Numerical Analysis Of Turbulent Forced-Convection Flow In A Channel With Staggered L-Shaped Baffles

Y. Menni, A. Azzi, C. Zidani and B. Benyoucef

Unit of Research on Materials and Renewable Energies - URMER -
Department of Physics, Faculty of Sciences, Abou Bekr Belkaid University,
BP 119-13000-Tlemcen, Republic of Algeria

Abstract
Characteristics of fluid flow and heat transfer are analyzed for a constant property fluid flowing turbulently through a two-dimensional horizontal rectangular cross section channel with staggered, transverse L-shaped baffles (STLBs) and a constant temperature along both walls. The Commercial CFD software FLUENT 6.3 is used to simulate the fluid flow and heat transfer fields. Air is the working fluid with the flow rate in terms of Reynolds numbers ranging from 12,000 to 30,000. The effects of the baffle L-shape as well as Reynolds numbers are examined. A detailed description of turbulent heat transfer flow behaviors around the STLBs was presented. In particular, contour plots of velocity and pressure fields, axial velocity profiles, local and average heat transfer coefficients, and friction loss evaluations were obtained at constant wall temperature condition along the top and bottom channel walls. The numerical results are validated with available rectangular-baffle measured data and found to agree well with measurement. The results reveal essentially, that the flow pattern of using STLBs is characterized by strong deformations and large recirculation regions. The highest values in the velocity and pressure fields are found near the top channel wall with an acceleration process that starts just after the second STLB. Also, an increase in the Reynolds number causes a substantial increase in the Nusselt number but the pressure loss is also very significant.

Keywords CFD ; Finite volume method ; Forced-convection ; L-shaped baffle ; Rectangular channel ; Turbulent flow.

1. Introduction
The use of baffles and fins in channels is commonly used for passive heat transfer enhancement strategy in single phase internal flow. Considering the rapid increase in energy demand, effective heat transfer enhancement techniques have become important task worldwide. Some of the applications of passive heat transfer enhancement strategies are in process industries, thermal regenerator, Shell-and-tube type heat exchanger, Internal cooling system of gas turbine blades, radiators for space vehicles and automobiles, etc. In literature, numerous studies on baffled channel heat transfer are reported, but only the relevant articles are cited here. Yuan [1] reported a numerical study for the characteristics of the periodically fully developed turbulent flow and heat transfer in a channel with transverse opposite-positioned fins. The influence of the thermal boundary condition of the fin to the heat transfer was verified. Yuan and others also studied experimentally the duct with periodic rectangular fins along the main flow direction [2] and the duct with winglet fins [3]. They can both increase heat transfer largely comparing with smooth duct. An experimental investigation was done by Habib et al. [4] to study the characteristics of the turbulent flow and heat transfer inside the periodic cell formed between segmented baffles staggered in a rectangular duct. The parameters of the experimental work were the Reynolds number and the baffle height. The results indicated that the pressure loss increases as the baffle height does, for a given flow rate. Also, the local and average heat transfer parameters increase with increasing Reynolds number and baffle height. Demartini et al. [5] presented the numeric and experimental analysis of the turbulent flow of air inside a channel of rectangular section, containing two rectangular baffle plates. Hot wire anemometry and the Finite Volume Method, by means of commercial program FLUENT 5.2 were applied in that research work. Tsay et al. [6] numerically investigated the heat transfer enhancement due to a vertical baffle in a backward-facing step flow channel. The effect of the baffle height, thickness and the distance between the baffle and the backward facing step on the flow structure was studied in detail for a range of Reynolds number varying from 100 to 500. They found that an introduction of a baffle into the flow could increase
In those studies, different aspect ratio channels augmentation both experimentally [15] and numerically perforated baffle concept for internal cooling. Later on, a number of research groups have utilized the duct and is 4.42-17.5 times for the half perforated baffles. The authors showed an enhancement of 79-169% in Nusselt number over the smooth duct for the fully perforated baffles. A combination of two baffles of same overall size was used in their experiment. The upstream baffle is attached to the top heated surface, while the position, orientation, and the shape of the other baffle are varied to identify the optimum configuration for enhanced heat transfer. A constant surface heat flux was applied from the top surface, but the bottom and the side surfaces were maintained at an adiabatic condition. The experimental results showed that the local Nusselt number distribution is strongly depended on the position, orientation, and geometry of the second baffle plate. The friction factor ratio goes up with an increase in the Reynolds number, but its value depends on the arrangement of baffles. Karwa and Maheshwari [14] presented results of an experimental study of heat transfer and friction in a rectangular section duct with fully perforated baffles (open area ratio of 46.8%) or half perforated baffles (open area ratio of 26%) at relative roughness pitch of 7.2-28.8 affixed to one of the broader walls. The authors showed 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. Later on, a number of research groups have utilized the perforated baffle concept for internal cooling augmentation both experimentally [15] and numerically [16-18]. In those studies, different aspect ratio channels and different porosity baffles were used. Other authors studied in detail the effect of the shape of baffles and orientations on the heat transfer enhancement in the heat exchanger channels. Laminar periodic flow and heat transfer in a two dimensional horizontal channel with isothermal walls and with staggered diamond-shaped baffles was investigated numerically by Sripratanapipat and Promvong [19]. Wang et al. [20] summarized computational and experimental results for research on the flow and heat transfer process of a rectangular channel embedded with staggered pin fins of various shapes (i.e., circular, elliptical, and drop-shaped) in a staggered arrangement. Guerroudj and Kahalerras [21] reported a numerical simulation of laminar mixed convective in a two-dimensional parallel plate channel provided with porous blocks of various shapes (i.e., rectangular, trapezoidal, and triangular-shaped). Benzenine et al. [22] presented 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 influence of baffle turbulators on heat transfer augmentation in a rectangular channel with Z-shaped baffles was investigated experimentally and numerically by Srirumreun et al. [23]. Numerical and experimental predictions of the flow and heat transfer in shell-and-tube heat exchangers helical baffles were investigated by Lei et al. [24], Dong et al. [25], and Wen et al. [26]. Other similar works can be found in literature as Promvonge [27,28], Promvonge and Kwankaomeng [29], Tamna et al. [30], and Jedsadaratanachai et al. [31] studied the heat transfer and flow over V-shaped baffles submitted to laminar and turbulent flows using numerical and experimental techniques. All these shapes increase the thermal transfer rate but created catastrophic pressure losses. The principal objective of the present study is to show the influence of STL Bs on the fluid flow and heat transfer characteristics when the Reynolds number effects are simultaneously present. This was decided after literature search has revealed that no work has been reported on the computation of the flow in L-baffled rectangular channels. This has motivated the present numerical simulation which is a contribution to the previous studies on improvement techniques of heat transfer. Throughout the study, a constant temperature is assumed from the entire wall of the computational domain. The simulation results of an L-shaped obstacle pair are also compared with conventional CFD simulations and the experimental data of previous researchers.
2. Physical model

