Enhancement of Heat Transfer in Steam Generator using Turbulentization at its unique Geometry

Joonyoung Sung, Jaeyoung Lee [*]
Department of Mechanical and Control Engineering, Handong Global Univ., Pohang, 791-708, Korea
[*]*Corresponding author: jylee7@handong.edu*

1. Introduction

Printed circuit heat exchangers (PCHEs) have been previously investigated regarding their ability to enhance heat transfer rates [1–4]. The performance of heat exchangers has been evaluated using both experimentation and computational fluid dynamics (CFD) simulation [5, 6]. Prior studies have attributed the observed or estimated changes in heat transfer or pressure to the geometric configuration of the heat exchangers [7–10]. Other studies considered PCHE performance and resultant heat transfer enhancements on the basis of PCHE shape characteristics (e.g., S-shape or airfoil) [11–15]. Previous researches on PCHEs commonly aimed to change the channel shape to enhance the heat transfer while reducing pressure loss in planar characteristics. However, PCHEs with a three dimensional channels are believed to have an advantage that could make mixing flow with enhanced heat transfer. In the present work, a new type of channel, i.e., a tangled fluid channel that can exhibit three-dimensional fluid flow behavior is investigated to figure out the enhancement of heat transfer with turbulentization generated by its unique geometry. This study expects to be based on the application on sodium intermediate heat exchanger.

2. Methods

2.1 PCHE geometry

The dimensions of the PCHE investigated in this study were 225 145 4.5 mm^3, and the heat exchanger was made of stainless steel. The PCHE consisted of three metal plates that were stacked sequentially. Fig. 1(a) shows the model created by computer-aided design, and Fig. 1(b) shows the real plates. As depicted in Fig. 1, the top and bottom plates were etched with a wavy shape. The middle plate was different from top and bottom plates; although the channel configuration was two-dimensional for the top and bottom plates, those of the middle plate made it possible to achieve a three-dimensional flow. The main feature of the middle plate was the numerous holes located at regular intervals. These holes mix the flow at a low flow rate. The ultimate aim of this PCHE was to create turbulent flow even though the flow regime was laminar.

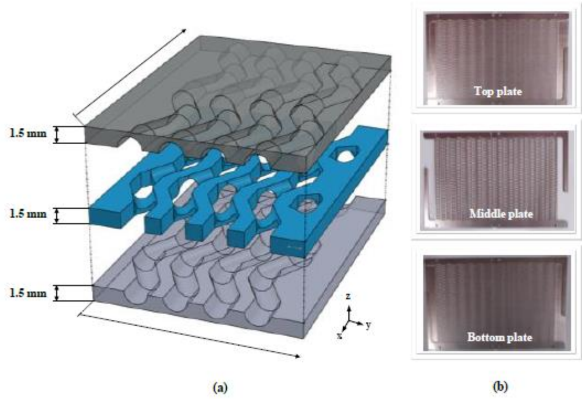

Fig. 1. (a) The exploded configuration in a part of the PCHE and (b) the divided plates of the PCHE.

2.2 Computational fluid dynamics simulation

In this study, a CFD simulation was performed using the ANSYS CFX 17.0 software. To determine the effect of the Reynolds number on turbulence occurrence, Reynolds numbers ranging from 50 to 2,000 were considered (the transition of flow characteristics is observed in this range). For steady-state simulation conditions, a consistent timescale was used for each test to ensure reliable results.

Figure 2 shows a magnified view of an individual unit component. It was hypothesized that any geometric effects would occur at the mixing zones (designated as Zone 7 in Fig. 2). Within each individual unit component, the flow bifurcates upon exiting Zone 1, follows the channel through Zones 2–6, and recombines upon entering Zone 7. In Zone 7, the inclined crossed-path flow causes a mixing effect. This flow pattern may result in turbulence even with low Reynold numbers. Because the geometry of each individual unit component was symmetric in the virtual y-z plane, only half of the unit component (i.e., Zones 2–6) was considered in the simulation. Thus, in this study, the degree of the turbulence and subsequent turbulentization was quantitatively determined for Zones 1–7 in Fig. 2. Because the turbulence was irregular, statistical rather than deterministic methods were used.

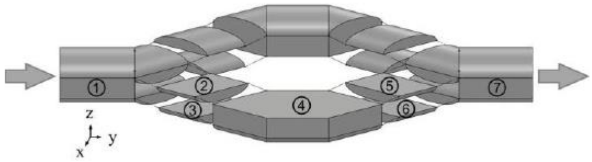

Fig. 2. Magnified view of an individual unit component.

