Thermo-fluid dynamic design optimization of a concentric tube heat exchanger

Tomohiro HIRANO*, **, Mitsuo YOSHIMURA*, Koji SHIMOYAMA* and Atsuki KOMIYA*

*Institute of Fluid Science, Tohoku University
2-1-1 Katahira, Aoba-ku, Sendai, Miyagi 980-8577, Japan
**Department of Aerospace Engineering, Tohoku University
6-6 Aramaki Aza Aoba, Aoba-ku, Sendai, Miyagi 980-8579, Japan
E-mail: tomohiro.hirano.t2@dc.tohoku.ac.jp

Received: 12 June 2019; Revised: 23 July 2019; Accepted: 24 October 2019

Abstract
This study proposes a shape optimization approach for the cross-sectional shape of the inner pipe of a counter-flow concentric tube heat exchanger. The cross-sectional shape of the inner pipe is expressed by an algebraic expression with a small number of parameters, and their heat transfer performance is evaluated by a commercial Computational Fluid Dynamics (CFD) solver. The optimization is conducted by the Non-Dominated Sorting Genetic Algorithm II (NSGA-II) assisted by the Kriging surrogate model, and the NSGA-II finds the optimal cross-sectional shape with many protrusions around the perimeter of the inner channel to improve the heat transfer performance. In this study, heat transfer performance is evaluated from the temperature drop at the outlet of the high-temperature fluid. The present optimization finds the optimal channel with many protrusions, which achieves lower outlet temperature than a circular channel even with the same heat transfer surface area. This result indicates that the number of protrusions plays important roles which contribute not only to increase heat transfer area but also to improve heat transfer performance.

Keywords: Shape optimization, Computational fluid dynamics, Counter-flow concentric tube heat exchanger, Genetic algorithm, Kriging model

Nomenclature

- $a$: design parameter in Eq. (1) [-]
- $A$: cross-sectional area [mm$^2$]
- $b$: design parameter in Eq. (1) [-]
- $c$: specific heat [J/(kg·K)]
- $C$: heat capacity rate [W/K]
- $d$: grid width [mm]
- $k$: thermal conductivity [W/(m·K)]
- $K$: overall heat transfer coefficient [W/(m$^2$·K)]
- $m$: design parameter in Eq. (1) [-]
- $m$: mass flow rate [kg/s]
- $n$: design parameter in Eq. (1) [-]
- $NTU$: number of transfer units [-]
- $p$: pressure [Pa]
- $r$: radial coordinate for the cross-sectional shape [mm]
- $Re$: Reynolds number [-]
- $S$: heat transfer area [m$^2$]
- $T$: temperature [K]
- $u$: $x$-directional velocity component [m/s]
- $v$: $y$-directional velocity component [m/s]
1. Introduction

In recent years, the additive manufacturing (AM) using 3D printers has attracted and has been utilized in various fields such as aerospace, medical, architecture, etc. (Gardan and Schneider, 2015; Gao et al., 2015). Figure 1 shows an aircraft part (the Airbus A320 nacelle hinge bracket) manufactured by AM (Tomlin and Meyer, 2011). By coupling topology optimization with AM, the optimum design weighs only 326 g at the end, which is 64% lighter than the original design (918 g) and results in 10 kg reduction per airplane. The main advantages of the additive fabrication concept used in AM are as follows: (1) Developing and manufacturing complex shapes are achieved, which have been difficult or impossible by conventional numerical control manufacturing such as cutting, drilling, and milling. (2) Materials can be saved since they are not processed by removal or deformation as in the conventional methods. (3) Changing of tools is unnecessary since the shape is modeled by hardening material powder with a heat source. (4) It is possible to quickly start modeling work using shape data.

In this study, we design heat exchanger for the purpose of application of the additive fabrication concept to thermal fluid device. For the sake of energy saving, heat exchangers are required to be more efficient and more compact. In this study, we focus on a counter-flow concentric tube heat exchanger as the design target, which exchanges heat between two fluids with different temperatures. Figure 2 shows a concentric tube heat exchanger model. Figure 3 plots the theoretical effectiveness against the Number of Transfer Units \( NTU = K \cdot S / C_{\min} \) for two types of concentric tube heat exchangers with different flow directions: (a) counter-flow and (b) parallel-flow (Kakaç et al., 2012). At the heat capacity rate ratio \( C_{\min} / C_{\max} = 1 \), the effectiveness of the parallel-flow heat exchanger converges to about 50% as the \( NTU \) (i.e. the heat transfer area) increases, while the effectiveness of the counter-flow heat exchanger keeps increasing theoretically. Therefore, higher heat transfer performance can be expected by the counter-flow heat exchanger.

