CFD-BASED INVESTIGATION OF HEAT TRANSFER CHARACTERISTICS OF FLUE GAS-WATER HEAT EXCHANGER PANELS PRODUCED WITH A NOVEL MANUFACTURING PROCESS

T. FUKUE[1], C. SPITAS[2], M. DWAIKAT[3] and M. ISHIZUKA[4]

[1] Department of Mechanical Engineering, Iwate University
4-3-5 Ueda, Morioka-Shi, Iwate 020-8551 Japan

[2] Department of Design Engineering, Delft University of Technology
Landbergstraat 15, 2628 CE Delft, The Netherlands

[3] Department of Civil Engineering, An-Najah National University
2520 Engineering Building Nablus, Palestine

[4] Toyama Prefectural University
5180 Kurokawa, Imizu-Shi, Toyama 939-0398, Japan

ABSTRACT

This study describes the characteristics of a panel geometry obtained by a novel manufacturing process for the development of ultra-compact flue gas-water condensing heat exchangers. In this process two stainless steel sheets are prepared and stacked and then brazed together in a special pattern outlining a desired configuration of water ducts and connecting manifolds. By injecting pressurized fluid to the clearance between the bonded sheets, the sheets are deformed and the enlarged clearance becomes a flow pass of water. By stacking several such bonded panels together, the clearances between them work as flue-gas passages. The principle of using this process is that panels can be produced more economically and with sufficient control over the parameters of the obtained geometry as well as the strength. Because of this manufacturing process, the cross-section of the water duct obtains a special non-circular shape, so the heat transfer characteristics must be specifically assessed for the new geometry, in order to qualify the new design. Through CFD analysis, we evaluate the heat transfer performance of the water duct in the novel heat exchanger by comparing with the heat transfer performance of a standard circular duct which has the same hydrodynamic diameter as the investigated water duct.

INTRODUCTION

In the recent years, an urgent demand of an exhaust heat exchanger which collects heat from thermal equipment’s flue gas is developing to improve thermal efficiency of thermal equipment and save energy(1). To incorporate an exhaust heat exchanger into thermal equipment at low cost, a new concept heat exchanger, which has high heat exchange performance while keeping down a production cost, is urgently demanded.

We are now developing a novel ultra-compact condensing flue-gas/water heat exchanger: UCCHE-A (Ultra-Compact Condensing Heat Exchanger, Type A)(2)(3). Figure 1 shows the whole image of the heat exchanger. It is designed for use with boilers firing natural gas or other sulphur-free fuel. The heat exchanger comprises a lightweight compact stainless-steel structure which can sustain 10 bar of water pressure.

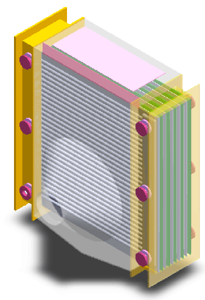


Figure 1. Whole Design of UCCHE-A.

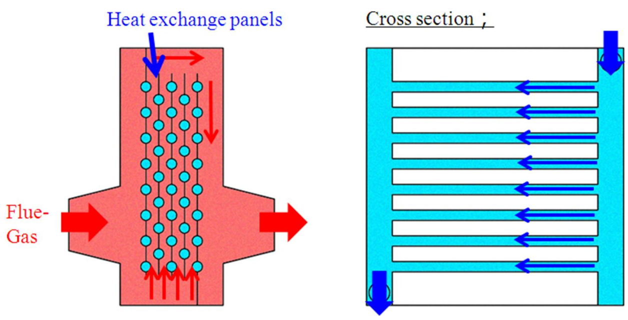


Figure 2. Inside Structure of UCCHE-A.

The total assembly is mainly composed of an enclosure and a heat exchange part as shown in Figure 2. To develop the heat exchanger without compromising versatility, a cheaper and high-reliability processing method have to be chosen. Especially, to sustain high water pressure while increasing heat exchange performance and decreasing production cost, the optimization of the processing method and the structure of the heat exchange part become important.

