Numerical modeling of the effect baffle inclination angle on flow and heat transfer along a horizontal channel

Amghar Kamal1*, Louhibi Mohamed Ali1, Salhi Najim1, Salhi Merzouki1
Laboratory of Mechanics and Energy, Department of Physics, Faculty of Sciences, Mohamed 1st University, Oujda, Morocco

(Received May 04 2017, Accepted April 18 2018)

Abstract. Forced convection is a phenomenon associated with the heat transfer fluid flow. The presence of forced convection affect concurrently on the hydrodynamic and thermal field. This mode of heat transfer in the horizontal channel is of relevance for many practical applications, such as electronic components, cooling, turbine blades and heat exchangers systems. In this paper a numerical study of forced convection turbulent in a two dimensional channel, with inclined baffle, is carried out. The baffle is placed on lower wall at the horizontal channel. The top, bottom walls and the baffle are heated at constant temperature $T_w$. The inlet temperature is maintained cold at $T_c$. The governing equations are solved using a finite volume method and were solved by ANSYS 15 Software. Special emphasis is given to detail the effect of the baffle inclination angle on the dynamics of velocity and heat transfer generated by forced convection. The results are given for the following control parameters, $Re = 8.73 \times 10^4$ and $Pr = 0.71$. The inlet and outlet opening hydraulic diameters are $D_h = 0.146m$. Three values of the baffle inclination angle are considered $\phi = 45^\circ, 60^\circ$ and $90^\circ$. The results show that when we gradually increase the inclination of baffle, the heat transfer increases. The results of this study are presented in terms of streamline contours, isotherms, and local Nusselt number for various baffle inclination angle.

Keywords: Forced convection, Numerical study, Heat transfer, $k - \varepsilon$, baffle effect.

1 Introduction

Heat exchanger are used in a wide range of 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 electronic systems, nuclear reactor, solar collectors and equipment chemical treatment.

In spite of appearances, the fluid flowing over baffles display a complicated steady motion and presents a good opportunity to understand these fundamental mechanisms of steady flow interactions occurring in various technologies.

This interest is due to industrial applications that present this type of geometries in various problems of industry. In the literature, the different shapes of inclination and location of baffles and fins has been the subject of several scientific studies. Considerable work has been done, in recent years, on the investigations of the flow and heat transfer processes especially in order to improve the accuracy of prediction of heat exchangers performances. Among, the many published research on heat sinks and available in literature, we cite the work of Patankar et al.\cite{12} these authors reported the first work on the numerical analysis of flow in forced convection in a channel and they presented the concept of flow are periodically developed. Nasiruddin and Kamran,\cite{11} examined numerically a heat exchanger channel by installing a baffle for the three different orientations (vertical, downstream side and upstream side) for study the effect of baffle size and orientation on the heat transfer enhancement. The results obtained show that for the vertical baffle, an increase in the baffle height causes a

* Corresponding author. E-mail address: amgharkamaler@gmail.com

Published by World Academic Press, World Academic Union
substantial increase in the Nusselt number but the pressure loss is also very significant. For the inclined baffles, the results show that the Nusselt number enhancement is dependent of the baffle inclination angle with the maximum and average Nusselt number 120% and 70% higher than that for the case of no baffle, Also, Prashanta and Sandip[15] were conducted an experimental investigation of frictional loss and heat transfer behavior 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, they found that there exists an optimum inclination to maximize heat transfer coefficients, so this experimental heat transfer analysis show that inclined the baffles can combine the heat transfer enhancement techniques. Among important, Berner et al.[4] studies baffle plates perpendicular to the flow direction where the obtained mean velocity and turbulence results in flow over baffles, and Habib et al.[9] investigated heat transfer and flow over perpendicular baffles of different heights and inclination. Recently, Dutta et al.[8] reported heat transfer enhancement with an inclined baffles. The literature that has been surveyed indicates that the number of studies concerning forced convection in channel provided the inclination of baffle in lower wall has increased in recent years. However, to the best of our knowledge, studies on these problems have been scarcely investigated. To address the limited number of existing studies on such a topic, this paper proposes a numerical approach to simulate a two-dimensional model of steady the effect of forced convection on heat transfer in channel rectangular and horizontal using the equations controlling the system are solved by a finite volume method and solved by SIMPLE.[14] algorithm was used for the velocity-pressure coupling. The investigation is carried out using a QUICK.[14] (Quadratic upwind differencing scheme) and Finite volume method. This numerical scheme was proposed by Patankar.[14] and was used by many authors, as an example, we can quote Leonard et al.[10] Amghar et al.[2] Bouchenafa et al.[5] and we can also lead the readers to others authors in the following references[3, 6].

