Journal of Informatics and Mathematical Sciences
Vol. 10, No. 4, pp. 673–684, 2018
ISSN 0975-5748 (online); 0974-875X (print)
Published by RGN Publications

http://www.rgnpublications.com

DOI: 10.26713/jims.v10i4.1065

Research Article

Laminar Flow and Heat Transfer inside a Bi-dimensional Planar Corrugated Channel

Rahima Benchabi[1,*] and Ahsene Lanani[2]

[1] *Department of Mechanical Engineering, University of Freres Mentouri, Constantine 1, 25000, Algeria*
[2] *Department of Mathematics University, of Freres Mentouri, Constantine 1, 25000, Algeria*
*__Corresponding author:__ rbenchabi@yahoo.fr

Abstract. The present study investigates the flow and heat transfer characteristics of a bi-dimensional planar corrugated channel. Computational fluid dynamics is used to study the effect of Reynolds number ($2000 = Re = 4000$) and two different values of channel height 5 and 10 mm on the heat transfer and flow developments. The obtained results are validated by comparing with the experimental data and there is a good agreement.

Keywords. Mathematical Model; CFD code; Laminar flow; Numerical Modeling; Correlations

MSC. 35Q30; 65Z05; 80A20; 51P05

Received: August 16, 2018 **Accepted:** November 11, 2018

This article was submitted to "*International Conference on Mathematics*" organized by Fatih Sultan Mehmet Vakıf University, Topkapı (Cevizlibağ) Campus, Istanbul, Turkey during July 3-6, 2018.

Academic Editors: Dr. Kenan Yildirim, Mus Alparslan University, Turkey
Dr. Gumrah Uysal, Karabuk University, Turkey
Dr. Reza Abazari, University of Tabriz, Iran

1. Introduction

In industry, the heat exchanger is an essential element of any energy conservation policy and therefore, by extension, environmental protection. Much of the thermal energy used in industrial processes passes at least once through a heat exchanger, both in the processes themselves or in the recovery thermal energy of these systems. Heat exchangers are mainly used in industrial sectors (chemicals, petrochemicals, food processing, power generation, etc . . .), transport (automotive, aeronautics) and in the residential and tertiary sector (heating, air conditioning, etc . . .). These heat exchangers are therefore very important devices for the thermicien and are an almost inevitable component in mastering technological energy. The major concerns of these exchangers is improving the heat exchange between the two fluids while generating the least possible pressure losses or reduce to the lowest possible level. The present work is conducted in the context to improve heat exchange with reduced pressure losses. Indeed, the resolution of the Navier-Stokes equations allows to simulate the flow of fluids and to model the heat transfer by convection along a corrugated wall. Nowadays, the study of flows along the uneven walls remains unresolved analytically except in simplified cases. Some numerical methods could give acceptable results in specific cases for such flows. Also, several experimental studies with several phenomena modeling strategies have been proposed, especially in the context of turbulent flows; because these fluid flows involved in many physical phenomena encountered in industrials processes. This sake of efficiency in heat exchangers makes it necessary to test many devices (blades, various obstacles, roughness, ripples, etc . . .). In this context, the objective of this work is the numerical study of the flow and heat transfer in channels with corrugated walls and to determine the influence of certain parameters related to the geometry on the thermo-hydraulic behavior. Also, the enhancement heat transfer is the process to improve the thermal performance of heat transfer devices. In general, the design of heat exchangers is needed to increase economies of energy. The use of corrugated plates is an appropriate method to increase the thermal efficiency. The corrugated surfaces are applied as turbulence organizer to increase the heat transfer. Many researchers have studied experimentally and numerically the flow and heat transfer in such corrugated channels.

Sunder and Trollander [12] considered the laminar bi-dimensional flow and heat transfer in a corrugated channel by using finite difference approximations.

A numerical study is conducted by Naphon [9] in a wavy channel with a constant heat flux imposed on the walls from 0.5 to 1.2kW/m^2, with three corrugations angles 20°, 40° and 60° flow rates for a Reynolds number ranging from 400 to 1600. The flow and heat transfer has been simulated by the $k-\varepsilon$ model and the finite volume method was used for the discretization of the equations. From this study, the authors found that the Nusselt number increases with increasing Reynolds number and the corrugation angle.