To solve the problem of the processing cost, we propose to produce the heat exchange part by using the following manufacturing process, as shown in Fig. 3: two stainless steel sheets are prepared. The sheets are stacked and then brazed together in a special pattern outlining a desired configuration of water ducts and connecting manifolds. The high pressure fluid is injected to the clearance between the bonded sheets. Due to the high pressure of the fluid, the

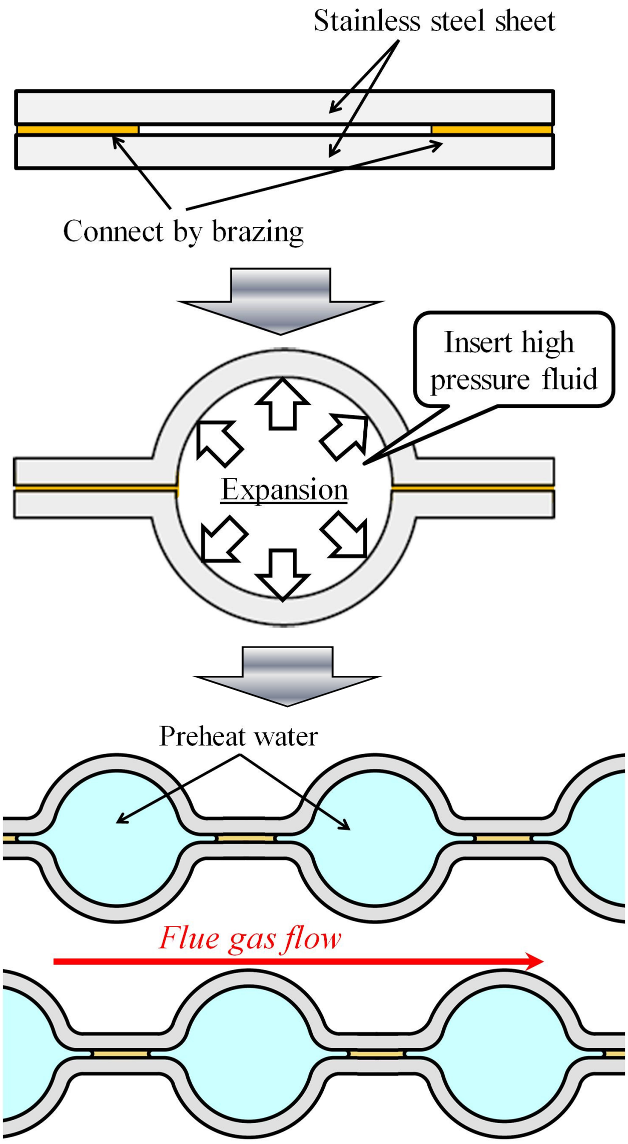


Figure 3. Image of how to make the heat exchange part.

sheets are plastically deformed and the flow pass for preheat water is composed. Depending on the manufacturing parameters, the produced cross-section is near-elliptical, as opposed to the standard circular shape used in some plate and all shell-and-tube heat exchangers. By stacking several such bonded panels together, the clearances between them work as flue-gas passages.

The principle of using this process is that panels can be produced more economically and with sufficient control over the parameters of the obtained geometry as well as the strength. However, to qualify the use of such a production method, the heat transfer characteristics of the proposed panel geometry should be investigated in order to evaluate the heat exchange performance of the novel heat exchanger.

With these as background, we are now investigating the flow and heat transfer phenomena in the novel heat exchanger which is produced by the proposed manufacturing process. In this paper, as the first investigation in order to clarify the heat exchange process and the performance of the heat exchanger, the flow and heat transfer characteristic of the flow pass of the water which is composed between the produced panels were investigated by using 3D Computational Fluid Dynamics (CFD) analysis. Because of the manufacturing process of the water duct, the cross-section of the water duct has a special non-circular near-elliptical shape. We evaluated the heat transfer performance of the water duct in the novel heat exchanger by comparing with the heat transfer performance of a standard circular duct which has the same hydrodynamic diameter as the proposed water duct.

Table 1. List of analytical target.

Type	Thick of the panel [mm]	Applied fluid pressure [MPa]	Span of the brazing [mm]	Hydraulic diameter: *d* [mm]
1	0.25	1	10	9.53
2	0.25	2	10	10.89
3	0.25	4	10	11.38
4	0.25	1	20	22.75
5	0.25	2	20	23.20
6	0.50	1	20	19.05
7	0.50	2	20	21.79

