Numerical Method for laminar fully developed flow in arbitrary cross section of ducts

Beghdadi Lotfi[#1], Belkacem Abdellah[#2]
[#]Labaoratory of ENERGARID, University of Bechar
BP 417 Route of Kenadsa, 08000 Bechar, Algeria
[1]beghdadi.lotfi@hotmail.fr
[2]belkacemabdellah915@yahoo.fr

Abstract—The present paper deals with the approximation of the solutions of partial differential equations that describe the phenomena of heat transfer and fluid flow, using a method based on Stokes' theorem and applied an unstructured computational mesh. The thus developed method will be applied in a problem of heat transfer charactiristics of laminar fully developed flow. After developing a code for calculating quantitative tests are planned to determine the accuracy of the method by the comparison with analytical solution or other studies that are already done.

Keywords—Numerical method, unstructured grid, heat transfer, fully developed flow, complex geometry.

I. INTRODUCTION

Study of thermal hydraulic characteristics for laminar fullydeveloped flow through ducts of different cross section assumes importance in design of heat exchangers of existing geometric configuration, as well as, for development of newer types of flow passages in them in order to achieve better thermal hydraulic performance. The primary objective of the heat exchanger designer is to work with duct geometries that yield (i) a high value of heat transfer area to volume ratio, (ii) a high value of heat transfer coefficient and (iii) a corresponding low value of friction factor. As a result, substantial research work is focused towards development of compact and efficient duct geometries, those satisfy the above criteria. In compact heat exchangers, due to smaller system dimensions, the hydraulic diameter is low, which, for most of the practical cases, yields essentially laminar flow. It is therefore essential to investigate the thermal hydraulic behaviour of ducted flows, particularly in the laminar regime. These studies find its relevance in different areas, such as, aerospace, nuclear, chemical and process industries, biomedical, electronics and instrumentation.

In most of the heat exchangers in service, especially in shell and tube type, generally circular duct is used. However, it may be noted here that a polygonal duct (number of sides ¼ n), offers higher surface area to volume ratio as compared to a circular duct. This ratio increases with the decrease in n. As a result, triangular ducts offer largest surface area to volume ratio and for the circular duct, for which n is infinite, it is the least. Hence, from the view point of compactness, the triangular duct should be the most preferred geometry and the square ducts should be the next choice. Circular ducts, on the other hand, should have the least priority. Offering maximum compactness, i.e., highest surface area to volume ratio, however, is not the sole criterion for selection of duct geometry. A designer should also look into the overall thermal hydraulic behaviour of the flow through the ducts. In this regard, it may be mentioned here that the Sinusoidal duct is one of the most frequently used duct cross sections for plate-fin exchanger design.

Extensive research work has been carried out by numerous researchers to study the thermal hydraulic behaviour of laminar flow through ducts of various geometries as the problem is of practical importance and relevant to heat exchanger industries. Upto 1978, exhaustive literature review has been presented by Shah and London [1]. Subsequently, Hartnett and Kostic [2] have compiled the existing experimental and numerical data for Newtonian and non-Newtonian fluid flow through rectangular ducts in both laminar and turbulent regimes. As more and more articles are still being published in this area, it is apparent that further monographs are essential on this topic. A detailed review of some literature covering various thermal hydraulic aspects of flow through ducts of different geometries, subjected to various possible boundary conditions, is left out of the scope of the present work. However, in the present context, literatures are reviewed only on works covering generalized techniques for flow and heat transfer through arbitrary shaped ducts.

As reported by Shah and London [1], there are various methods available for solution of relevant momentum and energy equations applicable for laminar fully-developed ducted flows, ranging from analytical treatments to computational solutions by finite difference/element techniques. Some of these methods are applicable only for simple geometries. For example, 'exact solution' is possible, only when the concerned geometry and the boundary conditions are relatively straightforward. On the other hand, some of the methods, like numerical solutions, are applicable to almost all kind of duct geometries and boundary conditions, although they are quite costly. As far as the analytical or semi-analytical methods are concerned, in the past, various researchers have used the ‘Conformal Mapping’ technique [3], the ‘Generalised Integral Transform’ technique [4], the ‘Variational Method’ [5] and the ‘Series Solution Method’ [6-13]. Among these various approximate semi-analytical methods, solutions by ‘Point-Matching-Methods’ and ‘Least Square Methods’, those fall into the general class of series solution method, require specialmention, as these methods are very general in nature and are applicable to fully-developed flow through ducts of very complex shape, as long as a constantwall heat flux type of boundary condition is used.

In general, for fully-developed ducted flow, if the dependent variables are suitably transformed, a Laplace and a Poisson equation can represent the momentum and the energy equations respectively. The energy equation, after decomposing the dependent variable (temperature) into complementary and particular solutions, can be further transformed to a Laplace equation. The general solution of Laplace equation is obtained by a linear combination of harmonic functions in the form of a truncated infinite series, consisting of N (starting from zero) terms. Therefore, the method involves solution of $n = 2N + 1$ unknown coefficients. The solutions exactly satisfy the governing equations, although the implementation of boundary conditions requires special treatment.[7]

Sparrow et al. [6,8] have applied the point matching method, using algebraic-trigonometric polynomials, for longitudinal flowover array of cylinders. Cheng and Jamil [9] have adopted the point matching technique to study flow and heat transfer in cylindrical ducts with diametrically opposite flat sides.