Fabbri [1] studied numerically the heat transfer in a channel composed of a smooth wall and a corrugated wall for a laminar flow. Velocities and temperature distributions are determined using a finite element method. The author compared the heat transfer performance of the corrugated-walled channel with those of the duct with smooth inner wall. The obtained results show that when no constraint is placed on the wall volume or on the pressure drop, the profile of the corrugated wall only maximize heat transfer at a time when the Reynolds and Prandtl numbers are not too low. Furthermore, the author found that the profile of the corrugated wall provides an increase of nearly 8% compared to the dissipated heat by a smooth channel profile.

In Lanani and Kadja [5], a numerical study was conducted by the authors to describe the flow and heat transfer in a corrugated duct. The results surround the influence of some geometrical and physical parameters namely channel shape and the Reynolds number (Re) in the dynamic and thermal behavior. The authors noted that the Nusselt number and the friction coefficient are influenced by the Reynolds number and the distance P. Also, they concluded that the heat transfer in a corrugated channel is better than the smooth ones.

Metwally and Manglik [7] developed a numerical study of laminar flow in a sinusoidal periodic channel. The study is based on the finite difference method for a range of Reynolds number ($10 \le Re \le 1000$), various viscous liquids ($Pr = 5.35, 150$) and an aspect ratio γ ($0 \le \gamma \le 1$). The authors found that the flow field is strongly influenced by low values of the Reynolds number and by the aspect ratio ($Re = 10$ and $\gamma = 0.25$); in this case the regime is laminar while for high values, the regime becomes turbulent and recirculation zones appear and are increasingly large.

Pehlivan [11] has studied experimentally heat transfer and air flow pressure drop in sinusoidal and triangular corrugated channels. The considered heat flux on the walls is uniform. Also, a range of the Reynolds number varying from 2000 to 9000 is considered. The author studied the effect of the corrugation angle, the channel height and the Reynolds number on the heat transfer and pressure losses. It has been found that increasing the angle and the height increases the heat transfer.

In Zhang and Chen [13], a numerical and experimental study was performed on convective heat transfer in a triangular wave channel with a uniform heat flux. The $k-\omega$ model is used for modeling turbulence in a range of Reynolds number varying from 500 to 5000. The authors have noticed that an increase in the Reynolds number leads to an increase of the Nusselt number. They also established correlations based on the measured results using the hot-wire anemometer, for the Nusselt number and the friction coefficient under uniform heat flux condition.

In the study of Molki [8], the aim is to optimize the hydraulic performance in a corrugated channel by varying the base angle γ of triangle from $0°$ to $30°$, for a range of Reynolds number ranging from 4000 to 30000. The author found that increasing the angle γ tends to increase the friction coefficient and the latter depends on the Reynolds number. However, the dependency decreases for large angles γ. He also concluded that the hydraulic performances are superior to those of smooth channel and noticed that for a very large base angle γ of the triangle and a very high Reynolds number, the hydraulic performances of the smooth corrugated channel are very close.

So as to predict the heat transfer and friction coefficients, Faghri and Asako [2] used the finite element method for a range of Reynolds numbers varying between 100 and 1500.

In Masoud and Khoshkov [6], the authors have made an experiment to optimize heat transfer and find optimum of the heat exchanger dimensions. The results showed that when the number of plates is less than 150, the optimum value of the wavelength for maximum heat transfer zone is between 6 and 10 mm.

A numerical study of flow and heat transfer of the air in both sinusoidal and rectangular channels with a hot temperature imposed on corrugated walls and a range of Reynolds number varying between 100 and 1000 is given in [10]. The authors found that increasing the Reynolds number and the channel height results in an increase of heat transfer in the considered channels.

The effect of two different values of the channel height (5mm and 10mm) with an angle $\alpha = 20°$ on the characteristics of heat transfer and friction coefficient of a corrugated channel, were examined by Islamoglu and Parmaksizoglu [4].

In Gradeck et al. [3], an experimental study was conducted by the authors to study the effects of hydrodynamic conditions on improved heat transfer for a single-phase flow in a corrugated channel. The study is performed for a wide range of Reynolds number ($0 < Re < 7500$) in order to obtain different flow regimes. Also, the measured local temperatures are used to assess the local and overall heat transfer coefficient of the heat corrugated exchanger. The authors concluded that the heat transfer in the corrugated channel is always higher than that of smooth channel.

