NUMERICAL ANALYSIS OF A TURBULENT FLOW IN A CHANNEL PROVIDED WITH TRANSVERSAL WAVED BAFFLES

by

Hamidou BENZENINE [a*], ***Rachid SAIM*** [b], ***Said ABBOUDI*** [c], ***and Omar IMINE*** [d]

[a] Department of Mechanical Engineering, Faculty of Mechanical Engineering,
University of Science and Technology, El M´naouar Oran, Algeria
[b] Laboratory of Energetic and Applied Thermal
Faculty of Technology, University Abou Bakr Belkaid, Tlemcen, Algeria
[c]IRTES-M3M, EA, University of Technology of Belfort Montbeliard, Sevenans site-France
[d]Laboratory of Aeronautics and Propulsive System, Faculty of Mechanical Engineering,
University of Science and Technology, M´naouar Oran, Algeria

Original scientific paper
DOI: 10.2298/TSCI111004099B

This article presents a computational analysis of the turbulent flow of air in a pipe of rectangular section provided with two waved fins sequentially arranged in the top and the bottom of the channel wall. The governing equations, based on the k-ε model with low Reynolds number used to describe the turbulence phenomena, are solved by the finite volume method. The velocity and pressure terms of momentum equations are solved by the SIMPLEC algorithm. The profiles of axial velocity, the velocity fields and the drag coefficient were obtained and presented for all the geometry considered and for selected sections, namely, upstream, downstream, and between two waved baffles. This contribution lead to results which were analyzed by the use of the solid, plane baffles, waved and inclined with active degrees of 0° up to 45° with a step equal to 15° and directed towards the left. Over the range of the study, the undulation of the baffles induced with an improvement on the skin friction of about 9.91% in the case of α = 15°, more than 16% in the other cases, and concerning the pressure loss, the undulation of the baffles was insured improvements starter from 10,43% in all cases compared with the baffles of plane form.

Key words: *baffles, channel, turbulent flow, waved inclined baffle, numerical solution, obstacle*

Introduction

Turbulent flow in complex geometries receives considerable attention due to its importance in many engineering applications and has been the subject of interest for many researchers. Some of these include the energy conversion systems found in some design of nuclear reactor, heat exchangers, solar collectors, and cooling of industrial machines and electronic components.

Considerable work has been done, in recent years, on the investigations of the flow and heat transfer processes at the shell-side are of special interest in order to improve the accuracy of prediction of heat exchangers performances. Important works are of particular interest in the improvement and the prediction of the flows around baffles. These studies are devised as well experimental and numerical techniques. An extensive experimental studies of turbulent

* Corresponding author; e-mail: b.hamidou@yahoo.fr

flow and heat transfer past baffles in heat exchangers has been performed by various authors. Wilfried and Deiying [1] examined experimentally turbulent flows throughout tubular heat--exchangers. The authors focused on the impact of the baffles on heat transfer, and the geometrical properties of the heat-exchanger on the overall thermal efficiency. Another experimental study conducted by Rajendra *et al.* [2] investigated heat transfer and friction phenomena in a rectangular asymmetrical channel provided with perforated baffles. The authors managed to raise the Nu by 73.7-82.7% using solid baffles. Another study was reported by Gupta *et al.* [3]. These authors used a helical-shape baffle in a mineral (Carbosep) membrane provided an increase of more than 50% in permeate flux compared with that obtained without a baffle at the same hydraulic dissipated power. Thermal and hydrodynamic parameters were examined numerically and experimentally by Lei *et al.* [4] for a flow passing through a channel with only one helical baffle. A comparative study between three different channels was conducted by the authors. In the first case, a channel without any baffles was examined. In the second case, the same channel with only one helical baffle was considered. In the third case, the same channel with two helical baffles was examined. Ko and Anand [5] have experimentally determined the mean heat transfer coefficients in a rectangular channel with porous obstacles. An important result from this work is that the use of porous baffles resulted in heat transfer enhancement as high as 300% compared to heat transfer in straight channel with no baffles. Tandiroglu [6] examined the effect of the geometric parameters on the steady turbulent flow passing through a pipe with baffles. The effect of the orientation and the distance between nine baffles on the improvement of heat transfer was highlighted in this work. Another experimental investigation was carried out by Molki and Mostoufizadeh [7] to evaluate heat transfer and pressure losses in a rectangular channel with baffles. The authors concluded that baffles raise the pressure losses and the coefficient of heat transfer as well. An investigation of the thermo-hydraulic parameters in a rectangular channel heated up by means of fins perforated to different heights is reported by Rajendra and Maheshwari [8]. The Reynolds number of the study ranges from 2700 to 11,150. The baffled wall of the duct is uniformly heated while the remaining three walls are insulated. The study shows an enhancement of 79-169% in Nusselt number over the smooth duct for the fully perforated baffles and 133-274% for the half perforated baffles while the friction factor for the fully perforated baffles is 2.98-8.02 times of that for the smooth duct and is 4.42-17.5 times for the half perforated baffles. In general, the half perforated baffles are thermo-hydraulically better to the fully perforated baffles at the same pitch. For all the configurations studied, the half perforated baffles at a relative roughness pitch of 7.2 give the greatest performance advantage of 51.6-75% over a smooth duct at equal pumping power. Demartini *et al.* [9] investigated air flow through a rectangular channel with two plate baffles. A comprehensive analysis of the velocity profiles and pressure gradients was carried out in this work. While Demartini's approach used a rectangular channel with plate baffles, the question remains whether baffles of different shapes will achieve the same results. Other authors studied in detail the effect of the size of baffles and orientations on the heat transfer enhancement in the tubes heat exchangers. Nasiruddin and Kamran [10] examined numerically three different orientations of baffles; the first case is a vertical baffle, the second inclined towards the downstream side, and the third inclined towards the upstream side. Also Prashanta and Sandip [11] were conducted an experimental investigation of frictional loss and heat transfer behaviour of turbulent flow in a rectangular channel with uniform flux heating from the upper surface is presented for different sizes, positions, and orientations of inclined baffles attached to the heated surface. Both solid and perforated baffles are used. Inclined perforated baffle combines three major heat transfer techniques, *e. g.* boundary layer separation, internal flow swirls, and jet impingement. Results indicate the existence of an optimum perforation density to maximize heat transfer coefficients and this optimum perforation exhibits a strong jet impingement technique from the lower confined channel along with other enhancement techniques of heat transfer.

