Predicting the effect of the rib pitch on thermal performance factor of small channels plate heat exchangers fitted with Y and C shapes obstacles

Sirine Chtourou¹ · Hassene Djemel² · Mohamed Kaffel³ · Mounir Baccar¹

Received: 23 October 2020 / Accepted: 5 March 2021 / Published online: 24 March 2021

© The Author(s) 2021 [OPEN](#)

Abstract

This study presents a numerical analysis of a laminar counter flow inside small channels plate heat exchanger fitted with Y and C shape obstacles. Using the Computational Fluid Dynamics CFD, an advanced and modern simulation technique, the influence of the geometrical parameters (such as geometry, rib pitch) on the flow characteristics, the thermal and the hydrodynamics performance of the PHE (plate heat exchanger) is investigated numerically. The main goal of this work is to increase the flow turbulence, enhance the heat transfer and the thermal efficiency by inserting new obstacles forms. The computational domain is a conjugate model which is developed by the Computer Aided Design CAD software Solidworks. The results, obtained with Ansys Fluent, show that the presence of the shaped ribs provides enhancement in heat transfer and fluid turbulence. The CFD analysis is validated with the previous study. The non-dimensional factors such as the Nusselt number Nu , the skin friction factor C_f and the thermo-hydraulic performance parameter THPP are predicted with a Reynolds number Re range of 200–800. The temperature and the velocity distribution are presented and analyzed. The Y ribs and the C ribs offer as maximum THPP values respectively about 1.44 and 2.6 times of a smooth duct.

Keywords Counter flow · Plate heat exchanger · CFD · Solidworks · Ansys Fluent · THPP

Nomenclature

A	Surface area (m^2)
L	Channel Length (m)
W_{ch}	Channel width (m)
H_h	Hot channel height (m)
H_c	Cold channel height (m)
D_h	Hydraulic diameter (m)
t	Plate thickness (m)
Re	Reynolds number
Nu	Nusselt number
C_f	Friction factor
$THPP$	Thermo-hydraulic performance
K	Thermal conductivity of the fluid (W/m K)
C_p	Specific heat of the fluid (J/(kgK))
ρ	Density of the fluid (Kg/m ³)
μ	Dynamic viscosity of the fluid (Pa s)

T	Temperature (K)
P	Pressure (Pa)
ΔP	Pressure drop (Pa)
H	Convective heat transfer coefficient (W/m ² K)
V	Fluid flow velocity (m/s)
U	Fluid x-component velocity (m/s)
v	Fluid y-component velocity (m/s)
w	Fluid z-component velocity (m/s)

Subscripts

h	Hot fluid
c	Cold fluid
s	Solid
in	Inlet
out	Outlet

✉ Sirine Chtourou, sirine.chtourou@enis.tn | ¹CFDTP, National School of Engineers of Sfax (ENIS), Sfax 3038, Tunisia. ²ISTSL Higher Institute of Transport and Logistics of Sousse, Sousse, Tunisia. ³IPEIN Preparatory Institute for Engineering Studies in Nabeul, Nabeul, Tunisia.

1 Introduction

The plate heat exchanger (PHE) is a device that transfers heat between fluids (liquid or gaseous) through an energy interaction. Indeed, the hot fluid gives part of its enthalpy to cold fluid. The PHE is characterized by its great diversity. First, it can be classified according to the transfer mode type. In addition, it can be categorized according to the contact type as direct contact and indirect contact. Moreover, Mota et al. [1] classified it according to the plate corrugations type as herringbone plate, zigzag plate protrusions and depression plate and washing board plate. Also, they showed that the PHE can be classified according to the flow arrangement type. Besides, this heat exchanger type can be classed according to its design.

Pirozfam and et al. [2] investigated the influence of the channel geometrical parameters on the PHE performance. Therefore, they simulated the hydrodynamics and thermal behavior of different forms of obstacles which were placed in the channel into a counter flow. The numerical result was obtained in a laminar flow by CFD software. They demonstrated that obstacles, improve the heat transfer. Indeed, the rectangular shape ensured the highest heat transfer rate with a great turbulence. It increased the Nusselt number and the friction factor due to its sharp corners. Also, they found that increasing the wave form of the corrugated plate can improve the heat transfer. Also, Ruoxu et al. [3] analyzed a counter flow parallel plate heat exchanger using the CFD software Comsol. They simulated a laminar flow with low Reynolds number in smooth ducts. A numerical and theoretical investigation of a counter flow parallel flow and multilayered plate heat exchangers were performed by Vera and Linan [4]. They developed analytical approximations and expressions for a laminar flow for the fully developed in a long PHE. Besides, Hossain and Islam [5] studied the thermal behavior of an internal cooling in different channel shapes. They showed the flow characteristics in 3 wavy channels (triangular, sine shaped, and arc shaped). In addition, the channel geometry influence on the counter flow thermal and hydraulic performance was carried out by Hassan and et al. [6]. A numerical simulation of different channel shapes such as trapezoidal, circular, rectangular, square and iso-triangular was conducted. They found that the increase of the channel number leads to the increase of pressure drop and the heat transfer. Also, the decrease of the channel volume can enhance the heat transfer. Their research showed that the circular shape has the best thermal performance. In addition, Zhan et al. [7] investigated numerically the effect of the channel geometrical factors on the thermal effectiveness and the counter flow system capacity. They