The system of interest is a two-dimensional isothermal wall rectangular cross section channel with a staggered transverse L-shaped baffle pair placed on both the upper and lower walls in staggered arrangement and pointing towards the upstream end as shown in Figure 1. A schematic view of the baffle shape is shown in Figure 2. The flow is two-dimensional, turbulent, incompressible and in steady state with no internal heat generation and neglecting viscous dissipation. The Prandtl number is taken equal to 0.71. All physical properties of the fluid and solid are considered to be constant.

The geometric dimensions of our problem have been based on the experimental work of Demartini et al. [5]. In their study, the experiment was conducted in a two-dimensional domain, which represents a rectangular duct of L = 0.554 m long and H = 0.146 m high, provided by two baffle plates, through which a steady flow of turbulent air. The first plate is attached to the top wall at a distance of Lin = 0.218 m and the second inserted to the bottom wall at Lout = 0.37 m from the entrance. The distance between the upper edge of the baffle and the wall was kept constant at h = 0.08 m. This corresponds to the area reduction of 54.794 % at the baffle edge. The thickness of the two baffles is a e = 0.01 m.

3. Mathematical modeling

Based on the above assumptions, the channel flow is governed by continuity, Navier-Stokes and energy equations, respectively

\[ \nabla \cdot \mathbf{V} = 0 \]  
\[ \rho \left( \mathbf{V} \nabla \mathbf{V} \right) = -\nabla P + \mu \nabla^2 \mathbf{V} \]  
\[ \rho C_p \left( \mathbf{V} \nabla T \right) = K \nabla^2 T \]  

where: \( \mathbf{V} \) is the velocity vector. \( P \) represents the pressure, \( \rho, \mu, K \) and \( C_p \) the density, the dynamics viscosity, the thermal conductivity and specific heat of fluid, respectively.