The aim of our work is the numerical study of two-dimensional flow, incompressible and Newtonian in a corrugated plane channel simulating a plate heat exchanger. The flow characteristics and heat transfer in the channel will be presented as curves. Furthermore, the obtained results in this study are validated with experimental results of Islamoglu [4] and they are in good agreement.

2. Mathematical Formulation

2.1 Introduction

Forced convection in complex geometries is important in many industrial applications, especially in heat exchangers. Considerable works have been done in recent years on the flow and heat transfer in plate heat exchangers.

In this work, we present the physical problem of stationary forced convection within a horizontal channel in the presence of transverse corrugations and the equations governing the flow in the corrugated channel. The mathematical equations on which this study is based and which govern the laminar flow are the continuity equation, the equations of motion and the energy equation.

2.2 Equations governing the flow

The continuity equations; momentum and energy are respectively given by:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j\right) = 0 \qquad (j = 1,2,3). \tag{2.1}$$

$$\frac{\partial\left(\rho u_j u_i\right)}{\partial x_i} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_j}\right)\right] + F_i \qquad (i;\ j = 1;2;3) \tag{2.2}$$

The left-hand term of equation (2.2) represents the transport rate by convection momentum. The first term on the right, the gradient of the pressure; the second transport rate of the amount of movement by diffusion, and the third represents the volume force.

$$\frac{\partial}{\partial x_j}\left(u_j t\right) = a\frac{\partial^2 t}{\partial x_j{}^2} + \varnothing \tag{2.3}$$

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \frac{\mu}{\rho}\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right] \tag{2.4}$$

where $\alpha = \frac{K}{\rho c p}$.

a: thermal diffusivity; K: thermal conductivity; cp: Specific heat at constant pressure; $\varnothing$: the viscous dissipation.

2.3 Mathematical model of the studied problem

The applied mathematical model to a corrugated configuration is written with simplifying assumptions. The air flow is assumed steady and two-dimensional; fluid physical properties are constant. Also, the fluid is incompressible and Newtonian. Given these assumptions, the equations become:

- Continuity equation

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{2.5}$$

- Equation of momentum following x

$$u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\mu}{\rho}\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right] \tag{2.6}$$

- Equation following y

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{\mu}{\rho}\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right] \tag{2.7}$$

2.4 Problem configuration

The geometry of the problem is shown in Figure 1. This is a corrugated plate in a heat exchanger similar to that used by Islamoglu [4]. The duct configuration is characterized by the spacing of the channel height H and the wavelength corrugation S. The channel heights are respectively equal to 5 mm and 10 mm and a corrugation angle equal to 20°. The total length of the channel included 10 corrugations.

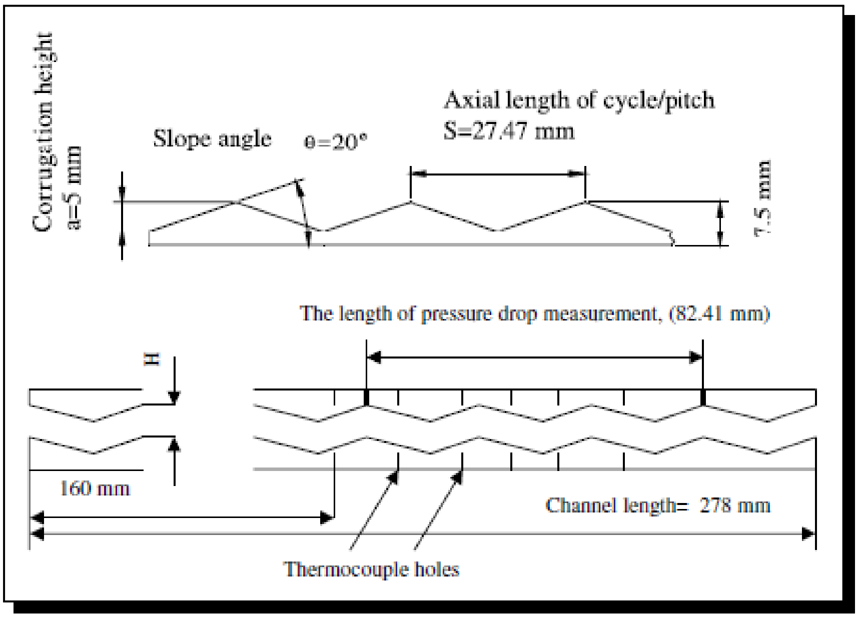


Figure 1. Geometry of the studied channel

2.5 Boundary conditions

Our flow is delimited by corrugated walls and the test fluid is air. The flow velocities on the channel entry are deducted

