Research Article

Behzad Ghobadi, Farshad Kowsary*, and Farzad Veysi

Optimization of heat transfer and pressure drop of the channel flow with baffle

<https://doi.org/10.1515/htmp-2021-0030>

received January 24, 2021; accepted July 26, 2021

Abstract: In this article, the numerical analysis has been carried out to optimize heat transfer and pressure drop in the horizontal channel in the presence of a rectangular baffle and constant temperature in two-dimension. For this aim, the governing differential equation has been solved by computational fluid dynamics software. The Reynolds numbers are in the range of $2,000 < Re < 10,000$ and the working fluid is water. While the periodic boundary condition has been applied at the inlet, outlet, and the channel wall, axisymmetric boundary condition has been used for channel axis. For modeling and optimizing the turbulence, $k-\omega$ SST model and genetic algorithm have been applied, respectively. The results illustrate that adding a rectangular baffle to the channel enhances heat transfer and pressure drop. Hence, the heat transfer performance factor along with maximum heat transfer and minimum pressure drop has been investigated and the effective geometrical parameters have been introduced. As can be seen, there is an inverse relationship between baffle step and both heat transfer and pressure drop so that for p/d equal to 0.5, 1, and 1.25, the percentage of increase in Nusselt number is 141, 124, and 120% comparing to a simple channel and the increase in friction factor is 5.5, 5, and 4.25 times, respectively. The results of modeling confirm the increase in heat transfer performance and friction factor in the baffle with more height. For instance, when the Reynolds number and height are 5,000 and 3 mm, the Nusselt number and friction factor have been increased by 35% and 2.5 times, respectively. However, for baffle

with 4 mm height, the increase in the Nusselt number and friction factor is 68% and 5.57 times, respectively. It is also demonstrated that by increasing Reynolds number, the maximum heat transfer performance has been decreased which is proportional to the increase in p/d and h/d . Moreover, the maximum heat transfer performance in 2,000 Reynolds number is 1.5 proportional to p/d of 0.61 and h/d of 0.36, while for 10,000 Reynolds number, its value is 1.19 in high p/d of 0.93 and h/d of 0.15. The approaches of the present study can be used for optimizing heat transfer performance where geometrical dimensions are not accessible or the rectangular baffle has been applied for heat transfer enhancement.

Keywords: heat transfer enhancement, $k-\omega$ SST, periodic boundary condition, genetic algorithm, Nusselt number

1 Introduction

Since there are different types of heat exchangers in the industry with wide range of applications, researchers have been recently motivated to improve their properties with different techniques to enhance heat transfer [1–4]. Researchers have used a variety of methods to increase the efficiency of heat transfer systems and to reach greater heat transfer on a smaller scale [5,6]. In general, the basis of these techniques is decreasing thermal resistivity by increasing the effective level of heat transfer or creating turbulence of the fluid flow in the channel [7,8]. Using an optimized channel which reduces the size of the heat exchanger and increases the heat transfer coefficient is the best way to enhance heat transfer. Nowadays, with increasing technology in engineering applications and industrial processors, heat exchangers with higher efficiency are essential. Heat exchangers have been designed based on increasing heat transfer rate and performance, decreasing pressure drop and volume, and the lowest cost. There are various methods for increasing the heat transfer rate and efficiency by reducing the volume. Ghyadh et al. [9] divided heat transfer enhancement techniques to three groups which are passive, active, and compound, described as follows:

* **Corresponding author: Farshad Kowsary**, Department of Mechanical Engineering, Faculty of Industrial and Mechanical Engineering, Qazvin Branch, Islamic Azad University, Qazvin, Iran, e-mail: fkowsari@ut.ac.ir, kowsary.f@gmail.com, tel: +98-9123853815

Behzad Ghobadi: Department of Mechanical Engineering, Faculty of Industrial and Mechanical Engineering, Qazvin Branch, Islamic Azad University, Qazvin, Iran

Farzad Veysi: Department of Mechanical Engineering, Razi University, Kermanshah, Iran

In the passive method, heat transfer enhancement takes place by creating turbulence in the flow or changing the flow regime without any external force which is always accompanied by a drop in pressure. This method includes changing the inner surface of the channel, installing heat transfer enhancer (HTE), and increasing the roughness by coatings method. Although this is a low-cost method, the cost of applying methods to overcome the pressure drop should be considered [10–12]. In the active method, external forces are used to increase the rate of heat transfer like what happens in the mechanical and rotary agitators, and magnetic or electrostatic fields. In this method, the turbulence is increased in the boundary layers by moving the fluid with a mechanical equipment or by rotating and vibrating the surface at high and low frequencies which causes heat transfer enhancement. Applying active method is not appropriate because of the high cost of external forces especially in critical situations [13]. Therefore, simultaneous use of these methods in one device has a greater impact on heat transfer. Since it is the combination of both active and passive methods, it can be defined as a new category i.e., combined optimization group [14].

There are great challenges in improving the efficiency of heat exchangers and always led researchers to explore new ways and review heat transfer enhancement methods from both passive and active aspects [15–28]. In the passive method, the increase in heat transfer occurs by using the HTE like ribs [29]. Budak et al. [30] investigated the increase in heat transfer by installing a flow turbine at the inlet of the heat exchanger which caused the flow to rotate. Besides, many researchers have studied the effect of ribs and their attack angle on nanofluid heat transfer [31–34]. The results show that the changes in attack angle are proportional to the Reynolds number and have significant effects on fluid mixture and heat transfer enhancement. Others analyzed heat and flow transfer in the micro-channel [35–38]. They compared the impact of using new designs and shapes of ribbed channels. The results show that the presence of ribs extremely depends on the values of Reynolds number. Some researchers also studied the effect of hydrodynamic parameters of nanofluid flow and heat transfer [39–41]. They studied the effect of different ratios of ribs and their height and revealed that the presence of ribs causes the increase in heat transfer to a significant amount.

Goodarzi et al. [42] studied the effect of geometric parameters of a channel on the nanofluid heat transfer. They analyzed channels with equal hydraulic length and diameter but with different cross sections such as square, trapezoid, and circle.