The least square method has been employed Ratkowsky and Epstein [10] and Hagen and Ratkowsky [11] to study laminar flow in regular polygonal ducts with circular centered cores and in cylindrical ducts with regular polygonal cores respectively. The least square approximation was first adopted by Sparrow and Haji-Sheikh [12] employing Gram-Schmidt orthonormalisation to study flow and heat transfer in arbitrary shaped ducts. However, they furnished results only for circular ducts and sectors.Subsequently, Shah [13] obtained various averaged parameters like friction factor in the form of fRe and Nusselt number for flow through ducts of various shapes, for example, rectangular, isosceles triangular, sinusoidal and equilateral triangle with rounded corners employing Golub’s method (using Householder reflections). It may be mentioned here that although the methods of Sparrow and Haji-Sheikh [12] and Shah [13] are similar to each other.

It is worthwhile to mention at this point that the present problem could also be solved using commercial or self developed (in-house) CFD codes. However, the advantages of the adopted numerical method, are as follows (as will be shortly apparent): (i) the present method is much faster compared to CFD solutions as it boils down to solving two least problems ; one for the velocity and the other for the temperature, where as, the CFD solution would require solution on either unstructured or on block structured curvilinear grids, (ii) the grid generation requires only generation of the boundary points and not the interior points and (iii) the velocity and temperature fields are known at any point in the domain, whereas the CFD solutions are actually obtained only on specified nodes.

In the present study, we have proposed a numerical using unstrictured grid and we have applyed in problem of sinusoidal duct and we have compared our numerical results with the results that was obtained by L. Zhi [14] and also we have considered the study of differents shapes of ducts to show the fiability of our method.

II. GOVERNING EQUATION :

A. Momentum transfer :

Sinusoidal duct is one of the most frequently used duct cross sections for plate-fin exchanger design. Fig. 2 shows the representing geometries for a duct: duct height 2a; width 2b; duct length, L; aspect ratio *a/b*. The duct is comprised of lower plate *AC*, and two sinusoidal fins *AD* and *DC*. The coordinate system for the two fins is s and y1, which is tangential and normal to the fins, respectively. Their directions change from point to pint along the curved fins. Fin curve can be expressed as a sinusoidal function :

$$y = a\left[1 - cos\left(\frac{\pi}{b}\,x\right)\right] \qquad (1)$$

The flow in the duct is considered to be laminar and hydrodynamically fully developed, but thermally and mass developing in the entrance region of the duct. The fluid is Newtonian with constant physical properties. Additionally, uniform plate temperature and uniform plate concentration boundary conditions are considered.

For fully developed laminar flow in ducts, the Navier–Stokes equations reduce to [15].

$$\mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) = \frac{dP}{dz} \qquad (2)$$

where μ is dynamic viscosity (Pa s), P is the pressure (Pa), z is the axial coordinate (m).

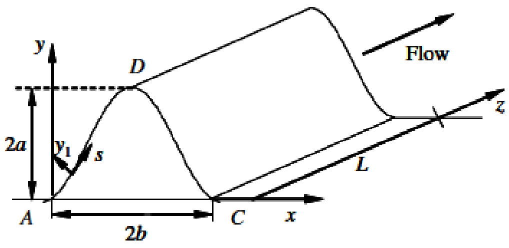

Fig. 1. Geometries of a sinusoidal duct.

The above equation can be normalized to:

$$\frac{\partial^2 u^*}{\partial x^{*2}} + \left(\frac{b}{a}\right)^2 \frac{\partial^2 u^*}{\partial y^{*2}} + \frac{4b^2}{d_h^2} = 0 \quad (3)$$

with a dimensionless velocity :

$$u^* = -\frac{\mu\, u}{(dP/dz) d_h^2} \quad (4)$$

where hydraulic diameter :

$$d_h = \frac{4A_c}{P_f} \quad (5)$$

where Ac is the cross-section area of the duct (m^2), P_f is the perimeter of the duct (m).

Dimensionless coordinates are defined by :

$$x^* = \frac{x}{2b} \quad (6)$$

$$y^* = \frac{y}{2a} \quad (7)$$

The characteristics of fluid flow in the duct can be represented by the product of the friction coefficient and the Reynolds number as:

$$(fRe) = \left(-\frac{d_h\left(\frac{dP}{dz}\right)}{2\rho u_m^2}\right)\left(\frac{\rho u_m d_h}{\mu}\right) = \frac{1}{2u_m^*} \quad (8)$$

Where u_m^* is the average dimensionless velocity on a cross section, and it is calculated by:

$$u_m^* = \frac{\int\int u^* dA}{A_c} \quad (9)$$

B. Heat transfer:

Energy conservation in the fluid can be expressed by [15]:

$$\rho_a c_p u \frac{\partial T}{\partial z} = \gamma\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \quad (10)$$

where T is fluid temperature (K).

