NUMERICAL STUDY OF THE DYNAMIC AND THERMAL FIELD OF A FLOW IN A SHELL AND TUBE HEAT EXCHANGER EQUIPPED WITH TWO BAFFLES

Ahmed YOUCEF [1], Rachid SAIM [2]

A research on the thermo-hydraulic performance of a fluid flow in a heat exchanger with two baffles was carried in this paper, we are present a numerical study, using the K-ε model to study the dynamic and thermal field of a forced convection water flow in a single-pass tube and shell heat exchanger for a Reynolds number margin "turbulent regime", the effect of the insertion of baffles was analyzed. The governing equations, continuity, Navier–Stokes and energy, are solved using the finite volume method and the SIMPLE algorithm. The temperature and velocity contours were obtained for two different planes (XY and XZ) for different treated cases. With a mass flow various to 0.5 kg/s at 2 kg/s, the heat transfer coefficient increases by 2,075%, the pressure drop increases by 16,31%, the pumping power increases by 65,22%.

Keywords: Baffles, CFD, Finite volume, Shell and tube heat exchanger.

Nomenclature

D_s Shell size (mm)
d Tube diameter (mm)
N_t Number of tubes
L Heat exchanger length (mm)
T Shell side inlet temperature (°K)
p Pressure (Pa)
P Pumping power (W)
B_c Baffle cut (%)
N_b Number of baffles
h Heat transfer coefficient ($W/(m^2K)$
G_k Turbulent kinetic energy production (kg/ms^3)
k Turbulent kinetic energy (m^2/s^2)
u Velocity components (m/s)
i,j Positions coordinates
$C_{1\varepsilon}$; $C_{2\varepsilon}$; C_μ Constants of transport equations
m Mass flow rate (kg/s)

[1] Unité de Recherche en Energies Renouvelables en Milieu Saharien, URERMS, Centre de Développement des Energies Renouvelables, CDER 01000 Adrar, Algeria e-mail : a_youcef83@yahoo.fr
[2] Laboratoire Energétique et Thermique Appliquée (ETAP), Département de Génie Mécanique, Faculté de Technologie, Université Abou Bekr Belkaid, Tlemcen

Greek letters

ρ Density (kg/m^3)
μ Dynamic viscosity (kg/ms)
μ_t Turbulent viscosity (kg/ms)
ε Viscous dissipation rate (m^2/s^2)
σ_k Turbulent Prandtl numbers for k
σ_ε Turbulent Prandtl numbers for ε
η Overall performance factor

1. Introduction

Forced convection in complex geometries finds its importance in various industrial fields and more particularly in nuclear reactors, heat exchangers, solar collectors, cooling of electronic components, shell and tube heat exchangers. Such work is of interest in improving the thermal performance of a shell and tube heat exchanger with baffle. In terms of study of the forms of chicanes, we quote the works of, Saim and al [1], Yuan and al [2], Acharya and al [3]. The influence of inter-chicane distance has been studied experimentally by Wilfried and al [4]. The study the effect of geometric parameters (orientations and spacings) on transient forced convection by Ahmet [5]. Patankar and al [6], reported a work on the numerical analysis of periodic flow fully developed in forced convection in a duct. Bemer and al [7], confirmed the results of the previous study for a laminar flow with Reynolds numbers less than 600. Webb and al [8], studied the fluid flow and heat transfer in a channel to two parallel plates with reversed baffles. They based their numerical model on the periodic conditions for the fully developed flow proposed by Patankar and al. A numerical investigation for a laminar forced convection fluid between two parallel plane walls with baffles was carried out by Kelkar and al [9]. The results prove that the flow is characterized by strong deformations and large recirculation regions. In general, the number of Nusselt and the coefficient of friction increase with the Reynolds number. Their results also show that the thermal performances increase with the increase of the size of baffles and with the decrease of the spacing between baffles. Cheng and al [10] studied forced convection between two parallel flat plates with transverse baffles that are not symmetrically placed. Their results indicated that the position relative to rows of baffles is a factor influencing at the flow field, particularly for baffles with large sizes. Cheng and al [11] also analyzed laminar flows in forced convection in the inlet region of a horizontal channel. Calculations for the semi-infinite channel in which one or two pairs of baffles are symmetrically attached to the respective walls in the input region have been analyzed. Hydraulic and thermal effects as a function of the location of normal baffles within a 3D channel have been studied numerically by Lopez and al [12,13]. An analysis of laminar forced convection was performed with baffles subjected to uniform heat flow. On the other hand, the upper foundations and the side walls are supposed to be adiabatic. Their results show that the three-dimensional effects on the friction factor of a channel with unit elongation and a blocking ratio of 0.5, increased with the increase in Reynolds number Re.