To ensure realistic accurate turbulent modeling, the performance of four different turbulent models, namely Spalart-Allmaras model, Standard \( k-\epsilon \) model, Shear Stress Transport (SST) \( k-\omega \) model, and Reynolds Stress model were evaluated by Nasiruddin and Kamran Siddiqui [7], solving Navier-Stokes equations. The comparison of the simulated results obtained from these turbulent models with the experimental data made the selection easy. In that study, the SST \( k-\omega \) model was found to be the one that most accurately predicts the flow and modification due to the baffle. The selected turbulent model is capable of calculating the rapidly evolving two-dimensional flow and also in predicting, interactions with the wall. Another advantage of the selected turbulent model is that the model equations behave appropriately in both the near-wall and far-field regions. The SST \( k-\omega \) model is defined by two transport equations, one for the turbulent kinetic energy, \( k \) and the other for the specific dissipation rate \( \omega \), as given below [32]

\[ \frac{\partial}{\partial x_i} \left( \rho k u_i \right) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \]  
\[ \frac{\partial}{\partial x_i} \left( \rho \omega u_i \right) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \]  

Where

\[ G_k = -\rho u_i \frac{\partial u_i}{\partial x_j} \]  
\[ G_\omega = \alpha \frac{\omega}{k} G_k \]  
\[ \Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \]  
\[ \Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \]

and

\[ Y_k = \rho C_p \left( \mathbf{V} \nabla T \right) \]  
\[ Y_\omega = \rho C_p \left( \mathbf{V} \nabla T \right) \]

In these equations, \( x \) and \( y \) are the spatial coordinates. \( G_k \) represents the generation of turbulence kinetic energy due to mean velocity gradients. \( G_\omega \) represents the generation of \( \omega \). \( \Gamma_k \) and \( \Gamma_\omega \) represent the effective diffusivity of \( k \) and \( \omega \), respectively. \( Y_k \) and \( Y_\omega \) represent the dissipation of \( k \) and \( \omega \) due to turbulence, respectively. \( D_\omega \) represents the cross-diffusion term. \( S_k \) and \( S_\omega \) are user-defined source terms.

The commercial CFD software FLUENT6.3 [33] is used to calculate the heat and fluid flow characteristics in the computational domain with STLBs. The governing equations were discretized by the QUICK-scheme [34],
decoupling with the SIMPLE-algorithm [34] and solved using the Finite Volume Method [34]. For closure of the
equations, the SST k-ω model [32] was used in the present
study. The mesh was generated by the pre-processor
software Gambit 2.3. Structured meshes, with type-
Quadrilateral elements were built and tested with the
Fluent 6.3 [33]. To ensure the independence of the mesh
with the numerical results, a serial of test was performed.
To control the update of the computed variables for all
iterations, under relaxation was varied between 0.3 and
1.0. The solutions were converged when the normalized
residual values were less than $10^{-7}$ for all variables but less
than $10^{-10}$ only for the energy equation.

3.1 Boundary conditions

The hydrodynamic boundary conditions are chosen
according to the experimental work of Demartini et al. [5]
while the thermal boundary conditions are set according to
the numerical work of Nasiruddin and Kamran Siddiqui [7]. Air was used as the working fluid for all simulation
runs. The boundary conditions were given as, (i) the air
entered the channel at ambient temperature with a
uniform one-dimensional velocity ($U_{in}, v = 0, T_{in} = 300
K$); (ii) the pressure at the inlet of the computational
domain was set equal to the zero (gauge); (iii) the
turbulence intensity was kept at $I = 2\%$ at the inlet; (iv) A
constant temperature of $102^\circ C$ ($T = 375\ K$) was applied
on the entire wall of the computational domain as the
thermal boundary condition. (v) impermeable boundary
and no-slip wall conditions are imposed at the walls; and
(vi) in the channel outlet it is prescribed the atmospheric
pressure ($P = P_{atm}$). The Reynolds number of the
experiments [5] is $Re = 8.73 \times 10^4$, defined as

$$Re = \frac{\rho \bar{U} D_h}{\mu}$$

(10)

where $\bar{U}$ is the entrance (reference) velocity, 7.8 m/s, and
$D_h$ is the hydraulic diameter of the channel, equal to 0.167
m. The total length of the channel is equivalent to $3.307 \times
D_h$, which is not sufficient for the flow development.
Therefore, no influence will result from the side walls, so
that the flow can be considered as being two-dimensional.