The above equation can be normalized to:

$$U\frac{\partial^2 \theta}{\partial z^*} = \frac{\partial^2 \theta}{\partial x^{*2}} + \left(\frac{b}{a}\right)^2 \frac{\partial^2 \theta}{\partial y^{*2}} \quad (11)$$

with a dimensionless temperature :

$$\theta = \frac{T - T_w}{T_i - T_w} \quad (12)$$

where in the equations, Ti is the inlet temperature of the fluid, and T_w is the wall temperature. Dimensionless axial position is defined by:

$$z^* = \frac{z}{Re\, Pr\, d_h} \quad (13)$$

Velocity coefficient U is defined by:

$$U = \frac{u^*}{u_m^*}\frac{4b^2}{d_h} \quad (14)$$

An energy balance in control volume in the duct [15] will give the equation for the estimation of the local Nusselt number as:

$$Nu_L = -\frac{1}{4\theta_b}\frac{d\theta_b}{dz^*} \quad (15)$$

III. Numerical Procedure

The Laplace's equation is integrated in space using a finite volume method that is developed for an unstructured grid made up of quadrilaterals [16],[17],[18],[19].

For the integration around finite volume, the derivations of the flow equation must be converted into closed line integrals using same formulation of the Stokes theorem, which is described by the following equation:

$$\oint_E \vec{F}\,.\overrightarrow{dr} = \iint_s rot\,\vec{F}\,.\vec{n}dS \tag{16}$$

Where $\overrightarrow{dr}$ is the elementary arc, dS is the elementary surface and is the normal vector to this surface. $\vec{F}$ can be the velocity U or the temperature T. The computational domain is discredited on a quadrilateral unstructured grid where each node is the centre of polygonal cell constituted of four elements; all computed variables are stored at the centres of the polygonal as:

A. Approximation of the first derives

The convective terms are calculated at the node P (fig.2). The nodal finite volume descritization scheme is used for the discretization of the convective terms that appear in the governing equation. The first differences are calculated as:

$$\left(\frac{\partial F}{\partial x}\right)_c = \frac{1}{A_C}\int_{Sc} F.dy = \frac{1}{A_C}\sum_{i=1}^{NC}\frac{F_{i+1}+F_i}{2}(y_{i+1}-y_i) \tag{17}$$

$$\left(\frac{\partial F}{\partial y}\right)_c = \frac{1}{A_C}\int_{Sc} F.dx = \frac{1}{A_C}\sum_{i=1}^{NC}\frac{F_{i+1}+F_i}{2}(x_{i+1}-x_i) \tag{18}$$

Where $A_C$is the area of the polygonal control volume (1,2,3,…NE), x,y are the coordinate of the polygonal vertices, and I refers to the vertices number of external polygonal control volume.

B. Approximation of the second derives

This terms must be calculated at the node P and this achieved by computing the second order derivatives at the same point. The required second differences may be computed as:

$$\left(\frac{\partial^2 F}{\partial x^2}\right)_c = \left[\frac{\partial}{\partial x}\left(\frac{\partial F}{\partial x}\right)\right]_C = \frac{1}{A_{CI}}\int_{SCI} F.dy = \frac{1}{A_{CI}}\sum_{i=1}^{NC}\left(\frac{\partial F}{\partial x}\right)_E (y_{i+1}-y_i) \tag{19}$$

$$\left(\frac{\partial^2 F}{\partial y^2}\right)_c = \left[\frac{\partial}{\partial y}\left(\frac{\partial F}{\partial y}\right)\right]_C = \frac{1}{A_{CI}}\int_{SCI} F.dx = \frac{1}{A_{CI}}\sum_{i=1}^{NC}\left(\frac{\partial F}{\partial y}\right)_E (x_{i+1}-x_i) \tag{20}$$

$A_{CI}$is the area of polygonal control volume (2,4,…NE) (fig.2) and I refer to the vertices number of internal polygonal control volume. Where, the first differences at the middle of the edge are defined as:

$$\left(\frac{\partial F}{\partial x}\right)_E = \frac{1}{A_E}\int_{SE} F.dy = \frac{1}{A_E}\sum_{i=1}^{4}\frac{F_{i+1}+F_i}{2}(y_{i+1}-y_i) \tag{21}$$

$$\left(\frac{\partial F}{\partial y}\right)_E = -\frac{1}{A_E}\int_{SE} F.dx = -\frac{1}{A_E}\sum_{i=1}^{4}\frac{F_{i+1}+F_i}{2}(x_{i+1}-x_i) \tag{22}$$

A_E is the area of the quadrilateral control volume ((1),(2),(3),(4))(Fig.2.) and the four vertices of quadrilateral control volume.

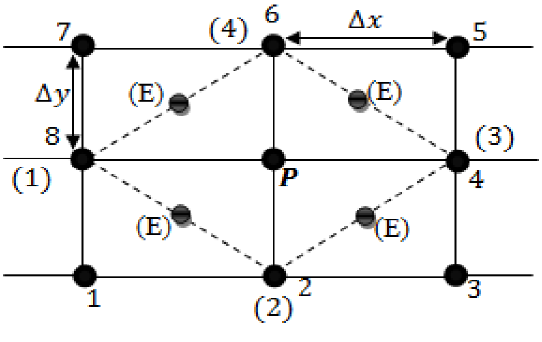

Fig. 2.The computational control volume structure.