The objective of this paper presents a numerical study of the heat transfer and flow developments in a horizontal channel with baffle mounted on lower wall. The temperature and velocity distributions are simulated by the finite volume method. The effects of inclination baffle on flow and heat transfer developments are studied.

The structure of this paper is as follows. In Section 3, where the problem statement and related mathematical formulation to be solved in the numerical simulation are introduced while supplementing them with the boundary conditions. Afterwards, the numerical procedure adopted here is detailed in Section 4. Section 5 deals with the results and their discussions, and the last section contain the conclusions for this paper.

2 Nomenclature

The following notation is used throughout the text:

\( T \) : Temperature, K  
\( c_1 \) : constant used in the standard \( k - \varepsilon \) model  
\( c_2 \) : constant used in the standard \( k - \varepsilon \) model  
\( c_\mu \) : constant used in the standard \( k - \varepsilon \) model  
\( \mu_t \) : turbulent viscosity, [Pa.s]  
\( L \) : length channel, m  
\( N_u \) : local Nusselt number  
\( P_k \) : kinetic energy production due to buoyancy, \( m^2 / s^2 \)  
\( \phi_e \) : value of \( \phi \) on the face ‘e’  
\( \phi_w \) : value of \( \phi \) on the face ‘w’  
\( T_{in} \) : inlet temperature, K  
\( Re \) : reynolds number  
\( S_\phi \) : source term for \( \phi \)  
\( H \) : channel height, m  
\( u_{int} \) : inlet velocity, \( m/s \)  
\( \sigma \) : thickness of the baffle  
\( P \) : pressure, \( Pa \)
3 Description of the problem

3.1 Statement of the problem

The physical problem considered in this study, is that a two-dimensional incompressible flow and turbulent in a horizontal, rectangular channel, of height $H$ and the length$L$, in which, the horizontal walls are carried and maintained at same temperatures $T = 373 K$ (Fig. 1). A baffle plane is used for generators of vortex, which is attached upon the lower wall of channel at a distance of $L_1$ from the upstream end of the tube. Three different baffle inclinations were considered in this study, which are referred as case (a), (b) and (c). In case (a), a baffle inclined from angle $\varphi_1 = 45^\circ$ was considered (see Fig. 1a), in case (b), a baffle inclined from angle $\varphi_2 = 60^\circ$ end was considered (see Fig. 1b) and in case (c), a vertical baffle end was considered (see Fig. 1c).

The fluid circulating in the channel is the air and its physical properties, except its density, are supposed to be constant, the dimensions of geometry are given in the Table 1.

At inlet the channel, the flow of fluid describe at uniform velocity, also the bottom and lower walls an artificial open boundary condition is imposed, when the temperature profile is constant value and the velocity is zero, hence, at outlet boundaries the usual zero gradient conditions are used for the velocity, pressure, temperature and $k - \varepsilon$ model.

<table>
<thead>
<tr>
<th>$L(m)$</th>
<th>$H(m)$</th>
<th>$h(m)$</th>
<th>$\delta(m)$</th>
<th>$L_1(m)$</th>
<th>$D_h(m)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.554</td>
<td>0.146</td>
<td>0.1</td>
<td>0.01</td>
<td>0.218</td>
<td>0.167</td>
</tr>
</tbody>
</table>

3.2 Mathematical model

With the above assumptions, the steady forced convection heat and mass transfer in horizontal channel can be described by the equations of continuity, momentum, thermal energy (in the absence of viscous dissipation)
Fig. 1: Schematic configuration, case (a) baffle inclined from $\varphi_1 = 45^\circ$, case (b), baffle inclined from $\varphi_2 = 60^\circ$, case (c) a vertical baffle