The majority of the previous studies were devoted to the dynamic and thermal behaviour of a flow in a channel provided with the planes generators of vortex. However, none the significant efforts were done to inquire the effect of the slope and the shape of the vortex generator on the dynamic structure of the flow and the reinforcement of transfer of heat. This is the aim of the present study in which the behaviour of an air turbulent flow inside channel with a rectangular section, containing two baffles of different shapes (plane and undulated) are studied. The characterization of the flow field in this channel is the intention to the comprehension of the dynamic behaviour presented by the velocity and variation of the friction factor in the channel. The pressure losses in the channel deduced from the drag coefficient are also treated in this article for two shapes and for different inclinations of waved baffles.

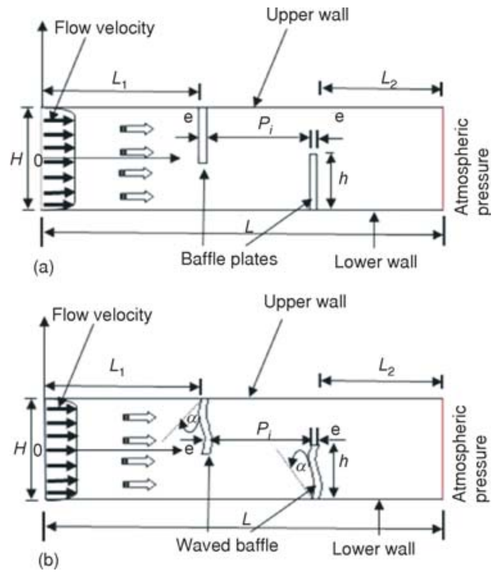

Figure 1. The geometry of the system under investigation; (a) plate baffles (b) waved baffles

Mathematical formulation

Problem statement

The geometrical configuration is presented in fig. 1(a) and (b). The system consists of air flow moving through a rectangular channel provided with two baffles. Two different shapes of baffles are used in this study, a first is plate baffle, fig. 1(a), and a second is waved, fig. 1(b), the corrugated baffle shape according to a local axis system, (fig. 2), is sinusoidal and is the following function: $x = A\sin 2\pi y/h$ where $A = 0.004$ m and $h = 0.08$ m.

In this numerical investigation, the following hypotheses are adopted:
- physical properties of air are constant,
- a profile of velocity is uniform at the inlet,
- the flow is assumed to be steady, and
- the flow is assumed to be 2-D.

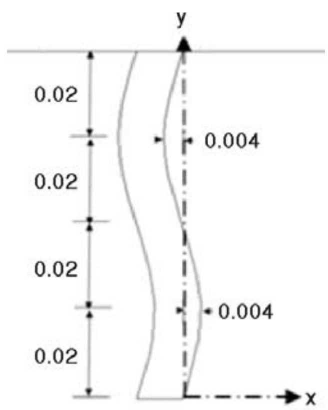

Figure 2. Dimensions of the corrugated baffle used in the problem (dimensions in m)

Governing equations

The general transport equation that describe the principle of conservation of mass and momentum can be written in the following conservative form Patankar [12]:

$$\frac{\partial}{\partial x}(\rho u\phi)+\frac{\partial}{\partial y}(\rho v\phi)=\frac{\partial}{\partial x}\left[\Gamma_\phi \frac{\partial \phi}{\partial x}\right]+\frac{\partial}{\partial y}\left[\Gamma_\phi \frac{\partial \phi}{\partial x}\right]+S_\phi \quad (1)$$

where ϕ stands for the dependent variables u, v, k, and ε, u, and v stand for the mean velocities towards the x- and y-axis, respectively, k and ε stand for kinetic energy and turbulent dissipation, respectively, Γ_ϕ and S_ϕ are the corresponding diffusion coefficient and source term, respectively, for general variable ϕ.