IV. RESULTS:

In the present study we have proposed a numerical method and considered a sinusoidal ducts and compared our numerical results with the results that are obtained by L. Z. Zhang [14] and also we have apply our method in different shapes of ducts to show the reliability of our numerical solution.

To assure the accuracy of the results presented, a grid independence test was performed for the duct to determine the effects of the grid size.

To further validate the numerical program, ordinary ducts of various cross sections are calculated under uniform temperature conditions for all walls. For hydrodynamically fully developed laminar flow in ducts, (*fRe*) is a constant. The local Nusselt numbers in the duct will decrease along the flow and reach stable values when the flow is thermally fully developed. The fully developed Nu values under uniform temperature conditions are denoted as *NuL*. The calculated values of (*fRe*) and *NuL* for these ordinary ducts are listed inTable 1 and Table 2.

The comparison is clearly shows that for simple geometries the results of the present study are in excellent agreement with those of the previous works.

- Comparison of *fRe:*

	fRe			
Cross Section	a/b	Ref [3,5]	L'étude de L. Z. Zhang	Notre étude
rectangle	0.125	20.5	20.32	20.62
	0.25	18.25	18.14	18.24
	0.5	15.50	15.30	15.55
	1.0	14.22	14.01	14.23
triangle	0.289	13.24	12.87	12.94
	0.5	13.30	12.91	13.15
	0.866	13.32	13.39	13.35
	1.866	13.09	12.96	13.11
sinusoïdale	0.5	11.20	11.70	11.19
	0.75	12.23	12.21	12.17
	1.0	13.00	12.96	13.06
	1.5	14.00	14.11	13.97

Table .1 fully developed (*fRe*) for common ducts of various cross sections.

- Comparison of Nusselt number:

	Nu			
Cross Section	a/b	Ref [3,5]	L'étude de L. Z. Zhang	Notre étude
rectangle	0.125	5.60	5.73	5.56
	0.25	4.44	4.55	3.95
	0.5	3.39	3.45	3.57
	1.0	2.97	3.06	2.98
triangle	0.289	2.30	2.26	2.15
	0.5	2.35	2.45	2.54
	0.866	2.50	2.59	2.60
	1.866	2.28	2.39	2.40
sinusoïdale	0.5	2.12	2.18	2.15
	0.75	2.33	2.37	2.40
	1.0	2.45	2.52	2.52
	1.5	2.60	2.57	2.59

