Heat transfer enhancement in a rectangular channel with pillars

Feng Jiao, Chengzhe Li, Songjiang Dai, Meilin Hu and Yongqing He

1 School of Chemical Engineering, Kunming University of Science and Technology, Kunming 650500, China
2 Yunnan PetroChina Kunlun Gas Co., Ltd, Kunming 650500, China
3 Chongqing Key Laboratory of Micro-Nano System and Intelligent Sensing, Chongqing Technology and Business University, Chongqing 400067, China
4 E-mail: yqhe@ctbu.edu.cn

Abstract. This paper numerically studied the heat transfer performance of water flowing in a rectangular channel with pillar structures. We systematically investigated the influence of the pitch, inclination angle, shape, and pillar arrangement on the flow and heat transfer characteristics. The results show that adopting spacing with S/d=2 can improve heat transfer performance sufficiently for a low Reynolds number case. Meanwhile, the noticeable enhancement can also be observed by making the pillar inclination direction consistent with the flow direction or staggered rhombic-shaped.

1. Introduction
It is laborious to enhance the heat transfer performance effectively by simply increasing the flow rate in laminar flow. The Nusselt number can be increased effectively only by using appropriate methods to guide the radial flow [1]. It has been pointed out that the insertion in the channel can enhance the heat transfer as well as the resistance. And most of the studies about heat transfer enhancement focus on uniform heat flux. The research on non-uniform heat flux in a narrow rectangular channel is even less. However, non-uniform heating is more practical, such as battery heat dissipation [2].

Guan et al. [3] investigated that heat transfer of de-ionized water over in-line and staggered microcylinder-groups have been numerically investigated with Reynolds number varying in the range from 25 to 150. Ahmad et al. [4] studied steady flow of a power law fluid through a tapered non-symmetric stenotic tube. Yadav et al. [5] studied that heat transfer enhancement in microchannel using extended surface has been carried out. It is found that heat transfer performance of Case I (upstream finned microchannel) is better than Case II (downstream finned microchannel). Case I even performs better than Case III (complete finned microchannel) at low Reynolds number. Turkyilmazoglu, M [6] studied the heat transfer of the exponential fin under the action of motion and heat generation (absorption), and obtained the calculation formula of the accurate temperature distribution. Liu et al. [7] experimentally studied the effects of pillars with different heights on the heat transfer performance and flowed characteristics of water in a channel. Qu et al. [8], Zhang et al. [9], Huang et al. [10] and Sweeney et al. [11] used numerical methods to study the flow and heat transfer characteristics of conventional scale and pillar groups in laminar, transitional and turbulent flows, and improved the prediction of flow and heat transfer of fins or pillar groups. Turkyilmazoglu, M [12] analyzed the composite heat and mass transfer mechanism and its effect on efficiency caused by the difference in
temperature and humidity ratio of different exponential porous wing structures. Gong et al. [13] studied the influence of the height and shape of pillar, and the thermal conductivity of radiator on the heat transfer performance of heat source. Turkylmazoglu, M [14] found that stretching degrades the efficiency, whereas shrinking provides better fin efficiency pointing to significant advantages in terms of fin design purposes. Liu et al. [15] experimentally studied the resistance characteristics and Nu of water flowing through pillars at different distances in the in-line and staggered arrangement. Taiho et al. [16] and Tamayol et al. [17] have also done experimental research on this model. Hamed et al. [18] studied the inertial flow deformation around a single cylinder in a microchannel, and proposed that the shape of the pillars be carved into complex geometric shapes for moving and separating fluids, performing solution exchange and achieving particle separation, which is of great significance for the automation of biology, chemistry and materials. Alfarawi S et al. [19] measured heat transfer and flow friction in a fully developed turbulent flow in a rectangular tube with three different geometrical ribs (semicircular ribs, rectangular ribs, and a mixture of two ribs) on the bottom wall.