The meshes used in the CFD simulation were controlled using an ANSYS meshing tool. Figure 3 shows the final meshes for the unit component. Mesh tests that check for differences in inlet and outlet temperatures and pressures were conducted for select areas of the unit component. Mesh characteristics, including inflation or element size, were adjusted until the simulation results showed <1% deviation from the previous mesh test result.

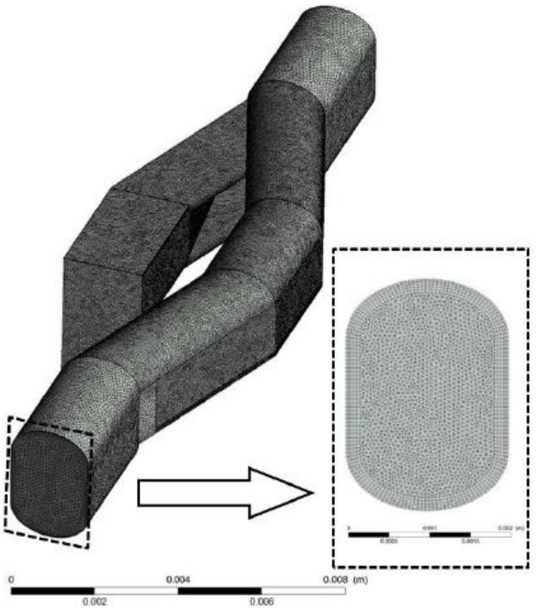

Fig. 3. Unit component mesh to support CFD simulation.

3. Results

Figure 4 depicts an individual unit component's velocity streamlines for different Reynolds numbers. The velocity streamline legend was unified using logarithmic scales for all Reynolds number ranges. The streamlines visually confirmed non-mixed laminar flow for Reynolds numbers <400. For Reynolds numbers >400, however, the streamlines gradually began mixing near the unit component's outlet (in Zone 7 in Fig. 2). As the streamline passed through the mixing zone, heat transfer was assumed to increase. Although this analysis visually confirmed the hypothesized mixing effect, further quantitative methods are required to determine the degree of turbulence and associated turbulentization.

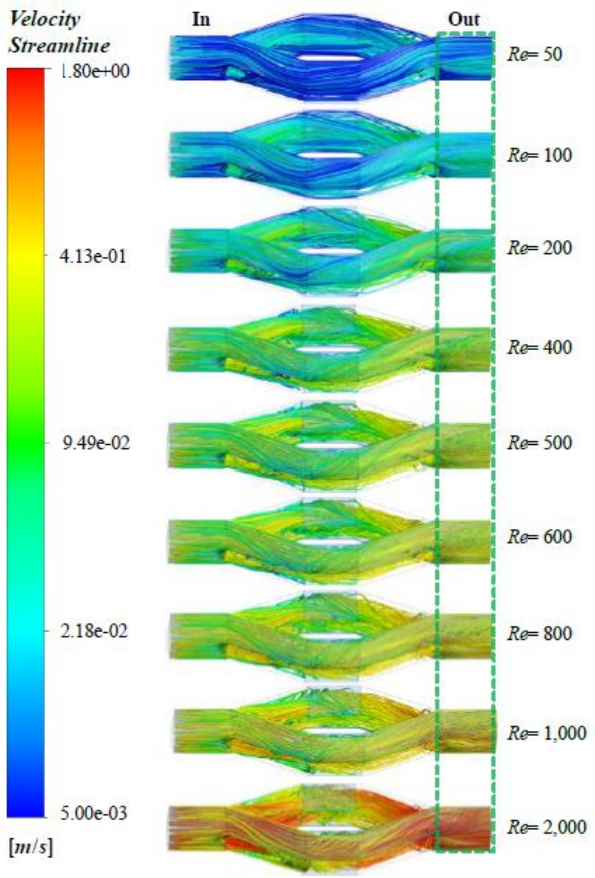

Fig. 4. An individual unit component's velocity streamlines for different Reynolds numbers.

To quantitatively determine the degree of turbulence and associated turbulentization, a CFD simulation was performed for Zones 1–7 of the unit component (see Fig. 2). The turbulentization was subsequently calculated using the results of the simulation. Figure 5 shows the calculated turbulentization for Zones 2–7 and for different Reynolds numbers. The turbulentization for Zone 1 is not included in Fig. 5 because the CFD simulation could not adequately describe the uniform flow phenomenon at the unit component's inlet.