The expressions of ϕ, Γ_ϕ, and S_ϕ are presented for:

– *the continuity equation*

$$\phi = 1 \tag{2}$$

$$\Gamma_\phi = 0 \tag{3}$$

$$S_\phi = 0 \tag{4}$$

– *the momentum equation in x-direction*

$$\phi = u \tag{5}$$

$$\Gamma_\phi = \mu_{\mathrm{e}} \tag{6}$$

$$S_\phi = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left[\mu_e\left(\frac{\partial u}{\partial x}\right)\right] + \frac{\partial}{\partial y}\left[\mu_{\mathrm{e}}\left(\frac{\partial v}{\partial x}\right)\right] \tag{7}$$

– *the momentum equation in y-direction*

$$\phi = v \tag{8}$$

$$\Gamma_\phi = \mu_e \tag{9}$$

$$S_\phi = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left[\mu_{\mathrm{e}}\left(\frac{\partial u}{\partial y}\right)\right] + \frac{\partial}{\partial y}\left[\mu_{\mathrm{e}}\left(\frac{\partial v}{\partial y}\right)\right] \tag{10}$$

– *the turbulent energy equation*

$$\phi = k \tag{11}$$

$$\Gamma_\phi = \mu_\ell + \frac{\mu_{\mathrm{t}}}{\sigma_k} \tag{12}$$

$$S_\phi = -\rho\varepsilon + G \tag{13}$$

– *the turbulent dissipation equation*

$$\phi = \varepsilon \tag{14}$$

$$\Gamma_\phi = \mu_\ell + \frac{\mu_{\mathrm{t}}}{\sigma_\varepsilon} \tag{15}$$

$$S_\phi = (C_1 f_1 G - C_2 f_2 \rho\varepsilon)\frac{\varepsilon}{k} \tag{16}$$

where, for a 2-D, the flow production term becomes:

$$G = \mu_t\left[2\left(\frac{\partial u}{\partial x}\right)^2 + 2\left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)^2\right] \tag{17}$$

$$\mu_{\mathrm{e}} = \mu_\ell + \mu_{\mathrm{t}} \tag{18}$$

$$\mu_{\mathrm{t}} = f_\mu \rho C_\mu \frac{k^2}{\varepsilon} \tag{19}$$

The turbulent constants correspond to those suggested by Launder and Spalding [13] and Chieng and Launder [14]. These constants are arranged in tab. 1.

Low-Reynolds number (LRN) k-ε model:

While basing itself on the study of Lam and Bremhorst [15], the LRN k-ε model used in this paper is described by Versteeg and Malalasekera [16]. The modelling damping functions f_1, f_2, and f_μ used for the k-ε model based on the LRN are presented as:

Table 1 .Turbulent constant in the governing equations

C_μ	$C_1 = C_3$	C_2	σ_k	σ_ε
0.09	1.44	1.92	1	1.3

$$f_\mu = [1 - \exp(-0.0165 R_y)]^2 \left(1 + \frac{20.5}{R_T}\right) \tag{20}$$

$$f_1 = 1 + \left(\frac{0.05}{f_\mu}\right)^3 \tag{21}$$

$$f_2 = 1 - \exp(-R_T^2) \tag{22}$$

where

$$R_T = \frac{\rho k^2}{\varepsilon \mu} \quad \text{and} \quad R_y = \frac{\rho \sqrt{k}\, y}{\mu} \tag{23}$$

The damping function (f_μ), which is a function of dimensionless wall-normal distance $R_y = k^{1/2} y/\nu$, is used to model the damping effect associated with pressure-strain correlations in the vicinity of walls.

Boundary conditions

A turbulent flow is considered. The quantities U, k, ε are obtained by using numerical calculations based on the k-ε model for low Re.

The boundary conditions are:

– at the inlet of the channel

$$u = U_{in}, \quad v = 0 \tag{24}$$

$$k_{in} = 0.005\, U_{in}^2 \tag{25}$$

$$\varepsilon_{in} = 0.1 k_{in}^2 \tag{26}$$

k_{in} stands for the admission condition for turbulent kinetic energy and ε_{in} is the inlet condition for dissipation.