An examination of the thermo-hydraulic performance in a shell and tube heat exchanger with seven tubes and six segmental baffles placed in overlapping. In this paper the dynamic and thermal behavior of fluid will be analyzed in detail using the commercial code FLUENT by varying the mass flow rate of water between 0.5 kg/s, 1 kg/s, 2 kg/s.

2. Problem definition

2.1 Geometry of the problem

The geometry studied in this paper is shown in Fig. 1, is a horizontal circular channel equipped with two baffles and seven tubes. The geometric dimensions are shown in table 1 and are based on the experimental study of Ender and al [14].

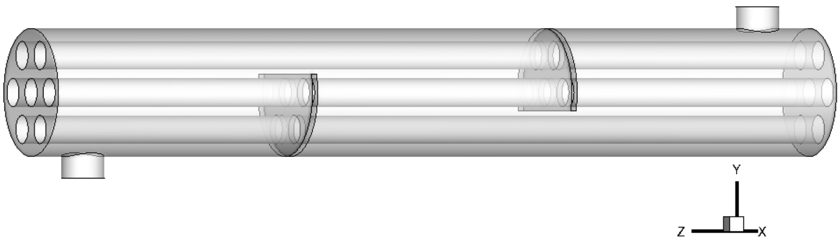

Fig. 1 Geometry of the system under investigation

Table 1

Geometric parameters

Parameter	Values
Shell size, D_s	90 mm
Tube outer diameter,d_o	20 mm
Tube bundle geometry and pitch	triangular, 30 mm
Number of tubes, N_t	7
Heat exchanger length, L	600 mm
Shell side inlet temperature, T	300°K
Baffle cut, B_c	36%
Number of baffles, N_b	2

2.2 Governing equations

the system of equations with respect to a ÿ artesian coordinate system is expressed as follows:

Continuity:

$$\frac{\partial(\rho.u_i)}{\partial x_i}=0 \tag{1}$$

Momentum:

$$\rho u_j\frac{\partial u_i}{\partial x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial u_i}{\partial x_j}-\rho\overline{u_i^{'}u_j^{'}}\right) \tag{2}$$

Energy:

$$\frac{\partial(\rho.u_iT)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\left(\frac{\mu}{\mathrm{Pr}}-\frac{\mu_t}{\mathrm{Pr}_t}\right)\frac{\partial T}{\partial x_j}\right) \tag{3}$$

Turbulence kinetic energy k:

$$\rho u_j\frac{\partial k}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k-\rho\varepsilon \tag{4}$$

Energy dissipation ε:

$$\rho u_j\frac{\partial \varepsilon}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}G_k-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{5}$$

Turbulent viscosity:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{6}$$

The turbulence production

$$G_k=-\rho\overline{u_i^{'}u_i^{'}}\frac{\partial u_j}{\partial x_i} \tag{7}$$

The model constants have the following values:

$C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92, C_μ=0.09, σ_k=1.0, σ_ε=1.3, et Pr_t=0.09.