validated their model experimentally. A new counter flow heat exchanger was developed by Kragh et al. [8]. They analyzed experimentally and theoretically its thermal efficiency. Moreover, Kim and Kim [9] developed a numerical procedure to find an optimal channel design with angled ribs. Four non-dimensional variables were chosen to optimize the internal cooling heat transfer. The numerical results were validated with experimental studies. Kim and Lee [10] studied an ideal conception of an obstacle placed in a cross flow with V-shaped ribs. Also, Mckillops and Dunkley [11] developed a correlation between the Nusselt number and the Reynolds number. They showed the effect of the channel geometry on the heat transfer coefficient. In order to the best configuration, Yilmaz et al. investigated a micro channel heat exchanger. Besides, Chamanroy et al. [12] investigated numerically the heat transfer characteristics and the pressure drop for a laminar flow in the wavy miniature heat sink (WMHS) and the straight miniature heat sink (SMHS) fitted with straight and wavy pin-fins. For a Reynolds number range Re 100–1000, the thermal performances are enhanced compared to the smooth cases. Due to the presence of wavy pin-fins, the heat transfer rate increases from 0.05 to 2.5 times and the pressure drop intensifies from 2.6 to 13.6 times compared to the smooth cases. They found that the WMHS equipped with wavy pin-fins and the SMHS fitted with straight pin-fins are the best configurations which provide the maximum the heat transfer and the minimum pressure loss. They validated experimentally their results. In addition, Hosseinirad et al. [13] studied numerically the presence of the Straight splitter and the Arched splitter Wavy splitter effect on the thermal hydraulic performance of the plate-pin-fin heat sinks (PPFHSs). They analyzed and investigated the impact of the arrangement and the shapes (square, rectangular) of the splitters on the thermal behavior with a Reynolds number Re range 50–250. They validated their numerical simulations with experimental. They discovered that with the arched and forward arrangement, the heat transfer rate is maximized. Indeed, they provide a thermal performance index enhancement with 36%. The effect of the pin-longitudinal-pitch and length on the thermal and hydraulic performance of the pin-fin heat sin was carried out by Rezaee et al. [14]. Different configurations such as PL-LH (variable-pin-length-with-low-to-high-arrangement), PL-HL (variable-pin-length-with-high-to-low arrangement), LP-LH (variable-longitudinal-pitch-with-low-to-high-arrangement) and LP-HL (variable-longitudinal-pitch-with-high-to-low-arrangement) were investigated numerically and experimentally. Compared to the smooth cases, the Nusselt number Nu and skin friction factor C_f enhanced. They found that the PL-HL and LP-HL are the best configurations which provide the optimum hydro-thermal performance.

The heat transfer enhancement of the PHE is based on increasing the fluid turbulence and the heat transfer area. The obstacles introduction in the channels creates the turbulence in the flow. Therefore, the researchers searched for an optimal design of PHE which ensures a high heat transfer rate (HTR). In this work, the impact of the geometrical parameters (shapes and rib pitch p) of the Y and C shape obstacles on the thermal behavior and the hydrodynamic performance of the PHE is studied numerically in terms of CFD simulation.

This paper covers a computational fluid dynamics examination of small channels PHE. It includes three significant steps: the pretreatment, the problem resolution and the post treatment. In the first step, the computational domain was defined and developed. Then, the mesh was conducted based on a refinement grid investigation. After that, the boundary conditions and the initial conditions were determined and defined. Also, the solid and fluid properties were specified. In the second level, based on the solution methods and convergence criteria, the differential equations are transformed into linear algebraic equations. At last, the post treatment is the evaluation and the analysis of the numerical results. This work displays the flow characteristics (the effect of the Reynolds number Re and the impact of the rib pitch p) and the performance characteristics in terms of Nusselt number Nu , Friction factor Cf and Thermo-Hydraulic Performance Parameter THPP.

2 CFD analysis of a counter flow plate heat exchanger

Three-dimensional CFD analysis of a counter flow through PHE fitted with Y and C shaped obstacles is conducted with Ansys Fluent. For this evaluation, the counter flow works under steady state conditions. The flow is considered as incompressible, single phase. It is thermally fully developed. The thermal and the physical properties of the working fluid and the plate material are maintained as constant. Besides, in the channel there are no slip boundary conditions. The channels are separated by insulated walls [3]. Ansys includes different tools such as the geometry construction, the mesh creation, the simulation (Fluent) and the post-processing. It is a volume modeler.

2.1 The computational model

For this examination, the computational domain is maintained the same of the previous works [2, 3]. The conjugate model studied is a counter flow parallel PHE. It was created by Solidworks 2017, CAD software. The computational domain consists of two square channels (hot and cold) separated by a plate which have the different forms of obstacles. Each channel dimensions are the same of the previous works [2, 3]. The internal cross-section of the duct is $20 \times 20\text{mm}^2$ where the obstacles are placed, as indicates the Fig. 1. All geometrical parameters of the computational channels are indicated in the Table 1 [2, 3]. The obstacles placed on the duct are assumed with constant rib height e and a variable rib pitch p , Fig. 2.

Fig. 1 Computational domains
a: without obstacles b: with obstacles

Table 1 Geometrical parameters of the computational channels

Parameter	Designation	Value (mm)
Channel width	W	20
Channel length	L	200
Channel height	H	20
Channel thickness	t	2

The height e of the Y or C ribs is maintained at 5 mm [2]. The relative roughness height is maintained constant at $e/D = 0.25$. Between two adjacent Y or C ribs, the pitch p is chosen at 10 mm, 15 mm, 20 mm, 25 mm. Then, the relative roughness pitch is kept variable with range of $2 < p/e < 5$. The hydraulic diameter of the counter flow is $D = 20$ mm. Both of the hot fluid and the cold fluid flow with different temperature and velocity. Different configurations of the Y roughness employed in the CFD simulation and examination are shown in Fig. 2. Their parameters are indicated in the Table 2.

2.2 Grid generation

The most importing step in the CFD examination is the grid generation. It divides the fluid domain into sub domains called elements (Tetrahedron, Hexahedron...). In order to ensure the simulation accuracy, the meshing should be with a high quality. Since, it influences the rate convergence and the computational time.

An-uniform meshing is employed. Grid independence is carried aiming to investigate the flow characteristics and the heat transfer. Six different grids are performed with different meshing elements number as indicated in the Table 3. The meshing sensibility investigation shows that the meshing bigger than 544,768 elements provides the same average Nusselt number Nu as indicates the Fig. 3. The Tetrahedral [2], Hexahedral [15] meshing were chosen for the previous analysis. While, in our case, we choose to work with prismatic meshing with 544,768 elements as shown the Fig. 4. This selected meshing grid provides accurate results with the minimum calculation time.