– at the walls

$$u = v = 0 \tag{27}$$

$$k = \varepsilon = 0 \tag{28}$$

– at the exit

$$p = p_{atm} \tag{29}$$

The relation ship between the drag coefficient and the pressure loss is defined as:

$$Cd = \frac{2\Delta p}{\rho u_0^2}\left(\frac{A_f}{A_0}\right) \tag{30}$$

Numerical resolution

The computer code FLUENT is used to calculate the fluid flow the computational domain, the governing equations of problem are solved by the finite volume method (FVM), based on the algorithm SIMPLEC [17], for the coupling pressure-velocity. Taking into account the characteristics the flow, the quick numerical scheme is applied to the interpolations. The iterative solution is continued until the residuals for all cells of calculation have become less than 10^{-5} for all dependent variables.

A structured grid element with the quadrilateral type is used because it considered being more adequate for the geometry suggested. Numerical simulations are tested by varying the number of elements of mesh. Stability of convergence of the model is achieved for all meshes.

Results and discussion

The obtained results are presented in this section. In the first, a comparison of the independence of the mesh and its influence on the results have been presented on the computational domain studied by Demartini *et al*., [9] who conducted an experimental study on a flat baffle. Taking to consideration their conditions, a comparison is made to validate our numerical results. After, the effect of corrugated baffle and its inclination on the dynamic behavior of the flow are investigated.

Sensitivity analysis of the mesh

A non-uniform mesh of (195 × 82) in the vertical and horizontal directions proved to be sufficient to modelling the system. The meshing size is comparatively small near the boundaries and the baffles (fig. 3), so a good estimate of the gradients can be obtained. Many meshing sizes (N_x, N_y) are tested to ensure the grid independence of results. The choice of this mesh is justified by the fact that the difference between the values found is less than 1%.

Figure 3. Mesh generated around the corrugated baffles vicinity

Validation of the model

For the numerical simulations presented in this work, we refer to the numerical and experimental work done by Demartini *et al.* [9] who studied the baffles with a plane shape. The geometric dimensions of the system are listed below:

- length of the channel $L = 0.554$ m,
- height of the channel $H = 0.146$ m,
- thickness of the baffle $e = 0.08$ m,
- distance between baffles $P_i = 0.142$ m,
- distance between the intake of the channel and the first baffle $L_1 = 0.218$ m,
- distance between the second baffle and the exit of the channel $L_2 = 0.174$ m,
- hydraulic diameter $D_h = 0.167$ m, and
- velocity of air particles at the inlet $U_0 = 7.8$ m/s.

Figures 4 and 5 show our results and those obtained by Demartini *et al.* [9], it represents the profiles of velocity at $X = 0.159$ and $X = 0.525$, respectively at Re equal to $8.73 \cdot 10^4$. It is clear that the results are in good agreement; this allows us to validate our numerical modeling.

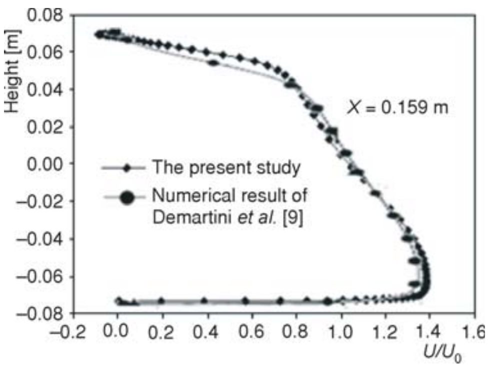

Figure 4. Comparison between the modeling results and Demartini *et al.* [9] results for $X = 0.159$

Figure 6 shows the axial velocity contours for the different cases studied in this paper at Re = $= 8.73 \cdot 10^4$. For rectangular baffles, fig. 6(a) is the same geometry studied by Demartini *et al*. [9], the other contours (fig. 6b-c) are those of waved baffles and inclined baffles with active degrees α = 0°-45 °.

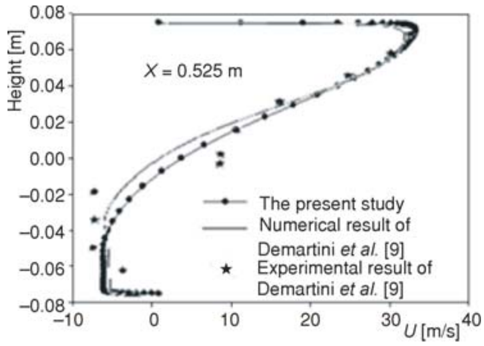

Figure 5. Comparison between the modeling results and Demartini *et al.* [9] results for X = 0.525

It is indicates clearly that the values of velocity are very low in the vicinity of the two baffles especially in the areas located downstream. This is characterized by the presence of the zones of re-circulation. One notices also the increase velocity in space between the tip of each baffle and the opposite walls of the channel. In the first part, this increase is generated by the singularity represented by the obstacles, also in the second part by re-circulation of flow which then results in an abrupt change of the direction of the fluid. It is also noticed that the elevated values of velocity appear close the top of the channel with a process of acceleration which starts just after the second baffle.