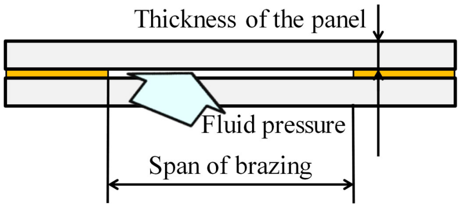
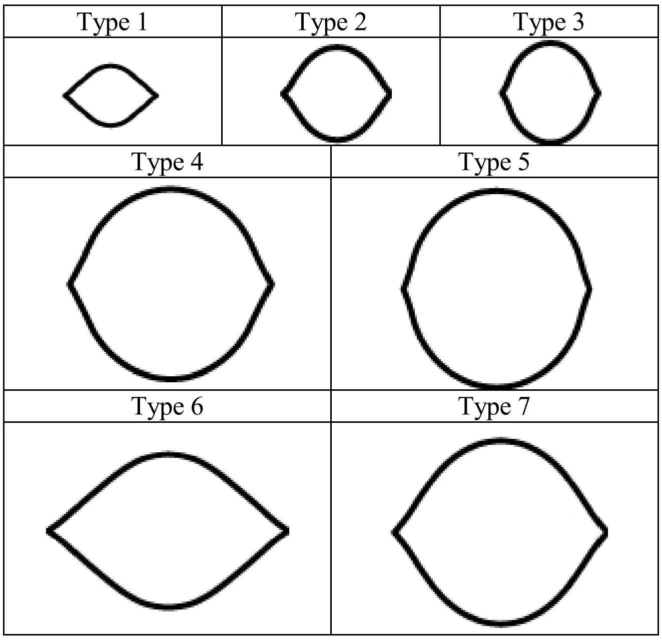


Table 2. Shape of each type of the test duct.



WATER DUCT SHAPE

Table 1 shows the list of the analytical model. Table 2 shows the shape of the cross section of the water duct of the heat exchanger. Here, “Applied fluid pressure” shows the pressure of the injected fluid in order to expand the panel and produce the water duct. “Span of the brazing” is the distance between the edges of the brazing material. The hydraulic diameter d [m] is defined by the following formula;

$$d = \frac{4A_{\mathrm{d}}}{C} \tag{1}$$

Here, A_{d} [m^2] is the cross section area of the water duct and C [m] is the perimeter of the cross section of the water duct. 7 types of the water duct were investigated in order to evaluate heat transfer characteristics and flow resistance characteristic. In Types 1 through 3 the thickness of the panel and the span of the brazing are kept the same, whereas the applied fluid pressure is changed. Due to the change of the pressure, the level of the expansion of the panel become different and the produced cross section area of the duct is changed. In Types 4 and 5, the thickness of the panel is the same as in Types 1 ~ 3, however the span of the brazing is changed. In Types 6 and 7, the span of the brazing part was the same as Types 4~ 5, whereas the thickness of the panel was double of Type 1 ~ 5. These parameters are summarized in Table 1 and the produced cross-sections are shown in Table 2. The shape of the water duct was preliminarily obtained by non-linear Finite Element Analysis (FEA), in consideration of material and geometric nonlinearities. Through the flow and heat transfer analysis in 7 types of the water duct, we investigated the net heat transfer performance of the proposed water duct. In addition, in order to compare the heat transfer characteristics with the general circular duct, we additionally performed the flow and heat transfer analysis of the circular ducts which has the same inside diameter as the hydraulic diameter of each type.

ANALYTICAL MODEL AND METHOD

Figure 4 shows the image of the analytical model used in this report. We tried to perform 3-dimentional CFD analysis in the water duct. The length of the analytical model L was 500 mm regardless of the duct type. With regard to the boundary condition of the analytical model, a uniform velocity was applied to the inlet of the duct. Temperature of the supplied water was set at 293 K. Constant pressure was applied to the outlet of the duct. Constant temperature (303 K) and a non-slip wall condition were applied to the wall of the duct. For this setup, we simulated the developing flow and heat transfer from the inlet of the duct through the outlet of the duct. Table 3 shows the analytical conditions. For CFD analysis, a commercial code, "ANSYS Fluent" developed by ANSYS Inc. was used. Water flow and forced convection in the water duct were calculated. Reynolds number of the water was set at between 300 and 1800 therefore we assumed that the flow in the duct was laminar flow. The steady analysis was performed. The mesh number was about 200,000 regardless of the type of the duct.

EVALUATION METHOD

In order to evaluate the flow and heat transfer characteristics in the water duct, we used the following average Nusselt number and Fanning friction factor.

The average Nusselt number between the duct inlet and the duct outlet Nu_{m} [-] was defined as the following formula.