The first approach to model our problem is the numerical method. The numerical simulation was performed using the computer code "Fluent" which uses the finite volume method. This method has considerable advantages, because it is simple, its reliability for the results, adapting to physical problems, its guarantee for the conservation of mass and momentum. The problem was treated for the resolution of two-dimensional case. In what follows, we describe the construction of the geometry of our study case, the generation of its mesh and the incorporation of boundary conditions as they were developed in the mesh given by the Gambit and Fluent solver. The creation of geometry and meshing is done under Gambit. This mesh offers comprehensive solutions for the Reynolds numbers and a hot fluid temperature is imposed on this entry. The channel walls are cold and have a uniform temperature equal to 293 K.

3. Numerical Modeling

The first approach to model our problem is the numerical method. The numerical simulation was performed using the computer code "Fluent" which uses the finite volume method. This method has considerable advantages, because it is simple, its reliability for the results, adapting to physical problems, its guarantee for the conservation of mass and momentum. The problem was treated for the resolution of two-dimensional case. In what follows, we describe the construction of the geometry of our study case, the generation of its mesh and the incorporation of boundary conditions as they were developed in the mesh given by the Gambit and Fluent solver. The creation of geometry and meshing is done under Gambit. This mesh offers comprehensive solutions for the most complex geometries. For our two-dimensional case, we have initially tried using a mesh with regular wavelength (uniform mesh); then used a finer mesh to the walls (refined mesh). It appeared that a refined mesh gives very satisfactory results and are in good agreement with those of the experience compared to results obtained with a uniform mesh. This type of mesh was so used in our following study. We used the Fluent CFD software; it allows to model fluid flow and heat transfer in complex geometries. This calculation code uses the finite volume method as discretization process. Integral equations that govern the flow, such as the continuity equation, the mass conservation and the energy equation, are resolved by this statistical method.

4. Results and Discussion

4.1 Reynolds number effect

Dynamic field

Figure 2 shows the contours of the average velocities for different Reynolds numbers with $H/S = 0182$, a central flow is seen with high velocities and decrease until it becomes zero near the solid walls and satisfying the non-slip condition. Thick layers dynamic limits are observed along the upper and lower wall where fluid velocities are low (decelerated fluid) and they are becoming thinner immediately upstream of the corners where velocities are important. We also note that the thickening rate of the boundary layer is increasingly important that the Reynolds number increases.