Dynamic configuration of the flow changes from one case to another, whatsoever with the change in the form of baffles or change the angle of inclination for corrugated baffle. For studying this dependence well, we plotted the velocity distribution for these sections: $X = 0.159$, $X = 0.255$, $X = 0.285$, $X = 0.310$, and $X = 0.525$ m.

The presence of the first baffle which is situated in the higher half of the channel induces in this region a strong reduction of velocity, paradoxically in the lower half, where an increase in the flow and especially in the vicinity of the passage under the baffle is provoked. It is also observed that the enhance of the baffle inclination angle on the left results in an enlarging of space between the top of the first baffle and the walls inferior what results then an increase velocity

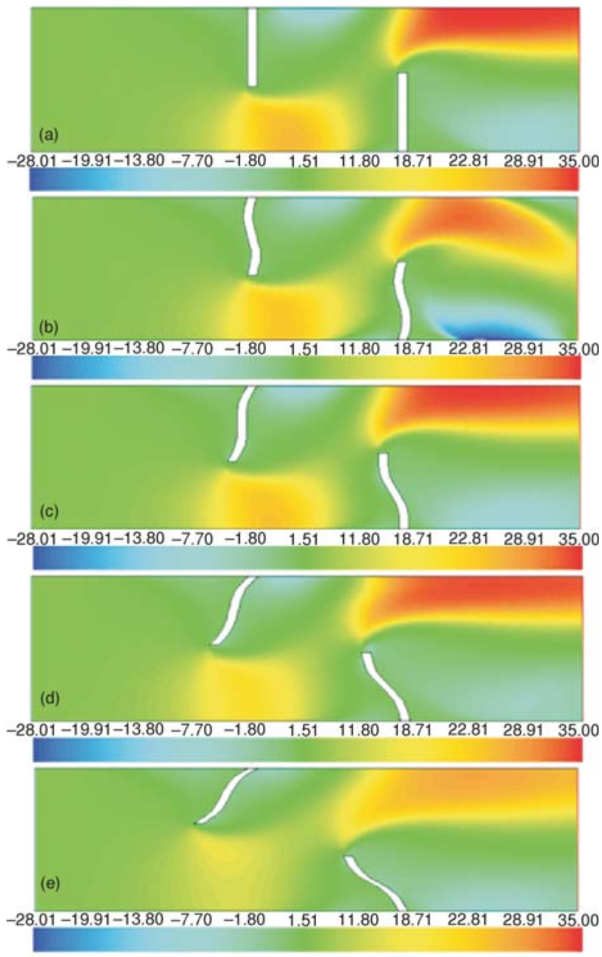

Figure 6. Axial velocity field in channel for two forms and for the orientations studied; (a) plan, (b) α = 0°, (c) α = 15°, (d) α = 30 °, (e) α = 45°

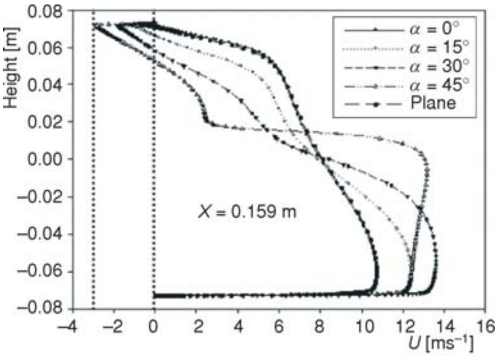

Figure 7. Profiles of axial velocity upstream of the first baffle for various studied cases

in the vicinity of the breach. This is clearly observed on fig. 7. Most surprising and impressive in this study is the effect of variation of the baffle angle inclination on the dynamic behaviour of the fluid even upstream the baffle.

In the intermediate zone, at positions equal to 0.255 m and 0.285 m of the entry, the flow is characterized by very high velocities in the lowest part of the channel. It can be seen that the maximum of velocity varies from one case to another.

In the inferior part of the channel, by comparing fig. 7 with figs. 8 and 9, it is noted that the acceleration of the fluid for the various cases tested upstream of the first baffle is relatively reversed downstream from it and this observation is due to the sensitivity of the dynamic behaviour of the fluid to the second baffle.

In the higher part of the channel, negative velocities indicate the presence of re-circulation of the flow behind the first baffle. In this zone, it may be that the reduction in the angle of inclination of the baffle towards the left decreases the size of the breach and results in an increase from the value axial velocity in the two directions (positive and negative) and thus an increase in size in the generated vortex.