Sheikh et al. [43] also used computational fluid dynamics to optimize the efficiency of a thermoelectric generator by

changing the baffle distribution of heat transfer. Their studies were divided into three groups. In the first group, they considered the effect of angle and thickness of the baffle. In the second group, the effect of height and the distance between the baffles were studied. Finally, in the third group, the different arrangements of the baffles were examined. Their results showed that the pressure drop in all models is within the permitted range. They also introduced a model with the highest power. Aghaei et al. [44] studied the influence of elliptical, horizontal, and vertical baffles on the heat transfer of a nanofluid and its entropy. Their results showed that placing the baffles horizontally increases the heat transfer rate. Mohebbi et al. [45] also investigated the heat transfer of nanofluid numerically by placing ribs with different shapes inside the channel. In addition to comparing heat transfer variation by increasing volume fraction and Reynolds number, they introduced an optimal geometric ratio for each rib shape. Goktepe and Atmaca [46] applied rectangular ribs between horizontal parallel plates to increase heat transfer. They used the k - ω SST model to specify the turbulence. They also investigated the effects of ribs on friction factor, Nusselt number, and heat transfer performance. Menni and Azzi [47] applied computational fluid dynamics to investigate the heat transfer of an airstream passing through a channel in which a diamond-shaped baffle was installed. In similar conditions, they compared the results with that of a baffle in a rectangular shape and found that the heat transfer optimized 4–29 times compared to that of unobstructed channel. Abdulhameed et al. [48] investigated the effect of baffle geometry on improving heat transfer by analyzing heat transfer and pressure drop in different geometries. Sahamifar et al. [49] applied pattern search algorithm to optimize the arrangement of channels and applied periodic method to model their work which was the same as the method applied in the present study.

A review of the previous studies shows that most studies have focused on finding suitable ways to enhance heat transfer and their comparison. However, there are few researchers who focused on optimizing the parameters that affect heat transfer. And among the studies, so far, the periodic boundary condition has not been used to reduce the computation time, and if it has been used, their number has been limited. And they have had to solve the whole domain model. In the present study, passive method has been used to improve heat transfer. The numerical study of heat transfer and pressure drop has been carried out by considering the geometric parameters and using the periodic boundary condition in the presence of the flow turbulence (baffle) inside the channel. In addition, the effect of baffle on heat transfer performance has been studied for different Reynolds numbers.

Figure 1: The Schematic of channel geometry and boundary conditions.

Finally, the geometric parameters have been optimized by using genetic algorithm.

2 Problem description

Figure 1 shows the geometry of the channel in two-dimensions and the baffle. The flow enters the channel with a constant velocity at a temperature of 300 K. The channel wall is subjected to a constant temperature of 400 K. Therefore, heat transfer takes place due to the differences between the temperatures of the inlet fluid and the wall. The length and the diameter of the channel are $L = 1,000$ mm and $D = 32$ mm, respectively. The effect of the applied baffles with the thickness of $w = 1$ mm and various heights and pitches on heat transfer and pressure drop have been investigated. As can be seen, except a small area at the beginning of the channel, the flow reaches the periodically fully developed conditions quickly and hence, modeling with the periodic boundary condition results in reasonable accuracy by reducing computational time. Sahamifar et al. [49] approved the accuracy of this approach.

Considering the periodic boundary condition, one part of the channel is modeled in Figure 2. As it is known, the whole domain is meshed with quadrilateral cells. However, due to the sensitivity of the calculations, a smaller lattice was used near the channel wall. In fact, due to the symmetry of the problem and the use of periodic boundary condition, one part of the channel is modeled.

3 Governing equations and numerical modeling

3.1 The governing equations

The physical properties of any fluid flow are based on three principles: the law of conservation of mass, the law of conservation of energy, and the momentum equation, all of which can be expressed by mathematical equations. Computational fluid dynamics is the science of determining a numerical solution for the governing equations of fluid flow and it is applied to find a numerical description for

Figure 2: The simulated geometry.

the flow under study. In the steady two-dimensional fluid flow model, the law of conservation of mass [50] (the continuity equation) is defined as follows:

$$\frac{\partial u_i}{\partial x_i} = 0, \quad i = 1, 2 \text{ or } i = x, y. \quad (1)$$

For the incompressible flow, the momentum equation [50] is as follows:

$$\rho \frac{\partial}{\partial x_j} (u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_i}{\partial x_j} \right) \right] + \frac{\partial}{\partial x_j} (-\rho \overline{u'_i u'_j}), \quad (2)$$

while the law of conservative energy [50] is as follows:

$$\frac{\partial}{\partial x_i} [u_i (\rho E + P)] = \frac{\partial}{\partial x_j} \left[\left(k + \frac{C_p \mu_t}{Pr_t} \right) \frac{\partial T}{\partial x_j} + u_i (\tau_{ij})_{\text{eff}} \right]. \quad (3)$$

The above equations are called Navier–Stokes equations in which ρ , P , and μ are density, pressure, and dynamic viscosity of the fluid, respectively. u_i is the velocity in x_i direction. Also, μ_t , T , k , and E are labeled for turbulent viscosity, temperature, thermal conductivity coefficient, and total energy, respectively. Pr_t is the turbulent Prandtl number. x_i represents the x and y coordinate axes and $i = 1, 2$ or x, y . The stress tensor $(\tau_{ij})_{\text{eff}}$ is represented as below [51]:

$$(\tau_{ij})_{\text{eff}} = \mu_{\text{eff}} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) - \frac{2}{3} \mu_{\text{eff}} \frac{\partial u_i}{\partial x_i} \delta_{ij}. \quad (4)$$

The transition equations governing the kinetic energy of turbulence (k) and the turbulence frequency (ω) are defined as [51]:

$$\frac{\partial}{\partial x_j} (\rho u_j k) = P_k - D_k + \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right), \quad (5)$$

$$\begin{aligned} \frac{\partial}{\partial x_j} (\rho u_j \omega) &= \frac{\gamma}{\nu_t} P_k - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) \\ &+ (1 - F_1) 2 \rho \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial K}{\partial x_j} \frac{\partial \omega}{\partial x_j}, \end{aligned} \quad (6)$$

where P_k indicates the kinetic energy of turbulence under the influence of the mean velocity gradient and D_k is the turbulent degradation of k , while F_1 is the perturbation function in ω . Γ_ω and Γ_k are the effective distribution of ω and k , respectively, and $\sigma_{\omega^2} = 1.168$. Calculating the heat transfer coefficient and Nusselt number are the best ways to evaluate heat transfer performance. The average heat transfer coefficient ($\bar{h}$) between the fluid and the channel wall is calculated by the following equation [52]:

$$\bar{h} = \frac{q}{\pi d P \Delta T_{\text{lm}}}, \quad (7)$$

where P is the pitch length and ΔT_{lm} is the difference of logarithmic mean temperatures which is defined by the equation below [52]:

$$\Delta T_{\text{lm}} = \frac{(T_w - T_i) - (T_w - T_o)}{\ln \left[\frac{(T_w - T_i)}{(T_w - T_o)} \right]}, \quad (8)$$

and the Nusselt number is ref. [52]:

$$Nu = \frac{hd}{k}, \quad (9)$$

where d is the internal diameter of the channel and k is thermal conductivity coefficient of the fluid in the bulk temperature.