3. Results and discussion

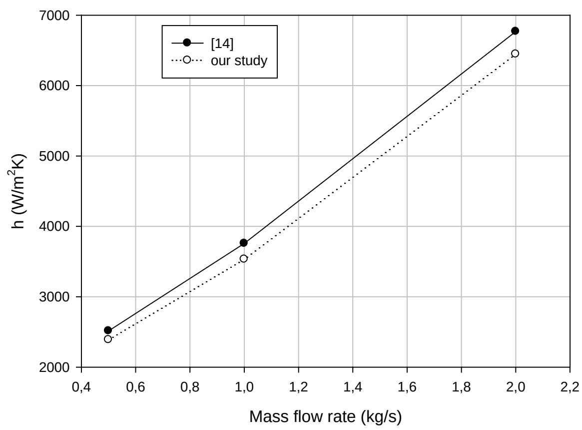

Fig.2 the heat transfer coefficient

The model with six baffles is validated with the data available in [14], it is found that the overall transfer coefficient of the fluid corresponds to the results of the literature and the difference between the two cases is about 4%.

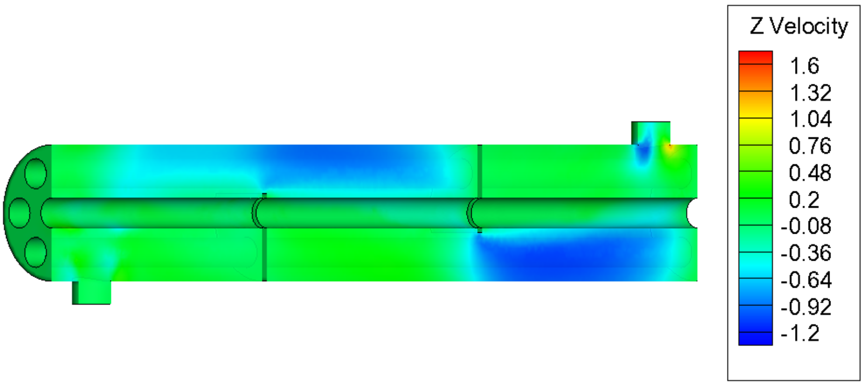

Fig. 3 Velocity contour at x = 0

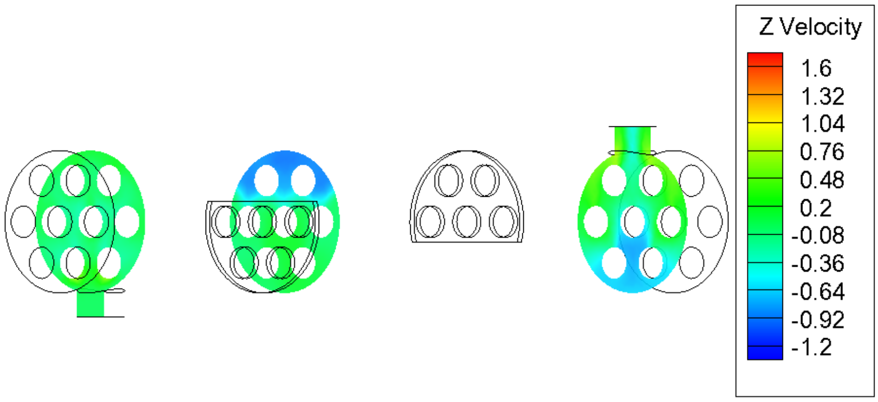

Fig. 4 Velocity Contour at the three selected sections: inlet, (x=L/3), outlet (x=L).

The velocity contour will give us an idea about the flow is especially on the cross sections, to facilitate the work we will give three different situations that present the variation of the distribution of the speed in the exchanger.

The structure of the water flow in the channel is shown in Figs. 3 and 4 which show the contours of the axial velocity.

The negative values of the velocity show in the figures present the opposite direction between the flow and the proposed axis.

From the numerical results it is noted that the values of the velocity are very low in the vicinity of the walls, because of the presence of the strong gradients of friction.