Table 2: Boundaries conditions

<table>
<thead>
<tr>
<th>Boundaries</th>
<th>Velocity inlet</th>
<th>Temperature inlet</th>
<th>Walls temperature</th>
<th>Baffle temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unit</td>
<td>m/s</td>
<td>K</td>
<td>K</td>
<td>K</td>
</tr>
<tr>
<td>Value</td>
<td>7.8</td>
<td>300</td>
<td>373</td>
<td>373</td>
</tr>
</tbody>
</table>

for a power law fluid and the two equation $k - \varepsilon$ used for turbulence model. Indeed, this numerical investigation is based on the following assumptions:

(i) Physical properties of fluid are constant;
(ii) A profile of velocity is uniform at the inlet;
(iii) The flow is assumed to be steady;
(iv) The fluid (air) is Newtonian and incompressible;
(v) The temperature of walls is constant (upper and lower);
(vi) The radiation heat transfer is negligible.

The general transport equation that describes the principle of conservation of mass, momentum, energy and equations of the standard $k - \varepsilon$, where we can be expressed in the following conservative form Patankar.\[14\]:
\[
\frac{\partial}{\partial x_j} (\rho u \phi) = \frac{\partial}{\partial x_j} (\tau_\phi \frac{\partial \phi}{\partial x_j}) + S_\phi.
\]  

(1)

Where, \( \phi \) stands for the dependent variables \( u, v, k, T \) and \( \varepsilon, u, \) and \( v \) stand for the mean velocities towards the \( x- \) and \( y- \)axis, respectively. \( k \) and \( \varepsilon \) stand for kinetic energy and turbulent dissipation, respectively. \( \tau_\phi \) and \( S_\phi \) are the corresponding diffusion coefficient and source term, respectively, for general variable.

Then, the governing equations for the present system can be expressed as follows:

Continuity:

\[
\frac{\partial}{\partial x_j} (\rho u_j) = 0.
\]  

(2)

Momentum:

\[
\frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} [(\mu + \mu_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)].
\]  

(3)

Energy:

\[
\frac{\partial}{\partial x_j} (\rho u_j T) = \frac{\partial}{\partial x_j} \left[ (\frac{\mu}{Pr} + \frac{\mu_t}{\sigma_T}) \frac{\partial T}{\partial x_j} \right].
\]  

(4)

To ensure realistic and accurate turbulent modeling, we have tried to present the \( k-\varepsilon \) model for turbulence, where this model is defined by two transport equations, one for the turbulent kinetic energy, \( k \) and the other for the specific dissipation rate \( \varepsilon \), are given as:

Turbulent kinetic energy \( k \) equation:

\[
\frac{\partial}{\partial x_j} (\rho u_j k) = \frac{\partial}{\partial x_j} [(\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j}] + P_k - \varepsilon.
\]  

(5)

Turbulent energy dissipation \( \varepsilon \) equation:

\[
\frac{\partial}{\partial x_j} (\rho u_j \varepsilon) = \frac{\partial}{\partial x_j} [(\mu + \frac{\mu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_j}] + \frac{\varepsilon}{k} (c_1 P_k - c_2 \varepsilon).
\]  

(6)

Where: \( \mu_t = \rho c_\mu \frac{k}{\varepsilon} \)

The turbulent constants correspond to those suggested by Launder et al.\(^1\) and Chieng et al.\(^7\). These constants are arranged in the table below (Table 3).

<table>
<thead>
<tr>
<th>( C_\mu )</th>
<th>( C_1 )</th>
<th>( C_2 )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\varepsilon )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1</td>
<td>1.3</td>
</tr>
</tbody>
</table>

These default values have been determined from experiments with air and water for fundamental turbulent shear flows including homogeneous shear flows and decaying isotropic grid turbulence. They have been found to work fairly well for a wide range of wall-bounded and free shear flows.

These Eqs. (??) are solved numerically for a Reynolds number from \( Re = 8.73 \times 10^4 \) and a Prandtl number, \( Pr = 0.71 \) where based at finite volume method and SIMPLE algorithm widely used numerical procedure to solve the Navier-stokes equations.