Table .1 fully developed (*Nu*) for common ducts of various cross section

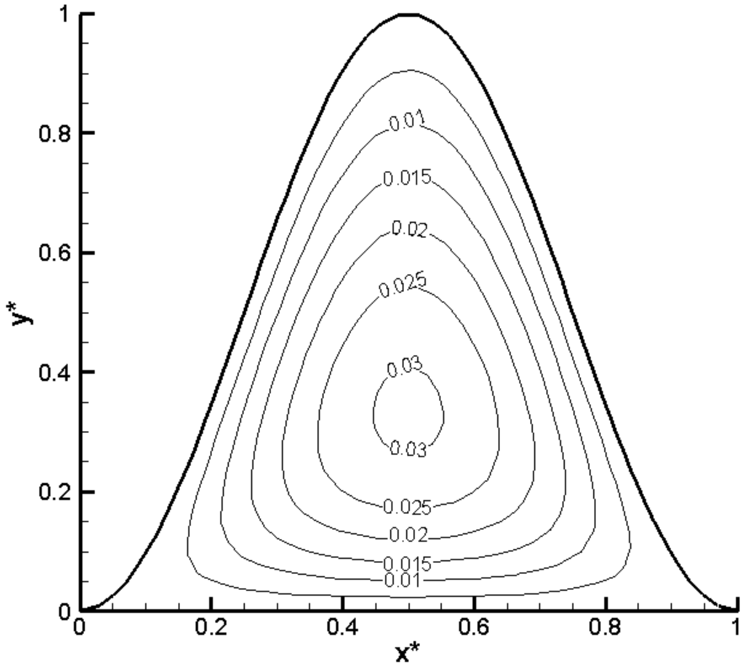

Fig. 3. dimensionless velocity profiles *u** on duct cross section, a/b=1.0

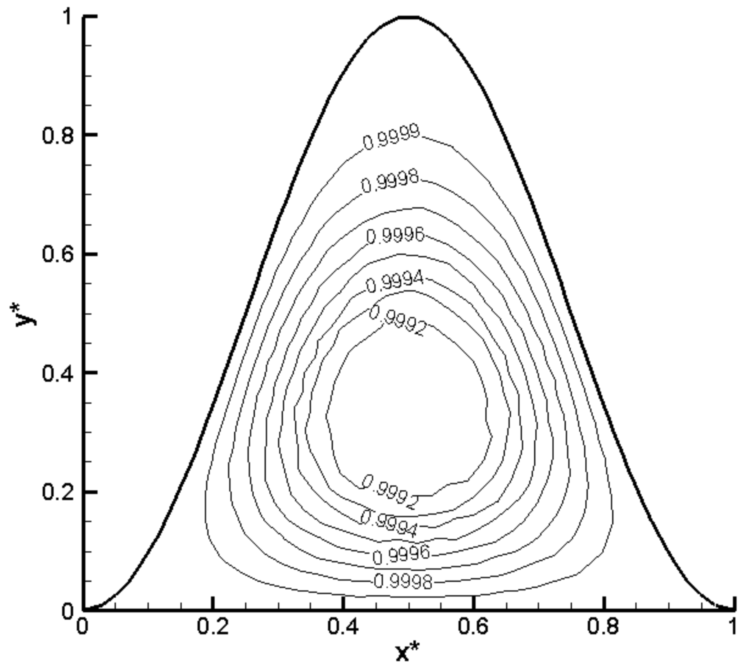

Fig .4. dimensionless temperature contours on duct cross section, a/b=1.0

The applicability of our method is demonstrated by considering various standard regular shapes (such as circular, square, rectangular,... etc, that are chosen for comparison). While the other shapes (such as elliptical, and semi elliptical or lenticular... etc) that are frequently used for heat transfer applications and are considering as irregular geometries, no analytical solutions are available. So, it is demonstrated how the present solution can accurately predict the friction factor, and the Nusselt number by treating them as arbitrary cross section.

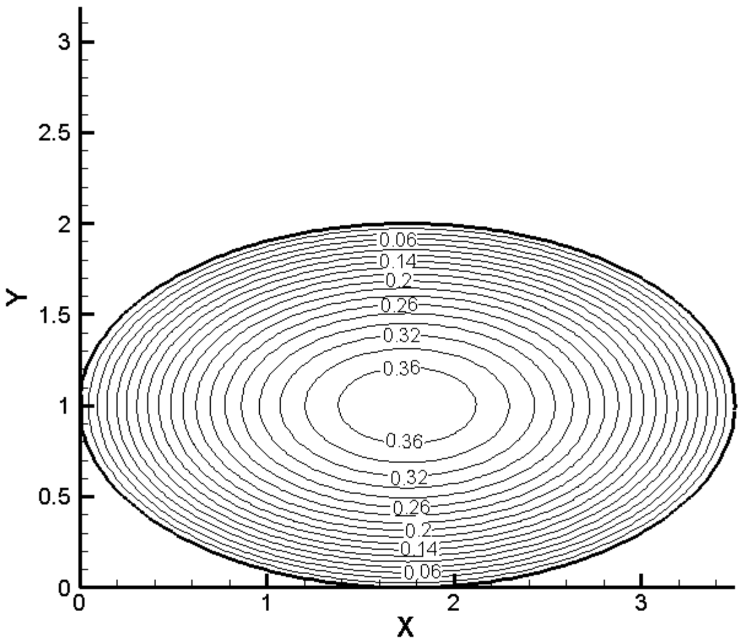
0.06
0.14
0.2
0.26
0.32
0.36
0.36
0.32
0.26
0.2
0.14
0.06
Y
X
0
0.5
1
1.5
2
2.5
3
0
1
2
3

Fig. (a)

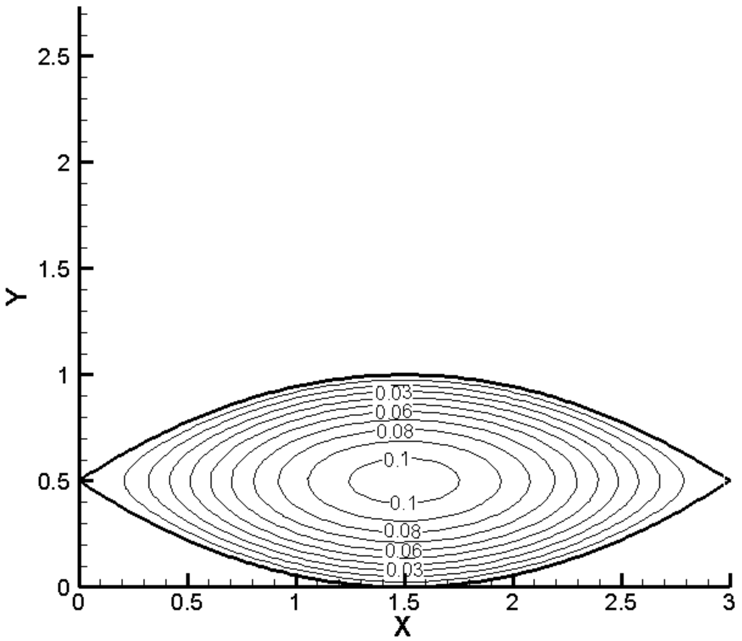
0.03
0.06
0.08
0.1
0.1
0.08
0.06
0.03
Y
X
0
0.5
1
1.5
2
2.5
0
0.5
1
1.5
2
2.5
3

Fig. (b)