It is empirically known that heat transfer performance of the heat exchangers is significantly affected by the shape of the cross-sectional area (e.g. circle, rectangle, trapezoid) (Hasan et al., 2009). However, the shape is usually determined by trial and error and there are still few studies that systematically design the cross-sectional shape of heat exchangers through optimization. Therefore, we optimize the cross-sectional shape of a counter-flow concentric tube heat exchanger by a shape optimization approach. In this study, we design the cross-sectional shape of the inner pipe. The purpose of this study is to investigate the effects of the cross-sectional shape on the heat transfer performance through the shape optimization technique.
2. Optimization problem

In this study, we consider a counter-flow concentric tube heat exchanger in which the inner pipe is filled with the high-temperature fluid and the outer pipe is filled with the low-temperature fluid. In both pipes, water (liquid) is given as a working fluid.

As seen in Fig. 4, the outer pipe is fixed as a circular pipe and only the inner pipe is optimized to minimize the outlet temperature of the inner channel. The cross-sectional shape of the inner pipe is expressed in a cylindrical coordinate system where the radius $r$ is given by the following equation (Gielis et al., 2003).

$$ r = \frac{1}{\sqrt{\left(\frac{1}{a} \cdot \cos \left(\frac{m}{4} \cdot \theta\right)\right)^{n_1} + \left(\frac{1}{b} \cdot \sin \left(\frac{m}{4} \cdot \theta\right)\right)^{n_2}}} \quad , (n_1, n_2, n_3, m, a, b \in \mathbb{R}) \quad (1) $$

This equation is able to express a cross-sectional shape flexibly with a small number of parameters. For the sake of simplicity, we do not consider external forces in the governing equations of thermo-fluid dynamics (given by Eqs. (6)-(10) later) and assume $a = b = 1$ in Eq. (1) to impose the shape symmetry. Therefore, we do not consider buoyancy in this study. Moreover, in order to keep the cross-sectional area of the inner channel constant during the optimization, the radius obtained by Eq. (1) is multiplied by $(A_{\text{target}}/A_0)^{1/2}$, where $A_{\text{target}}$ is the prescribed cross-sectional area and $A_0$ is the original cross-sectional area calculated from the radius obtained by Eq. (1).

The pipe length is 280 mm, the wall thickness between the inner and the outer pipe is 0.5 mm, the material of the wall is aluminum ($k = 190.5 \text{ W/(mK)}$), the inlet temperature is 373 K for the high-temperature fluid and 293 K for the low-temperature fluid, the inlet velocity is 3 mm/s for both pipes, and the Reynolds number ($Re$) is about 70 based on the diameter as the characteristic length. In general, the effectiveness decreases at higher $Re$ because the residence time...
inside the pipe is reduced by higher flow velocity. Besides, in applications the effects of the cross-sectional shape on the effectiveness are small at $Re > 100$ while the effects are significant at $Re < 100$ (Hasan et al., 2009). Thus in this study, we set $Re < 100$ to examine the possibility of improving the performance by a variety of the cross-sectional shapes.

The optimization problem in this study is defined in Table 1. Heat transfer performance is evaluated from the temperature drop at the outlet of the high-temperature fluid. In other words, the objective function is to minimize the area-weighted average outlet temperature of the high-temperature fluid.

In addition, the following two constraints are imposed: (1) The cross-sectional area of the inner channel and outer channel area fixed to constant. Therefore, inlet flow rate is same for all designs. (2) The maximum radius of the inner pipe is bounded to avoid the inner pipe touches the outer pipe.