Hence, the boundary conditions associated to the system of the Eqs. (??) are detailed as bellow:
• $x = 0$ and $0 \leq y \leq H$

$$u = u_{\text{int}} = 7.8 \text{m/s}, v = 0, T = T_{\text{in}}, k = 0.005u_{\text{in}}^2, \varepsilon_{\text{in}} = 0.1k_{\text{in}}^2.$$  \hfill (7)

• $y = 0$ and $0 \leq x \leq L$

$$u = v = 0, T = T_w.$$  \hfill (8)

• $x = L$ and $0 \leq y \leq H$

$$\frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = \frac{\partial T}{\partial x} = \frac{\partial k}{\partial x} = \frac{\partial \varepsilon}{\partial x} = 0.$$  \hfill (9)

• $y = H$ and $0 \leq x \leq L$

$$u = v = 0, T = T_w.$$  \hfill (10)

• $x = L_1$ and $0 \leq y \leq h$

$$T = T_w.$$  \hfill (11)

• $x = L_1 + \delta$ and $0 \leq y \leq h$

$$T = T_w.$$  \hfill (12)

• $y = H$ and $L_1 \leq x \leq L_1 + \delta$

$$T = T_w.$$  \hfill (13)

The local heat transfer coefficient is defined as:

$$N_{\mu x} = \frac{h_x \times H}{k} = \frac{q_w \times D_h}{k(T_w - T_b)}.$$  \hfill (14)

### 4 Numerical procedure

The governing conservation equations are discretized in space using the finite volume method (FVM), when the convection diffusion terms were treated with a Quick scheme. The resulting algebraic equations, with the associated boundary conditions, are then solved using the line by line method. As the momentum equation is formulated in terms of the primitive variables ($U, V$ and $P$), the iterative procedure includes a pressure correction calculation method, namely SIMPLE modified.\textsuperscript{[14]} to solve the pressure-velocity coupling. Noted that the convergence criterion for temperature, pressure, and velocity is given as:

$$\frac{\sum_{j=1}^{m} \sum_{i=1}^{n} \left| \eta_{ij}^{k+1} - \eta_{ij}^k \right|}{\sum_{j=1}^{m} \sum_{i=1}^{n} \left| \eta_{ij}^{k+1} \right|} \leq \varepsilon_\chi,$$
where both m and n are the grid points numbers in X and Y directions, respectively. \( \eta \) is any of the computed field variables, \( k \) is the iteration number, \( i \) and \( j \) are nodes according to X and Y directions, and \( \chi \) is a dependent variable. In this study, the velocity components and temperature were driven to \( \varepsilon_u = \varepsilon_v = \varepsilon_\theta = 10^{-6} \) and for pressure \( \varepsilon_p = 10^{-6} \). The performance of the using code via the forced convection turbulent in horizontal channel is established by comparing its predictions with other numerical results, and by verifying the grid independence of the present results.

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.

Various computations were performed for inclination baffle and \( Re = 8.73 \times 10^4 \) for different uniform grids \( (N_x \times N_y) \) in order to examine the grid independence. Indeed, the maximum difference between the values of \( Nu \) obtained for the 120 \( \times \) 80 grid and the finest 160 \( \times \) 100 grid was less than 0.78\%. Consequently, to optimize appropriate grid refinement with computational efficiency, the grid 120 \( \times \) 80 was chosen for all the further computations.

Under steady conditions the leading term would vanish. Patankar.[13, 14] discusses each term in such equations, as well as the application of boundary conditions.

To solve the relevant equations by a finite volume method (FVM), a grid is first generated to cover the domain of interest. A typical control volume within such a grid is denoted by the shaded area in Fig. 2. Eq. (1) is integrated over this volume, and each term in the resulting integral balance is approximated in terms of the discrete values of at the nodal points (i.e., of \( P, E, N \), etc.). The average value of the source term is approximated by the gradient of pressure and the central differencing approximation has been used to represent the diffusion terms. Hence, the expression of eq. (1) becomes:

\[
F_e \phi_e - F_w \phi_w + F_n \phi_n - F_s \phi_s = D_e (\phi_E - \phi_P) - D_w (\phi_P - \phi_W) + D_n (\phi_N - \phi_P) - D_s (\phi_P - \phi_S) + \int_S S_c dv. \tag{15}
\]