Fig. 2 Y and C configurations with different rib pitch p : Y shaped ribs b: C shaped ribs

Table 2 Geometrical parameters of different configurations used in the small channels PHE

Configurations	Hydraulic parameter D (mm)	Rib height e (mm)	Relative roughness e/D	Rib pitch p (mm)	Relative roughness pitch p/e
Smooth	20	5	0.25	10	2
Y510				10	2
Y515				15	3
Y520				20	4
Y525				25	5
C510	20	5	0.25	10	2
C515				15	3
C520				20	4
C525				25	5

Table 3 Different meshing grids independence for the CFD simulations

Gridnumber	1	2	3	4	5	6
Elements	10,494	68,096	370,105	544,768	555,848	756,200
Nu average	4.58	6.21	9.4	10.15	10.15	10.15

Fig. 3 Grid independence results for the average Nusselt number Nu

2.3 Governing equations

In order to solve the steady state problem for a fully developed flow, the fundamental physical principals are resolved. For a three-dimensional analysis, the governing equations can be written in a Cartesian tensor as,

Continuity equation:

$$\frac{\partial u_j}{\partial x} + \frac{\partial v_j}{\partial y} + \frac{\partial w_j}{\partial z} = 0$$

Momentum equation:

$$(1) \quad u_j \frac{\partial T_j}{\partial x} + v_j \frac{\partial T_j}{\partial y} + w_j \frac{\partial T_j}{\partial z} = \frac{k_j}{\rho_j c_{p_j}} \left(\frac{\partial^2 T_j}{\partial x^2} + \frac{\partial^2 T_j}{\partial y^2} + \frac{\partial^2 T_j}{\partial z^2} \right) \quad (7)$$

$$u_j \frac{\partial u_j}{\partial x} + v_j \frac{\partial u_j}{\partial y} + w_j \frac{\partial u_j}{\partial z} = \frac{\mu}{\rho} \left(\frac{\partial^2 u_j}{\partial x^2} + \frac{\partial^2 u_j}{\partial y^2} + \frac{\partial^2 u_j}{\partial z^2} \right) - \frac{1}{\rho} \frac{\partial p_j}{\partial x} \quad (2)$$

$$u_j \frac{\partial v_j}{\partial x} + v_j \frac{\partial v_j}{\partial y} + w_j \frac{\partial v_j}{\partial z} = \frac{\mu}{\rho} \left(\frac{\partial^2 v_j}{\partial x^2} + \frac{\partial^2 v_j}{\partial y^2} + \frac{\partial^2 v_j}{\partial z^2} \right) - \frac{1}{\rho} \frac{\partial p_j}{\partial y} + g \quad (3)$$

$$u_j \frac{\partial w_j}{\partial x} + v_j \frac{\partial w_j}{\partial y} + w_j \frac{\partial w_j}{\partial z} = \frac{\mu}{\rho} \left(\frac{\partial^2 w_j}{\partial x^2} + \frac{\partial^2 w_j}{\partial y^2} + \frac{\partial^2 w_j}{\partial z^2} \right) - \frac{1}{\rho} \frac{\partial p_j}{\partial z} \quad (4)$$

Energy equation [16–18]: The conservation equation of energy is defined in a steady heat flow [16],

$$k_s \nabla^2 T + q = 0 \quad (5)$$

At the interface Solid–fluid, the heat flux is defined with the Newton law of heating and cooling.

$$-k \frac{\partial T}{\partial y}|_{y=wall} = h(T_{wall} - T_m) \quad (6)$$

Afzal et al. [16] assumed in their evaluation that at the interface Solid–Fluid bodies, the temperatures are the same. Besides, they supposed that heat flux is continued. Where T_w is the wall temperature and T_m is the fluid domain mean temperature. Hence, the energy equation for the fluid domain is determined as,

Fig. 4 Uniform grid for the computational domain of the small channels PHE

While for the solid wall, the energy equation can be defined as,

$$\frac{\partial^2 T_s}{\partial x^2} + \frac{\partial^2 T_s}{\partial y^2} + \frac{\partial^2 T_s}{\partial z^2} = 0 \quad (8)$$

$j=c$ for cold channel and $j=h$ for hot channel. Where ρ is density, g is gravitational acceleration, μ is dynamic viscosity, C_p is the heat capacity, T is the temperature, p is pressure u, v, w are respectively the velocity components in x, y, z directions and q is the heat generation. The flow velocity depends on the Reynolds number Re which indicated,

$$Re = \frac{\rho \times V \times D}{\mu} \quad (9)$$

Between the following water stream in the square duct and the separated plate, the average value of Nusselt Number Nu is calculated by using the following equation:

$$Nu = \frac{h \times D}{k} \quad (10)$$

$$Q = h \times (T_w - T_{bulk}) \quad (11)$$

$$\rightarrow Nu = \frac{Q \times D}{k \times (T_w - T_{bulk})} \quad (12)$$

It is non-dimensional parameter which is a ratio between the convective transfer and the conductive transfer [25]. It characterizes the flow heat transfer intensity. Where Q is the total surface heat flux, T_w is the temperature of wall and T_{bulk} is the average temperature cross the section of the laminar flow, h is the convective coefficient of heat transfer (W/m^2K), k is the thermal conductivity and D_h

is the hydraulic diameter. It is a ratio between the cross-sectional area A and the wetted perimeter of the cross section P . It can be written as the following equation,

$$D_h = \frac{4A}{P} \quad (13)$$

The second important parameter is the skin friction factor C_f which investigates the flow characteristics. It is written as,

$$C_f = \frac{2 \times \Delta P \times D}{\rho \times L \times V^2} \quad (14)$$

To evaluate the heat transfer enhancement of shaped ribs in small channels PHE, Kamali and Binesh [19], Kumar and Saini [20], Webb and Eckert [21], Promvong and Thianpong [22], defined the thermal enhancement factor TEF or the thermo-hydraulic performance parameter $THPP$. It is a comparison between the results obtained with shaped ribs and with the smooth duct. It is computed as,

$$THPP = \frac{\frac{Nu}{Nu_s}}{\left(\frac{C_f}{C_{f_s}}\right)^{\frac{1}{3}}} \quad (15)$$