Moreover, friction factor is specified by Darcy–Weisbach equation as [52] follows:

$$f = \frac{\Delta P}{\frac{1}{2} \rho U^2 \frac{L}{D}}, \quad (10)$$

where U , L , and D are the average fluid velocity, channel length, and channel diameter, respectively. f is the friction factor and ΔP is the amount of pressure drop.

Moreover, Reynolds number [52] is calculated as follows:

$$Re = \frac{\rho U d}{\mu}, \quad (11)$$

where ρ and μ are the density and dynamic viscosity of the fluid, respectively.

The relationship between the Reynolds number and the mass flow is [52] as follows:

$$Re = \frac{4 \dot{m}}{\pi D \mu}. \quad (12)$$

3.1.1 Heat transfer performance parameter

Replacing a straight optimized channel increases the heat transfer in the same Reynolds numbers. However, the cost of using heat exchangers with the optimized channel is increased because of the pressure drop which limits their practical applications. Therefore, a parameter called heat transfer performance factor must be defined for calculating the performance of heat exchanger. Thus, in order to obtain optimal conditions and considering pressure drop along with the increase in heat transfer, the heat transfer performance factor (η) should be calculated as follow [53]:

Table 1: The thermophysical properties of water and material

Material	Density ($\text{kg}\cdot\text{m}^{-3}$)	Specific heat ($\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$)	Thermal conductivity ($\text{W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$)	Viscosity ($\text{N}\cdot\text{m}\cdot\text{s}^{-1}$)
Water	998.2	4,182	0.6	0.001003
Aluminum	2,718	870	201.6	—

$$\eta = \frac{(\text{Nu}_E/\text{Nu})}{(f_E/f)^{1/3}}, \quad (13)$$

where Nu_E and f_E are the Nusselt number and the friction factor of the channel in the presence of baffles and flow turbulence, respectively. Nu and f are Nusselt number and friction factor, respectively, when no baffle is used.

Since fluid flow is one of the main effective parameters in the heat transfer process, different Reynolds numbers and, accordingly, different mass flows are considered at 300 K inlet flow temperature. Both channel and baffle are made of aluminum and the working fluid is water. The thermophysical properties of water and material are presented in Table 1.

3.2 Numerical modeling

In order to perform calculations and numerical analysis, Ansys Fluent software has been used. The system of governing equations for all cells of the computational network is solved for the velocity of flow, temperature, and pressure. In addition, a two-dimensional model is modeled to study fluid flow and heat transfer parameters. The temperature of the channel wall is 400 K and the Reynolds numbers are in the range of $2,000 < \text{Re} < 10,000$. A part of the selected channel with the diameter of 32 mm can be modeled by periodic boundary condition, the axisymmetric of the turbulent flow inside the channel, and the periodicity of the flow in the geometry of the channel. Channel geometry includes periodic inlet and outlet boundaries, channel wall, and symmetric boundary. The different boundary conditions are listed in Table 2.

Table 2: Different boundary conditions used in the applied modeling

Boundary	Assigned as
Inlet	Periodic
Outlet	Periodic
Tube walls	Wall
Tube axis	Axis

The modeling of Navier–Stokes which is the governing equations of fluid flow and heat transfer is studied in Ansys Fluent software by finite control volume method. Besides, the heat transfer of the flow is modeled by the energy equation. Solving these equations is carried out by segregated solver method. Also, the implicit method has been used to linearize the equations. The coupling of pressure and velocity is done by simple algorithm, while the second order upwind scheme is applied to solve momentum equations, turbulent dissipation rate, and energy equation. Meanwhile, the k – ω SST turbulence model has been used where the turbulence kinetic energy and turbulence rate are represented by k and ω , respectively. The under-relaxation factor and residual value (convergence criteria) are two main iteration parameters that should be adjusted before the beginning of the modeling.

The segregated solver is calculated via under-relaxation factor for controlling the variables in each iteration. In Ansys Fluent software, the value of prerequisite under-relaxation factor for all variables is close to the optimal state for the highest possible value. Although these values are helpful, it would be more accurate if the under-relaxation factor is reduced. According to the accuracy of numerical solution and the use of flow turbulence, the friction factor is obtained from the smooth channel geometry achieved from Blasius and Petukhov relations [52] in equations (14) and (15), respectively:

$$f_m = 0.316 \text{Re}^{-0.25}, \quad (14)$$

$$f_m = (0.790 \ln \text{Re} - 1.64)^{-2}. \quad (15)$$

On the other hand, the Nusselt number is extracted from the Dittus-Boelter and Gnielinski relations [52] in equations (16) and (17), respectively. The comparisons of these results are illustrated in Figures 3 and 4.

$$\text{Nu} = 0.023 \text{Re}^{0.8} \text{Pr}^{0.3}, \quad (16)$$

$$\text{Nu} = \frac{\left(\frac{f}{8}\right)(\text{Re} - 1,000)\text{Pr}}{1 + 12.7\left(\frac{f}{8}\right)^{0.5}\left(\text{Pr}^{\frac{2}{3}} - 1\right)}. \quad (17)$$

Figure 3 shows the changes in the friction factor of the baffle-free geometric model compared to the experimental relationships for different Reynolds numbers. As can be seen, increasing Reynolds number up to the

Figure 3: The comparison of friction factor for baffle-free geometric model with experimental relations.

defined range decreases the friction factor. So, the large values of the friction factor are achieved for low Reynolds numbers. Figure 4 shows the Nusselt number variations for the baffle-free geometric model compared to the experimental relationships for different Reynolds numbers. As can be seen, in general, as the Reynolds number increases, the Nusselt number also increases. Here the effective parameter is the increase in heat transfer coefficient in proportion to the increase in Reynolds number.