The value of \( \phi \) on the faces \( e, w, n \) and \( s \) can be given by scheme “Quick”:
The algebraic approximation of the integral balance for the $P$ control volume (see Fig. 2) becomes:

$$
F_e \phi_e - F_w \phi_w + F_n \phi_n - F_s \phi_s = D_e (\phi_E - \phi_P) - D_w (\phi_P - \phi_W) + D_n (\phi_N - \phi_P) - D_s (\phi_P - \phi_S) + \int \limits_V S_\phi \, dv.
$$

(16a)

Or more simply:

$$
a_p \phi_P = a_E \phi_E + a_W \phi_W + a_N \phi_N + a_S \phi_S + a_{EE} \phi_{EE} + a_{WW} \phi_{WW} + a_{NN} \phi_{NN} + a_{SS} \phi_{SS} + \int \limits_V S_\phi \, dv.
$$

(16b)

Where indices $nb$ refer to adjacent cells to the cell $P$. $\alpha$ are coefficient including the convection and the diffusion terms of $\phi$, whereas $S_\phi$ is a source term.

Where the summation is over the appropriate neighbor points. The coefficient $a_p$ is given by

$$
a_p = a_E + a_W + a_N + a_S + a_{EE} + a_{WW} + a_{NN} + a_{SS}.
$$

(17)

The cell values of the variables $a_{nb}$ can be written as:

$$
\begin{align*}
a_E &= D_e - \frac{3}{8} a_e F_e + \frac{1}{8} (1 - a_w) F_w + \frac{6}{8} (1 - a_e) F_e \\
a_w &= D_w + \frac{3}{8} a_e F_e + \frac{6}{8} a_w F_w - \frac{3}{8} (1 - a_w) F_w \\
a_n &= D_n - \frac{3}{8} a_N F_n + \frac{1}{8} (1 - a_s) F_S + \frac{6}{8} (1 - a_n) F_n \\
a_S &= D_s + \frac{3}{8} a_n F_n + \frac{6}{8} a_S F_S - \frac{3}{8} (1 - a_s) F_S \\
a_{EE} &= -\frac{1}{8} (1 - a_e) F_e \\
a_{WW} &= -\frac{1}{8} a_w F_w \\
a_{NN} &= -\frac{1}{8} (1 - a_n) F_n \\
a_{SS} &= -\frac{1}{8} (1 - a_s) F_s
\end{align*}
$$

And $b$ is the source term (the rate of heat generation per unit volume), where defined by the variation of pressure between two positions successive in the mesh.

Where: $\phi = u$, the term $b$ is defined by $[P(I - 1, J) - P(I, J)](y_{j+1} - y_j)$.
However, the maximum vortex observed downstream of the baffle vertical is explained by the variation of the inclination angle $\phi$ approximately up to $90^\circ$. For comparison, the maximum vorticity for the case with no baffle was $\phi_3 = 90^\circ$. However, the maximum vortex observed downstream of the baffle vertical is explained by the fact that the air is more blocked when the baffle vertical, for this, the vortices becomes more intense and increases in volume. However, the variation in the vortex length for different values of $\phi$ was within 5%. The maximum vorticity in the core of the vortex region was observed to be in the range of $20 \sim 30$. For comparison, the maximum vorticity for the case with no baffle was approximately $2s_1$. Therefore, the vortices with longer streamwise extent and higher vorticity will contribute more to the mixing and thus, the heat transfer.

The thermal distribution is shown in Fig. 6 for different values of $\phi$ in terms of isotherms when length of the baffle $h = 0.08m$. Clearly three distinct position of baffle can be identified in Fig. 6. These figures show that the variation of the inclination angle $\phi$ has a significant influence on the flow and the temperature fields. The plot shows that the fluid temperature for cases (d); (e); (f) in the baffle region is significantly high for case (f) as compared to that in the same region for cases (d) and (e). Indeed, for case (f), the isotherms are very dense near the vertical baffle isothermal and walls of the horizontal channel when the heat is transferred from the plate to

$$\phi = v$$, the term $b$ is defined by: $\{P(I, J - 1) - P(I, J)\}(x_{i+1} - x_i)$

A similar equation applies for each control volume in the grid. Details related to the derivation, the application of boundary conditions, the treatment of nonlinearities, etc., have been provided by Patankar[13, 14] and studied by Amghar et al.[2].