2.4 Boundary conditions

The computational domain of a counter flow PHE is a square duct with inlet, outlet, symmetry and walls as indicates the Fig. 5. It can be assumed as a conjugate model. In the upper channel, the cold fluid enters with temperature of 300 K and in the lower channel, the hot fluid enters with temperature of 330 K [2]. The separated plate has obstacles

Fig. 5 Boundary conditions of the small channels PHE

Table 4 Mathematical formulations of the applied boundary conditions

Lower channel		Hot side fluid		
Position 1	$x=0$	$y=0$	$z=0$	
Boundary conditions	$\frac{\partial T_h}{\partial x} = 0$	$\frac{\partial T_h}{\partial y} = 0$	$T_h = T_{hi}$	
Position 2	$x=W_{ch}$	$y=H_h$ solid–fluid interface	$z=L$	
Boundary conditions	$\frac{\partial T_h}{\partial x} = 0$	$-k_h \frac{\partial T_h}{\partial y} = -k_s \frac{\partial T_s}{\partial y}$	$\frac{\partial T_h}{\partial z} = 0$	
Upper channel		Cold side fluid		
Position 1	$x=0$	$y=H_h+t$ solid–fluid interface	$z=0$	
Boundary conditions	$\frac{\partial T_c}{\partial x} = 0$	$-k_c \frac{\partial T_c}{\partial y} = -k_s \frac{\partial T_s}{\partial y}$	$\frac{\partial T_c}{\partial z} = 0$	
Position 2	$x=W_{ch}$	$y=H_h+t+H_c$	$z=L$	
Boundary conditions	$\frac{\partial T_c}{\partial x} = 0$	$\frac{\partial T_c}{\partial y} = 0$	$T_c = T_{ci}$	
		Solid wall boundary condition		
Position 1	$x=0$	$y=H_h$	$z=0$	
Boundary conditions	$\frac{\partial T_s}{\partial x} = 0$	$-k_h \frac{\partial T_h}{\partial y} = -k_s \frac{\partial T_s}{\partial y}$	$\frac{\partial T_s}{\partial z} = 0$	
Position 2	$x=W_{ch}$	$y=H_h+t$	$z=L$	
Boundary conditions	$\frac{\partial T_s}{\partial x} = 0$	$-k_c \frac{\partial T_c}{\partial y} = -k_s \frac{\partial T_s}{\partial y}$	$\frac{\partial T_s}{\partial z} = 0$	

which are located in the both hot and cold channels. The upper and the lower walls are assumed symmetries. The others walls are maintained isolated.

The water is considered as working fluid for both hot and cold channels [2]. The fluid enters with a constant velocity which depends on the Reynolds number Re . This numerical simulation was conducted with a Reynolds number Re range of 200–800. The no slip boundary condition is assumed for all solid surfaces constituting the channels. ($u_c = v_c = w_c = 0$) At the inlet of the hot and the cold channels, the velocities are imposed ($w_c = w_{ci}$) or ($w_h = w_{hi}$). Furthermore, to model the flow exists, the pressure outlet boundary in Ansys Fluent are used. The thermal boundary conditions for heat transfer

Table 5 Thermo-physical properties of the hot and the cold fluids

Properties	Fluids	Steel
Density ρ (kg/m ³)	998.2	2719
Specific heat C_p (J/kgK)	4182	871
Thermal conductivity k (W/mK)	0.6	202.4
Viscosity μ (kg/ms)	0.001003	

calculations are resumed in the table below (Table 4) [16, 17]. The thermal properties of the working fluids are mentioned in the Table 5.

2.5 Solution method

In order to investigate the heat transfer and the fluid flow, the computational fluid dynamics CFD code is used [23]. It is based on the finite volume method FVM [24]. It allows to convert the governing equations to the algebraic equations. Therefore, Ansys Fluent, CFD software, is chosen to solve the Navier Stokes equations. Then, it allows visualizing the velocity, the pressure and the temperature distribution. The simple algorithm is employed in order to solve the conservation of mass, momentum and energy [21]. The convergence solutions for the residuals of mass, energy, and momentum conservation equations are selected as constant and equals, respectively to 10^{-3} , 10^{-6} , 10^{-6} [2].

Afzal et al. [18] announced that the obtaining of significant and accuracy results needs an important computational time, millions of mesh elements. They confirmed that the increase in the computational domain size produces an increment in the simulation time. Then, it is necessary to employ a CPU with powerful performance. The evaluation of these small channels PHE (conjugate model) with the CFD code requires high informatics performance (computer capacity and memory). Due to the computing technology progress, the simulation was conducted using a computing station with 20Go RAM and Intel i7 4770 processor.

3 Results and discussions

3.1 Validation with a simple smooth channel

A smooth duct with the same dimensions of the channel was considered as a validation model. It is a laminar flow over a smooth duct with a low Reynolds number Re . The air as working fluid enters with a constant velocity of 0.146 m/s and a temperature of 250 K. The walls of the channel are maintained isothermal with a temperature of 400 K. The accuracy of the simulation is presented in terms of the velocity and the Nusselt Number profiles for a fully developed flow. Handbook of heat transfer [25] indicates that for a laminar flow in square duct, with isothermal walls, the Nusselt number Nu for fully developed is assumed a constant equal to 2.98.

Laminar velocity profile The accuracy of the numerical method is ensured by the velocity field flow evolution in the smooth duct with a laminar flow. The velocity profiles at the midpoint of the smooth channel and at the inlet are mentioned in the Fig. 6. Indeed, at the inlet of the channel, the velocity is constant and equals to 0.146 m/s ($Re = 200$). At the midpoint of the channel, after the entrance zone, the velocity profile has the parabolic aspect. The flow is hydro dynamically fully developed for $Re < 2300$. The fully

Fig. 6 Velocity profiles at the midpoint of the smooth channel and at the inlet for a simple smooth channel

Fig. 7 Nusselt number Nu profile for simple smooth channel

developed begins at 200 mm. That's why the channel dimensions are $20 \times 20 \times 200 \text{ mm}^3$.

Nusselt number Nu and the friction factor f profiles

At the inlet, in the entrance zone, the Nusselt number Nu is higher as shown the Fig. 7. Then, in the fully developed zone, it established an asymptotic value equal to 2.95.