Figure 5 shows qualitative characteristics of each design parameter used in Eq. (1). For $n_1/n_2 > 10, n_1/n_3 > 10$, a circular pipe is obtained. For $n_2/n_1 > 10, n_3/n_1 > 10$, the inner pipe is larger than the outer pipe. $m$ corresponds to the number of protrusions since it dominates the period of $\theta$. As the curvature of the protrusions increases, it is difficult to generate a mesh for Computational Fluid Dynamics (CFD) resolving laminar boundary layer, and the convergence of a CFD solution is remarkably deteriorated. Based on the above discussion, we determine the design variables and their ranges as listed in Table 1.

Generally, the cross-sectional shape of a channel affects the pressure drop. For simplicity, this study represents and optimizes the cross-sectional shape with protrusions, which are exactly aligned with the channel flow direction. The present simple representation method is not sufficient to evaluate the influence of the cross-sectional shape on the pressure drop (see the future work discussed in Section 5). Hence, this study does not consider the pressure drop as a quantity of interest.

Table 1  Optimization problems.

| Objective function | Constraints                  | Design variables |
|--------------------|------------------------------|------------------|
| Minimize $\frac{1}{A_{h,o}} \int T_{h,o} dA_{h,o}$ | $A_h = A_l = 314.15$ mm$^2$ | $1 \leq n_1 \leq 10$
|                    | $r_c = 14.142$ mm           | $1 \leq n_2 \leq 10$
|                    | $r_{h,max} < 13$ mm         | $1 \leq n_3 \leq 10$
|                    |                              | $0 < m \leq 8$   |

Fig. 4  A cross-sectional shape of the inner pipe expressed by Eq. (1) and the outer pipe fixed as a circular pipe.

Fig. 5  Cross-sectional shapes of the inner pipe with different combinations of the design parameters used in Eq. (1).
3. Numerical Methods

3.1 Optimization

Genetic algorithm (GA) is an optimization scheme based on the theory of evolution of organisms. GA selects individuals from the current generation as parents, generates new individuals as children by the crossover and mutation of the parents, and inherits better individuals to the next generation. In engineering optimization problems, GA offers several advantages over the gradient-based methods. First, GA can be applied to any design optimization problems without considering differentiability and convexity of objective functions. Second, GA can solve the optimization problems without any specific knowledge or background of the problems. Moreover, GA stochastically explores the optimum by a population whereas the gradient-based methods explores deterministically by one solution from a user-defined initial position. These features provide GA with further advantage that GA is more likely to find a global optimum rather than local optima.

In this study, the Non-Dominated Sorting Genetic Algorithm II (NSGA-II) (Holland, 1992; Deb et al., 2002; Goldberg et al., 1992; Deb and Agrawal, 1995; Deb and Goyal, 1996; Deb, 2001) is employed for exploration because this algorithm is effective and widely accepted for many optimization problems. The detail of NSGA-II is described as follows (Fig. 6). Initially, a parent population \( P_{t-1} \) with the size of \( N \) is created randomly. Here, \( t \) indicates the number of generation. Each feasible solution is assigned a rank (the solution with lower rank is better) according to its objective function value. On the other hand, each infeasible solution is assigned a rank which is higher than the minimum rank for the feasible solutions. Between two infeasible solutions, the solution with a smaller constraint violation has a better rank. Then, after choosing \( N \) solutions with lower rank in the parent population, recombination and mutation are conducted to create an offspring population \( Q_t \) with the size of \( N \). In order to introduce elitism, first, a combined population \( R_t = P_{t}Q_t \) with the size of \( 2N \) is formed. Then, the solutions in \( R_t \) are sorted according to the ranks based on objective function values and constraint violation. Now, \( N \) solutions are chosen from \( R_t \) in the order of their ranks and make up a new population \( P_{t+1} \). The procedure as described above is for one generation. The non-dominated solutions with the lowest rank are explored by repeating this procedure for a certain number of generations.

NSGA-II is a population-based optimizer that needs to evaluate an objective function for many solutions in the population in parallel. Hence, we also apply a surrogate model to estimate the objective function at a reduced computational cost. This model expresses the response of an objective function to design variables with an algebraic equation, which is derived by the true objective function values sampled at several points in the design variable space. The surrogate model enables us to estimate the objective function value at an arbitrary point in the design variable space instantaneously.