As can be seen, the results of the numerical solution are in good agreement with the correlation.

Also, in order to validate the results, the values obtained for the friction factor and the Nusselt number for baffle-free geometry were compared with similar values in the experimental work of Ramadhan et al. [54] and a good agreement was observed.

The residual value (convergence criterion) states the end of the solution. Ansys Fluent software initially calculates

the temperature, pressure, and velocity of the flow in each cell and then the modeling process is continued until the convergence criterion and exact equilibrium mass and energy are met. The convergence criterion for all variables is 10^{-6} . The modeling is repeated until the exact convergence conditions are reached.

4 Optimization definition

The objective function specified in this research is the heat transfer performance (η) (equation (13)) and the design variables of this function are dimensionless pitch p/d and dimensionless baffle height h/d . Furthermore, maximizing was done in different Reynolds numbers which means that Reynolds number is one of the optimization parameters.

Figure 4: The comparison of Nusselt number of baffle-free geometric model with experimental relations.

4.1 Genetic algorithm

In order to achieve heat transfer optimization after modeling, the heat transfer performance of the system is considered as the objective function for maximizing heat transfer and minimizing pressure drop by using a genetic algorithm. One of the most effective methods in optimizing one or more objective function is using the genetic algorithm and this is why genetic algorithm is called function optimizer which can be used for optimizing objective functions in different problems. It should be noticed that the range of applications that use genetic algorithms is very wide. One of the advantages of optimizing with the genetic algorithm method is that it only deals with the value of the function and hence, the exact equation is not needed for selecting the function. In fact, choosing a suitable way to find the numerical value of the function by entering the variables is enough. This is in contrast with other optimization methods where the exact objective function is essential. For example, if the value of a function for different variables is found by one of the designing methods, there is no need to know the relationship between the variables and the function. Hence, the genetic algorithm method finds the optimal value with great accuracy. As another example, if there is any software for analyzing engineering problem that takes variables and gives the answer, one can obtain the optimal value of the function for the input variables by genetic algorithm without knowing what happens during the procedure. This is an advantage which rarely exists in other optimization methods. According to the possibilities of Ansys Fluent software, several experiments can be performed for optimization by considering different methods in order to reduce the computation time. This can be done by choosing the genetic algorithm as an optimization method. Optimization in Ansys Fluent without having a complete solution is possible by defining the input parameter and having access to the output values quickly. The accuracy of this optimization method depends on the various parameters such as the complexity of the variables, the number of points in the experiment, and the type of optimization method. To get the most effective answer from the facilities of Ansys Fluent software, a population of different models with different configurations is produced which is related to

the first population of genetic algorithm. Subsequent populations are achieved by mutating and crossing over the previous population. This method of optimization is more reliable and has higher accuracy.

5 Results

5.1 Grid independence study

To evaluate mesh independent solution, six grids with different number of cells are modeled for channel containing baffles with p/d of 0.5 and h/d of 0.125. The variations of Nusselt number are investigated when Reynolds number is 5,000. The results of this modeling are shown in Table 3. As can be seen, by increasing the number of cells in a grid with more than 3,471 cells, no significant changes appear in the Nusselt number. Moreover, the difference between two grids with 3,471 and 4,670 cells is very small (less than 1%) and as a result, the network with 3,471 cells is a good candidate for numerical simulation.

5.2 The effect of pitch and height of the baffle

5.2.1 Investigating the effect of baffle pitch

Modeling the flow of water inside the channel with smooth and baffled geometries with p/d ratios of 0.5, 1, and 1.25 is carried out to study its effect on friction factor, Nusselt number, and the heat transfer performance parameter, as shown in Figure 5. As can be seen, the installation of a baffle inside the channel causes the enhancement of heat transfer and pressure drop. It was also confirmed that smaller p/d results in more heat transfer and pressure drop, while less heat transfer and pressure drop occurred in larger p/d . In fact, the baffle pitch is inversely proportional to the heat transfer and friction factor so that increase in the pitch in the constant Reynolds number causes the decrease in heat transfer and pressure drop. On the other hand, in the constant p/d ,

Table 3: Results of grid independence study

Number of cells	509	1,074	2,273	3,471	4,670	5,867
Nu	50.876	56.947	57.938	58.8757	58.8961	58.897

Figure 5: The effect of baffle pitch on Nusselt number and friction factor.

increase in the Reynolds number causes the increase in heat transfer and the decrease in friction factor. Therefore, comparing to smooth channel, the percentage of increase in Nusselt number for the p/d ratios of 0.5, 1, and 1.25, is 141, 124, and 120%, respectively. Also, comparing to the smooth channel, the friction factor of these p/d ratios increases by 5.5, 5, and 4.25 times, respectively. The increase in heat transfer and pressure drop is mainly related to the presence of baffle and turbulence. As shown in Figure 6, the curve of heat transfer performance for the baffled channel with p/d ratio of 1.25 is higher than the curves of heat transfer performance with other pitches which is because of high pressure drop in baffled channels with smaller pitches.

5.2.2 Investigating the effect of baffle height

The presence of baffles with different heights affects the friction factor, Nusselt number, and heat transfer performance which are shown in Figure 7 where the fluid is water and the channel has constant pitch of p/d ratio of 1.25 with 3, 4, and 5 mm heights. As can be seen, with the increase in the baffle height, both the Nusselt number and the friction factor increase. However, its effect on friction factor is more. For instance, in 5,000 Reynolds number, the percentage of increase in Nusselt number for the baffle with 3 mm height is 35 % and the friction factor is increased by 2.5 times. In the same Reynolds number, however, for the baffle with 4 mm height, the percentage

Figure 6: The effect of baffle pitch on heat transfer performance.