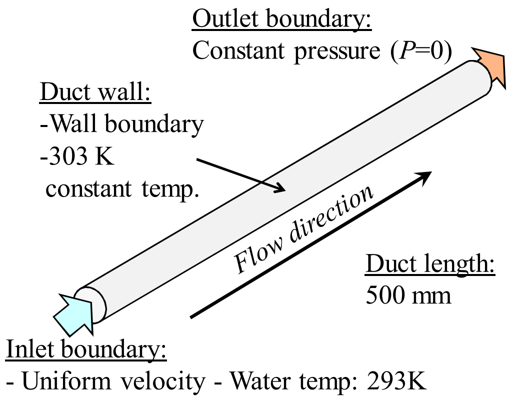


Figure 4. Analytical model and boundary conditions.

Table 3. Analytical Conditions.

CFD Codes	ANSYS Fluent
Mesh Number	About 200,000
Laminar/Turbulent	Laminar Flow Analysis
Steady/Transient	Steady Analysis
Reynolds number	300 ~ 1800
Working fluid	Water
Water duct length L	500 mm

$$Nu_{\mathrm{m}} = \frac{h_{\mathrm{m}} d}{\lambda} \tag{2}$$

Where λ [W/(m·K)] is thermal conductivity of the water. h_{m} [W/(m^2·K)] is the average heat transfer coefficient between the duct inlet and the duct outlet. h_{m} is obtained from the following formula.

$$h_{\mathrm{m}} = \frac{Q}{\Delta T_{\mathrm{lm}} A_{\mathrm{h}}} \tag{3}$$

Where A_{h} (= $C \times L$) is the heat transfer area in the water duct. ΔT_{lm} [K] is the logarithmic-mean temperature difference which is defined as the following formula:

$$\Delta T_{\mathrm{lm}} = \frac{T_{\mathrm{out}} - T_{\mathrm{in}}}{\ln \dfrac{T_{\mathrm{wall}} - T_{\mathrm{in}}}{T_{\mathrm{wall}} - T_{\mathrm{out}}}} \tag{4}$$

Where T_{wall} [K] is the wall temperature (303 K), T_{in} is the inlet temperature of the water (293 K) and T_{out} is the outlet temperature of the water. Q [W] is the heat value which the water is received in the duct. Q was estimated by using the following formula.

$$Q = A_{\mathrm{d}} U_{\mathrm{d}} \rho c \left(T_{\mathrm{out}} - T_{\mathrm{in}} \right) \tag{5}$$

Where U_{d} [m/s] is the mean velocity at the cross section of the duct, ρ [kg/m^3] is density of the water and c [J/(kg·K)] is specific heat of the water. By using the analysis, we obtained T_{out} and evaluated heat transfer performance by using the proposed Nu_{m}.

In order to investigate pressure drop characteristic, we used the following Funning friction factor C_f.

$$C_f = \Delta P \cdot \frac{d}{2 L \rho U_{\mathrm{d}}^{\,2}} \tag{6}$$

ΔP [Pa] is the pressure drop between the duct inlet and the duct outlet. Reynolds number of the duct flow was defined as the following formula.

$$Re = \frac{U_{\mathrm{d}} d}{\nu} \tag{7}$$

Where ν [m^2/s] is kinematic viscosity of the water. By using the following relationships, we evaluated the flow and heat transfer in the duct while changing the type of the cross section.

ANALYTICAL RESULT

To begin with, we describe the analytical result when the thickness of the panel is 0.25 mm and the span of the brazing is 10 mm (Type 1 ~ 5).

Figures 5 and 6 show the relationship among Re, Nu_{m} and C_f respectively. In these graphs, the results in the case of the circular duct which has the same diameter as the hydraulic diameter of each types of the heat exchanger ducts. We can see that the average Nusselt number becomes higher when Reynolds number increases. Moreover, the Fanning friction factor becomes lower when Reynolds number increases. Regardless of the type of the duct, the value of the Nusselt number and Fanning friction factor becomes almost the same and these values change dependent on the Reynolds number. The difference of the average Nusselt number and the friction coefficient between Type1 and Type 3 was about 10 % in the case of Re = 1800. Therefore we can say that the flow and heat transfer characteristic of the water duct which is produced in the novel heat exchanger is not affected significantly by the shape of the duct's cross section.