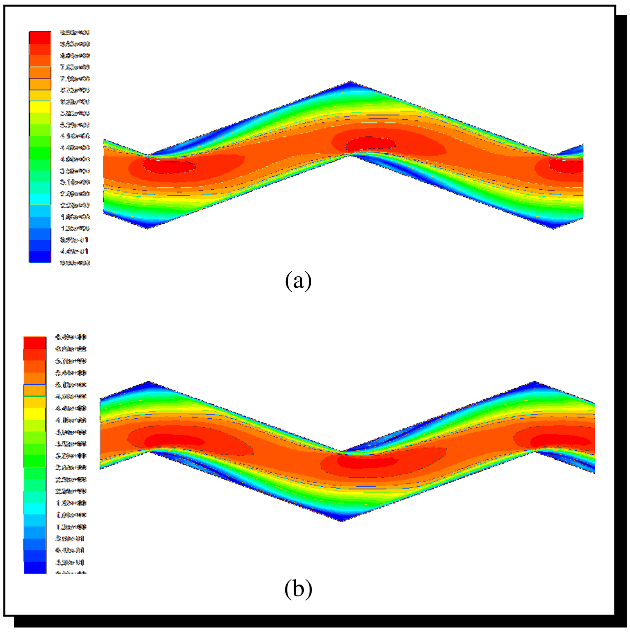


Figure 2. Contours of the average velocities with $H/S = 0.182$ for $H = 5$ mm

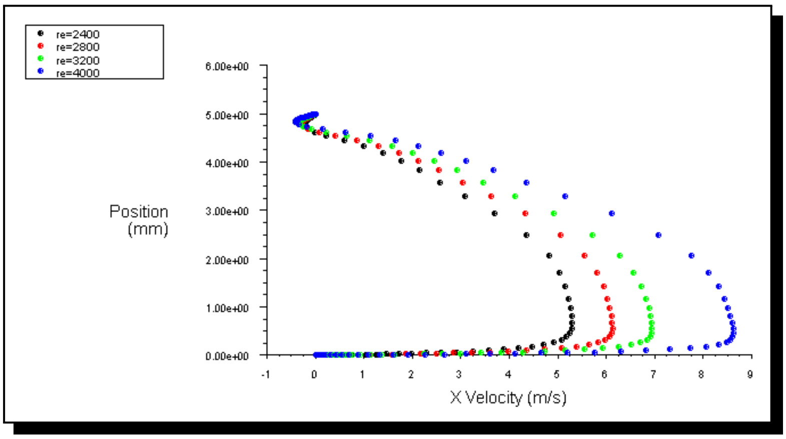
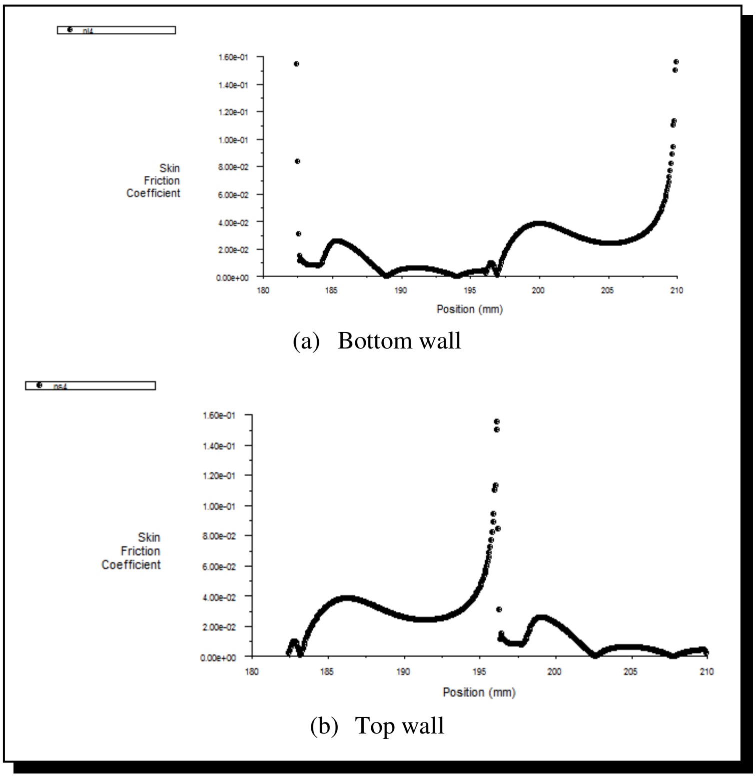


Figure 3. Profiles of the horizontal component of the velocities with $H/S = 0.182$ for different Reynolds $x = 182.41$ mm

The profiles of the horizontal velocity for different Reynolds are plotted in Figure 3, at $x = 182.41$ mm station, above. This figure shows the appearance of negative values of velocity for different Reynolds numbers. This is due to the creation of recirculation zones which are especially important as increasing the Reynolds number.

Figure 4 shows the profiles of local friction coefficient on the bottom and top walls for different Reynolds numbers. It is noted minimum values of the friction wedge and on a large part of the upper and lower wall; this is due to the creation of the recirculation zones and the maximum is observed at the corner (top wall) and at the attachment points of the fluid to the walls.



(a) Bottom wall

(b) Top wall

Figure 4. Variation in local friction with $H/S = 0.182$ for $H = 5$ mm, $Re = 4000$ for a periodic corrugation

Thermal field

Figure 5 represents the contours of static temperature for different Reynolds numbers with $H/S = 0.182$. It is noted that the temperatures are highest at the entrance of the channel and there is a heat transfer fluid to the cold walls of the channel. Thick thermal boundary layers are observed along the upper and lower wall; this is consistent with the thickening rate of the dynamic boundary layer in the fluid separation regions. The boundary layer is very thin in the connecting areas that which is consistent with the high velocities in these areas.