5 Results and discussions

Numerical simulations are performed to examine, the influence of the inclined transversal baffle on flow and temperature field inside the channel. In particular the streamlines and the fluid temperature contours as well as the convective Nusselt number distribution, were treated along the channel and for various baffle inclination with active degrees and it should be noted that for the flow, all solutions are dependent on time, then, in the following study curves and streamline fields are given at an arbitrary instant.

5.1 Streamlines and isotherms

The impact of the baffle orientation on the structure of the near wall flow is depicted in Fig. 5. The plots show the streamlines for different baffle inclinations at $Re = 8.73 \times 10^4$ and this study is performed using air as the working fluid with number of Prandtl ($Pr = 0.71$). In all cases, a strong vortex is observed downstream of the baffle, which was induced due to the flow separation. The vortex was located close to the solid wall and its height was approximately equal to the extent of the flow blockage by the baffle, which is equal to 0.08 m for the cases shown in Fig. 5. A vorticity based thresholding scheme was used to detect the physical extent of the vortex was studied out by Hussain and Hayakawa, 1987. The results show that the baffle inclination has an effect on the length and the magnitude of the maximum vorticity. The comparison of the vortex length for different baffle orientations shows that the vertical baffle generates the several vortices. For case (c), it was observed that the vortex length increases with $\phi$ up to $\phi_3 = 90^\circ$. However, the maximum vortex observed downstream of the baffle vertical is explained by the fact that the air is more blocked when the baffle vertical, for this, the vortices becomes more intense and increases in volume. However, the variation in the vortex length for different values of $\phi$ was within 5%. The maximum vorticity in the core of the vortex region was observed to be in the range of $20 \sim 30$. For comparison, the maximum vorticity for the case with no baffle was approximately $2s_1$. Therefore, the vortices with longer streamwise extent and higher vorticity will contribute more to the mixing and thus, the heat transfer.

The thermal distribution is shown in Fig. 6 for different values of $\phi$ in terms of isotherms when length of the baffle $h = 0.08m$. Clearly three distinct position of baffle can be identified in Fig. 6. These figures show that the variation of the inclination angle $\phi$ has a significant influence on the flow and the temperature fields. The plot shows that the fluid temperature for cases (d); (e); (f) in the baffle region is significantly high for case (f) as compared to that in the same region for cases (d) and (e). Indeed, for case (f), the isotherms are very dense near the vertical baffle isothermal and walls of the horizontal channel when the heat is transferred from the plate to

Fig. 4: Section of a Cartesian grid showing placement of control-volume boundaries

(a) Control volume for $P, T, k$ and $\varepsilon$
(b) Control volume for $u_e$
(c) Control volume for $v_n$

A similar equation applies for each control volume in the grid. Details related to the derivation, the application of boundary conditions, the treatment of nonlinearities, etc., have been provided by Patankar[13, 14] and studied by Amghar et al.[2].

5 Results and discussions

Numerical simulations are performed to examine, the influence of the inclined transversal baffle on flow and temperature field inside the channel. In particular the streamlines and the fluid temperature contours as well as the convective Nusselt number distribution, were treated along the channel and for various baffle inclination with active degrees and It should be noted that for the flow, all solutions are dependent on time, then, in the following study curves and streamline fields are given at an arbitrary instant.