Hereafter, we will compare the difference of flow and heat transfer characteristics between the heat exchanger duct and the general circular duct. From the result, we observe that the friction coefficient remains almost the same regardless of the shape of the cross section if the hydraulic diameter is the same. On the other hand, even if the hydraulic diameter is the same, the average Nusselt numbers of the circular duct become slightly higher than the value of the heat exchanger duct. This may be caused by the contracted area in the heat exchanger duct. This can be observed near the brazed position. Around the brazed position, the panel is constricted during forming by the brazing and the level of the expansion is inhibited. Therefore the clearance between the panels remains narrow near the brazed edge, the effect of the friction on the flow becomes higher and the water flow velocity near the brazed edge is inhibited. This causes the decrease of the average Nusselt number in the heat exchanger duct.

Next, we will confirm the analytical result when the span of the brazing and the thickness of the stainless panel are changed. Figures 7 though 10 show the analytical result obtained for Type 4 through Type 7. The almost same tendency of the average Nusselt number and the Fanning friction factor as seen in the results for Type 1 ~ 3 can be obtained. In this case, both the average Nusselt number and the friction coefficient become higher than the result of Types 1, 2 and 3. This is caused by the increase of the contact area between the water and the wall. In the case of Type 4 through Type 7, the span of the

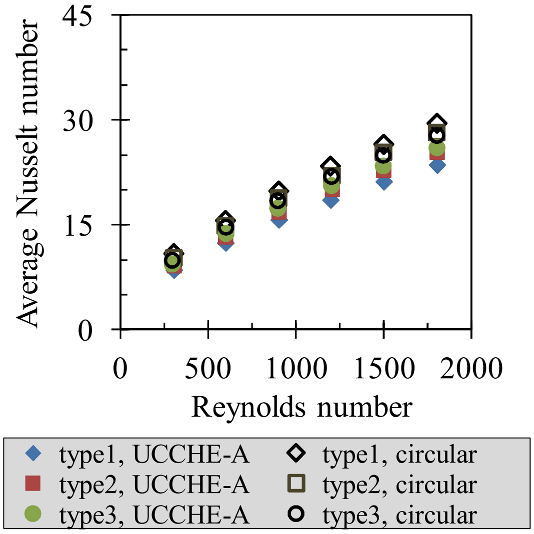


Figure 5. The relationship between Re and Nu_{m} when the thickness of the panel is 0.25 mm and the span of the brazing is 10 mm.

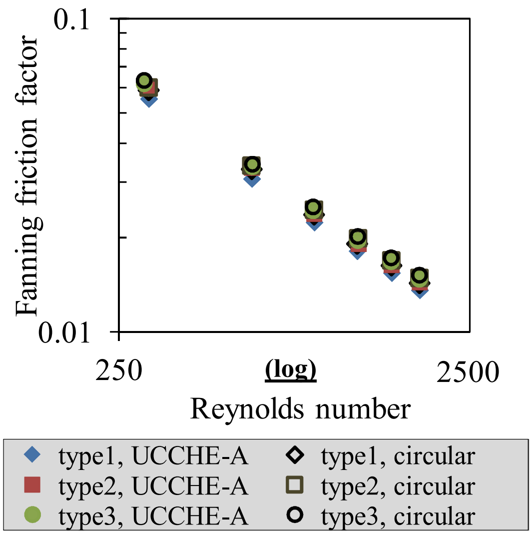


Figure 6. The relationship between Re and C_f when the thickness of the panel is 0.25 mm and the span of the brazing is 10 mm.

brazing position is double that of Type 1 ~ 3. Therefore the area of the cross section becomes larger and the contact area between the wall and the water also becomes larger. This affects the heat transfer performance and the friction factor.

Here, in the case of Type 4 and 5, the difference of the heat transfer and the friction coefficient between the heat exchanger duct and the circular duct is small. From Table 2, we observe that the shape of the cross section of the heat exchanger duct is almost circular. Therefore it is to be expected that there is little difference in flow and heat transfer between the circular duct and the heat exchanger duct. On the other hand, in the case of Type 1 through 3 and Type 6 and 7, even if the hydraulic diameter is the same, if the shape of the cross section is different, the difference of the Nusselt number and the Fanning friction