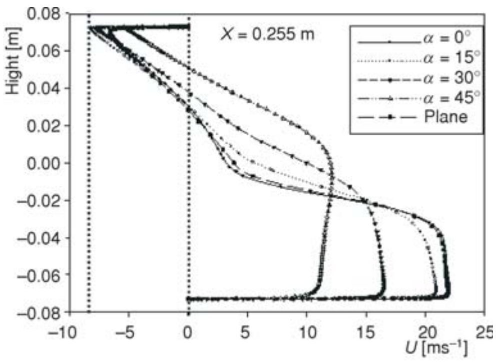

Figure 8. Profiles velocity between the first and the second baffle corresponds studied to X = 0.255 m for various cases

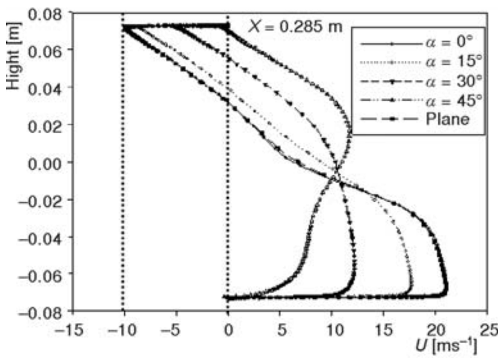

Figure 9. Profiles velocity between the first and the second baffle corresponds studied to X = 0.285 m for various cases

Figure 10 shows the velocity profiles at position given by X = 0,310 m *i. e.* to 0,06 m before the second baffle. At this section, the results show that for all the treated cases, velocity is reduced in the lower part of the channel, while in the higher part is increased. This acceleration of the fluid noticed towards this free segment above the second baffle is readable in both cases $\alpha = 30°$ and $\alpha = 45°$, while for the others are still in the change of the direction of the flow (zone of recycling), therefore one to say little that the use of two baffles purely vertical (plane or corrugated) results in a significant delay, then it offers time necessary to us in order to ensure the heat exchange desired by contribution the other steepness.

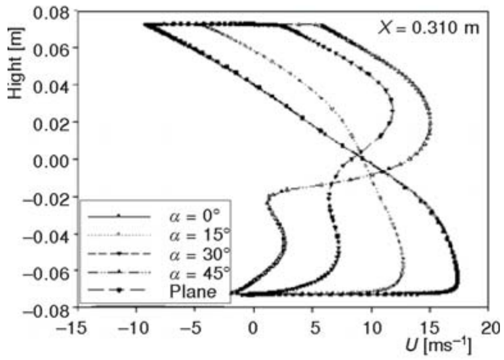

Figure 10. Profiles velocity upstream of the second baffle for various studied cases

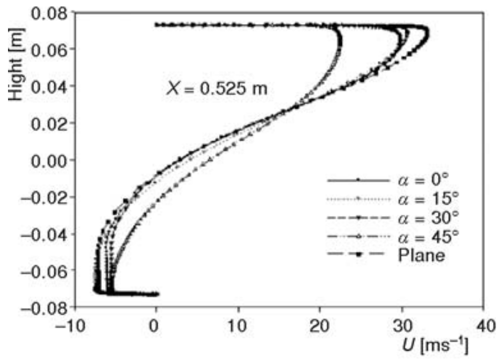

Figure 11. Profiles velocity at the exit to the channel for various studied cases

Profile velocity after the second undulated baffle, close to the exit of the channel is presented on fig. 11 at the position X = 0,525 m, *i. e.* to 29 mm before the exit of the channel. The values of velocities for all the studied cases are higher than velocity inlet. These values are generated by the presence of the strong re-circulation of the flow in the back face of the second baffle. Strong variation of velocity is characterised by the negative and positive values. Table 2 presents the relationship between the maximum axial velocity at the exit of the channel and the velocity of reference for the five cases of the baffle used in this work. The results prove that in a rectangular channel provided with two plane or undulated baffles, vertical or inclined with a going angle of 0° up to 30° degree, one ensures that velocity at the exit measures more than three times and half the velocity of reference.

Table 2. Comparison maximum axial velocity at the exit section with the reference velocity for all the treated cases

Case	U_0 [ms^{-1}]	$U_{\text{max exit}}$ [ms^{-1}]	$U_{\max}/U_0$
Plane	7.8	34.37	4.41
α = 0°	7.8	27.90	3.58
α = 15°	7.8	31.41	4.03
α = 30°	7.8	28.55	3.66
α = 45°	7.8	21.97	2.82

According to fig. 12, it is apparent that the coefficient of friction along the two walls varies slightly upstream the first baffle whatever is the shape or the angle of baffle. Indeed, this coefficient increases between the two baffles and in the exit of the channel. It may be, because the high velocity of the flow at the exit. Taking for example fig. 12(a), appears which shows the variation of the friction coefficient along the high wall, of which the maximum value in the case of the plane baffles equal to 1.45 whereas in the cases of the baffles undulated for α = 0°, α = = 15°, α = 30°, α = 45° we finds, respectively, maximum values of friction coefficient equal to 0.96, 1.25, 1.00, and 0.63 differently say reductions of 33,55%, 14,14%, 31,34%, and of 56,58% compared to the plane baffles.