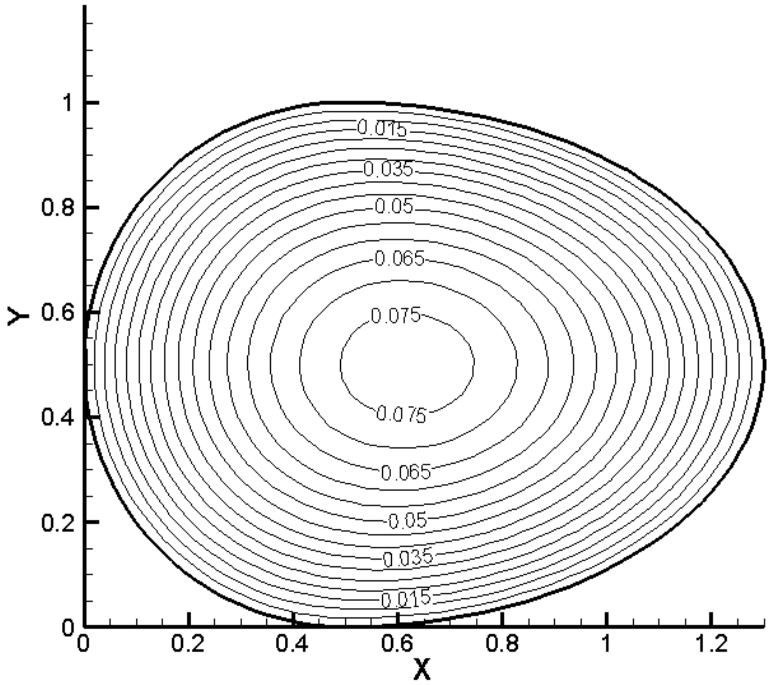

Fig. (c)

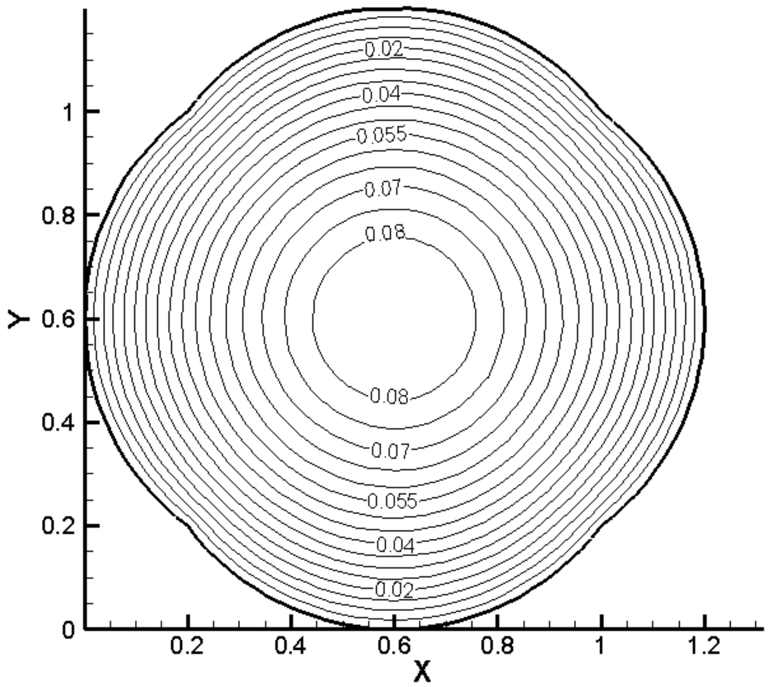

Fig. (d)

Fig. 5. Velocity contours in different shapes (a) elliptical (b) lens shaped (c) semi elliptical (d) discontinuous corrugated.

As noted earlier, various standard shapes such as rectangular and square cross-sections, can be easily obtained from the proposed numerical method. Similarly other frequently used channels for heat transfer and micro-channel applications such as (a) elliptical (b) lens shaped (combination of two circular arcs), (c) semi-elliptical or lenticular (which consiste of semi-circular and a semi elliptical duct joined together where *a*, and *b* are semi major axis, and semi elliptical portion respectively), (d) discontinuous corrugated wall; that can be easily be analysed, where the fig. 5 (a,b,c,d) show the velocity contours profiles for this complicated and generic shapes. Whereas the velocity contours follow the shape of the ducts.

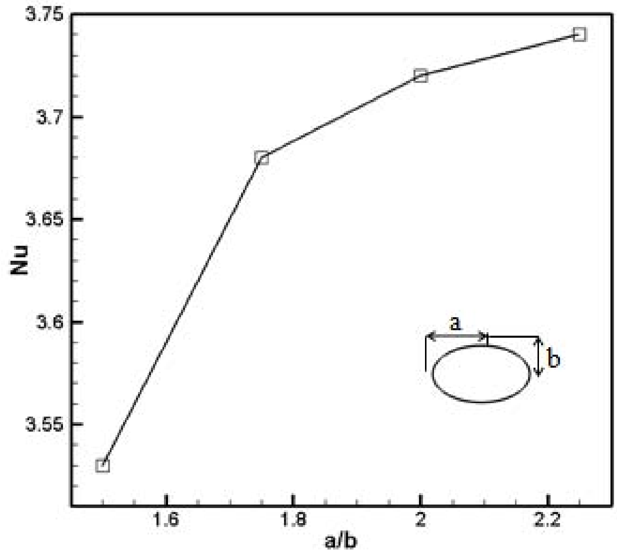

Fig. (a')