The skin friction coefficient, $C_f$, is given by

$$C_f = \frac{\tau_w}{\frac{1}{2} \rho \bar{U}^2}$$

(12)

The numerical friction factor was computed from the
pressure drop, $\Delta P$, across the length of computational flow
domain, $L$, having the hydraulic diameter, using Darcy
Weisbach formula. That is,

$$f = \frac{(\Delta P/L)D_h}{\frac{1}{2} \rho \bar{U}^2}$$

(13)

The local Nusselt number, $Nu$, is evaluated as follows

$$Nu = \frac{h_x D_h}{\lambda_f}$$

(14)

The average Nusselt number, $Nu_{av}$, can be obtained by

$$Nu_{av} = \frac{1}{L} \int Nu \, dx$$

(15)

where $h_x$ represents the local convective heat transfer
coefficient.

3.2 Grid sensitivity

The mesh was generated by the Pre-processor software
Gambit 2.3. The mesh was refined at all solid boundaries,
with volumes growing in geometrical progression with the
increasing distance from the wall, as given by the expression

$$a_n = a_1 (q^{n-1})$$

(16)

where $a_n$ is the length of the last volume from the wall, $a_1$
is the length of the first volume adjacent to the wall, $q$ is the
growth ratio and $n$ is the number of volumes. This
expression is valid for the regions near the walls. For the
regions more distant from the walls, the mesh is uniform,
as reported by Demartini et al. [5]. A fine non-uniform
mesh with the minimum cell number of $(220 \times 95)$ and
the maximum cell number of $(420 \times 235)$ were provided
through the walls. It is found that the variation in $Nu$ and $f$
values for the staggered L-shaped baffles at $P_i = 0.142$ and
$Re = 8.73 \times 10^4$ is marginal when increasing the number of
cells from $(220 \times 95)$ to $(420 \times 235)$. Hence, there is no
such advantage in increasing the number of cells beyond
this value. Considering both convergent time and solution
precision, the grid system of $(220 \times 95)$ cells (in X and Y
directions, respectively) was adopted for the current
computation. The grid density was kept higher in the
vicinity of the heated wall and the STLBs to capture the
hydraulic and thermal boundary layers.

![Figure 3. Validation plot of the axial velocity profile distribution with reported data](image-url)
4. Results and discussion

To verify our numerical simulation, a comparison was made with the numerical and experimental data obtained by Demartini et al. [5] in the case of a rectangular channel, where two baffle plates were placed in opposite walls. The geometry of the problem is a simplification of the geometry of baffles found in shell-and-tube heat exchangers.

A quantitative comparison between both the experimental and numerical velocity profiles after the lower wall baffle, near the channel outlet is shown in Figure 3. For the case of axial velocity profiles at a position $x = 0.525$ m, measured downstream of the entrance, there is a good agreement between the present study and previous work [5]. These results give confidence that the numerical scheme used was accurate.

The two-dimensional horizontal constant temperature-surfaced rectangular cross section channel flow structure in the presence of STLBs could be easily discerned by considering the velocity contour plots in Figure 4(a-d). The figures present axial velocity fields of turbulent channel flow through STL Bs using the SST $k-\omega$ model for $Re = 12,000, 18,000, 24,000$, and $30,000$, respectively. The largest variations in the velocity fields occur in the regions near to the staggered STL Bs, as expected. The peak velocity values are seen near the heated top channel wall with an acceleration process that starts just after the lower wall STL B, while the velocity is observed to be very low at the locations corresponding to the zones of counter rotating flow in the regions downstream of both STL Bs as seen in the figure. In the regions between the tip of both STL Bs and the channel walls, the velocity is increased. The effect of the flow rate on the fluid velocity is also depicted in the figure which shows that the velocity value increases with the increase of Reynolds number.