The variation of the local Nusselt number on the top wall to a periodic corrugation is shown in Figure 6 for different Reynolds numbers. In consistency with that previously discussed, the maximum values of Nusselt number correspond to the corners where velocities are very important reflecting the existence of a high heat transfer in those areas because the temperature gradient is very high along the normal to the wall; this leads a significant heat flux. However, immediately downstream of these corners and also at the corners (zones with low velocity); the Nusselt number reaches minimum values since the boundary layer tends to peel off, the velocities are low, therefore the temperature gradient along the normal to the wall is low.

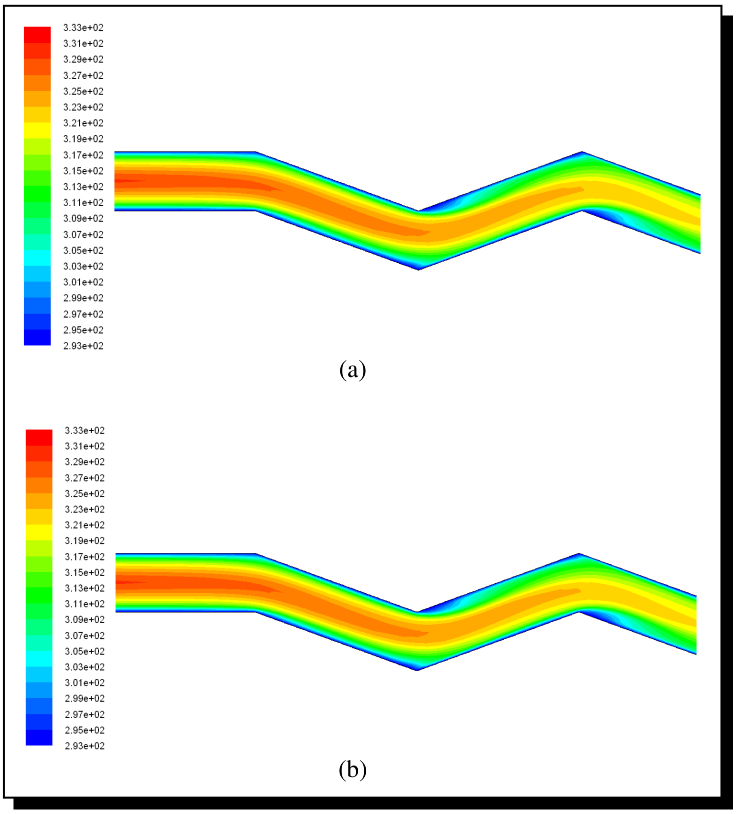


Figure 5. Contours of static temperature with $H/S = 0.182$ for $H = 5$ mm (a) Re=4000 (b) Re=2800

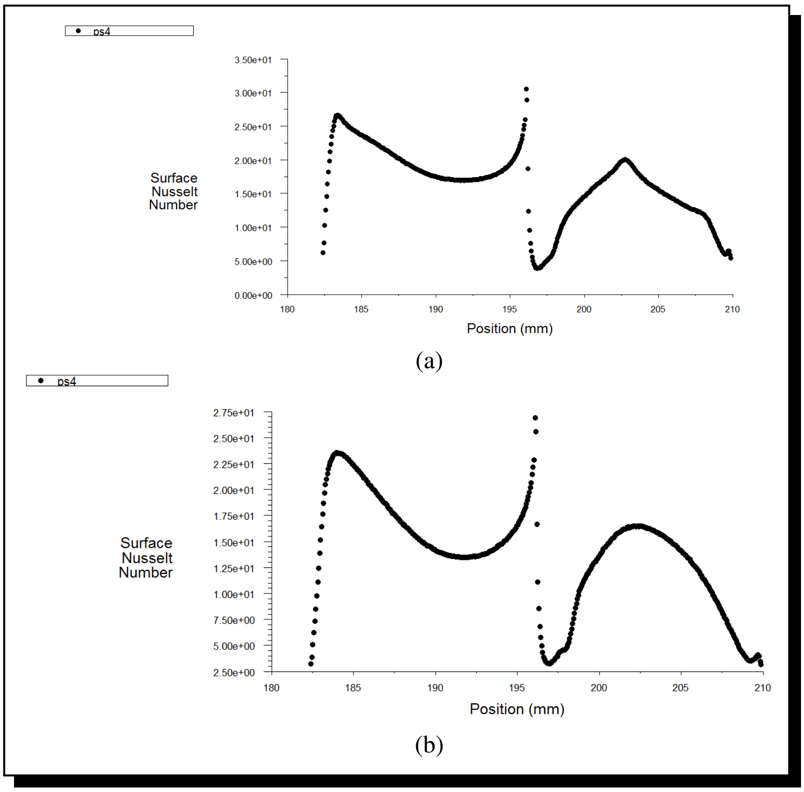


Figure 6. Variation in number of local Nusselt with $H/S = 0.182$ for $H = 5$ mm (a) Re=4000 (b) Re=2800.