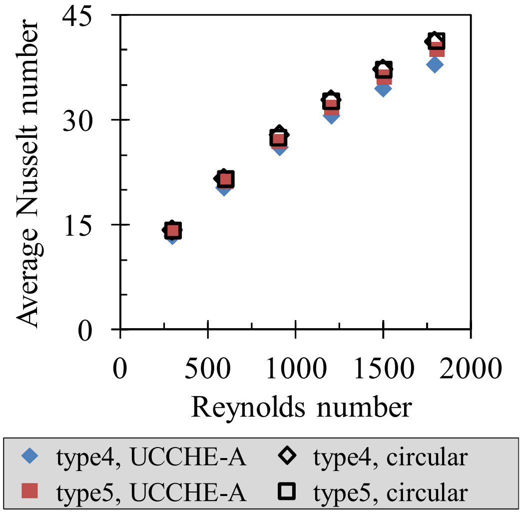


Figure 7. The relationship between *Re* and Nu_m when the thickness of the panel is 0.25 mm and the span of the brazing is 10 mm.

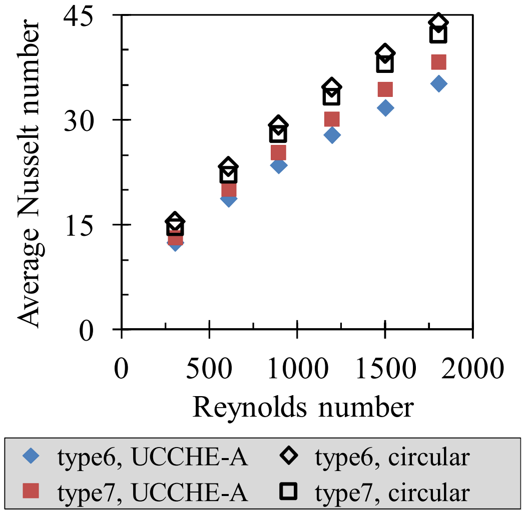


Figure 9. The relationship between *Re* and Nu_m when the thickness of the panel is 0.25 mm and the span of the brazing is 10 mm.

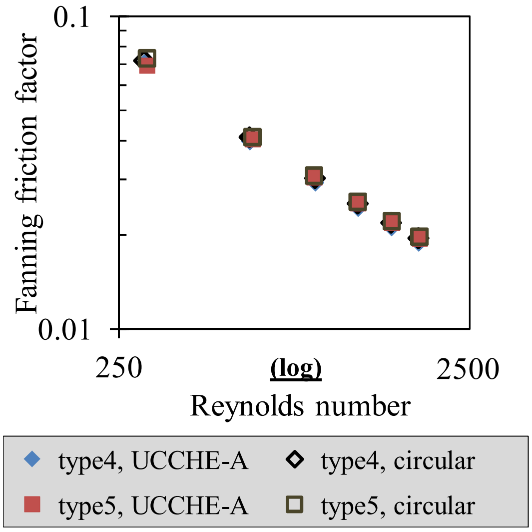


Figure 8. The relationship between *Re* and C_f when the thickness of the panel is 0.50 mm and the span of the brazing is 20 mm.

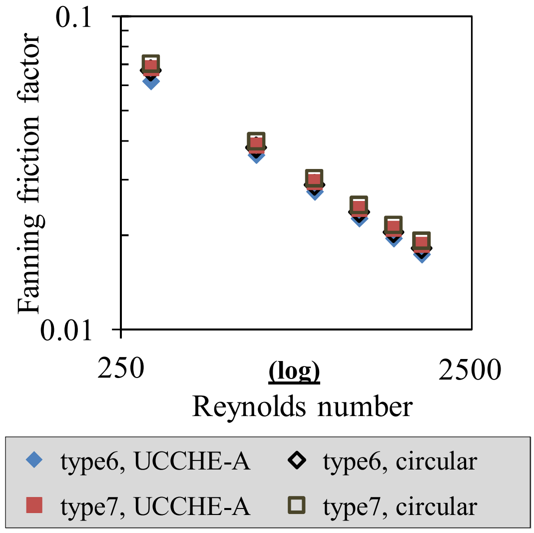


Figure 10. The relationship between *Re* and C_f when the thickness of the panel is 0.50 mm and the span of the brazing is 20 mm.

factor becomes slightly larger. In these types, the produced cross-section is near-elliptical and the decrease of the flow velocity is caused near the brazing position as mentioned above. Because of this, the flow phenomena become different from the circular duct, resulting in a difference in heat transfer.