The structure of the near wall flow in the channel with two solid STL Bs which are arranged on the top and bottom channel walls in a periodically staggered way can be displayed by considering the axial velocity profile plots in transverse stations for four different Reynolds number values ($Re = 12,000, 18,000, 24,000$, and $30,000$) as depicted in Figure 5(a-g) at various axial positions, $x = 0.159, 0.189, 0.255, 0.285, 0.315, 0.345$, and $0.525$ m, respectively. At Reynolds numbers of $12,000$ to $30,000$, the air flows under, between and above the STL Bs which will certainly causes a higher heat transfer than that of the lower Reynolds number. Between and downstream the STL Bs there is appearance of larger vortices whose size changes with the value of the Reynolds number. The comparison of axial velocity profiles at different axial stations shows that the influence of the deformation of the flow field increases as the air approaches the upper wall STL B, increasing the velocity of the flow approaching the passage under the baffle (see $x = 0.159$ m and $x = 0.189$ m in the figure). The plots also show that as the flow is accelerated and redirected near the first STL B, a very small recirculation zone is formed in the vicinity of the upper left corner (see Fig. 5a and b). Downstream of the first STL B, as a result of sudden expansion in the cross-section, the airflow separates, a larger clockwise vortex is formed behind the upper wall STL Bs in the upper part of the channel and flow reattachment is then established (see Fig. 5c and d).
In the region opposite the first STLB, the flow is characterized by very high velocities, approaching 237.7% of the inlet velocity, which is 1.049 m/s at the lowest value of Re number, as shown in Fig. 5(c and d). A similar behavior is observed near the STLB mounted on the lower wall with counterclockwise vortices at the upstream (see Fig. 5e and f) and downstream STLB (see Fig. 5g). The plots if Figure 5(e and f) shows that as the airflow

Figure 5. (Continued)
approaches the second STLB, its velocity is reduced in the lower part of the channel, while in the upper part is increased. In these locations ($x = 0.315$ m and $x = 0.345$ m) the negative velocities indicate the presence of a small recirculation zone at the lower left corner behind the lower wall STLB. The axial velocity profile distribution obtained for different values of Reynolds number after the lower wall STLB, near the channel outlet is also shown in Figure 5. At a position $x = 0.525$ m, 0.145 m after the second STLB and 0.029 m before channel outlet, the value of the velocity reaches 4.443 m/s, 4.233 times higher than the entrance velocity at the lowest value of Re number, as shown in Figure 5g. In the lower part of the channel, a strong counterclockwise vortex is observed downstream of the considered baffle, which was induced due to the flow separation. The vortex is located close to the solid wall and its height was approximately equal to the extent of the flow blockage by the STLB, which is equal to 0.08 m (see Fig. 5g).

The presence of the STLBs influences not only the velocity field but also the pressure distribution in the whole domain examined. The contour plots of dynamic pressure are shown in Figure 6 (a-d) for different Re values. The plots in Figure 6 show very low dynamic pressure values adjacent to the STLBs. In the regions downstream of both STLBs, recirculation cells with very low dynamic pressure values are observed. This fact is associated to the negative velocities at the upstream and downstream both STLBs shown in Figure 5. In the regions between the tip of the STLBs and the channel walls, the dynamic pressure is increased.

It indicates that the highest dynamic pressure value can be observed at the area of high velocities especially at the area near the upper channel wall with an acceleration process that starts just after the second STLB.

Airflow is from left to right. Dynamic pressure values in Pa.

Figure 6. Two-dimensional dynamic pressure field in the L-baffled channel at different Reynolds numbers: (a) Re = 12,000, (b) Re = 18,000, (c) Re = 24,000, and (d) Re = 30,000.

Figure 7. Two-dimensional temperature field in the L-baffled channel at different Reynolds numbers: (a) Re = 12,000, (b) Re = 18,000, (c) Re = 24,000, and (d) Re =
30,000. Airflow is from left to right. Fluid temperature values in K.

The temperature field is an important indicator to reflect the performance of a baffled channel. The contour plots of temperature field distributions in the given channel with the guidance of two-STLBs and a constant wall temperature condition along the top and bottom walls for the tested Reynolds number range of 12,000-30,000 are shown in Figure 7(a-d), respectively. In the figure, the lowest temperature value can be observed where the flow impinges the channel wall, especially in the region opposite the STLB tip where the velocity in this region is somewhat high while the highest one is found at the STLB corner area where the corner recirculation zone occurs, especially area behind the STLBs. The figure shows that the fluid temperature in the recirculation region is significantly high as compared to that in the same region of no STLB. It is also shown that in the recirculation regions, the fluid temperature is increased by approximately 7.5% due to the insertion of STLBs. The most intense in the temperature field is that occurring downstream of the lower-wall STLB, responsible for the high flow velocities observed at the outlet of the channel, creating a negative velocity profile which introduces mass inside the test channel through the outlet (see Fig. 5g).