Cho et al. [20, 21] experimentally investigated the effect of the heat flux condition and geometry of the microchannel heat sink on the temperature distribution. Turkylmazoglu, M [22] found that the temperature felt by the measured object in the continuously heated fluid was higher when it moved upward than when it moved downward. Ahmed et al. [23] studied the effect of using subchannels in a liquid cooled heat sink for minimizing the effect of hotspots generated on a chip or circuit. Davide et al. [24] presented a new experimental technique to detect the onset of dryout and to determine the dryout quality in a nonuniformly heated microchannel. Heat transfer tests of subcooled water flow boiling in a circular channel with a twisted tape insert under high and non-uniform fluxes have been performed by Zhu et al. [25]. Turkylmazoglu, M [26] solved the coupled energy equation of the thermal phenomena of particle solids and cooling fluid in the parallel plate moving bed heat exchanger by analytical method. Kazem [27] presented the results of an experimental examination of the effect of non-uniform wall temperature on local heat transfer coefficient in a rotating smooth-walled square channel. Xu et al. [28] designed two novel two-layer mini-channel heat sinks to improve the uniformity of surface temperature distribution.

With the rapid development electronic industry, electronic components are developing toward miniaturization. The heat dissipation space is limited, and the heat dissipation of electronic components is not uniform. On this basis, the structural optimization of the pillars in the rectangular channel are simulated numerically, and the relationship between the temperature field and the eddy distribution is studied.

2. Mathematical model

2.1. Model description

(a) In-line arrangement  
(b) Staggered arrangement  
(c) The side of the channel

Figure 1. Geometric dimensions of models.
The physical and geometric structure of the pillar is shown in the Figure 1. \( \theta \) is inclination angle of the pillar. In the example, \( d = 0.5 \text{ mm}, L = 24 \text{ mm}, W = 4.5 \text{ mm}, H = 1 \text{ mm}, h = 0.75 \text{ mm}, \) and \( S_0 = 5 \text{ mm} \). Different situations of \( S/d, \theta \), shape and arrangement of pillars are numerically simulated. Detailed dimensions of the pillar groups are listed in Table 1.

**Table 1.** Detailed dimension of the calculated chips of pillar groups.

| No. | \( S/d \) | \( \theta \) | Shape | Arrangement |
|-----|----------|----------|-------|-------------|
| Case1 | 1.5 | 90°  | Circular | In-line |
| Case2 | 2.0  | 90°  | Circular | In-line |
| Case3 | 3.0  | 90°  | Circular | In-line |
| Case4 | 2.0  | 90°  | Circular | Staggered |
| Case5 | 3.0  | 90°  | Circular | Staggered |
| Case6 | 2.0  | 60°  | Circular | In-line |
| Case7 | 2.0  | 120° | Circular | In-line |
| Case8 | 2.0  | 90°  | Ellipse  | In-line |
| Case9 | 2.0  | 90°  | Rhombus  | In-line |
| Case10| 2.0 | 90°  | Ellipse  | Staggered |
| Case11| 2.0 | 90°  | Rhombus  | Staggered |

2.2. **Numerical methods**

Three-dimensional numerical investigation is performed by ANSYS Fluent. To simplify the calculation, the following assumptions were made: (1) The water is considered incompressible with constant properties. The flow is laminar and steady state. (2) The conductivity of the wall is ignored. (3) The fluid flow is fully developed. (4) There is no temperature jump and velocity slip on the wall of the channel. (5) The buoyancy and radiation effects are neglected.

The governing equations (continuity equation, momentum equation and energy equation) of incompressible steady state flow in a channel are as follows [4]:

\[
\frac{\partial (\rho u_i)}{\partial x_i} = 0; \quad \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}\left(\rho u_j C_p T - k \frac{\partial T}{\partial x_j}\right) = 0
\]

(1)

The laminar flow model and finite volume method are used to discretize the equation. The coupling of pressure and velocity is implemented by SIMPLE algorithm. The standard pressure and the second order upwind discretization scheme for momentum and energy are employed in the model. Furthermore, the appropriate sub-relaxation factor is selected and the Boussinesq approximation is activated as the density calculation method. Set up the thermal expansion coefficient of liquid. The convergence criterion of 10^-8 is chosen for all calculated parameters.

2.3. **Boundary conditions**

The boundary conditions for the numerical model consisting of walls (including heating walls and adiabatic walls), a velocity inlet, a full developed outlet and two symmetry planes are expressed as follows:

![Figure 2. Non-uniform heating in channel.](image-url)
Considering the non-uniformity of heat generated by electronic components, normal distribution of the heat flux on the wall is adopted, as shown in the Figure 2. To certificate the numerical results with the experimental results in the Reference [29], the water inlet temperature is 25°C.