From these discussions, we can conclude that the effect of the shape of the water duct on the flow and heat transfer is relatively small when the span of the brazing is the same. Especially, when the cross section shape of the duct is near the circular, the obtained flow and heat transfer characteristic is the almost same as in a circular duct with the same hydraulic diameter. However, when the shape of the cross section becomes near-elliptical from the circular such as type-1, 2, 6 and 7, the obtained heat transfer characteristics become relatively smaller in comparison to a circular duct with the same hydraulic diameter.

CONCLUSIONS

In order to develop a novel ultra-compact condensing heat exchanger which can be produced by a novel manufacturing process, we investigated the flow and heat flow pass of the water were investigated in order to clarify transfer phenomena in the heat exchanger analytically. In this paper, the flow and heat transfer characteristic of the heat exchange process and the performance of the heat exchanger. We evaluated the heat transfer performance of the water duct in the heat exchanger by comparing with the heat transfer performance of a standard circular duct which has the same hydrodynamic diameter as the proposed water duct.

The flow and heat transfer characteristic of the water duct which is produced in the novel heat exchanger is not affected significantly by the shape of duct's cross section if the span of the brazing is the same. However, depending on the type of the duct, the heat transfer performance decreases from the circular duct which has the same hydraulic diameter. When the cross section shape of the duct is near-circular, the obtained flow and heat transfer characteristic is almost same as the circular duct which has the same hydraulic diameter. On the other hand, when the produced cross-section is near-elliptical, the decrease of heat transfer performance causes. Therefore the optimization of the shape of the water duct becomes important.

As our future work, we should confirm the accuracy of the analysis by means of experimental framework. In addition, in order to clarify the net heat exchange performance of the heat exchanger, we should investigate the flow and heat transfer on the flue-gas side panel. However, as shown in Fig. 3, the shape of the flue-gas side passage is complex. Prior research has been reported on the complex flow and heat transfer on a wavy wall[4] or around a wavy obstruction which simulates a hill[5]. In order to investigate the complex flow and heat transfer without calculation error, we have to choose the suitable analysis methods. After getting enough information about the heat transfer and the flow of both flue-gas and water ducts, we will try to optimize the structure of the heat exchanger. In addition, we will try to develop the rapid design method of the heat exchanger[6][7] in order to improve the efficiency of the heat exchanger design and decrease the production cost.

REFERENCES

(1) Wakashima, S., Hoshi, A. and Yamada, N. (2012): "Numerical Analysis of Small Organic Rankine Cycle System Combined with Latent Heat Thermal Storage (in Japanese)", *Proceedings of the 49th National Heat Transfer Symposium in Japan*, Paper No., E114.

(2) Spitas, C., Kazilas V., Spitas V. (2009): An Ultra-Compact Gas-Liquid Heat Exchanger for High Pressure Differentials and Low Pressure Losses, Patent GR1006579

(3) Spitas, C. and Song, Y. (2010): "Thermal Analysis of the UCCHE-A Heat Exchanger using an Element Method with Localised Flow Properties", *Proceedings of the 21st International Symposium on Transport Phenomena*.

(4) Kanasaki, T., Tanaka, K., Yamauchi, T., Nakagawa, S. and Ishizuka, M. (2008): "Turbulent Flow and Heat Transfer in a Channel with a Wavy Wall", *Proceedings of the 2nd International Forum on Heat Transfer*.

(5) Yanaoka, H., Inamura, T., Suenaga, Y. and Kobayashi, Y. (2007): "Numerical Simulation of Vortex Structures and Heat Transfer behind a Hill in a Laminar Boundary Layer (in Japanese)", *Transactions of the Japan Society of Mechanical Engineers*, 73-736, pp. 2537-2544.

(6) Hirose, K., Fukue, T., Nakagawa, F., Ito, R., Wauke, T., Hoshino, H. and Terao, H. (2013): "Basic Study on Transient Temperature Response of Papers in a Thermal Transfer Printer", *Proceedings of the ASME InterPack'13 Conference*, Paper No. IPACK2013-73088.

(7) Fukue, T., Ishizuka, M., Nakagawa, S., Hatakeyama, T. and Nakayama, W. (2010): "Resistance Network Analysis of Airflow and Heat Transfer in a Thin Electronic Equipment Enclosure with a Localized Finned Heat Sink", *Proceedings of the 14th International Heat Transfer Conference* (2010), Paper No. IHTC14-22979.