3.2 Validation with Ruoxu et al. [3] work

It is a comparison between the present work and the previous numerical study of Ruoxu jia et al. [3]. In this case, it is a CFD examination of a laminar counter flow through Smooth Square two ducts with length of 200 mm. The computational domain and the boundary conditions are maintained the same as Ruoxu et al. [3]. The hot fluid

enters with a temperature of 330 K and the cold fluid enters with 300 K. The temperature profiles in the centerline according to the duct length are indicated in the Fig. 8a. The Fig. 8b represents the temperature evolutions in both hot and cold domains according to Y.

While, the Fig. 9a and b describe the temperature and the velocity distributions according to the various planes parallel to the inlet and the outlet. There is a good agreement between the present work and the previous numerical work in terms of the temperature and velocity contours. Therefore, the numerical model is more reliable.

3.3 Validation with Piroozfam et al. [2] work

To ensure the accuracy of the CFD results, we tried to validate the previous analysis work of Piroozfam [2]. The previous numerical work was validated by the previous experimental work of Wie [26]. A laminar counter flow with rectangular obstacles placed on the separated plate was numerically simulated. The same dimensions and the same boundary conditions as Piroozfam [2] were performed. The Fig. 10 confirmed the validity of the CFD tool analysis. There is a good agreement between the previous work [2] and our CFD present simulation. The maximal deviation

between the previous numerical work and the present CFD work differences is 1.46%.

3.4 Flow characteristics

In this part, CFD analysis are performed in order to evaluate the pressure drop and the heat transfer through small channels PHE fitted with Y and C shaped obstacles. The numerical method is validated by the previous numerical results of Piroozfam et al. [2]. In this evaluation, a cold fluid with a temperature of 300 K, a hot fluid with 330 K and a Reynolds number range of 200–800 are employed for counter laminar and single phase flow.

The Figs. 11 and 12 present the temperature and the velocity distribution in the hot and the cold sides for respectively Y and C shaped obstacles (rib pitch P of 15 mm) with a Reynolds number Re of 200 at different planes parallel to the plane (x – z). For both Y and C shaped obstacles, the heat transfer area is increased and the heat transfer is enhanced. Besides, as shown in the velocity contours, the thermal boundary layer is interrupted.

Following the expected path, the fluid enters in the channels. The trajectory of the fluid is parallel to the walls. As, it encounters the obstacles Y or C, the fluid diverges

Fig. 8 Temperature profiles a: in the centerline according to the duct length b: in both hot and cold domain according to Y

Fig. 9 Temperature and velocity distributions according to the various planes parallel to the inlet and the outlet a: temperature distribution b: velocity distribution

Fig. 10 Numerical results for rectangular obstacles using the CFD code (Fluent)

and the fluid velocity increases. Around the obstacles, the fluid flow turbulence intensity is augmented. At the walls of the channels, the velocity is zero. Away from the walls, it increases progressively. At the level of the obstacles, the thermal boundary layer is broken and destabilized [27, 28]. Due to the Y and C obstacles, the turbulence in the flow is created. Besides, the increase of heat transfer area leads to the increase of the heat transfer rate. These obstacles play the role of the generator of vortices.

3.5 Effect of Reynolds number Re

The presence of the obstacles in the duct creates turbulence in the fluid flow. Therefore, the heat transfer is enhanced. These obstacles have the role of generators of vortices. Indeed, around them, the flow is separated and the laminar sub layer is broken. The local mixing of the flow is immersed. Then, the disturbances are created in the flow. In fact, the flow pressure cross the duct is increased. Besides, the fluid flow resistance is intensified also. Therefore, the pumping power is augmented. The only goal is

Fig. 11 Temperature and Velocity distributions in the cold and the hot sides at different planes of Y shaped ribs Y515 (rib pitch P of 15 mm) with Reynolds number Re of 200

to obtain the maximum heat transfer and the minimum flow resistance.

Figures 13, 14, 15 and 16 represent the temperature and the velocity distribution in the mid plane parallel to the plane (y - z) respectively for the Y and C shapes obstacles with a Reynolds number Re range of 200–800. Indeed, with a constant rib pitch P of 10 mm, the increase of the Reynolds number Re leads to the increment of the fluid velocity cross the flow. Therefore, the heat transfer enhancement is increased. Moreover, the laminar sublayer thickness is decreased by the augmentation of the Reynolds number Re . In fact, around the obstacles, the vortices are created. Then, the increase of the velocity provides the intensification of the flow turbulence. At the regions next to the Y and C shapes obstacles, the temperature variation is maximum as indicated in the temperature contours. As can be

seen, with the Y shape obstacles, the laminar sub layer is more broken than the flow with the C shape obstacles.

3.6 Effect of pitch between two successive ribs:

The effect of the pitch between two successive ribs on the velocity distribution is presented in two different mid planes as show the Figs. 17 and 18. With a constant Reynolds number Re of 600, various configurations are numerically investigated with rib pitch range of [10, 15, 20, 25] mm. The Y and C shape obstacles fitted in the mid plate of the small channel PHE influence the fluid motion. Then, these affect the thermal displacement phenomenon. This variation affects strongly the velocity distribution for the Y shape obstacles more than the C shape obstacles.

Fig. 12 Temperature and Velocity distributions in the cold and the hot sides at different planes with C shaped ribs C515 (rib pitch P of 15 mm) with Reynolds number Re of 200

The increase of the rib pitch from 15 to 25 mm with a constant Reynolds number Re of 600, leads to the decrease of velocity cross the duct. Indeed, the intensification of the pitch from 10 to 15 mm causes the increase of the flow velocity. The flow becomes detached when the distance between two successive ribs increased. Then, it attached to the walls again. Near the Y obstacles, the vortices are forms. Therefore, the flow turbulence increased and the average heat transfer coefficient

enhanced. The shapes of the vortexes have a direct impact on the heat transfer improvement.

The presence of the shaped obstacles in the small channels as generators vortexes, affects the flow path. Near the obstacles, the hydrodynamic and thermal boundary layer is cut and again reformed. Then, the heat transfer increased and the thermal resistance reduced.