Heating walls:

\[ u, v, w|_{wall,h} = 0, \frac{\partial T}{\partial n}_{wall,h} = q; \]  

(2)

Adiabatic walls:

\[ u, v, w|_{wall,h} = 0, \frac{\partial T}{\partial n}_{wall,h} = 0; \]  

(3)

Inlet:

\[ u|_{inlet} = u_m, v, w = 0, T|_{inlet} = 298.15K; \]  

(4)

Outlet:

\[ \frac{\partial u}{\partial n}_{outlet} = 0, \frac{\partial v}{\partial n}_{outlet} = 0, \frac{\partial w}{\partial n}_{outlet} = 0, \frac{\partial T}{\partial n}_{outlet} = 0. \]  

(5)

The heat flux distribution function is as follows:

\[ q = q_{m} \cdot e^{\frac{-3 \times 10^2 x^2}{2}} + q_{m} \cdot e^{\frac{-3 \times 10^2 y^2}{2}} \]  

(6)

where the \( q_{m} \) is the maximum heat flux on the heating surface, 0.21 MW/m\(^2\) is chosen in this paper; \( x \) and \( y \) are the axial and radial lengths respectively.

2.4. Parameter definitions

The qualitative temperature of water (\( T_m \)) is defined as the average temperature of inlet and outlet temperatures. The Reynolds number (\( Re \)), the average Nusselt number (\( Nu \)), heat transfer coefficient (\( h \)), friction coefficient (\( f \)) and thermal performance factor (\( \eta \)) in the rectangular channel are defined as follows:

\[ T_m = \frac{T_{in} + T_{out}}{2}; Re = \frac{d_e \cdot u_{max}}{\mu}; Nu = \frac{h \cdot d_e}{\lambda}; h = \frac{Q}{A \cdot \Delta T}; f = \frac{2 \cdot d_e \cdot \Delta p}{\rho u_{max}^2}; \eta = \frac{Nu}{Nu_{ref}} \]  

(7)

where \( q \) is the average heat flux in the tube, \( \Delta T \) is the difference between the average temperature of the inner wall of the tube and the qualitative temperature, \( \Delta P \) is the pressure drop at the inlet and outlet of the channel and \( u_{max} \) is the maximum velocity of the water.

For in-line arrays, the flow resistance in laminar flow can be calculated by Gaddis-Gnielks formula:

\[ f = \frac{280 \pi}{\text{Re} \left[ \frac{1}{4} \left( \frac{S}{d} \right)^2 \right]} \left[ \left( \frac{S}{d} \right)^2 - 0.6 \left( \frac{S}{d} \right)^{1.6} + 0.75 \right] \]  

(8)

2.5. Model validation

The model is meshed by Gambit. The sample of the grid is shown in Figure 1(a). In order to validate the independence of solution on the grid, grid independence test is made to ensure the relative errors in both the averaged Nusselt numbers and friction factor between such grids are less than 2% for each model.

\( Nu \) number is calculated according to the number of grid elements 489176 to 181708. The deviation of \( Nu \) is 1.8%. Therefore, 489176 are used to discretize the computational domains.
To validate the reliability of the numerical method being used, the corresponding numerical results are compared with the experimental results [5, 29] and the values of Equation (8) in Figure 3. As shown in the left figure, the maximum error between the simulation results and the experimental values is 1.7% and the maximum error between the simulation results and the Equation (8) is 2.8%. As shown in the right figure, there is good agreement between the predicted and experimental results [29] about the water outlet temperature. It indicates that the numerical model is reliable.

![Figure 3. Validation of the model.](image)

3. Results and discussion

3.1. Streamline distributions

![Figure 4. Streamline distributions at Re=75 (in-line array).](image)

Figure 4 shows the streamline distributions in in-line arranged pillars with different pitches when Re=75. There is no vortex along the flowing direction behind the two rows of pillars closes the wall due to the strong interaction among pillars at S/d=1.5. However, after the fluid has bypassed all the ore columns, the tail flow return area becomes very large. The reason is that the velocity of water on both sides is fast and the pressure decreases. The pressure difference between the column and the central area drives the fluid to both sides.

When S/d=2, almost all the pillars wake regions appear vortex, which indicates that the influence of the pipe wall on the column and the weakened influence of the columns on each other combined to affect the flow, causing some of the water to flow from the sides to the column void area. While S/d=3, there is only one row of the middle rows has vortex phenomenon, which can be seen from the Figure 4(c) that the pillar gap is too large. The interaction with the pillars and wall is prominent.