Figure 7: The effect of baffle height on Nusselt number and friction factor.

of increase in Nusselt number is 68% and the friction factor increases by 5.57 times. For 10,000 Reynolds number, comparing to smooth channel, the percentage of increase in Nusselt number in 3 and 4 mm baffle heights is 54 and 79%, respectively, while the increase in friction factor is 7.5 and 9 times, respectively. On the other hand, for the constant height, increasing Reynolds number tends to increase the Nusselt number and decrease friction factor. By analyzing the heat transfer performance diagram and investigating the effect of baffle height, it is obvious that the curve of heat transfer performance for smaller height is located at the higher level which is due to the less increase in pressure drop. Furthermore, because of the flow turbulence created by increasing Reynolds numbers, the influence of baffle on heat transfer is reduced. The results are shown in Figure 8.

5.3 Optimization

After performing various simulations and investigating the sensitivity of heat transfer performance to geometric parameters, maximizing heat transfer performance is carried out as an objective function using genetic algorithm. In this regard, the dimensionless pitch p/d and height h/d of the baffle are studied as the variables of design and Reynolds number. Another important parameter is the Prandtl number which is related to water in the present study. Optimization can be used to obtain the best heat transfer performance at a given pitch and height. For this purpose, the changes in Nusselt number, friction factor, and heat transfer performance parameters in terms of dimensionless p/d are calculated and the results are compared. The outcomes of optimization also result in

Figure 8: The effect of baffle height on heat transfer performance.

Figure 9: The changes in the maximum heat transfer performance (η^*) versus the Reynolds number in different pitches.

Figure 11: The changes in maximum heat transfer performance (η^*) versus the Reynolds number in different heights.

different parameters like maximum heat transfer performance (η^*), $(p/d)^*$, and $(h/d)^*$ versus Reynolds number as shown in the diagrams of Figures 9, 10, 11 and 12. As can be seen, by increasing Reynolds number, the maximum heat transfer performance (η^*) is decreased by changes in $(p/d)^*$ because the increase in pressure drop depends on the increase in Reynolds number. In addition, for low Reynolds numbers, the maximum heat transfer parameter is placed at a higher level due to a lower pressure drop. In Figure 10, by increasing Reynolds number, the maximum value of (p/d) i.e., $(p/d)^*$ initially increases and then it changes slightly and remains almost constant which means that the effect of baffle pitch is reduced in high Reynolds number and the main cause of the pressure drop is the Reynolds number. It can be concluded that the maximum heat transfer performance of baffle is obtained in the high pitch and low Reynolds numbers. Meanwhile, if the baffle step decreases, the pressure drop is affected by low baffle pitch and increases again.

Also, in Figure 11, it is found that by increasing the Reynolds number, the maximum heat transfer performance (η^*) decreases by changes in the h/d ratio and

the reason is the increase in the pressure drop in proportion to the increase in the baffle height. Studying Figure 12 shows that with increasing Reynolds number, the maximum (h/d) i.e., $(h/d)^*$ decreases and the reason is that with increase in Reynolds number, the pressure drop increases. It can be concluded that the maximum heat transfer performance of the baffle is achieved at low baffle height and low Reynolds numbers. However, if the height of the baffle increases, the pressure drop will increase again. Also, for maximizing the efficiency of the heat exchanger, the height of the baffle must be reduced.

6 Discussion

With the help of modeling, the effect of Reynolds number, geometric parameters, and the flow turbulence on the Nusselt number and pressure drop have been investigated inside the channel. The presence of flow turbulence in the present study results in better fluid turbulence and

Figure 10: The changes in optimal dimensionless pitch $(p/d)^*$ versus Reynolds number.

Figure 12: The changes in optimal dimensionless baffle height $(h/d)^*$ versus Reynolds number.

heat transfer. The points to be drawn from this modeling are as follow: (1) Comparing to the smooth channel, the Nusselt number is higher at the presence of baffle turbulence which indicates that adding baffle causes more turbulence for the flow inside the channel followed by a significant influence on heat transfer optimization. (2) By increasing the Reynolds number and consequently increasing the mass flow rate of the fluid inside the channel, the flow turbulence and Nusselt number have increased. (3) Adding turbulent flow inside the channel causes the increase in resistivity to flow which increases the pressure drop inside the channel. (4) With increasing Reynolds number, the pressure drop increases but the friction factor is decreased. As mentioned above, applying a flow turbulence increases the pressure drop significantly. These variations are also observed for heat transfer so that with increasing Reynolds number, the heat transfer increases. (5) As it is clear from the diagram of heat transfer performance in terms of pitch, with increase in flow turbulent pitch, heat transfer and friction factor decreased but the decrease in friction factor is more. So, the maximum value of heat transfer performance is related to the turbulence with high pitch. (6) As can be seen from the diagram of heat transfer performance in terms of the baffle height, the heat transfer and friction factor increase with the increase in the baffle height, but the increase in the friction factor is more. So, the maximum value of the heat transfer performance is in low height. As discussed previously, the heat transfer performance increases by increasing the pitch and decreasing the height of flow turbulence. However, in order to determine the optimal state, the maximum dimensionless parameters of heat transfer performance (η^*), $(p/d)^*$, and $(h/d)^*$ are studied in terms of Reynolds number. (7) As can be seen from the diagram of heat transfer performance, optimization in terms of pitch and height of the flow turbulence, in an optimal condition, the value of η^* for most modeling regions is more than 1 which confirms the priority of heat transfer optimization to the increase in pressure drop. (8) From the results of the baffle pitch optimization, it is found that for every Reynolds number, there is a maximum heat transfer performance value. Moreover, by increasing Reynolds number, the maximum value of heat transfer performance decreases and occurs at a higher ratio of $(p/d)^*$. (9) Optimizing the baffle height and studying the changes in (η^*) and $(h/d)^*$ in terms of Reynolds number demonstrate that there is maximum heat transfer performance for each Reynolds number. By increasing the Reynolds number, the maximum value of the heat transfer performance decreases and occurs in a lower $(h/d)^*$.

7 Conclusion

In the present study, computational fluid dynamics was applied in two-dimensional geometry to investigate the effect of installing HTE (baffle) in the channel with water as the working fluid.