In the plane median parallel to plane $(x-y)$, the velocity contours (Figs. 17, 18) show that the thermal boundary

Fig. 13 Temperature distributions in the mid plane parallel to the plane (y–z) with Y shapes obstacles for Reynolds number range of 200–800 and a constant rib pitch P of 10 mm (configuration Y510)

layer is interrupted around the obstacles Y and C. The presence of Y and C ribs as obstacles on the separated plate will generate turbulence in the flow. Indeed, the flow is dispersed around the obstacles. The hydrodynamic and thermal boundary layers are distributed. It has been cut and reformed. Then, the heat transfer rate is improved. Therefore the vortices generators change the flow structure and primary profile [27–29]. At the level of the obstacles, the flow becomes interrupted and tortuous. Then, the flow shear stress is enhanced. Indeed, the raise of the Reynolds

number Re causes the augmentation of velocity and the diminish of the temperature distribution progression in both hot and cold channels. The heat transfer enhancement of the PHE is based on the passive techniques as, the introduction of the geometrical disruptive in the flow. This last works as generator vortex. Fiebig et al. [30–32] affirmed that the generator vortices disrupt the flow and modify the flow structure. Therefore the flow turbulence intensity increased and the heat transfer rate grows. In both the hot and the cold channels, the fluid distribution

Fig. 13 (continued)

over the separated plate is non-uniform. In fact, this last affects the heat transfer rate and the thermal performance in the PHE. The Fig. 19 indicates that the presence of the obstacles in the channels enhanced the heat transfer between the cold and the hot fluids. Indeed, on the side of the hot small channel, the Y shape obstacles lead to a significant decrease in the temperature.

3.6.1 The non-dimensional parameters

3.6.1.1 Nusselt number Nu The average Nusselt number Nu for different configurations Y and C ribs are indicated in Figs. 20 and 21. They describe the Nusselt number Nu evolution for various configurations. The parameter Nu can be calculated by the expression.

Fig. 14 Temperature distributions in the mid plane parallel to the plane (y–z) with C shapes obstacles for Reynolds number range of 200–800 and a constant rib pitch P of 10 mm (configuration C510)

$$Nu = \frac{hD}{k} \quad (16)$$

where, D is the Hydraulic diameter, k is the fluid thermal conductivity and h is the convective heat transfer coefficient. For a constant rib pitch, the increase of the Reynolds number Re leads to the augmentation of the Nusselt number Nu due to the intensification of the velocity. The counter flow through small plate heat exchanger with a pitch of 10 mm for Y ribs and a pitch of 25 mm for C ribs,

have the maximum value of the non-dimensional parameter Nusselt number Nu with a Reynolds number Re range of 200–800. The maximum value of the factor Nu of Y ribs and C ribs are found respectively about 16.43 and 15.7 with a Reynolds number Re of 800.

3.6.1.2 Skin friction factor C_f The Fig. 22 and 23 show the average skin friction factor C_f evolution according to Reynolds number range of 200–800, with Y and C ribs. It is calculated by the following expression,

Fig. 14 (continued)

$$C_f = \frac{2 \times \Delta P \times D}{L \times \rho \times V^2} \quad (17)$$

where, ΔP is the pressure drop, L is the length of the channel, ρ is the fluid density and V is the mean flow velocity.

The presence of the Y and C shapes obstacles in the small channel, not only enhance the heat transfer but also, it causes an increase in the pressure drop in the small channels. it affects the flow path. In fact,

the increase of the Reynolds number Re leads to the decrease of the skin friction factor C_f . For a constant Reynolds number Re , the increase of the rib pitch causes the decrease of average friction factor C_f . Comparing to smooth duct, the minimum value of average friction factor C_f is obtained with Reynolds number Re of 800 and a pitch of 25 mm for the Y ribs and 20 mm for C ribs as indicated in Figs. 22 and 23.

Fig. 15 Velocity distributions in the mid plane parallel to the plane (y–z) with Y shapes obstacles for Reynolds number range of 200–800 and a constant rib pitch P of 10 mm (configuration Y510)

3.6.1.3 Thermo-hydraulic performance parameter THPP In order to discover the overall heat exchanger performance, the parameter thermo-hydraulic performance $THPP$ is employed. It is expressed as following [2, 19–21]

$$THPP = \frac{Nu}{Nu_s} \left(\frac{Cf}{Cf_s} \right)^{\frac{1}{3}} \quad (18)$$

The Figs. 24 and 25 present the impact of the Reynolds number Re on the thermo-hydraulic performance parameter $THPP$. They describe the $THPP$ evolution for both Y

Fig. 15 (continued)

and C ribs with rib pitch range of 10–25 mm and a Reynolds number Re range 200–800. Indeed, the index $THPP$ increases with the raise of Reynolds number Re . Beyond a critical Reynolds Number Re_c , the factor $THPP$ decreases.

For Y rib configuration, the maximum value of the thermo-hydraulic performance parameter $THPP$ is about 1.44 and achieved with the Reynolds number Re of 800

and the rib pitch of 10 mm. For the C rib configuration, the thermal efficiency factor $THPP$ is maximized with Reynolds number Re of 600 and a rib pitch of 20 mm. It is about 2.6 times the smooth duct.

The Fig. 26, presents a comparison of the thermal performance between the Y ribs and the C ribs with the same rib pitch, and geometrical parameters. The

Fig. 16 Velocity distributions in the mid plane parallel to the plane (y–z) with C shapes obstacles for Reynolds number range of 200–800 and a constant rib pitch P of 10 mm (configuration Y510)

thermo-hydraulic performance parameter THPP is maximized with Reynolds number of 800 for the Y ribs and a Reynolds number Re of 600 for the C ribs. With Reynolds number inferior to 400, the C ribs configuration has the highest thermal efficiency. While, beyond the Reynolds number Re of 400, the Y ribs has the best overall thermal performance.

The Fig. 27, describes the effect of the Reynolds number Re on the thermo-hydraulic performance parameter THPP. It presents a comparison between the THPP evolution for both Y ribs and C ribs and the previous works [2] for a Reynolds number Re range 200–800.