5.1 Streamlines and isotherms

The impact of the baffle orientation on the structure of the near wall flow is depicted in Fig. 5. The plots show the streamlines for different baffle inclinations at $Re = 8.73 \times 10^4$ and this study is performed using air as the working fluid with number of Prandtl ($Pr = 0.71$). In all cases, a strong vortex is observed downstream of the baffle, which was induced due to the flow separation. The vortex was located close to the solid wall and its height was approximately equal to the extent of the flow blockage by the baffle, which is equal to 0.08 m for the cases shown in Fig. 5. A vorticity based thresholding scheme was used to detect the physical extent of the vortex was studied out by Hussain and Hayakawa, 1987. The results show that the baffle inclination has an effect on the length and the magnitude of the maximum vorticity. The comparison of the vortex length for different baffle orientations shows that the vertical baffle generates the several vortices. For case (c), it was observed that the vortex length increases with $\phi$ up to $\phi_3 = 90^\circ$. However, the maximum vortex observed downstream of the baffle vertical is explained by the fact that the air is more blocked when the baffle vertical, for this, the vortices becomes more intense and increases in volume. However, the variation in the vortex length for different values of $\phi$ was within 5%. The maximum vorticity in the core of the vortex region was observed to be in the range of $20 \sim 30$. For comparison, the maximum vorticity for the case with no baffle was approximately $2s_1$. Therefore, the vortices with longer streamwise extent and higher vorticity will contribute more to the mixing and thus, the heat transfer.

The thermal distribution is shown in Fig. 6 for different values of $\phi$ in terms of isotherms when length of the baffle $h = 0.08m$. Clearly three distinct position of baffle can be identified in Fig. 6. These figures show that the variation of the inclination angle $\phi$ has a significant influence on the flow and the temperature fields. The plot shows that the fluid temperature for cases (d); (e); (f) in the baffle region is significantly high for case (f) as compared to that in the same region for cases (d) and (e). Indeed, for case (f), the isotherms are very dense near the vertical baffle isothermal and walls of the horizontal channel when the heat is transferred from the plate to

WJMS email for subscription: info@wjms.org.uk
the fluid by forced convection. Another solution characterized by the presence of the vortices in downstream of baffle are significantly by the fluid does not find sufficient space to circulate rapidly. Hence, the increase of heat transfer in the baffle zone. Corresponding isotherms show a good heat exchange along the lower and top part of the hot walls. They are distorted around the baffles and we can notify that the major part of the open lines pass between the hot wall and the baffles isothermal.

![Streamlines for different baffle inclination angle](image)

Fig. 5: Streamlines for different baffle inclination angle

5.2 Local Nusselt number

An important outcome of the computation is the local Nusselt number distribution along a horizontal channel with provided the baffle on the lower the wall. The effect of baffle inclination angle $\phi$ on the local Nusselt number distribution at $Re = 8.73 \times 10^4$ is presented in Fig. 7. For the angles $\phi = 45^\circ, 60^\circ$, we note that the curves of the local Nusselt number along the channel and lower $x = 0.218m$, are quite identical. This can be explained by the fact that the heat transfer is the same upstream of the baffle.

In the pure forced convection case (Fig. 7), and when the inclination angle $\phi$ varies, one notes that the local Nusselt number along the channel from $x = 0.218m$ is more important and increased for $\phi = 90^\circ$ (Fig. 7) than other cases. This augmentation can be interpreted by increasing the vortices in downstream of baffle vertical with significantly by the fluid does not find sufficient space to circulate rapidly and consequently an improvement of heat transfer.

Moreover, the inclination of baffle seems to have significant effects on the distribution of the local Nusselt number in the horizontal channel mounted the baffle.

6 Conclusion

In this paper, the turbulent flow of air and the heat transfer in a channel differentially heated by an inclined baffle have been investigated by a 2-D quick scheme and solved by a finite volume method and the velocity pressure fields are linked by SIMPLE algorithm. The obtained results lead to the following conclusions:

*WJMS email for contribution: submit@wjms.org.uk*
In this paper, the turbulent flow of air and the heat transfer in a channel differentially heated by an inclined baffle have been investigated by a 2-D quick scheme and solved by a finite volume method and the velocity pressure fields are linked by SIMPLE algorithm. The obtained results lead to the following conclusions:

- For number of reynolds $8.73 \times 10^4$, the presence of the inclined baffle has very important local effects on heat transfer and fluid flow.
- Even using a coarse mesh, the importance of our code could be reliably applied to other complex geometry.
- The local Nusselt number increasing with increasing the baffle inclination angle and that very important when $\varphi = 90^\circ$ at $x = 0.19m$.

Fig. 7: Effect of baffle inclination angle on the local Nusselt number
References


WJMS email for contribution: submit@wjms.org.uk