4.2 Height effect

Dynamic field

The contours of the average velocity in the corrugated channel for varying heights (different form ratios $H/S = 0.182$ and $H/S = 0.364$) and are shown in Figure 7 for $Re = 4000$. It is found that the biggest velocities are centrally located and particularly at the reduced sections where the principle of mass conservation (continuity equation) imposes an acceleration of the fluid.

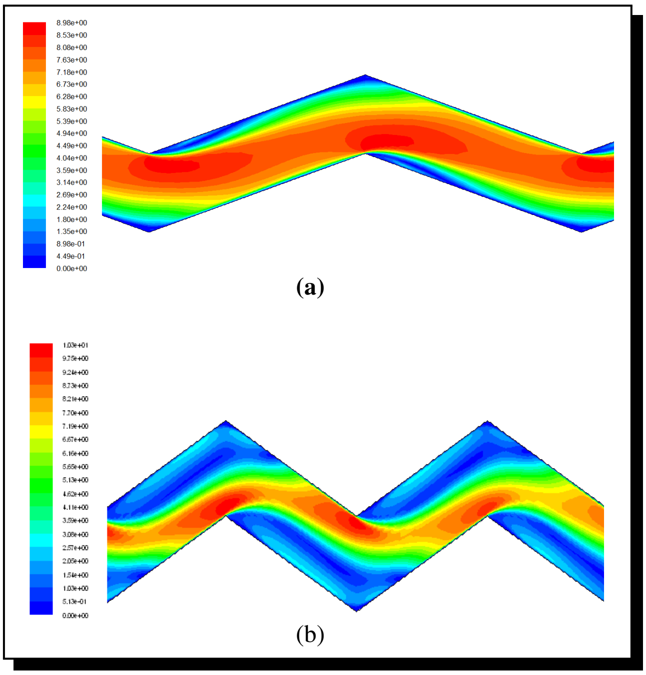


Figure 7. Contours of the average velocity for $Re = 4000$ (a) H = 5mm (b) H =10 mm

Thermal field

Figure 8 exposes the temperature contours in the corrugated channel for different aspect ratios for $Re = 4000$. It is noted that the temperatures are highest at the entrance of the channel and that there is a heat transfer fluid to the cold walls of the channel. We also note that more channel height increases and the heat transfer fluid to the walls is intense.

5. Conclusion

In this study, a numerical simulation of two-dimensional flow with heat transfer was conducted using Fluent CFD software that is based on the finite volume method. The obtained results for a refined mesh are in good agreement with experimental results. Furthermore, this study has allowed us to see the influence of the Reynolds number and channel height on the thermal-hydraulic behavior and we deduced the following results:

- The produced simulations for different Reynolds numbers show the presence of recirculation zones within the channel and that increase with increasing these numbers.
- The local Nusselt number reaches the maximum at the corners where there is connection corresponding to the areas where velocities are important and the values are minimum