In the annular passage the speed is markedly high compared to the speed in the shell. The difference in speed is due to the difference between the two passage sections on the one hand and the presence of the water recirculation zone by the obstacle which results in a sudden increase in the dynamic pressure of the water.

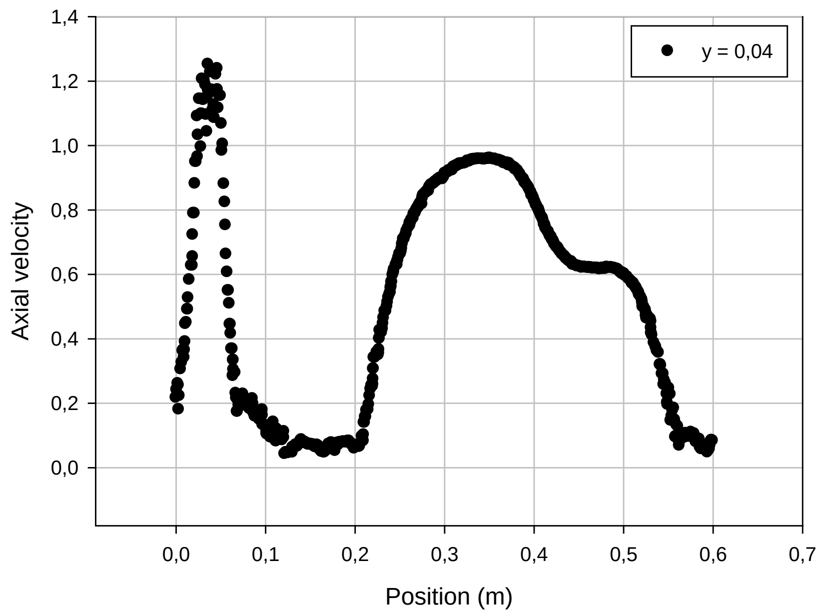

Fig 5: axial velocity profile at y = 0.04.

A presentation of the velocity distribution along the exchanger at the section y = 0.04 m illustrated in this figure. The velocity increase at the level of the passage of the first baffle is decreased because it changes its direction because of the baffle.

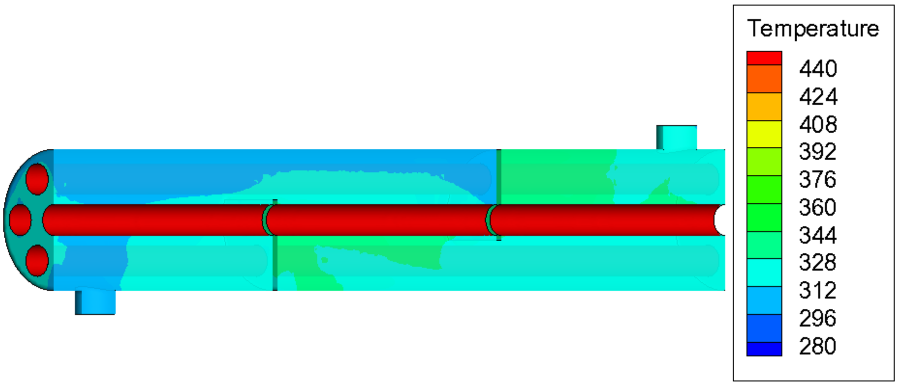

Fig 6: Temperature contours at x = 0.

The distribution of the temperature along the heat exchanger can be seen laterally on the x = 0 plane. The total temperature field shows a low temperature in the first flow passage and an average temperature in the second passage.