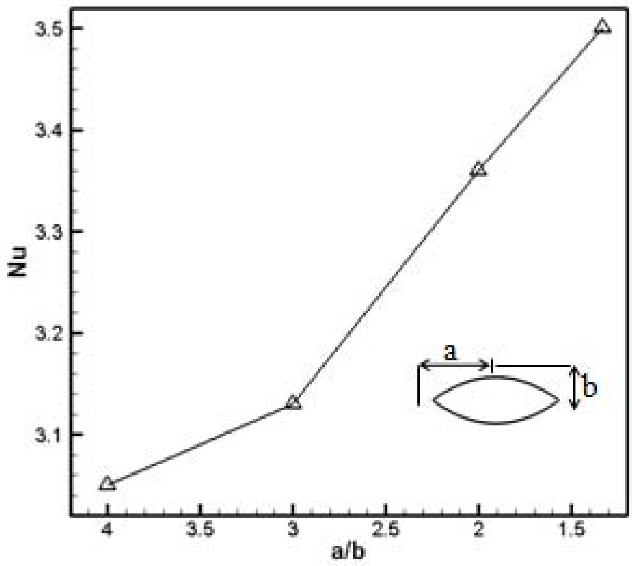

Fig. (b')

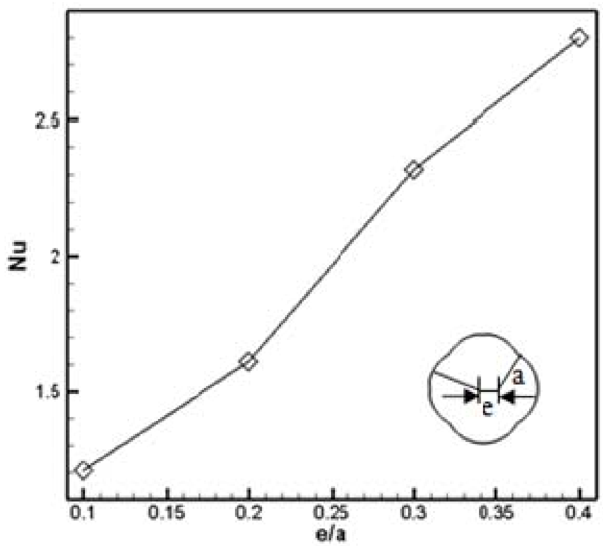

Fig. (c')

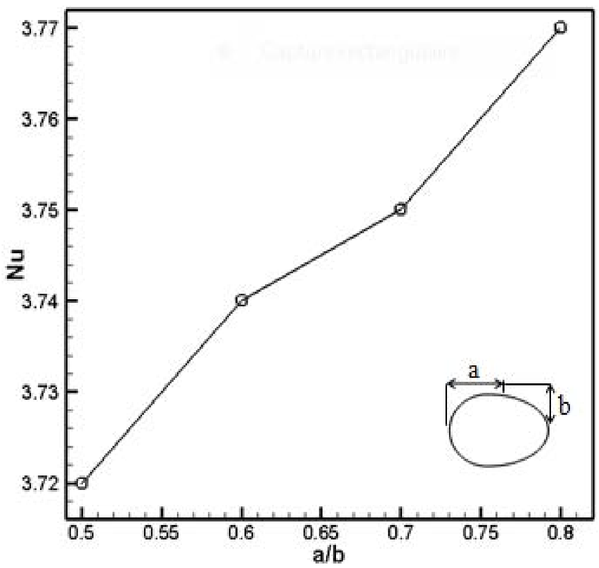

Fig. (c')

Fig. 6. Variation of *Nu* number with the aspect ratio *a/b*.

- Presentation of Nusselt number:

Figure .6 shows that as expected the variation of Nusselt number with the ratio of the diagonals (a/b). It is observed that the *NuL* increase rapidly with the increasing of the ratio a/b in all the type of duct geometries and the angular variation gradually deviates from uniform distribution; being minimum at the ends of major axis and maximum at the ends of major axis.

V. CONCLUSION

The numerical method proposed seems able to solve different problem of heat transfer and fluid flow especially when we have problem with geometries pretty complexes.

The problem of fully developed flow: the comparison of *fRe* and *Nu* number of the present study with the existing results show excellent agreement. Although, the local Nusselt number show different behaviour but it increase rapidly with the increase of the ratio a/b in all the types of geometries. Also, this study helped us for the development of newer types of flow passages in order to achieve better thermal hydraulic performances. So, from he advantages of the adopted numerical method, are as follows (as will be shortly apparent): (i) the present method is much faster compared to CFD solutions as it boils down to solving two least problems ; one for the velocity and the other for the temperature, where as, the CFD solution would require solution on either unstructured or on block structured curvilinear grids, (ii) the grid generation requires only generation of the boundary points and not the interior points and (iii) the velocity and temperature fields are known at any point in the domain, whereas the CFD solutions are actually obtained only on specified nodes.