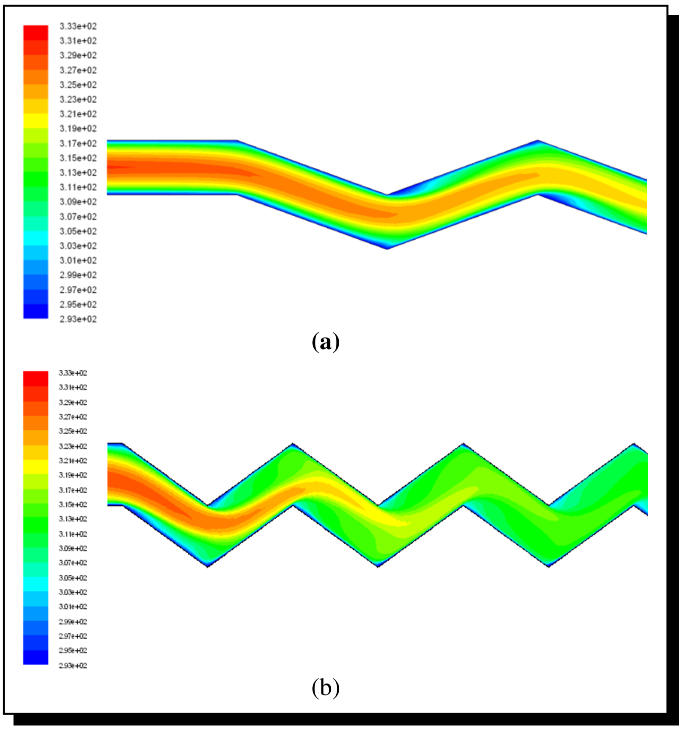


Figure 8. Contours of static temperature for $Re = 4000$ (a) H = 5mm (b) H =10mm

over a large part because of the presence of recirculation zones channel which generates less heat transfer.

- The Nusselt number and friction coefficient are influenced by the Reynolds number, an increase of the latter causes an increase in the Nusselt number and a reduction in friction.
- An increase in the Nusselt number is also observed with the increase in height of the channel.
- Increasing the height also results in an increase of the friction coefficient.

Finally, this study showed that the use of corrugated channels in the dynamic vein remains an effective means of improving its performance. All the presented results show that the corrugations help (allow) to create disturbances in the flow at the same time they cause pressure losses.

Competing Interests

The authors declare that they have no competing interests.

Authors' Contributions

All the authors contributed significantly in writing this article. The authors read and approved the final manuscript.

References

[1] G. Fabbri, Heat transfer optimization in corrugated wall channels, *International Journal of Heat and Mass Transfer* **43** (2000), 4299 – 4310.

[2] M. Faghri and Y. Asako, Finite-Volume Solutions for Laminar Flow and Heat Transfer in Corrugated Duct, *Journal of Heat Transfer* **109** (1987), 627 – 634.

[3] M. Gradeck, B. Hoareau and M. Lebouche, Local analysis of heat transfer inside corrugated channel, *International Journal of Heat and Mass Transfer* **48** (2005), 1909 – 1915.

[4] Y. Islamoglu and C. Parmaksizoglu, The effect of channel height on the enhanced heat transfer characteristics in a corrugated heat exchanger channel, *Applied Thermal Engineering* **23** (2003), 979 - 987.

[5] R. Lanani-Benchabi and M. Kadja, Contribution à l'étude thermique et hydraulique d'un canal plan corrugué, *Revue des Energies Renouvelables* **10**(3) (2007), 381 – 386.

[6] A. Masoud and R.H. Khoshkhoo, Effects of wavelength on the thermal performance of a plate heat exchanger, *Elixir Thermal Engineering* **60** (2013), 16603 – 16608.

[7] H.M. Metwally and R.M. Manglik, Enhanced heat transfer due to curvature-induced lateral vortices in laminar flows in sinusoidal corrugated-plate channels, *International Journal of Heat and Mass Transfer* **47** (2004), 2283 – 2292.

[8] M. Molki, Heat transfer characteristics of corrugated ducts with variable corrugation angle, *Heat Exchangers* **6** (1988), 2879 – 2884.

[9] P. Naphon, Laminar convective heat transfer and pressure drop in the corrugated channels, *International Communications in Heat and Mass Transfer* **34** (2007), 62 – 71.

[10] V. Ozbolat, N. Tokgoz and B. Sahin, Flow characteristics and heat transfer enhancement in 2D corrugated channels, *International Journal of Mechanical Science and Engineering* **7**(10) (2013), 796 – 800.

[11] H. Pehlivan, Experimental investigation of convection heat transfer in converging–diverging wall channels, *International Journal of Heat and Mass Transfer* **66** (2013), 128 – 138.

[12] B. Sunder and S. Trollander, Periodic Laminar flow and heat transfer in a corrugated two dimensional channel, *International Communications in Heat and Mass Transfer* **16** (1989), 215 – 225.

[13] L.-Z. Zhang and Z.-Y. Chen, Convective heat transfer in cross-corrugated triangular ducts under uniform heat flux boundary conditions, *International Journal of Heat and Mass Transfer* **54** (2011), 597 – 605.