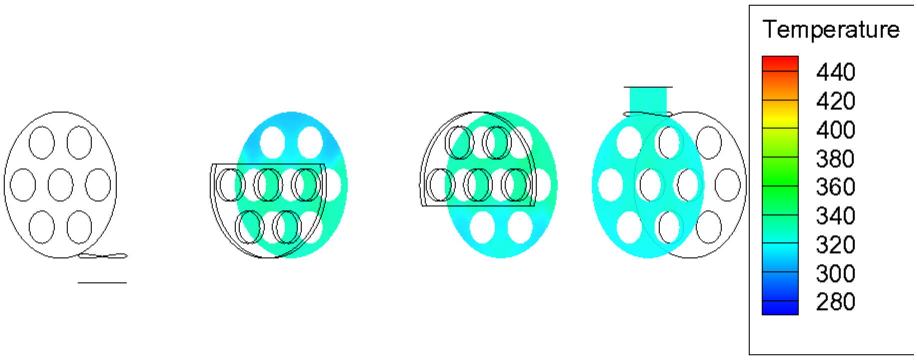

Fig. 7: Temperature contours at z = 0.01, z = 0.03, z = 0.05.

In Figure 7, we can see that the profile of the temperature and therefore the heat transfer is not uniform over the entire length and finds that it is less important near the entrance and higher up close to the outlet of the shell. The hottest areas are located near the wall of the inner tubes.

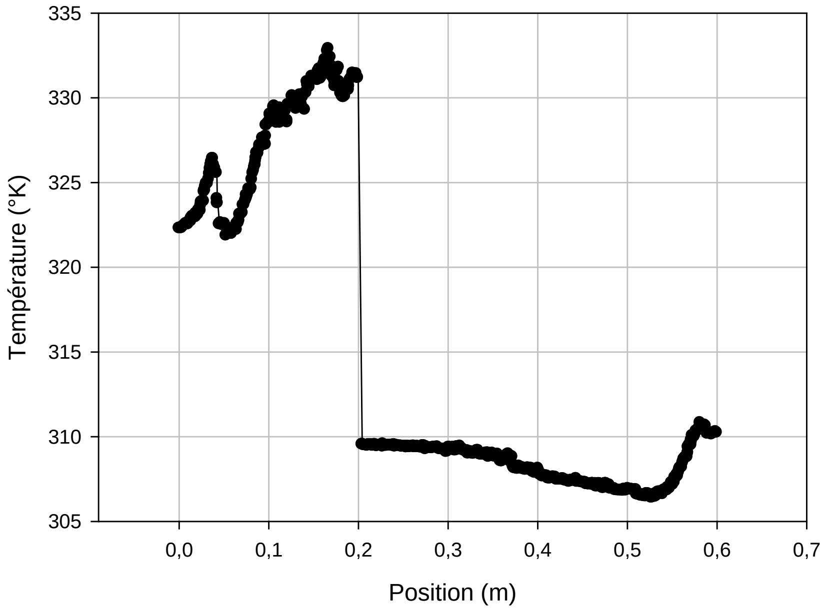

Fig 8. Temperature distribution at y = 0.04.

A presentation of the temperature distribution along the heat exchanger at the section y = 0,04 m illustrated in this figure. The temperature of the fluid increases along the shell in the direction of flow and finds that the higher value located after the second baffle near the outlet. The baffle imposes an increase of the fluid temperature from 310 °K to 332 °K.