- The results showed that the addition of HTE (baffle) in the channel increased the rate of heat transfer and the pressure drop.
- For baffles with the same heights, maximum increase in Nusselt number and pressure drop was found in baffles with fewer pitches.
- The heat transfer performance of the baffle with higher pitch was more because of the reduction in pressure drop.
- The inverse relationship between baffle pitch and heat transfer or pressure drop was found comparable to smooth channel. A: the percentage of increase in Nusselt number was 141, 124, and 120% for the p/d ratios of 0.5, 1, and 1.25, respectively. B: For these p/d ratios, comparing to smooth channel, the friction factor increased by 5.5, 5, and 4.25 times, respectively.
- For the constant baffle pitch, however, the Nusselt number and pressure drop increased with increase in the baffle height. Since the pressure drop increased more, the heat transfer performance decreased.
- For smaller height, the heat transfer performance was at the higher level. For instance, for 5,000 Reynolds number, the increase in Nusselt number in baffle with 3 mm height was 35% and the friction factor increased by 2.5 times. Besides, for the baffle with 4 mm height, the increase in Nusselt number was 68% and the friction factor increased by 5.57 times.
- With increasing Reynolds number the increase in $(p/d)^*$ and decrease in $(h/d)^*$ were observed.
- For constant height, the maximum heat transfer performance of baffle was obtained in high pitch and low Reynolds numbers.
- For the constant pitch, the maximum heat transfer performance of baffle was achieved in low baffle height and low Reynolds numbers.
- As a remarkable result, it was found that compared to low Reynolds numbers, the maximum heat transfer performance decreased in high Reynolds numbers.
- The results of the optimization also showed that with increasing Reynolds number, the maximum heat transfer performance decreased which was proportional to the increase in p/d ratio and the decrease in h/d ratio.
- Maximum heat transfer performance of 1.5 was observed when Reynolds number was 2,000, at p/d ratio of 0.61 and h/d ratio of 0.36. Also, for Reynolds number equal to

10,000, the maximum heat transfer performance value was 1.19 at high p/d ratio of 0.93 and h/d ratio of 0.15.

- When the geometric dimensions of the problem under study are not available, the introduced optimal ratios can be used to maximize the heat transfer performance.

Funding information: The authors state no funding was involved.

Author contributions: Behzad Ghobadi and Farshad Kowsary initiated and performed the computation, Behzad Ghobadi, Farshad Kowsary and Farzad Veysi analyzed data. Finally, all authors wrote the paper and reviewed the revised manuscript and approved the submission.

Conflict of interest: The authors report no conflict of interests. The authors alone are responsible for the content and writing of the article.

Appendix

D	channel diameter (m)
C_p	specific heat capacity ($\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$)
L	channel length (m)
t	channel thickness (m)
R	channel radius (m)
h	channel height (m)
p	baffle pitch (m)
T	temperature (K)
u	velocity in x direction ($\text{m}\cdot\text{s}^{-1}$)
v	velocity in y direction ($\text{m}\cdot\text{s}^{-1}$)
k	thermal conductivity ($\text{W}\cdot\text{m}^{-2}\cdot\text{K}^{-1}$)
q	heat transfer (W)
Nu	Nusselt number
f	friction factor
ΔP	pressure drop
Re	Reynolds number
Pr	Prandtl number
$\dot{m}$	mass flow rate ($\text{kg}\cdot\text{s}^{-1}$)
W	baffle thickness (m)
ΔT_{lm}	logarithmic mean temperature difference

Greek letter

μ	kinematic viscosity ($\text{kg}\cdot\text{s}^{-1}\cdot\text{m}^{-1}$)
η	heat transfer performance parameter
ρ	density ($\text{kg}\cdot\text{m}^{-3}$)
τ	stress (Pa)

Subtitles

w	channel wall
b	bulk
E	enhanced
i	unit vector
t	turbulence

References

- [1] Gugulothu, R., N. S. Somanchi, V. K. Reddy, and K. Akkiraju. A review on enhancement of heat transfer in heat exchanger with different inserts. *Materials Today: Proceedings*, Vol. 4, No. 2, 2017, pp. 1045–1050.
- [2] Qasim, M., M. S. Kamran, M. Ammar, M. A. Jamal, and M. Y. Javaid. Heat transfer enhancement of an automobile engine radiator using ZnO water base nanofluids. *Journal of Thermal Science*, Vol. 29, 2020, pp. 1010–1024.
- [3] El-Said, E. M., M. Abdulaziz, and M. M. Awad. A numerical investigation on heat transfer enhancement and the flow characteristics in a new type plate heat exchanger using helical flow duct. *Cogent Engineering*, Vol. 4, 2017, id. 1396638.
- [4] Sheikholeslami, M. and D. D. Ganji. Nanofluid flow and heat transfer between parallel plates considering Brownian motion using DTM. *Computer Methods in Applied Mechanics and Engineering*, Vol. 283, 2014, pp. 651–663.
- [5] Abro, K. A., A. A. Memon, Sh. H. Abro, and I. K. I. Tlili. Enhancement of heat transfer rate of solar energy via rotating Jeffrey nanofluids using Caputo–Fabrizio fractional operator: An application to solar energy. *Energy Reports*, Vol. 5, 2019, pp. 41–49.
- [6] Zhao, N., S. Li, and J. Yang. A review on nanofluids: data-driven modeling of thermalphysical properties and the application in automotive radiator. *Renew Sustain Energy*, Vol. 66, 2016, pp. 596–616.
- [7] Kumar, P., D. Chaudhary, P. Varshney, U. Varshney, S. M. Yahya, and Y. Rafat. Critical review on battery thermal management and role of nanomaterial in heat transfer enhancement for electrical vehicle application. *Journal of Energy Storage*, Vol. 32, 2020, id. 102003.
- [8] Al-damook, A. and F. S. Alkasmoul. Heat transfer and airflow characteristics enhancement of compact plate-pin fins heat sinks – a review. *Propulsion and Power Research*, Vol. 7, No. 2, 2018, pp. 138–146.
- [9] Ghyadh, N., S. S. Ahmed, and H. J. Al Talqani. Review paper of heat transfer enhancement techniques of RIBS. *International Journal of Research and Analytical Reviews*, Vol. 7, No. 2, 2020, pp. 164–172.
- [10] Kumar, B., G. P. Srivastava, M. Kumar, and A. K. Patil. A review of heat transfer and fluid flow mechanism in heat exchanger tube with inserts. *Chemical Engineering and Processing: Process Intensification*, Vol. 123, 2018, pp. 126–137.