This study employs the Kriging surrogate model (Jones et al., 1998), which is suitable for approximating nonlinear functions. For an objective function \( f(\mathbf{x}) \) of design variables \( \mathbf{x} \), the Kriging model can estimate the uncertainty in the objective function \( \hat{s}^2(\mathbf{x}) \) as well as the function itself \( \hat{f}(\mathbf{x}) \). Based on these estimates, it is possible to calculate the Expected Improvement (EI) of the objective functions, which helps us to specify the location of the next sample point to be added for updating the model. The NSGA-II searches the location where the EI value is maximized on the Kriging model instead of the original objective function value. In other words, it is possible to simultaneously search the global optimal solution and improve the accuracy of the surrogate model. To minimize \( f(\mathbf{x}) \), the improvement \( I(\mathbf{x}) \) and the EI \( E[I(\mathbf{x})] \) are expressed as follows.

\[
I(\mathbf{x}) = \max[f_{\min} - y, 0] \\
E[I(\mathbf{x})] = \int_{-\infty}^{f_{\min}} (f_{\min} - y)\phi(y)dy
\]

Here, \( f_{\min} \) is the minimum value of \( f(\mathbf{x}) \) found so far, \( y \) is the random variable \( N[\hat{y}(\mathbf{x}), \hat{s}^2(\mathbf{x})] \), and \( \phi(y) \) is the probability density function of \( y \).

The optimization procedure in this study is shown in Fig. 7: (1) Generate some combinations (sample points) with different design variables in the design variable space and evaluate the objective function values (the true values) by CFD. (2) Construct the Kriging surrogate model approximating the response of an objective function to design variables. (3) Select the maximum EI value point as an additional sample point. (4) Evaluate the objective function value at the additional sample point by CFD. (5) Check the convergence by the cross-validation which expresses the accuracy of the
surrogate model. If not converged, the algorithm returns to (2).

38 initial sample points for model approximation are generated by Latin Hypercube Sampling (LHS) (McKay et al., 2000; Stein, 1987; Owen, 1992) that enables to sample the design space comprehensively with fewer sample points than simple random sampling in Monte Carlo studies. Population size is 128 and number of generations is 100 in the current NSGA-II.

![Fig. 6 Procedure of NSGA-II.](image)

![Fig. 7 Flow chart of the optimization process.](image)

3.2 CFD analysis

In this study, the objective function is evaluated by the pressure-based solver in a commercial CFD software ANSYS FLUENT 17.2.

In order to solve CFD sufficiently, it is necessary to pay attention to stiffness. A typical example is a boundary layer. We need to resolve the boundary layer sufficiently to capture an important phenomenon such as separation dominating the flow field, since the flow velocity in the direction along the object changes rapidly. In the vicinity of the object surface, it is necessary to distribute the number of mesh that can sufficiently resolve the boundary layer in the vertical direction of the object surface. Generally, minimum grid width is prepared to be 1/50 or less of the boundary layer thickness $\delta$ developed on the object surface. Therefore, minimum grid size $\Delta d_{min}$ is given as follows (Fujii, 1994):

\[
\delta \approx \frac{5.0}{\sqrt{Re}}
\]

\[
\frac{\Delta d_{min}}{r} \approx \frac{0.1}{\sqrt{Re}}
\]

From the above, the number of cells is about six million and minimum grid width is 0.1 mm in the vertical direction of the object surface to solve the thermal boundary layer sufficiently in this study. Tables 2 and 3 show the results of the grid independence tests in the angular direction (around the cross-sectional shape) and the axial direction (parallel to the flow direction), respectively, for a circular pipe heat exchanger. These results show that the value of $T_{h,o}$ does not change much when the grid size is reduced to 0.628 mm in the angular direction (mesh configuration 2-2) and 0.560 mm in the axial direction (mesh configuration 3-2). Therefore, the grid size of mesh configuration 3-2 is employed in this study.