The average heat transfer results are done on the heated top and bottom surfaces and presented in terms of a non-dimensional Nusselt number ration, $Nu/\overline{Nu}$, along the tested channel walls. $Nu$ is the Nusselt number for fully developed flow in a smooth channel at the same Reynolds number, and is given by

$$Nu_0 = 0.023 \Re^{0.8} \Pr^{0.4}$$

(17)

The $Nu_0$ is used as a reference to minimize the Reynolds number effect in the presented results. The Nusselt number ratio essentially indicates the amount of enhancement in heat transfer obtained by the flow guidance turbulators over the smooth rectangular channel.
Figure 9 shows the normalized average Nusselt number versus the Reynolds number (based on the hydraulic diameter) for the channel with upper and lower wall-mounted STLBs. The figure shows as expected, that the heat transfer rate increased with the Reynolds number. Both channel walls show similar trend but with different values. The largest variations are found at the heated top surface of the channel, due to the strong velocity gradients in that region. It is also noted that the use of the STLBs lead to extremely considerable increase in the convective average Nusselt number rate in comparison with the plain channel with no baffle. The heat transfer rate value for the STLBs is found to be higher by about 8.637 -28.081 times over the plain channel with no baffle, depending on the wall surface and the Reynolds number values.

Figure 10(a and b) shows the axial distribution plots of local skin friction coefficient profiles along the lower and upper channel walls in the range of Reynolds number investigated, respectively. We will start analyzing the effect of the STLB presence before discussing the influence of flow Reynolds number. Similarly to the results in Figure 8, the largest $C_f$ value is found in the region opposite the upper and lower wall STLBs, due to the strong velocity gradients in that region while the smallest $C_f$ value is found, firstly, upstream of the top wall STLB in the upper part of the channel and, secondly, downstream of the bottom wall STLB in the lower part of the channel, due to the orientation of the airflow by these deflectors. However, the skin friction coefficient is increased again at the locations corresponding to the zones of recirculation as seen in the figure. It indicates that the highest skin friction coefficient can be observed at the top heated surface of the channel in the area of high velocity especially at the top face of the STLB mounted on the lower wall of the channel. The variation of isothermal skin friction coefficient value with Reynolds number for the STLBs is also shown in Figure 10. It is clear from this figure that the $C_f$ tends to increase with raising the Reynolds number value, as expected.

Figure 11 shows the variation of the dimensionless pressure drop given by the friction factor ratio, $f/f_0$, with Reynolds number ($12,000 \leq Re \leq 30,000$). Here $f$ is the friction factor in a fully developed smooth channel at the same Reynolds number, and it can be presented as

$$f_0 = \frac{1}{(0.79 \ln Re - 1.64)^2}$$  \hspace{1cm} (18)

As expected, obviously it can be observed that values of $f/f_0$ become higher with increasing values in Reynolds number.
In the figure, the L-shaped baffled channel flows give higher values of friction factor than that for smooth channel flow due to the induction of high recirculation or vortex flow and thin boundary layer in the baffled channel, leading to higher temperature gradients, heat transfer and pressure drop.

![Figure 11. Normalized friction factor versus the Reynolds number at the surface of the top and bottom channel walls](image)

The maximum values for both upper and lower channel walls are found to be about 0.722-2.918 and 2.784-10.554 times above those for the smooth duct with no baffle, respectively, depending on the Re values. Thus the flow blockage due to the existence of baffle as well as the role of turbulence degree in the core region is a key factor to cause an extreme pressure drop.