ACKNOWLEDGMENT

The present study was supported by ENERGARID laboratory, especially the group "thermo-fluidic".

REFERENCES

[1] R.K. Shah, A.L. London, Laminar flow forced convection in ducts. Adv. Heat Transfer(Suppl. 1) (1978).
[2] J.P. Hartnett, M. Kostic, Heat transfer to Newtonian and non-Newtonian fluids in rectangular ducts. Adv. Heat Transfer vol. 19 (1989).
[3] U.A. Sastry, Heat transfer by laminar forced convection in a pipe of curvilinear polygonal section. J. Sci. Eng. Res. vol. 7 (1963) 281e292.
[4] R.M. Cotta, Integral Transform in Computational Heat Transfer and Fluid Flow. CRC Press, Boca Raton, Florida, USA, 1993.
[5] E.M. Sparrow, R. Siegel, A variational method for fully developed laminar heat transfer in ducts. Trans. ASME, J. Heat Transfer vol. 81 (1959) 157e167.
[6] E.M. Sparrow, A.L. Loeffler Jr., Longitudinal laminar flow between cylinders arranged in regular array. AICHE J. vol. 5 (1959) 325e330.
[7] S. Ray, D. Misra, Laminar fully developed flow through square and equilateral triangular ducts with rounded corners subjected to H1 and H2 boundary conditions. Int. J. Thermal Sciences vol 49 (2010) 1763-1775
[8] E.M. Sparrow, A.L. Loeffler Jr., H.A. Hubbard, Heat transfer to longitudinal laminar flow between cylinders. Trans. ASME, J. Heat Transfer vol. 83 (1961) 415e422.
[9] K.C. Cheng, M. Jamil, Laminar flow and heat transfer in circular ducts with diametrically opposite flat sides and ducts of multiply connected cross sections. Can. J. Chem. Eng. vol. 48 (1970) 333e334.
[10] D.A. Ratkowsky, N. Epstein, Laminar flow in regular polygonal ducts with circular centered cores. Can. J. Chem. Eng. vol. 46 (1968) 22e26.
[11] S.L. Hagen, D.A. Ratkowsky, Laminar flow in cylindrical ducts having regular polygonal shaped cores. Can. J. Chem. Eng. vol. 46 (1968) 387e388.
[12] E.M. Sparrow, A. Haji-Sheikh, Flow and heat transfer in ducts of arbitrary shapes with arbitrary thermal boundary conditions. Trans. ASME, J. Heat Transfer vol. 88c (1966) 351e358.
[13] R.K. Shah, Laminar flow friction and forced convection heat transfer in ducts of arbitrary geometry. Int. J. Heat Mass Transfer vol 18 (1975) 849e862.
[14] L.Z. Zhang, Heat and mass transfer in plate-fin sinusoidal passages with vapor-permeable wall materials. Int.J. Heat Mass Transfer vol 50 (2008) 618-629.
[15] R. Taylor, R. Krishna, in: Multicomponent Mass Transfer, John Wiley & Sons Inc., New York, 1993.
[16] G.K. Despotis, and S. Tsangaris, Fractional step method for the solution of incompressible Navier-Stockes equations on unstructured triangular meshes, Int. J. Num. Meth. Fluids, vol. 20, pp. 1273–1288, (1995).
[17] G. K. Despotis, and S. Tsangaris, A finite volume method for the Solution of the extruDate swell problem on unstructured triangular meshes, Int. J. Num. Meth. Heat Fluid Flow, vol. 6, pp. 65-83, (1996).
[18] S. Boivin, F. Cayré, and J. M. Hérard, A Finite volume method to solve the Navier-Stokes equations for incompressible flows on unstructured meshes Int. J. Therm. Sci., vol.39 , pp. 806–825, (2000).
[19] N. Piskounov, calcul intégral et différentielle tome 2, union Soviétique 11 ème Edition (1987).

AUTHOR PROFILE

Beghdadi Lotfi was born in Bechar-Algeria in 1988. He is currently a Ph.D. student in the University of Tahri Mohammed, Bechar (Algeria). He received his BS Degree in 2009 in mechanical engineering from the university of Bechar (Algeria). He received his MS Degree in in mechanical engineering from the university of Bechar (Algeria) in 2011. He is a member in Thermo-Fluidic equip in the laboratory of ENERGARID in the University of Tahri Mohammed, Bechar (Algeria). His research interests are natural convection heat transfer and its applications, numerical heat transfer and fluid flow.

Pr. Belkacem Abdellah was born in Bechar-Algeria in 1964. He is currently a professor in the Department of Technology at Bechar Technical University where he teaches fluid mechanics and thermodynamics related courses. He received his BS Degree in energetic at Oran Science and Technology University of Oran (Algeria) in 1988. He obtained his MS Degree from the University of Bechar in 1995 and his doctorate Degree in 2007 from the University of Science and Technology Houari Boumediene, Algeria, in the area of natural convective heat transfer. He received a professor Degree in 2015 from the university of Tahri Mohammed, Bechar (Algeria). His research interests are convective heat transfer and its applications in energy systems, numerical heat transfer and nanofluid.