The heat exchanger operates under steady state conditions. The fluids do not undergo phase change while moving through the heat exchanger. The governing equations of thermo-fluid dynamics are given as follows:
Equation of continuity:
\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0
\]  \( (6) \)

Steady incompressible Navier-Stokes equations:
\[
\frac{u}{\partial x} + \frac{v}{\partial y} + \frac{w}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right)
\]  \( (7) \)
\[
\frac{u}{\partial x} + \frac{v}{\partial y} + \frac{w}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right)
\]  \( (8) \)
\[
\frac{u}{\partial x} + \frac{v}{\partial y} + \frac{w}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right)
\]  \( (9) \)

Steady energy equation:
\[
\frac{\partial T}{\partial x} + \frac{\partial T}{\partial y} + \frac{\partial T}{\partial z} = \alpha \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right)
\]  \( (10) \)

Here, \( \nu \) and \( \alpha \) are assumed to be constant, i.e., independent of temperature. For temperature, the second-kind boundary condition for an adiabatic wall is applied to the outer surface of the outer pipe in the normal direction. The third-kind boundary condition for convective heat transfer is applied to the inner surface of the outer pipe in the normal direction and the outer surface of the inner pipe in the normal direction. For velocity, the no slip boundary condition is assumed on the channel walls. The inlet velocity is set to 3 mm/s for both pipes, the inlet temperature is 373 K for the high-temperature fluid and 293 K for the low-temperature fluid. The initial inlet gauge pressure is 0 Pa for both pipes, and the outlet gauge pressure is 0 Pa for both pipes. The inlet gauge pressure is set to 0 Pa as the initial condition and then varies automatically as the CFD simulation progress, such that flow can pass through the channel with a constant velocity (3 mm/s). In ANSYS FLUENT, it is available to create a couple wall boundary at the non-conformal interfaces. Here, the non-conformal interfaces are boundaries between cell zones (the high-temperature region and the low-temperature region in this study) in which the mesh model node locations are not identical. In such a case, fluid flow cannot pass across the interface, which is a solid wall, but heat transfer can occur through the wall. We specify the material type and wall thickness for the thin-wall thermal resistance calculations and the solver can calculate heat transfer directly from the solution in the adjacent cells.

Table 4 shows the CFD schemes used in this study. The pressure-based solver employs an algorithm that belongs to a general class of methods called the projection method. In the projection method, wherein the constraint of mass conservation of the velocity field is achieved by solving a pressure equation. The pressure equation is derived from the continuity and the momentum equations in such a way that the velocity field, corrected by the pressure, satisfies the continuity. Since the governing equations are nonlinear and coupled to one another, the solution process involves iterations wherein the entire set of governing equations is solved repeatedly until the solution converges. ANSYS FLUENT uses a control-volume-based technique to convert a general scalar transport equation to algebraic equation that can be solved numerically. This control volume technique consists of integrating the transport equation about each control volume, yielding a discrete equation that expresses the conservation law on a control-volume basis.
4. Result and Discussion

The Kriging model is updated 10 times through the sequential optimization shown in Fig. 7. Figure 8 shows the leave-one-out-cross-validation result, which plots the true value and the Kriging-estimated value of the objective function evaluated at each sample point. In Fig. 8, closer plots to the line (true value = estimated value) mean higher approximation accuracy of the Kriging model. It is found that the accuracy of the Kriging model is improved after adding 10 sample points. In addition, most of the additional sample points are concentrated on the low-temperature region while the initial sample points are distributed in the range from 329 K to 333 K. Therefore, it can be confirmed that the present optimization can improve the heat transfer performance toward the optimum. Since the grid dependency depends on the geometry of the flow channel, the discussions on the results for the optimal shape are important. Tables 5 and 6 show the results of the grid independence tests in the angular direction (around the cross-sectional shape) and the axial direction (parallel to the flow direction), respectively, for the optimal shape heat exchanger. As shown in Table 5 and Table 6, it is found that the grid independence tests in Table 2 and Table 3 are valid.

![Cross-validation result of the Kriging model.](image)

![Fig. 8 Cross-validation result of the Kriging model.](image)

![Perimeter = 113 mm](image)  
 ![112 mm](image)  
 ![106 mm](image)
As shown in Fig. 8, the additional sample yields the inner pipe shape with many protrusions. One of the reasons is that the heat transfer area has increased since the shape with many protrusions has a large perimeter in its cross section. As noted in Fig. 3, the effectiveness of the heat exchanger is larger as the $NTU$ (i.e. the heat transfer area) is larger. Therefore, it is assumed that the shape with many protrusions leads to high efficiency.