Therefore, the use of corrugated walls can lead to a slight variation of the coefficient of friction and provides an improvement over the pressure losses.

On the one hand, the use of the baffles is significant for the improvement of heat exchange but in return, the presence of the baffle in the flow causes an additional pressure drop. So this is a duality to controlling between the thermal performance and the hydraulic performance. It is noted that the undulation of the walls of baffles reduced the action flow because the curve in the wall corrugated increases the factor of slip, and the curve of the baffle used is in the same direction of the flow. Thus, it finds that the plane form of baffles induced a higher pressure loss.

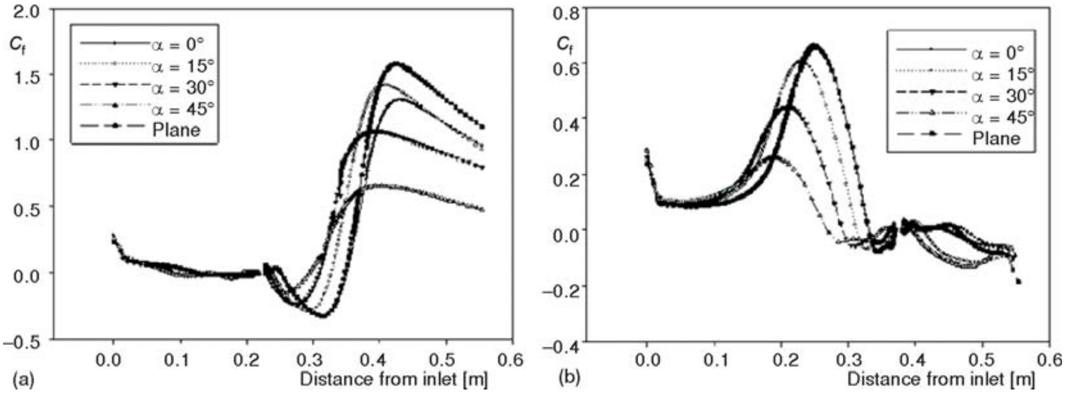

Figure 12. Variation of friction coefficient along the two walls of the channel for all treated cases (a) Top wall of channel, (b) Lower wall of channel

The study of dynamics of fluid in the geometries mentioned, involves two resistances: resistance due to viscous friction on the canal walls and obstacles and resistance form of the obstacle is due to the pressure difference between upstream and downstream of the obstacle. Both introduce the concept of resistance coefficient of drag. Table 3 shows the drag coefficients for all the cases studied. The undulating shape of the baffles ensures a significant reduction in losses pressure comparing with the flat shape of baffle.

Table 3. Drag coefficients and theirs improvements obtained for various treated cases

Case	Plane	α = 0°	α =15°	α =30°	α =45°
Drag coefficient (C_d)	19.54	16.76	17.50	13.60	7.03
C_d improvement		14.21%	10.43%	30.38%	64.00%

Conclusions

A numerical investigation to study a turbulent flow of air through a rectangular section was conducted. Two baffles were introduced into the field to produce vortices, to improve the mixture and thus, the transfer of heat. The effects of the form of baffles (plane and corrugated) and of orientation on the dynamic behaviour of the flow were studied in detail.

The numerical results, obtained by the FVM, are validated and presented to analyze the dynamic behaviour of a turbulent flow using the low Reynolds number k-ε model. The profiles and the distribution of axial velocities show a relatively intense re-circulation zone above the facets of each baffle from left on the right of the channel. The highest disturbance is obtained upstream second baffle.

This study showed that the undulation of the baffles induced with an improvement on the skin friction of about 9.91 % in the case of α =15°, more than 16% in the other cases. Concerning the pressure loss the undulation of the baffles was insured improvements starter from 10,43% in all cases compared with the baffles of plane form.

We have also tested four cases of slopes for the corrugated baffles going from 0° up to 45°, with a step equal to 15°. It may be concluded that the purely vertical use of the waved baffles (α = 0) in the geometry studied, ensures the optimal size of the zone of re-circulation and thus necessary time for guarantee the improvement of heat exchange. Also this case ensures us a very high velocity in the exit of the channel, measures more than four times the reference velocity, and most significant is to reduce less the action flow induces on the pressure losses.