Derived from the turbulence model, turbulentization is determined by a numerical value of 1 [16]. When its numerical value is >1, the turbulence kinetic energy produced is more dominant than the turbulence kinetic energy dissipated (recall that turbulentization is defined as the ratio of the turbulence kinetic energy produced to the turbulence kinetic energy dissipated). Some caution may be warranted, however, when defining turbulence under these conditions. The thermal-hydraulic performance, reflected as heat transfer or pressure loss, resembles turbulent rather than laminar flow in the dominant region of turbulence kinetic energy production. In this study, however, non-mixing zones such as Zone 4 became a mixing zone when multiple unit components were serially connected to form a long-length tangled fluid channel. This finding suggests that turbulentization may be more prevalent.

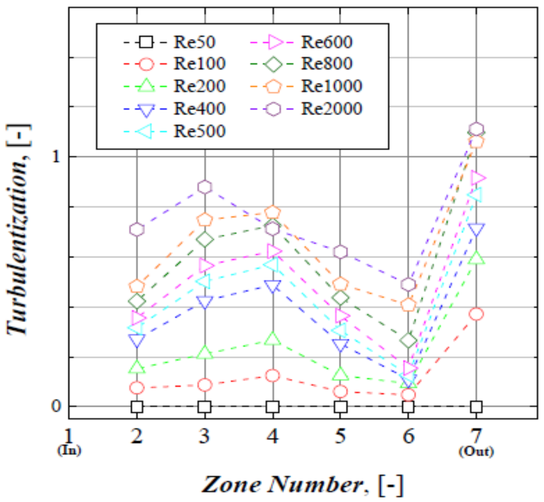

Fig. 5. An individual unit component's calculated turbulentization for different Reynolds numbers and for Zones 2–7.

Figure 6 shows heat transfer (expressed as a Nusselt number) and pressure loss (expressed as various friction factors) as a function of the Reynolds number for an individual unit component. The second friction factor is classically defined for laminar flow in a circular tube with a Reynolds number <2,300 and is determined as 64/*Re*.

The heat transfer rate changed at a Reynolds number of 200; the slope of a heat transfer rate increased for Reynolds numbers >200. Also at a Reynolds number of 200, the slopes of the two friction factors (*f* and *f* = 64/*Re*) substantially diverged. These graphical results suggested turbulent flow for Reynolds numbers >200, which is well below the Reynolds number of 2,300 commonly used to define turbulent flow. This discrepancy suggests the need for further investigation (qualitative and quantitative) regarding the determination of laminar or turbulent flow.

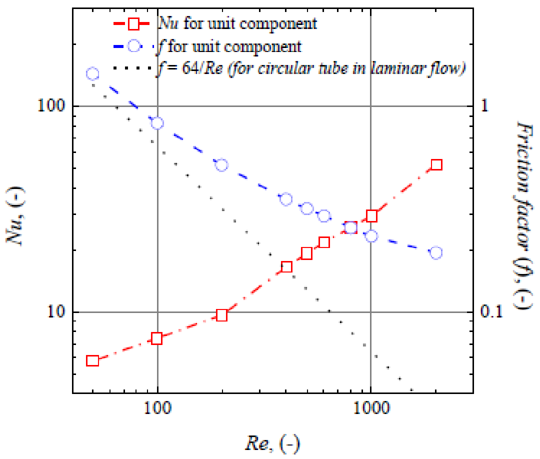

Fig. 6. Heat transfer and pressure loss as a function of the Reynolds number for an individual unit component.

4. Conclusion

In this study, we evaluated the flow characteristics in a single unit component of a tangled fluid channel using both qualitative and quantitative methods. Analysis using non-dimensional parameters and graphical visualizations were used to qualitatively determine the flow characteristics in a single unit component. Results indicated that the heat transfer rate (expressed as a Nusselt number) changed at a Reynolds number of 200; the slope of a heat transfer rate increased for Reynolds numbers >200. Also at a Reynolds number of 200, the slopes of the two friction factors (*f* and *f* = 64/*Re*) representing pressure loss substantially diverged. These qualitative results suggested turbulent flow for Reynolds numbers >200, which is well below the Reynolds number of 2,300 commonly used to define turbulent flow. In addition, visual assessment of the unit component's velocity streamlines for different Reynolds numbers indicated mingled flows in the mixing zone for Reynolds numbers >400.