In addition, there is another important reason. Figure 9 shows the temperature contours at the outlet of the optimal inner channel and Fig. 10-13 shows those of typical inner channels with the same heat transfer surface area as the optimal channel. Here remember that inlet flow rate is same for all designs. From those figures, therefore, it is confirmed that the optimal channel with more protrusions is more efficient than the typical channels despite the same heat transfer surface area. In other words, it is found that only increasing the heat transfer area is not enough to improve heat transfer performance and the number of protrusions with sufficient height plays important roles. In the optimal design, since each protrusion is locally surrounded by the low-temperature fluid, the temperature of the high-temperature fluid can drop significantly inside the protrusions. Figure 14 shows a flow-directional sequence of the temperature contours in the optimal inner channel. It is found that the temperature of the high-temperature fluid gradually decreases from the inlet to the outlet. Therefore, we conclude that it is more favorable not only to increase the heat transfer area but also to arrange more protrusions in the cross-sectional shape for the improvement in heat transfer performance.

| Mesh configuration | Grid size in the angular direction [mm] | Grid size in the axial direction [mm] | $T_{h,o}$ [K] |
|--------------------|----------------------------------------|----------------------------------|---------------|
| 5-1                | 1.26                                   | 0.280                            | 325.701       |
| 5-2                | 0.628                                  | 0.280                            | 324.582       |
| 5-3                | 0.209                                  | 0.280                            | 324.580       |

| Mesh configuration | Grid size in the angular direction [mm] | Grid size in the axial direction [mm] | $T_{h,o}$ [K] |
|--------------------|----------------------------------------|----------------------------------|---------------|
| 6-1                | 0.628                                  | 0.933                            | 325.497       |
| 6-2                | 0.628                                  | 0.560                            | 324.582       |
| 6-3 (5-2)          | 0.628                                  | 0.280                            | 324.582       |

Fig. 9 Temperature contours at the outlet of the optimal inner channel (average temperature $= 324.58$ K).

Fig. 10 Temperature contours at the outlet of a circular inner channel with the same heat transfer surface area as the optimal channel (average temperature $= 331.93$ K).
Fig. 11  Temperature contours at the outlet of a triangle inner channel with the same heat transfer surface area as the optimal channel (average temperature = 332.28 K).

Fig. 12  Temperature contours at the outlet of a rectangle inner channel with the same heat transfer surface area as the optimal channel (average temperature = 331.62 K).

Fig. 13  Temperature contours at the outlet of a pentagon inner channel with the same heat transfer surface area as the optimal channel (average temperature = 332.13 K).

Fig. 14  A flow-directional sequence of the temperature contours in the optimal inner channel.

A more efficient shape is expected to be available by increasing the upper limit of the parameter \( m \) in Eq. (1) which dominates the number of protrusions. However, larger \( m \) may lead to the following issues to be addressed: (1) A thin shape at each protrusion may conflict with the manufacturing limit of 3D printers. (2) We should reconsider proper mesh resolution so that it can solve the laminar boundary layer on spiky protrusions. In other words, it is difficult to do that for the present method in larger \( m \) because of failing to assemble two mesh models of inner channel and outer channel.

5. Conclusions

In this study, we investigated the possibility of improving heat transfer performance of the counter-flow concentric tube heat exchanger based on the shape optimization technique. NSGA-II explored solutions with larger heat transfer area. Moreover, the comparison of five typical channels with the same heat transfer area but with different cross-sectional shapes revealed that the number and the size of protrusions also play an important role to improve heat transfer performance. Consequently, we demonstrated an advantage of the present design optimization, such that the optimal
channel overcomes the typical channels in terms of heat transfer performance.

As stated in Section 2, the present simple shape representation is not sufficient to evaluate the influence of the cross-sectional shape on the pressure drop. It should be evaluated for a channel where flow passes over protrusions, such as a heat exchanger with helical protrusions, and should be studied in future work.

In addition, it has been widely reported in literature that helical coil heat exchangers offer advantageous over straight tube heat exchanger due to the compact structure and high heat transfer coefficient (Prabhanjan et al., 2002). They are used in a variety of industrial applications such as power generation, air conditioning, refrigeration, food industry, etc. (Futagami and Aoyama, 1988; Abdalla, 1994; Xin et al., 1996). Therefore, another future work is to consider the optimization for a helical coil heat exchanger toward much higher heat transfer performance.