- [11] Kareem, Z. S., M. N. Mohd Jaafar, T. M. Lazim, S. Abdullah, and A. F. Abdulwahid. Passive heat transfer enhancement review in corrugation. *Experiment Thermal and Fluid Science*, Vol. 68, 2015, pp. 22–38.
- [12] Dewan, A., P. Mahanta, K. S. Raju, and P. S. Kumar. Review of passive heat transfer augmentation techniques. *Proceedings of Institution of Mechanical Engineers*, Vol. 218, 2004, pp. 509–527.
- [13] Léal, L., M. Miscevic, P. Lavieille, M. Amokrane, F. Pigache, F. Topin, et al. An overview of heat transfer enhancement methods and new perspectives: Focus on active methods using electroactive materials. *International Journal of Heat and Mass Transfer*, Vol. 61, 2013, pp. 505–524.
- [14] Song, Y., J. Wang, Y. Liu, and J. Zhao. Analysis of heat transfer influences on gas production from methane hydrates using a combined method. *International Journal of Heat and Mass Transfer*, Vol. 92, 2016, pp. 766–773.
- [15] Gholamalizadeh, E., F. Pahlevanzadeh, K. Ghani, A. Karimipour, T. K. Nguyen, and M. R. Safaei. Simulation of water/FMWCNT nanofluid forced convection in a microchannel filled with porous material under slip velocity and temperature jump boundary conditions. *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol. 30, No. 5, 2019, id. 00300.
- [16] Abdulrazzaq, T., H. Togun, H. Alsulami, M. Goodarzi, and M. R. Safaei. Heat transfer improvement in a double backward-facing expanding channel using different working fluids. *Symmetry*, Vol. 12, No. 7, 2020, id. 1088.
- [17] Nguyen, Q., S. N. Sedeh, D. Toghraie, R. Kalbasi, and A. Karimipour. Numerical simulation of the ferro-nanofluid flow in a porous ribbed microchannel heat sink: investigation of the first and second laws of thermodynamics with single-phase and two-phase approaches. *Journal of Brazilian Society of Mechanical Sciences and Engineering*, Vol. 42, 2020, id. 492.
- [18] Toghraie, D., M. Mahmoudi, O. A. Akbari, F. Pourfattah, and M. Heydari. The effect of using water/CuO nanofluid and L-shaped porous ribs on the performance evaluation criterion of microchannels. *Journal of Thermal and Calorimetry*, Vol. 135, No. 1, 2019, pp. 145–159.
- [19] Mendez, M., M. S. Shadloo, and A. Hadjadj. Heat transfer analysis of a transitional boundary layer over a concave surface with Görtler vortices by means of direct numerical simulations. *Physics of Fluids*, Vol. 32, No. 7, 2020, id. 074111.
- [20] Sarafraz, M. M., M. R. Safaei, M. Goodarzi, B. Yang, and M. Arjomandi. Heat transfer analysis of Ga-In-Sn in a compact heat exchanger equipped with straight micro-passages. *International Journal of Heat and Mass Transfer*, Vol. 139, 2019, pp. 675–684.
- [21] Goodarzi, H., O. Ali Akbari, M. M. Sarafraz, M. M. Karchegani, M. R. Safaei, and G. H. Ahmadi Sheikh Shabani. Numerical simulation of natural convection heat transfer of nanofluid with Cu, MWCNT, and Al_2O_3 nanoparticles in a cavity with different aspect ratios. *Journal of Thermal Science and Engineering Applications*, Vol. 11, No. 6, 2019, id. 4043809.
- [22] Meng, H., J. Qi, Z. Zheng, F. Shi, and Y. Lei. Numerical simulation of nickel-based alloys' welding transient stress using various cooling techniques. *High Temperature Materials and Processes*, Vol. 39, 2020, pp. 633–644.
- [23] Wang, Y. C., P.-J. Liu, G.-P. Luo, Z. Liu, and P.-F. Cao. Optimization of heat treatment of glass-ceramics made from blast furnace slag. *High Temperature Materials and Processes*, Vol. 39, 2020, pp. 539–544.
- [24] Hua Chen, H., J. Chen, W. Wang, and H. Lin. Research on electromagnetic sensitivity properties of sodium chloride during microwave heating. *High Temperature Materials and Processes*, Vol. 39, 2020, pp. 54–62.
- [25] Li, T., Sh.-i Shimasaki, S. Narita, and S. Taniguchi. Energy dissipation rate in an agitated crucible containing molten metal. *High Temperature Materials and Processes*, Vol. 36, No. 10, 2017, pp. 993–1000.
- [26] Hasanpour, A., M. Farhadi, and K. A. Sedighi. Review study on twisted tape inserts on turbulent flow heat exchangers: The overall enhancement ratio criteria. *International Communication in Heat and Mass Transfer*, Vol. 55, 2014, pp. 53–62.
- [27] Garg, M. O., H. Nautiyal, S. Khurana, and M. K. Shukla. Heat transfer augmentation using twisted tape inserts: A review. *Renewable and Sustainable Energy Reviews*, Vol. 63, 2016, pp. 193–225.
- [28] Alam, T. and M. H. Kim. A comprehensive review on single phase heat transfer enhancement techniques in heat exchanger applications. *Renewable and Sustainable Energy Reviews*, Vol. 81, 2018, pp. 813–839.
- [29] Kumar, A. and M.-H. Kim. Thermal hydraulic performance in a solar air heater channel with multi V-type perforated baffles. *Energies*, Vol. 9, No. 7. id. 564.
- [30] Budak, N., H. Yucel, and Z. Argunhan. Experimental and numerical investigation of the effect of turbulator on heat transfer in a concentric-type heat exchanger. *Experimental Heat Transfer*, Vol. 29, No. 3, 2016, pp. 322–336.
- [31] Cheloi, N. A., O. Ali Akbari, and D. Toghraie. Computational fluid dynamics and laminar heat transfer of water/Cu nanofluid in ribbed microchannel with a two-phase approach. *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol. 29, No. 5, 2019, id. 0243.
- [32] Parsaiemehr, M., F. Pourfattah, O. Ali Akbari, D. Toghraie, and G. H. Sheikhzadeh. Turbulent flow and heat transfer of water/ Al_2O_3 nanofluid inside a rectangular ribbed channel. *Physica E: Low-dimensional System and Nanostructures*, Vol. 96, 2018, pp. 73–84.
- [33] Heydari, A., O. Ali Akbari, M. R. Safaei, M. Derakhshani, A. A. A. Alrashed, R. Mashayekhi, et al. The effect of attack angle of triangular ribs on heat transfer of nanofluids in a microchannel. *Journal of Thermal Analysis and Calorimetry*, Vol. 131, No. 6, 2017, pp. 2893–2912.
- [34] Pourfattah, F., M. Motamedian, G. H. Sheikhzadeh, D. Toghraie, and O. Ali Akbari. The numerical investigation of angle of attack of inclined rectangular rib on the turbulent heat transfer of water- Al_2O_3 nanofluid in a tube. *International Journal of Mechanical Sciences*, Vol. 131–132, 2017, pp. 1106–1116.
- [35] Gholami, M. R., O. Ali Akbari, A. Marzban, D. Toghraie, G. H. Ahmadi Sheikh Shabani, and M. Zarringhalam. The effect of rib shape on the behavior of laminar flow of oil/MWCNT nanofluid in a rectangular microchannel. *Journal of Thermal Analysis and Calorimetry*, Vol. 134, 2018, id. 1628.
- [36] Behnampour, A., O. Ali Akbari, M. R. Safaei, M. Ghavami, A. Marzban, G. H. Sheikh Shabani, et al. Analysis of heat transfer and nanofluid fluid flow in microchannels with rectangular and triangular shaped ribs. *Physica E:*