As shown in the Figure 5, pillars in the staggered arrangement can make fluid flow more disorderly, and bring different heat transfer effects. The staggered arrangement results in the interaction of adjacent pillars. The number of vortex and pillars in the wake region of pillars is not as regular as that in-line array. In the case of S/d=1.5 and 2, vortex phenomenon is seldom seen in the wake area on the back of pillar, whereas in the case of S/d=3, almost every pillar wake area has shedding pattern, which
leads to an increase in pressure drop. This is because the interaction between pillars is weakened due to the large pitches.

Figure 5. Streamline distributions at Re=75 (staggered array).

Figure 6. The influence of different pitch of pillars on Nu and f.

Figure 6 shows the effect of pitches on Nu and f in staggered arrangement. It is found that the heat transfer performance is the best when S/d = 2, and the worst when S/d = 1.5. This is because the flow is need affecting by pillars and the wall, and the strong change of liquid mixing between different regions will enhance the heat transfer ability. The small pitch will weaken the disturbance of the pillar by making a large amount of water flow through both sides; the large pitch will weaken the effect of the wall. The resistance coefficient increases with the increase of the pitch between the pillars, which is due to the increase of the pitch between the pillars and the increase of the area affected by the pillars in the whole channel. Compared to the other two cases, the one with S/d=2 has a very good heat transfer effect.

3.2. Heat flux

Figure 7 shows the temperature distributions of channels with three kinds of pillars under uniform heat flux (UHF) and non-uniform heat flux (NHF) at \( u_{in} = 0.0411 \text{ m/s} \), \( T_{in} = 298.15 \text{ K} \) and \( q_{in} = 0.21 \text{ MW/m}^2 \). As shown in the figure, we can see that pillars can change the temperature distribution. UHF high temperature region appears in the latter half of the channel, while NHF high temperature region appears in the center of the channel, and the temperature at the inlet is much lower than that of UHF. This is because that the heat flux is concentrated in the center of the channel. When the fluid flows through the concentrated heated wall, it begins to be heated efficiently. Compared with the wall temperature distribution of three kinds of pillars under NHF, the temperature distribution of rhombic pillars in high heat source area is relatively uniform, and there is no temperature jump. This has a better effect on solving the temperature non-uniformity, and can reduce the hot spot risk, thereby reducing the channel stress.
3.3. The shape of pillars

The effect of pillars in staggered array with different shapes on $Nu$ and $f$ is shown in Figure 8. Heat transfer performance of rhombic pillars is the best. Because the boundary layer becomes thinner when water flows through the pillars. The disturbance of the rhomboid microcolumn structure to the fluid is enhanced. And the cross-section area of the channel suddenly becomes larger after flowing through the rhombic. Then the sudden change of flow velocity is more likely to produce low pressure, resulting in intense collision of liquid, forming vortex or secondary flow, thus enhancing the heat transfer. The elliptical pillars are too smooth to cause sudden change, the phenomenon is not obvious, and the flow is relatively stable. Owing to the small size of its short axis, the cross-section area of the elliptical pillars flowing through both sides is relatively large, and the velocity of flow increases slowly, so it is not easy to produce the collision between fluids. It also can be seen that the wake area of the elliptical pillars is relatively narrow and its velocity is relatively large. So, it is not easy to form a vortex. When
the water flows through the rhombic, the velocity increases, while the velocity in the wake region is slow, which is prone to generate vortex.

### 3.4. Effect of pillar inclination

Figure 9 shows the effect of the inclination of pillar on $Nu$ and $f$ in staggered arrangement. $Nu$ decreases with the increase of $\theta$, and $f$ increases with the increase of $\theta$. This is because the inclined direction of the pillar is consistent with the direction of flow velocity, which has a certain guiding effect. The flow of fluid through the pillar not only produces disturbance, but also hinders the flow when the inclined direction of the pillar is opposite to the velocity direction, resulting in the need for more energy to overcome the viscous force on the wall, and the resistance coefficient increases with the increase of the inclination of the pillar.

The thermal performance factor of different $\theta$ is shown in the Figure 10. When $\theta$ is $60^\circ$, the thermal performance factor is biggest. It indicates that the thermal performance is best at this case.

![Figure 9. Effect of $\theta$ on $Nu$ and $f$.](image)

![Figure 10. The thermal performance factor of different $\theta$.](image)

### 4. Conclusions

The effects of geometrical parameters of pillars on fluid flow and heat transfer characteristics in rectangular channels were compared under non-uniform heat flux. The following conclusions are obtained:

(a) Non-uniform heat flux will cause high non-uniform temperature distribution on the heating wall. Pillars at the channel central heat source can effectively change the non-uniformity of wall temperature and reduce the hot spot risk, thus reducing the channel stress.
(b) The near-wall passageway and the pillar gap passageway affect the fluid flow together. The stronger the interaction between pillars, the less vortex will appear in the wake region near the wall pillars.