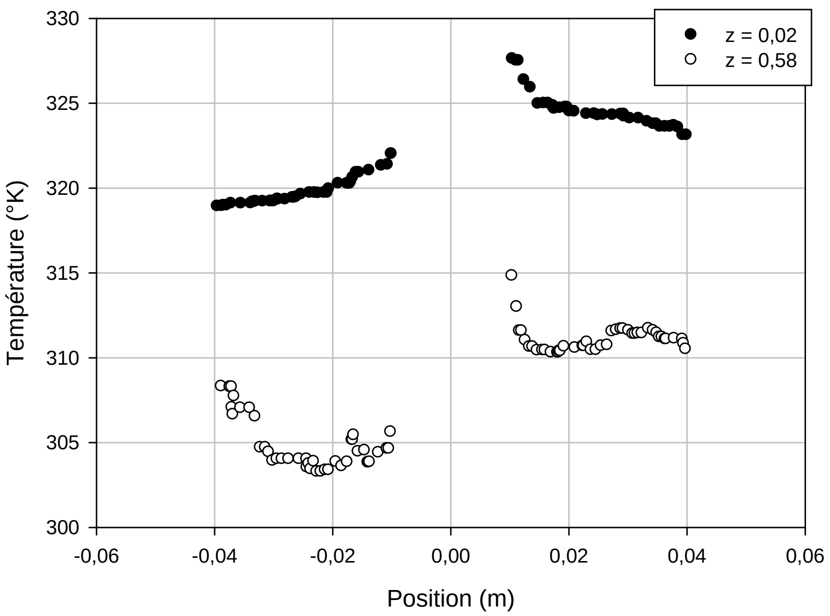

Fig 9: Temperature distribution at z = 0.02 and z = 0.58.

In this figure we have a presentation of the temperature curve at two very important sections z = 0,02m and z = 0,58m. The temperature of the fluid increases near the central tube and decreases when away from that. The temperature of the fluid increases by 312 °K for z = 0, 58 m (inlet) at 324 °K for z = 0,02m (outlet).

Table 3

performance in heat exchanger

Mass flow rate (kg/s)	T (°K)	h (W/m^2°K)	p (pa)	P (W)
0,5	328,47	1235,63	101,17	0,05119
1	325,15	1779,55	407,27	0,412
2	322,1	2564,16	1649,91	3,339

In this table a presentation of the various thermo-hydraulic parameters of the heat exchanger versus the mass flow. The heat transfer coefficient, the pressure drop and the pumping power increase with increasing the mass flow.

With a mass flow various to 0.5 kg/s at 2 kg/s, the heat transfer coefficient increases by 2,075%, the pressure drop increases by 16,31%, the pumping power increases by 65,22%.

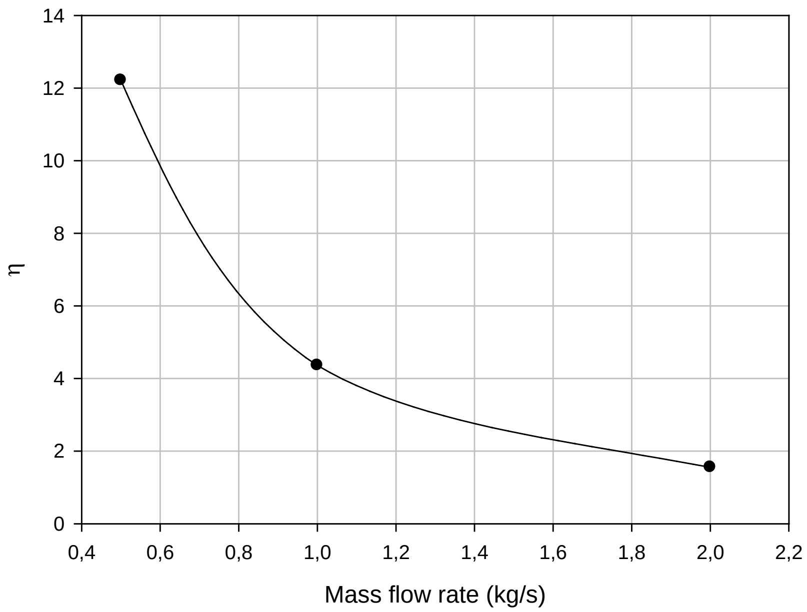

Fig. 10 Overall performance factor

In Fig. 10 shows the change of the performance to different mass flow rates. When the mass flow rate increase the performance decreases because the maximum observed value of the performance index for mass flow 0.5 kg/s is 12.21 means that the heat transfer coefficient is 12.21 times the greater that the pressure drop, at 2kg/s the ratio between these two decisive parameters is 1.58, so the heat transfer coefficient is greater than 1.56 times for the pressure drop. The performance evaluation index decreased because the contact time to the tubes is decreased.

4. Conclusion

The practical realization of a model designed is very expensive, CFD is a tool that allows to simulate processes and thus eliminate the cost of developing a prototype based on the study.