For the single unit component, results indicated that turbulentization in each simulated zone increased as the Reynolds number increased. The mixing zone (Zone 7) had markedly higher turbulentization values; in this zone, the turbulence kinetic energy produced is higher than the turbulence kinetic energy dissipated. A microscopic investigation of the mixing zone's flow indicated that turbulent flow and associated turbulentization occurred for Reynolds numbers ≥500 and was highest along the side of the outlet.

The computational fluid dynamics simulations and numerical turbulentization values were used to quantitatively characterize the turbulent flow in a single unit component. The concept of turbulentization was introduced to support turbulence quantification.

ACKNOWLEDGMENTS

This work was supported by the Nuclear Safety Research Program through the Korea Foundation Of Nuclear Safety (KOFONS), granted financial resource from the Nuclear Safety and Security Commission (NSSC), Republic of Korea (No. 1305008- 0416-SB130).

REFERENCES

[1] Q. Li, G. Flamant, X. Yuan, P. Neveu, L. Luo., Compact heat exchangers: A review and future applications for a new generation of high temperature solar receivers, Renewable and Sustainable Energy Reviews 15 (2011) 4855-4875

[2] S.V.Garimella and C.B.Sobhan, Transport in microchannels -a critical review, Annual Review of Heat Transfer v.13.30 (2003) 1-50

[3] M. Asadi, G. Xie, B. Sunden, A review of heat transfer and pressure drop characteristics of single and two-phase microchannels, International Journal of Heat

and Mass Transfer 79 (2014) 34-53

[4] N.Garcia-Hernando, A.Acosta-Iborra, U.Ruiz-Rivas, M.Izquierdo, Experimental investigation of fluid flow and heat transfer in a single-phase liquid flow micro-heat exchanger , International Journal of Heat and Mass transfer 52 (2009) 5433-5446

[5] W. Qu, I. Mudawar, Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink, Internation Journal of Heat and Mass Transfer 45 (2002) 2549-2565

[6] Y. Mishan, A. Mosyak, E. Pogrebnyak, G. Hetsroni, Effect of developing flow and thermal regime on momentum and heat tranfer in micro-scale heat sink, Internation Journal of Heat and Mass Transfer 50 (2007) 3100-3114

[7] Merle C. Potter, David C. Wiggert, Bassem H. Ramadan, Mechanics of fluids, Fourth ed., Cengage Learning, 2011, pp.105.

[8] S.Kandlikar, S. Joshi, S. Tian, Effect of channel roughness on heat transfer and fluid flow characteristics at low Reynolds numbers in small diameter tubes, Heat Trnasfer Engineering 24 (2003) 4-16

[9] X.F.Peng, G.P.Peterson, and B.X.Wang, Heat transfer characteristics of water flowing through microchannels, Experimental Heat Transfer 7 (1994) 265-283

[10] D. Pfund, D. Rector, A. Shekarriz, A. Popescu, J. Welty, Pressure drop measurements in a microchannel, American Institute of Chemical Engineers 46 (2000) 1496-1507

[11] N. Tsuzuki, Y. Kato, T. Ishiduka, High performance printed circuit heat exchanger, Applied Thermal Engineering 27 (2007) 1702-1707

[12] T. Ngo, Y. Kato, K.n Nikitin, T. Ishizuka, Heat transfer and pressure drop correlations of microchannel heat exchangers with S-shaped and zigzag fins for carbon dioxide cycles, Experimental Thermal and Fluid Science 32 (2007) 560-570

[13] N. Tsuzuki, M. Utamura, T. Ngo, Nusselt number correlations for a microchannel heat exchanger hot water supplier withS-shaped fins, Applied Thermal Engineering 29 (2009) 3299-3308

[14] X. Xu, T. Ma, L. Li, M. Zeng, Y. Chen, Y. Huang, Q. Wang, Optimization of fin arrangement and channel configuration in an airfoil fin PCHE for supercritical CO2 cycle, Applied Thermal Engineering 70 (2014) 867-875

[15] T. Ma, F. Xin, L. Li, X. Xu, Y. Chen, Q. Wang, Effect of fin-end wall fillet on thermal hydraulic performance of airfoil printed circuit heat
exchanger, Applied Thermal Engineering 89(2015) 1-9

[16] R. Narashima and K. R. Sreenivasan,, Relaminarization of Fluid Flows, Advances in Applied Mechanics 19 (1979) 221-309