(c) The pitch, inclination, shape and arrangement of pillars will affect the heat transfer characteristics of the channel. At low $Re$, the heat transfer performance can be effectively improved by changing the distance between pillars to be close to the wall and the inclined direction to be consistent with the flow direction. The rhombus pillars and staggered arrangement can be used to improve the heat transfer performance.

(d) The inclination of pillar also can affect the heat transfer characteristics of the channel. And when $\theta=60^\circ$, the thermal performance factor has the best value.

Acknowledgements
The authors gratefully acknowledge the financial support from the National Natural Science Foundation of China (Grant No. 11962010 and 11502102) and Open Fund of Chongqing Technology and Business University (Grant No. KFJJ2019064).

References
[1] Xianhe D, Yuxiang H and Haimin L 2012 Chem. Ind. & Eng. Pro. 31 2390-94
[2] Ling L 2012 Computers & Fluids 35 855-62
[3] Guan N, Liu Z and Zhang C W 2012 Heat & Mass Transf. 48 1141-53
[4] Ahmad R, Farooqi A, Zhang J Z and Ail N 2019 Appl. Math. & Nonlinear Sci. 1 249-60
[5] Yadav V, Baghel K, Kumar R and Kadam S T 2016 Int. J. Heat Mass Transf. 93 612-22
[6] Turkyilmazoglu M 2018 Int. J. Heat Mass Transf. 116 346-51
[7] Liu Z, Zhang C and Guan N 2011 Exp. Therm. Fluid Sci. 35 226-33
[8] Qu W and Siu-Ho A 2009 Int. J. Heat Mass Transf. 52 1853-63
[9] Zhang P F, Wang J J and Huang L X 2006 Appl. Ocean Res. 28 183-92
[10] Huang Z, Olson J A, Kerekes R J and Green S I 2006 Computers & Fluids 35 485-91
[11] Sweeney C and Meskell C 2004 J. Fluids Struct. 18 501-12
[12] Turkyilmazoglu M 2014 Int. J. Refrig.-Rev. Int. Froid. 46 158-64
[13] Gong S, Chen L, Xie Z, Feng H and Sun F 2016 Sci. China-Technol. Sci. 59 631-39
[14] Turkyilmazoglu M 2015 Energy Conv. Manag. 91 199-203
[15] Liu Z, Wang Z and Zhang C 2013 Heat & Mass Transf. 54 1052-67
[16] Yeom T, Simon T, Zhang T, Zhang M, North M and Cui T 2016 Int. J. Heat Mass Transf. 92 617-27
[17] Tamayol A, Yeom J, Akbari M and Bahrami M 2013 Int. J. Heat Mass Transf. 58 420-26
[18] Amini H, Sollier E, Masaeli M, Xie Y, Ganapathysubramanian B, Stone H A and Carlo D D 2013 Nat. Commun. 4 1826
[19] Alfarawzi S, Abdel, Moniem S A and Bodalal A Int. J. Therm. Sci. 118 123-38
[20] Cho E S, Choi J W, Yoon J S and Kim M S 2010 Int. J. Heat Mass Transf. 53 2159-68
[21] Cho E S, Choi J W, Yoon J S and Kim M S 2010 Int. J. Heat Mass Transf. 53 1341-48
[22] Turkyilmazoglu M 2016 Appl. Therm. Eng. 98 280-287
[23] Al-Waaly A A Y, Paul M C and Dobson P 2017 Appl. Therm. Eng. 115 558-74
[24] Col D D and Bortolin S 2012 Int. J. of Therm. Sci. 57 25-36
[25] Zhu G, Bi Q, Cai L, Yan J and Lv H 2018 Appl. Therm. Eng. S1359431118301704
[26] Turkyilmazoglu M 2019 J. Heat Transf.-Trans. ASME. 141 11
[27] Farhadi K 2007 Chem. Eng. Sci. 62 3672-84
[28] Xu S, Yang L, Li Y, Wu Y and Hu X 2016 Heat & Mass Transf. 52 1755-63
[29] Wang N 2014 MA. Sc. South China University of Technology