The maximum value of the thermo-hydraulic performance parameter THPP for the C rib with a rib pitch of 10 mm is 1.399 with a Reynolds number Re 400. In fact, the thermo-hydraulic performance parameter THPP increase until a Reynolds number 400 then, it decreases. While for the Y ribs configuration, the maximum value of the thermo-hydraulic performance parameter THPP is 1.44 with a Reynolds number Re of 800.

Indeed, the increase of Reynolds number Re leads to the decrease of the thermo-hydraulic performance parameter THPP for Reynolds number superior to 400. The presence of the Y and C shaped ribs in the counter flow leads to more thermo-hydraulic performance

Fig. 16 (continued)

parameter THPP than the previous works (rectangular which is the best configuration in their analysis). Hence, the heat transfer is enhanced, and the thermal performance of the small channel PHE is improved. Therefore, the new proposed designs Y and C ribs provide the highest performance.

4 Conclusion

The presence of the obstacles in a laminar counter flow through small channels enhances the thermal and the hydraulic performance of a plate heat exchanger. The influence of the shape obstacles, the rib pitch on the thermal efficiency and the heat transfer enhancement is studied numerically with a Reynolds number range

Fig. 17 Velocity distributions with the Y shapes obstacles for pitch P range of [10 15 20 25] mm and Reynolds number Re of 600 in the mid plane parallel to the plane (y - z)

200–800. The Y and C shaped obstacles which are placed on the separated plate, cause modification on the flow direction and structure. Indeed, the profile of the flow is perturbed. Moreover, the thermal boundary layer is interrupted. Therefore, the heat transfer is enhanced. The CFD evaluation and examination can be summarized,

- The average Nusselt number Nu increases with the increase of Reynolds number Re with a constant rib pitch P . While, the average skin friction factor Cf decreases with the increment of the Reynolds number Re .
- Up to the critical Reynolds number Re , the thermo-hydraulic performance parameter THPP intensifies

Fig. 17 (continued)

with the augmentation of the Reynolds number Re . Beyond the critical Reynolds number Re , the factor THPP decreases. The maximum value of the thermo-hydraulic performance parameter THPP for the Y ribs configuration is about 1.44 times of smooth duct with Reynolds number Re of 800 and a rib pitch of 10 mm.

For the C rib channel, with Reynolds number Re of 600 and a rib pitch of 20 mm the thermo-hydraulic performance parameter THPP is maximized with a value of 2.6 times the smooth duct.

- The CFD analysis show that the small channel plate heat exchanger fitted with Y shape obstacles is the optimum

Fig. 18 Velocity distributions with the Y shapes obstacles for pitch range of [10 15 20 25] mm and Reynolds number Re of 600 in the mid plane parallel to the plane (x–y)

configuration. It procures the highest thermo-hydraulic performance by providing the maximum heat transfer rate, and the minimum pressure drop. Indeed, the Y shape obstacles are inspired by the human respiratory system structures [33]. That's why, these new proposed designs Y and C shaped obstacles (Vortex generators VG) enhance the thermal and hydrodynamic efficiency of the PHE.

In the future research, the impact of the geometrical parameters such as the rib height h , the transversal pitch P_t , the longitudinal pitch P_l , the fractal steps number N ... and the working fluid nano-fluid (Al_2O_3 /water, Fe_3O_4 /water...) on the novel PHE thermal efficiency can be studied and evaluated.

Fig. 18 (continued)

Fig. 19 Temperature distributions with the Y and C shapes obstacles for a constant rib pitch of 25 mm and Reynolds number Re of 800 in different planes

Fig. 20 Average Nusselt number Nu with the Y configuration according to Reynolds number Re range of 200–800 for different rib pitch $P=10$ –15–20–25 mm

Fig. 22 Average Skin friction factor Cf with the Y configuration according to Reynolds number Re range of 200–800 for different rib pitch $P=10$ –15–20–25 mm

Fig. 21 Average Nusselt number Nu with the Y configuration according to Reynolds number Re range of 200–800 for different rib pitch $P=10$ –15–20–25 mm

Fig. 23 Average Skin friction factor Cf with the C configuration according to Reynolds number Re range of 200–800 for different rib pitch $P=10$ –15–20–25 mm

Fig. 24 Thermo hydraulic performance parameter $THPP$ with the Y configuration according to Reynolds number Re range of 200–800 for different rib pitch $P = 10\text{--}15\text{--}20\text{--}25$ mm

Fig. 26 Thermo hydraulic performance parameter $THPP$ with the Y and C configurations according to Reynolds number Re range of 200–1000 with a constant rib pitch $P = 10$ mm

Fig. 25 Thermo hydraulic performance parameter $THPP$ with the C configurations according to Reynolds number Re range of 200–800 for different rib pitch $P = 10\text{--}15\text{--}20\text{--}25$ mm

Fig. 27 Thermo-hydraulic performance parameter $THPP$ according to Reynolds number Re range of 200–800 for C and Y ribs configurations

Declaration

Conflict of interest The authors declare that they have no conflict of interest in connection with this paper.

Open Access This article is licensed under a Creative Commons Attribution 4.0 International License, which permits use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons licence, and indicate if changes were made. The images or other third party material in this article are included in the article's Creative Commons licence, unless indicated otherwise in a credit line to the material. If material is not included in the article's Creative Commons licence and your intended use is not permitted by statutory regulation or exceeds the permitted

use, you will need to obtain permission directly from the copyright holder. To view a copy of this licence, visit <http://creativecommons.org/licenses/by/4.0/>.