A numerical study of the dynamic and thermal behaviour of turbulent forced convection water flow in a heat exchanger composed of tubes and baffles was presented. The governing equations are solved using the finite volume method and the turbulence model k-ε. The evolution of the axial velocity, the distribution of the temperature in selected sections are presented and analysed for thermal conditions.

An increase in the velocity at the inlet between the inner tubes of the heat exchanger and after the baffles generated first by the presence of the recirculation zones which accompany by a sudden change in the direction of flow of the water. An inverse proportionality between the evolution of the axial velocity of the flow and the distribution of the temperature in each cross section.

R E F E R E N C E S

[1]. *H. Benzenine, R. Saim, S. Abboudi, O. Imine,* « Numerical simulation of the dynamic turbulent flow field through a channel provided with baffles: comparative study between two models of baffles: transverse plane and trapezoidal », Revue des Energies Renouvelables Vol. 13 N°4, p (639 – 651), 2010.

[2]. *Yuna, Z.X., Tao, W.Q. and Wang, Q.W.,* « Experimental investigation of heat transfer enhancement in ducts with winglet fins », First Int. Conference of Engineering Thermophysics, 21-23 August, Beijing, China, p (457-463) ,1999.

[3]. *Acharya, S., Myrum, T. A., qiu, X., and Sinha, S.,* « Developing and Periodically Developed Flow », Temperature and Heat Transfer in a Ribbed Duct, Int. J. Heat Mass Transfer, vol. 40, p (461–479), 1997.

[4]. *Wilfried Roetzel, W.Lee. Deiving* « Effet of baffle/shel leakage flow on heat trasfert in shell-and-tube heat exchanger », Experimental Thermal and Fluid Science, N° 8, p (10-20), 1994.

[5]. *Ahmet Tandiroglu,* « Effect of flow geometry parameters on transient heat transfer for turbulent flow in a circular tube with baffle inserts », , International Journal of Heat and Mass Transfer 49, p (1559-1567), 2006.

[6]. *S.V. Patankar, E. M. Sparrow,* « Fully developed flow and heat transfer in ducts having stream wise-periodic variations of cross-sectional area », Journal of Heat Transfer, Vol. 99, p (180-6), 1977.

[7]. *C. Bemar, F. Durst, DM. Mceligot,* « Numerical study of fluid flow and heat transfer in a parallel plate channel with baffles », Journal of Heat Transfer, p (106-743), 1984.

[8]. *B.W. Webb, S. Ramadhyani,* « Conjugate heat transfer in a channel with staggered ribs», Int. J. Heat Mass Transfer 28, p (1679-1687), 1985.

[9]. *K.M. Kellar, S.V. Patankar,* « Numerical prediction of flow and heat transfer in parallel plate channel with staggered fins », Journal of heat transfer. 109, p (25-30), 1987.

[10]. *Cheng Chin-Hsiang et Huang Wen-Hsiung,* « Numerical prediction for laminar forced convection in parallel-plate channels with transverse fin arrays » , International Journal of Heat and Mass Transfer, Vol. 34, N° 11, p (2739-2749), 1991.

[11]. *Cheng Ch, Huang Wh*. International Journal Heat Mass Transfer, p (1315), 1991.

[12]. *J.R. Lopez, N.K. Anand, L.S. Flecher,* « A numerical analysis of laminar convective heat transfer in a three-dimensional channel with baffles », in: Proceedings of the Third U.S. National Congress on Computational Mechanics, Dallas, TX, vol.1, p (406-416), 1995.

[13]. *J.R. Lopez, N.K. Anand, L.S. Flecher,* « Heat transfer in a three-dimensional channel with baffles », Numerical Heat Transfer.30, p (89–205), 1996.

[14]. *Ender O, Ilker T,* “Shell side CFD analysis of a small shell-and-tube heat exchanger” (), Energy Conversion and Management 51 (1004–1014), 2010.