### 4. Conclusion

A numerical study has been conducted to examine the fluid flow and heat transfer characteristics in a two-dimensional horizontal constant temperature-surfaced rectangular cross section channel with lower and upper wall-mounted L-shaped baffles in the turbulent regime from $Re = 12,000$ to $30,000$. The aim at using the STLBs is to create vortex flows having a significant influence on the flow pattern leading to higher heat transfer enhancement in the given channel. The computations are based on the Finite Volume Method, and the SIMPLE-algorithm has been implemented. The effects of the baffle L-shape geometry as well as Reynolds numbers are examined. The obtained results are compared with available experimental data from the literature and good agreement is obtained. The velocity profiles, and pressure and temperature fields over the baffled channel are the most notable characteristics of the effects of the STLB on the mainstream flow. These effects are: mainstream flow separation, recirculation, and secondary flow. The comparison of axial velocity profiles at different Reynolds numbers shows that as the airflow is accelerated and redirected near the STLBs, a very small recirculation zone is formed in the vicinity of the upper left corner. Downstream, as a result of sudden expansion in the cross-section, the flow separates, a larger clockwise recirculation zone is formed behind the upper wall STLB and flow reattachment is then established. A similar phenomenon is found near the STLB mounted on the lower wall with counterclockwise recirculation zones at the upstream and downstream STLB. The length of these recirculation zones strongly depend on the flow Reynolds number value. The appearance of these recirculation flows can help to increase higher the heat transfer in the channel with two STLBs because of transporting the fluid from the core to the near wall regimes, and in general, an increase in the flow Reynolds number causes a substantial increase in the flow velocity, leading to a high temperature gradient along the heating channel walls but the pressure loss is also very significant.

The above results suggest that a significant heat transfer enhancement in a heated channel can be achieved by introducing STLBs into the flow. The study can be extended for different number, sizes, positions, arrangements, orientations, and inclination angles, of the L-shaped baffles at different boundary conditions for temperature and velocity.

### AUTHOR’S CONTRIBUTIONS

Each author of this manuscript made considerable contributions in developing the mathematical modeling, data-analysis and contributed to the writing of this manuscript.

### NOMENCLATURE

- $a_n$: Length of the last volume from the wall, m
- $a_1$: Length of the first volume adjacent to the wall
- $C_p$: Specific heat at constant pressure, J/kg °K
- $C_f$: Skin friction coefficient
- $D_h$: Hydraulic diameter of rectangular channel, m
- $e$: L-shaped baffle thickness, m
- $f$: Friction factor
- $f_0$: Friction factor in smooth channel at the same Reynolds number
- $G_k$: Turbulent kinetic energy generation due to mean velocity gradient
- $G_{\omega}$: Kinetic energy generation due to buoyancy
- $H$: Channel height, m
- $h$: L-shaped baffle height, m
- $h_c$: Local convective heat transfer coefficient, W/m²K
- $k$: Turbulent kinetic energy, m²/s²
- $L$: Length of rectangular channel in x-direction, m
- $L_{in}$: Distance upstream of the first L-shaped baffle, m
- $L_{out}$: Distance downstream of the second L-shaped baffle, m
- $n$: Number of volumes.
"Nu\text{z}\) Average Nusselt number
"Nu\text{z}\) Local Nusselt number
"Nu\text{b}\) Nusselt number for fully developed pipe flow at the same Reynolds number
\(p\) Pressure, Pa
\(p_{\text{atm}}\) Atmospheric pressure, Pa
\(Pr\) Prandtl number
\(\eta\) Growth ratio
\(Re\) Reynolds number based on the channel hydraulic diameter
\(S\) L-shaped baffle distance or spacing, m
\(S_{s}, S_{o}\) Source term of \(k\) and \(\omega\)
\(T\) Temperature, °C
\(T_{\text{in}}\) Inlet fluid temperature, °C
\(T_{\text{wall}}\) Wall temperature, °C
\(\bar{U}\) Mean axial velocity of the section, m/s
\(u, v\) Velocity component in \(x\)- and \(y\)-direction, m/s
\(\bar{u}\) Velocity component in \(x\)-direction, m/s
\(\bar{V}\) Velocity vector, m/s
\(x, y\) Cartesian coordinates, m
\(Y_{x}, Y_{\omega}\) Dissipation of \(k\) and \(\omega\)

**Greek symbols**
\(\omega\) Specific dissipation rate, m\(^2\)/s
\(\Gamma, \Gamma_{\omega}\) Effective diffusivity of \(k\) and \(\omega\)
\(\rho\) Fluid density, kg/m\(^3\)
\(\mu\) Dynamic viscosity, Kg/m s
\(\mu\) Fluid dynamic viscosity, Kg/m s
\(\tau\) Wall shear stress, Kg/s m
\(\lambda\) Fluid thermal conductivity, W/m °C
\(\Delta P\) Pressure drop, Pa

**Subscript**
\(\text{atm}\) Atmospheric
\(f\) Fluid
\(i, j\) Refers coordinate direction vectors
\(\text{in, out}\) Inlet, outlet of the computational domain
\(t\) Turbulent
\(w\) Wall
\(x\) Local

### 6. REFERENCES