References

1. Mota FA, Carvalho EP, Ravagnani MA (2015) Modeling and design of plate heat exchanger. In: Heat Transfer Studies and Applications, IntechOpen.
2. Piroozfam N, Shafaghi AH, Razavi SE (2018) Numerical investigation of three methods for improving heat transfer in counter-flow heat exchangers. *Int J Therm Sci* 133(2018):230–239
3. Jia R, Hu J (2014) Analysis of a counter flow parallel-plate heat exchanger

4. Vera M, Liñán A (2010) Laminar counterflow parallel-plate heat exchangers: exact and approximate solutions. *Int J Heat Mass Transf* 53(2010):4885–4898
5. Hossain Z, Islam S (2007) Numerical investigation of fluid flow and heat transfer characteristics in sine, triangular, and arc-shaped channels. *Thermal Sci* 11:17–26. <https://doi.org/10.2298/TSCI0701017H>
6. Hasan MI, Rageb AA, Yaghoubi M, Homayoni H (2009) Influence of channel geometry on the performance of a counter flow microchannel heat exchanger. *Int J Therm Sci* 48:1607–1618
7. Zhan C, Duan Z, Zhao X, Smith S, Jin H, Riffat S (2011) Comparative study of the performance of the M-cycle counter-flow and cross-flow heat exchangers for indirect evaporative cooling—paving the path toward sustainable cooling of buildings. *Energy* 36:6790–6805
8. Kragh J, Rose J, Nielsen TR, Svendsen S (2007) New counter flow heat exchanger designed for ventilation systems in cold climates. *Energy Buildings* 39:1151–1158
9. Design optimization of rib-roughened channel to enhance turbulent heat transfer - ScienceDirect, (n.d.). <https://www.sciencedirect.com/science/article/pii/S0017931004002509> (accessed April 6, 2019).
10. Design Optimization of Internal Cooling Passage with V-shaped Ribs: Numerical Heat Transfer, Part A: Applications: Vol 51, No 11, (n.d.). <https://www.tandfonline.com/doi/abs/https://doi.org/10.1080/10407780601112860>. Accessed April 6, 2019.
11. McKillop AA, Dunkley WL (2002). Heat Transfer. <https://doi.org/10.1021/ie50609a020>
12. Chamanroy Z, Khoshvaght-Aliabadi M (2019) Analysis of straight and wavy miniature heat sinks equipped with straight and wavy pin-fins. *Int J Thermal Sci* 146:106071
13. Hosseiniad E, Khoshvaght-Aliabadi M, Hormozi F (2019) Effects of splitter shape on thermal-hydraulic characteristics of plate-pin-fin heat sink (PPFHS). *Int J Heat Mass Transfer* 143:118586
14. Rezaee M, Khoshvaght-Aliabadi M, Arani AA, Mazloumi SH (2019) Heat transfer intensification in pin-fin heat sink by changing pin-length/longitudinal-pitch. *Chem Eng Process-Process Intensification* 141:107544
15. Mushtaq H (2009) Numerical simulation of counter flow micro-channel heat exchanger with different channel geometries and working fluids. https://www.researchgate.net/publication/325658512_Numerical_simulation_of_counter_flow_micro_channel_heat_exchanger_with_different_channel_geometries_and_working_fluids
16. Afzal A, Samee AM, Razak RA, Ramis MK (2020) Thermal management of modern electric vehicle battery systems (MEVBS). *J Thermal Anal Calorim* 2020:1–15
17. Afzal A, Mohammed Samee AD, Abdul Razak RK, Ramis MK (2019) Effect of spacing on thermal performance characteristics of Li-ion battery cells. *J Therm Anal Calorim* 135(2019):1797–1811. <https://doi.org/10.1007/s10973-018-7664-2>
18. Afzal A, Ansari Z, Faizabadi AR, Ramis MK (2017) Parallelization strategies for computational fluid dynamics software: state of the art review. *Arch Computat Methods Eng* 24(2017):337–363. <https://doi.org/10.1007/s11831-016-9165-4>
19. Kamali R, Binesh AR (2008) The importance of rib shape effects on the local heat transfer and flow friction characteristics of square ducts with ribbed internal surfaces. *Int Commun Heat Mass Transfer* 35(2008):1032–1040
20. Kumar S, Saini RP (2009) CFD based performance analysis of a solar air heater duct provided with artificial roughness. *Renewable Energy* 34(2009):1285–1291
21. Webb RL, Eckert ERG (1972) Application of rough surfaces to heat exchanger design. *Int J Heat Mass Transfer* 15(1972):1647–1658
22. Promvong P, Thianpong C (2008) Thermal performance assessment of turbulent channel flows over different shaped ribs. *Int Commun Heat Mass Transfer* 35(2008):1327–1334
23. ANSYS FLUENT 12.0 Theory Guide. https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/main_pre.htm. Accessed 3 Mar 2021.
24. Patankar S (1980) Numerical heat transfer and fluid flow. CRC Press, Boca Raton
25. Bejan A, Kraus AD (2003) Heat transfer handbook. Wiley, New York
26. Wei X (2004) Stacked microchannel heat sinks for liquid cooling of microelectronics devices. PhD Thesis, Georgia Institute of Technology.
27. Doo JH, Ha MY, Min JK, Stieger R, Rolt A, Son C (2013) An investigation of cross-corrugated heat exchanger primary surfaces for advanced intercooled-cycle aero engines (Part-II: Design optimization of primary surface). *Int J Heat Mass Transfer* 61(2013):138–148
28. Doo JH, Ha MY, Min JK, Stieger R, Rolt A, Son C (2012) An investigation of cross-corrugated heat exchanger primary surfaces for advanced intercooled-cycle aero engines (Part-I: Novel geometry of primary surface). *Int J Heat Mass Transfer* 55(2012):5256–5267
29. Kim H-M, Kim K-Y (2004) Design optimization of rib-roughened channel to enhance turbulent heat transfer. *Int J Heat Mass Transfer* 47(2004):5159–5168
30. Fiebig M (1995) Embedded vortices in internal flow: heat transfer and pressure loss enhancement. *Int J Heat Fluid Flow* 16:376–388
31. Fiebig M, Kallweit P, Mitra NK (1986) Wing type vortex generators for heat transfer enhancement. in: International Heat Transfer Conference Digital Library, Begel House Inc.
32. Fiebig M, Kallweit P, Mitra N, Tiggelbeck S (1991) Heat transfer enhancement and drag by longitudinal vortex generators in channel flow. *Exp Thermal Fluid Sci* 4:103–114
33. Gürel B, Akkaya VR, Göltaş M, Şen ÇN, Güler OV, Koşar Mİ, Keçebaş A (2020) Investigation on flow and heat transfer of compact brazed plate heat exchanger with lung pattern. *Appl Thermal Eng* 175:115309. <https://doi.org/10.1016/j.applthermaleng.2020.115309>

Publisher's Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.