- Low-Dimensional Systems and Nanostructures*, Vol. 91, 2017, pp. 15–31.
- [37] Heydari, M., D. Toghraei, and O. Ali Akbari. The effect of semi-attached and offset mid-truncated ribs and water/TiO₂ nanofluid on flow and heat transfer properties in a triangular microchannel. *Thermal Science and Engineering Progress*, Vol. 2, 2017, pp. 140–150.
- [38] Rezaei, O., O. Ali Akbari, A. Marzban, D. toghraie, F. Pourfattah, and R. Mashayekhi. The numerical investigation of heat transfer and pressure drop of turbulent flow in a triangular microchannel. *Physica E: Low-dimensional Systems and Nanostructures*, Vol. 93, 2017, pp. 179–189.
- [39] Varzaneh, A. A., D. Toghraie, and A. Karimipour. Comprehensive simulation of nanofluid flow and heat transfer in straight ribbed microtube using single-phase and two-phase models for choosing the best conditions. *Journal of Thermal Analysis and Calorimetry*, Vol. 139, 2020, pp. 701–720.
- [40] Ali Akbari, O., D. Toghraie, A. Karimipour, M. R. Safaei, M. Goodarzi, H. Alipour, et al. Investigation of rib's height effect on heat transfer and flow parameters of laminar water–Al₂O₃ nanofluid in a rib-microchannel. *Applied Mathematics and Computation*, Vol. 290, 2016, pp. 135–153.
- [41] Gravndyan, Q., O. Ali Akbari, D. Toghraie, A. Marzban, R. Mashayekhi, R. Karimi, et al. The effect of aspect ratios of rib on the heat transfer and laminar water/TiO₂ nanofluid flow in a two-dimensional rectangular microchannel. *Journal of Molecular Liquids*, Vol. 236, 2017, pp. 254–265.
- [42] Goodarzi, M., I. Tlili, Z. Tian, and M. R. Safaei. Efficiency assessment of using graphene nanoplatelets-silver/water nanofluids in microchannel heat sinks with different cross-sections for electronics cooling. *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol. 30, No. 1, 2019, id. 0730.
- [43] Sheikh, R., S. Gholampour, H. Fallahsohi, M. Goodarzi, M. Mohammad Taheri, and M. Bagheri. Improving the efficiency of an exhaust thermoelectric generator based on changes in the baffle distribution of the heat exchanger. *Journal of Thermal Analysis and Calorimetry*, Vol. 143, 2021, pp. 523–533.
- [44] Aghaei, A., G. H. A. Sheikhzadeh, M. Goodarzi, H. Hasani, H. Damirchi, and M. Afrand. Effect of horizontal and vertical elliptic baffles inside an enclosure on the mixed convection of a MWCNTs-water nanofluid and its entropy generation. *The European Physical Journal Plus*, Vol. 133, 2018, id. 486.
- [45] Mohebbi, K., R. Rafee, and F. Talebi. Effects of Rib shapes on heat transfer characteristics of turbulent flow of Al₂O₃-water nanofluid inside ribbed tubes. *Iranian Journal of Chemistry and Chemical Engineering*, Vol. 34, No. 3, 2015, pp. 61–77.
- [46] Goktepli, I. and U. Atmaca. Computational study on the effect of the staggered Ribs on heat transfer phenomena between the horizontal plates. *Journal of Science & Engineering*, Vol. 8, 2021, pp. 7–17.
- [47] Menni, Y. and A. Azzi. Numerical analysis of thermal and aerodynamic fields in a channel with cascaded baffles. *Periodica Polytechnica Mechanical Engineering*, Vol. 62, 2018, pp. 16–25.
- [48] Abdulhameed, J. J., D. Saad, and M. S. Kassim]. Effect of baffles geometry on heat transfer enhancement inside corrugated duct. *International Journal of Mechanical Engineering and Technology*, Vol. 10, No. 3, 2019, pp. 763–773.
- [49] Sahamifar, S., F. Kowsary, and M. H. Mazlaghani. Generalized optimization of cross-flow staggered tube banks using a subscale model. *International Communications in Heat and Mass Transfer*, Vol. 105, 2015, pp. 46–57.
- [50] Nejad, S. M. *Fundamentals of turbulent flows and turbulence modeling*, Danesh Negar Pub. In Persian, 2009.
- [51] Menter, F. R. Two- equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598–1605.
- [52] Incropera, F. P., P. D. Witt, T. L. Bergman, A. S. Lavine. *Fundamentals of heat and mass transfer*, John-Wiley & Sons. United State. 1998.
- [53] Webb, R. L. Performance evaluation criteria for use of enhanced heat transfer surfaces in heat exchanger design. *International Journal of Heat and Mass Transfer*, Vol. 24, No. 4, 1981, pp. 715–726.
- [54] Ramadhan, A. I., W. H. Azmi, R. Mamat, and K. A. Hamid. Experimental and numerical study of heat transfer and friction factor of plain tube with hybrid nanofluids. *Case Studies in Thermal Engineering*, Vol. 22, 2020, id. 100782.