Nomenclature

A – amplitude
A_f – fluid flow cross-section
A_0 – wetted surface
C_1 – constant used in the standard k-ε model
C_2 – constant used in the standard k-ε model
C_μ – constant used in the standard k-ε model
e – width of baffles, [m]
f_1, f_2, f_μ – the modelling damping functions used for the LRN k-ε model
G – the flow production term
H – height of air tunnel in pipe, [m]
h – baffle height, [m]
k – turbulent kinetic energy, [m^2s^{-2}]
L – channel length, [m]
L_1 – distance upstream of the first baffle, [m]
L_2 – distance downstream of the second fin, [m]
p – pressure, [Pa]
P_i – distance between two baffles, [m]
Re – Reynolds number $(= \rho D_H U_0/\mu)$
R_T, R_y – constant used in (LRN) k-ε model
S_ϕ – limit of source for the general variable
U_0 – inlet velocity, [ms^{-1}]
u, v – fluid velocity in the x- and y-direction, [ms^{-1}]

Greek symbols

Γ – diffusion coefficient
ε – dissipation rate of turbulence energy, [m^2s^{-1}]
μ_l, μ_t – laminar, turbulent viscosity, [Pa·s]
μ_e – effective viscosity, [Pa·s]
ν – kinematics viscosity, [m^2s^{-1}]
ρ – density of the air, [kgm^{-3}]
ϕ – stands for the dependent variables u, v, k and ε

Subscripts and Superscripts

e – effective
f – fluid
in, out – inlet, outlet of the test section
t – turbulent
w – wall

References

[1] Wilfried, R., Deiying, W. L., Effect of Baffle/Shell Leakage Flow on Heat Transfer in Shell-And-Tube Heat Exchanger, *Experimental Thermal and Fluid Science, 8* (1994), 1, pp. 10-20
[2] Rajendra, K., *et al.*, Experimental Study of Heat Transfer Enhancement in an Asymmetrically Heated Rectangular Duct with Perforated Baffles, *International Communications in Heat and Mass Transfer, 32* (2005), 1-2, pp. 275-284
[3] Gupta, B. B., *et al.*, A Helical Baffle for Cross-Flow Micro Filtration, *Journal of Membrane Science, 102* (1995), 6, pp. 31-42
[4] Lei, Y.-G., *et al.*, Design and Optimization of Heat Exchangers with Helical Baffles, *Chemical Engineering Science, 63* (2008), 17, pp. 4386-4395
[5] Ko, K.-H., Anand, N. K., Use of Porous Baffles to Enhance Heat Transfer in a Rectangular Channel, *International Journal of Heat and Mass Transfer, 46* (2003), 22, pp. 4191-4199
[6] Tandiroglu, A., 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* (2006), 9-10, pp. 1559-1567
[7] Molki, M., Mostoufizadeh, A. R., Turbulent Heat Transfer in Rectangular Ducts with Repeated-Baffle Blockages, *International Journal of Heat and Mass Transfer, 32* (1989), 8, pp. 1491-1499
[8] Rajendra, K. , Maheshwari, B. K., Heat Transfer and Friction in an Asymmetrically Heated Rectangular Duct with Half and Fully Perforated Baffles at Different Pitches, *International Communications in Heat and Mass Transfer, 36* (2009), 3, pp. 264-268
[9] Demartini, L. C., *et al.*, Numeric and Experimental Analysis of the Turbulent Flow through a Channel with Baffle Plates, *J. of the Braz. Soc. of Mech. Sci. & Eng, 26* (2004), 2, pp.153-159
[10] Nasiruddin, M. H., Kamran, S., Heat Transfer Augmentation in a Heat Exchanger Tube Using a Baffle, *International Journal of Heat and Fluid Flow, 28* (2007), 2, pp. 318-328
[11] Prashanta., D., Sandip., D., Effect of Baffle Size, Perforation, and Orientation on Internal Heat Transfer Enhancement, *International Journal of Heat and Mass Transfer, 41* (1998), 19, pp. 3005-3013
[12] Patankar, S. V., *Numerical Heat Transfer and Fluid Flow*, Hemisphere, New York, USA, 1980

[13] Launder., B. E., Spalding., D. B., The Numerical Computation of Turbulent Flow, *Computer Methods in Applied Mechanics and Engineering, 3* (1974), 2, pp. 269-289
[14] Chieng, C. C., Launder, B. E,. On the Calculation of Turbulent Heat Transport Downstream from an Abrupt Pipe Expansion, *Numerical Heat Transfer, 3* (1980), 2, pp. 189-20
[15] Lam, C. K .G., Bremhorst, K., A Modified Form of the k-ε Model for Predicting Wall Turbulence, *ASME J. Fluids Engng., 103* (1981), 3, pp. 456-460
[16] Versteeg, H. K., Malalasekera, W., An Introduction to Computational Fluid Dynamics, the Finite Volume Method, Addison Wesley Longman Limited, London, 1995
[17] van Doormal, J. P., Raithyb, G. D., Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flow, *Numerical Heat Transfer, 7* (1984), 2, pp. 147-163

Paper submitted: October 4, 2011
Paper revised: April 28, 2012
Paper accepted: May 26, 2012