Optimization procedure

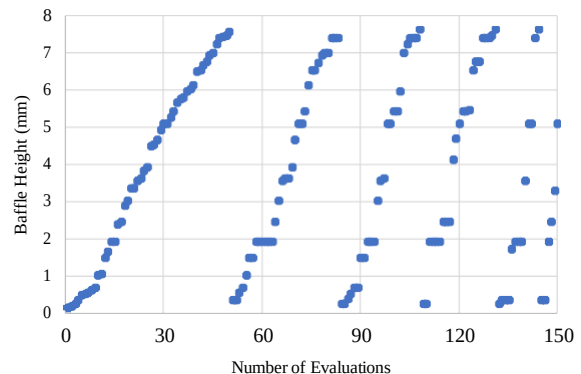
In present work, a surrogate-based optimization model and single objective genetic algorithm (SOGA) were implemented as the optimizers. The purpose of algorithm is to minimize the standard deviation of the flow rate at the outlets. The optimization block automatically generates baffle dimensions and 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].

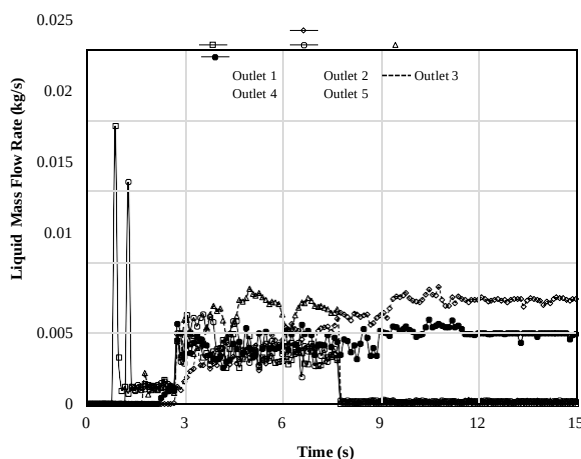
RESULTS AND DISCUSSIONS



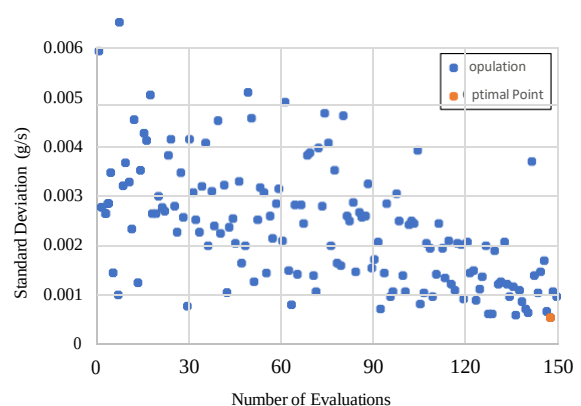
(a) Void fraction at the outlets



(a) Baffle height



(b) Liquid mass flow rate



(b) Standard deviation



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 is 0. As seen in the figure, after few first seconds, the liquid flow rate is stable and most of liquid is 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 procedure 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



compact parallel flow heat exchangers, part II: Modified inlet header”. *Applied Thermal Engineering*, vol. 31, pp. 3235–3242, 2011.

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

- [1] C. Mueller, J.P. Chiou. “Review of various Types of Flow Maldistribution in Heat Exchangers”. *Heat Transfer Engineering*, vol. 9, no. 2, pp. 36-50, 1988.
- [2] S. Lalot, P. Florent, S.K. Lang, A.E. Bergles. “Flow maldistribution in heat exchangers”. *Applied Thermal Engineering*, vol. 19, no. 8, pp. 847–63, 1999.
- [3] A. Jiao, R. Zhang, S. Jeong. “Experimental investigation of header configuration on flow maldistribution in plate-fin heat exchanger”. *Applied Thermal Engineering*, vol. 23, pp. 1235–1246, 2003.
- [4] A.M. Khattak, G.A. Khan, Y. Khan, M.I. Khattak, F. Jamal. “Characterization of Soft B Separation Axioms in Soft Bi-Topological Spaces”. *Matrix Science Mathematic*, vol. 2, no. 2, pp. 11-17, 2018.
- [5] Z.H. Yan. “Modeling and Kinematics Simulation of Plane Ten-Bar Mechanism of Warp Knitting Machine Based on Simcape/Multibody”. *Acta Mechanica Malaysia*, vol. 2, no. 2, pp. 15-18, 2018.
- [6] N.F. Basir, S. Kasim, R. Hassan, H.N. Mahdin, A.A. Ramli, M.F. Md Fudzee, M.A. Salamat. “Sweet8bakery Booking System”. *Acta Electronica Malaysia*, vol. 2, no. 2, pp. 14-19, 2018.
- [7] E.O. Omini, O.M. Akpang. “Cavity Detection Under Re-Enforced Concrete Floor Using Ground Penetration Radar”. *Engineering Heritage Journal*, vol. 2, no. 2, pp. 11-18, 2018.
- [8] J. Wen, Y. Li. “Study of flow distribution and its improvement on the header of plate-fin heat exchanger”. *Cryogenics*, vol. 44, pp. 823–831, 2004.
- [9] C.C. Wang, K.S. Yang, J.S. Tsai, I.Y. Chen. “Characteristics of flow distribution in compact parallel flow heat exchangers, part II: Modified inlet header”. *Applied Thermal Engineering*, vol. 31, pp. 3235–3242, 2011.
- [10] O. Tonomura, S. Tanaka, M. Noda, M. Kano, S. Hasebe, I. Hashimoto. “CFD-based optimal design of manifold in plate-fin microdevices”. *Chemical Engineering Journal*, vol. 101, pp. 397–402, 2004.
- [11] L. Wang, Y. Fan, L. Luo. “Lattice Boltzmann method for shape optimization of fluid distributor”. *Computers & Fluids*, vol. 94, pp. 49–57, 2014.
- [12] J. Facão. “Optimization of flow distribution in flat plate solar thermal collectors with riser and header arrangements”. *Sol. Energy*, vol. 120, pp. 104–112, 2016.
- [13] N.V. Queipo, R.T. Haftka, W. Shyy, T. Goel, R. Vaidyanathan, P. Kevin Tucker. “Surrogate-based analysis and optimization”. *Progress in Aerospace Sciences*, vol. 41, pp. 1–28, 2005.
- [14] P. Fei, P.S. Hrnjak. “Adiabatic developing two-phase refrigerant flow in manifolds of heat exchangers”. 2004.
- [15] R.K. Shah, D. Sekulić. “Fundamentals of heat exchanger design”. John Wiley & Sons, ISBN 0-471-32171- 0, 2003.
- [16] E.W. Lemmon, M.L. Huber, M.O. McLinden. “REFPROP8 (Reference Fluid Thermodynamic and Transport Properties)”. NIST Standard Reference Database 23, Version 8.0, National Institute of Standards and Technology, 2007.