References

Abdalla, M. A., A four-region, moving-boundary model of a once-through, helical-coil steam generator, Annals of Nuclear Energy, Vol. 21, No. 9 (1994), pp. 541-562.

Deb, K., Multi objective optimization using evolutionary algorithms, John Wiley and Sons, Vol. 16 (2001).

Deb, K., Agrawal, R. B., Simulated binary crossover for continuous search space, Complex Systems, Vol. 9 (1995), pp. 115-148.

Deb, K., Goyal, M., A combined genetic adaptive search (GeneAS) for engineering design, Computer Science and informatics, Vol. 26 (1996), pp. 30-45.

Deb, K., Pratap, A., Agarwal, S., Meyarivan, T., A fast and elitist multiobjective genetic algorithm: NSGA-II, IEEE transactions on evolutionary computation, Vol. 6, No. 2 (2002), pp. 182-197.

Fujii, K., Numerical Methods for Computational Fluid Dynamics, University of Tokyo Press (1994) (in Japanese).

Futagami, K., Aoyama, Y., Laminar heat transfer in a helically coiled tube, International Journal of Heat and Mass Transfer, Vol. 31 (1988), pp. 387-396.

Gao, W., Zhang, Y., Ramanujan, D., Ramani, K., Chen, Y., Williams, C.B., Wang, C.C.L., Shin, Y.C., Zhang, S., Zavattieri, P.D., The status, challenges, and future of additive manufacturing in engineering, Computer-Aided Design Vol. 69 (2015), pp. 65-89.

Gardan, N., Schneider, A., Topological optimization of internal patterns and support in additive manufacturing, Journal of Manufacturing Systems, Vol. 37 (2015), pp. 417-425.

Giels, J., Beirinckx, B., Bastiaens, E., Superquadrics with rational and irrational symmetry, Proceedings of the eighth ACM symposium on Solid modeling and applications, ACM (2003), pp. 262-265.

Goldberg, D.E., Deb, K., Clark, J.H., Genetic algorithms, noise, and the sizing of populations, Complex Systems, Vol. 6 (1992), pp. 333-362.

Hasan, M.I., Rageb, A.A., Yaghoubi, M., Homayoni, H., Influence of channel geometry on the performance of a counter flow microchannel heat exchanger, International Journal of Thermal Sciences, Vol. 48 (2009), pp. 1607-1618.

Holland, J.H., Adaptation in natural and artificial systems: an introductory analysis with applications to biology, control, and artificial intelligence, MIT press (1992).

Jones, D.R., Schonlau, M., Welch, W.J., Efficient global optimization of expensive black-box functions, Journal of Global optimization, Vol. 13 (1998), pp. 455-492.

Kakać, S., Liu, H., Pramanjaroenki, A., Heat exchangers: Selection, Rating, and Thermal Design, CRC Press (2012).

McKay, M.D., Beckman, R.J., Conover, W.J., A comparison of three Methods for selecting values of input variables in the analysis of output from a computer code, Technometrics, Vol. 42, No. 1 (2000), pp. 55-61.

Owen, A.B., A central limit theorem for Latin hypercube sampling, Journal of the Royal Statistical Society, Series B (Methodological), Vol. 54, No. 2 (1992), pp. 541-551.

Prabhanjan, D.G., Raghavan, G.S.V., Rennie, T.J., Comparison of heat transfer rates between a straight tube heat exchanger and a helically coiled heat exchanger, International Communications in Heat and Mass Transfer, Vol. 29, No. 2 (2002), pp. 185-191.

Stein, M., Large sample properties of simulations using Latin hypercube sampling, Technometrics, Vol. 29, No. 2 (1987), pp. 143-151.

Tomlin, M., Meyer, J., Topology optimization of an additive layer manufactured (ALM) aerospace part, Proceeding of the 7th Altair CAE technology conference (2011), pp. 1-9.
Xin, R. C., Awwad, A., Dong, Z.F., Ebadian, M.A., An investigation and comparative study of the pressure drop in air-water two-phase flow in vertical helicoidal pipes, International journal of heat and mass transfer, Vol. 39, No. 4 (1996), pp. 735-743.