Heat and fluid flow in an uneven heated chimney

C. Hemmer a, b, C.V. Popa a, *, A. Sergent c, d, G. Polidori a

a GRESPI-Thermomécanique, Université de Reims Champagne-Ardenne, Moulin de la Housse, BP 1039, 51687 Reims Cedex 2, France
b CAMPA, route de Soissons, 51170 Fismes, France
c LIMSI, CNRS, Université Paris-Saclay, Rue John Von Neumann, 91400 Orsay, France
d Sorbonne Universités, UPMC Univ Paris 06, UFR d’Ingenierie, 4 place Jussieu, 75005 Paris, France

A R T I C L E   I N F O

Article history:
Received 9 April 2015
Received in revised form
13 April 2016
Accepted 14 April 2016

Keywords:
Natural convection
Numerical simulation
Vertical channel
Heat source location
Asymmetric heating
Chimney

A B S T R A C T

A 2D numerical study of the natural convection flow in an asymmetrically heated vertical plane channel is carried out in the boundary layer regime for three modified Rayleigh numbers (Ra*) between 2.25 \times 10^6 and 9 \times 10^6. This configuration is seen to be a simplified model of an electrical heater. This study focuses on the influence of different spatial arrangements of the heat sources on both heat transfer and flow dynamics, and especially on the back flow. In order to simplify the analysis, the working fluid is water, making thermal radiation negligible. It is shown that, at constant heat power and exchange surface, two distant heat sources are more efficient to put fluid into motion and to transfer heat.

© 2016 Elsevier Masson SAS. All rights reserved.

1. Introduction

The design of new electric heaters is subordinated to the respect for European standards which the designers have to take into account [1]. One of the main constraints, which determines the internal choice and arrangement of the various components of radiators, is the control of the temperature on accessible surfaces of the heater for obvious safety reasons [2]. One of the lines of thought envisaged by the manufacturers is to consider various heat sources positioned vertically inside the box of the heater, under the same incident electric power [3]. It is in this context that we should consider the present study, where the radiator box is then likened to a vertical open-ended channel within whom the knowledge of the thermo-convective mechanisms becoming established is necessary.

This important step in the design of electric heaters includes several difficulties: the interaction between different modes of heat transfer (conduction, convection and radiation), the shape and the position of the electrical resistances. In order to simplify the design of electric heaters, this paper deals with the numerical investigation of the natural convection flow, neglecting radiation, in a vertical plane channel heated asymmetrically which represents a simplified electric heater.

Beyond the present study initially motivated by an industrial application in the electric heating, the heated vertical channel is also representative of several problems such double-skin facade or Trombe wall, the chimney effects and solar panel to name a few. For example, several studies [4] [5], have shown that the implementation of the concept of double-skin facade in a building with low inertia seems essential to improve the thermal comfort. Despite a great number of experimental and numerical studies concerning convective heat transfer in vertical channel heated or cooled, there is still a lack of fundamental knowledge in the main dynamic and thermal behaviors of such double skin facades in order to maximize their efficiency and minimize resulting energy losses. Numerical simulations of natural convection in agreement with experimental results can allow understanding complex phenomena involved in these cases. Although many experimental studies on the vertical channel have been conducted [6–8], the majority of studies have been limited to thermal measurements and studies concerning the flow dynamics are not so numerous in the literature [9–11]. Common to all these studies is the observation of an upward flow along the heated wall and a downward one along the opposed adiabatic wall which is Ra* dependent. For example, Elenbaas [6] has determined, in a vertical channel, several
kinds of dynamical flow behaviors depending on the modified Rayleigh number (Ra*), which is based on the channel width (b) and the ratio (A/b) where A is the heated length at wall. These authors highlighted that for a low modified Rayleigh number (Ra*<100), the flow regime is fully developed over the entire width of the channel whereas for a high-modified Rayleigh number (Ra*>1000), the flow regime is of boundary layer-type along the heated wall with the presence of a reverse flow observed near the unheated wall. This flow structure was also observed by Manca et al. [12].

For such free convection coupled problem the knowledge of quantitative dynamic quantities is essential for the implement of numerical simulation codes. It is the reason why, in literature, numerical studies [13–15] have tempted to correlate the geometrical and heating channel conditions with the dynamics and structure of the flow. Christian et al. [16] performed a numerical study of the natural convection flow in a ventilated façade, which was modeled by a vertical channel heated with a constant imposed temperature. The authors showed for Ra* > 100 that convection is the principal mode of heat transfer. Corresponding correlations for the average Nusselt number were highlighted. Dehghan et al. [17] have studied natural convection in a vertical slot with two heat source elements. The influence of both the heat generation rate and the separation distance between the sources has been investigated. It has been shown that wall conduction was important, and for substrates with a finite thermal conductivity, it is essential that the conjugate analysis be employed. The authors showed that the thermal interference between the heat sources was reduced increasing Ra*. In this study it was also observed that increasing the spacing between the sources had the effect to increase velocities within the channel. This led to an enhancement in the convective cooling of the upper located electronic component. A higher cold mass flow rate entering the cavity was observed when the spacing between the sources increased, which resulted in an increase in the convection heat transfer from both components. Fossa et al. [18] and Menezzo et al. [19] studied the case of a non-uniformly heated channel at large numbers of modified Rayleigh channel. They studied a channel with an alternating periodic heating either on both sides or on a single. They measured the wall temperatures to determine the heat transfer in each zone. The authors underlined a new geometrical parameter in the non-uniform heating case. This geometrical parameter has been defined as the distance between the inlet channel and the beginning of the heating zone. They showed that an alternation between heated and unheated zones could enhance the heat transfer up to 20%, compared to a configuration of uniform heating at one wall. From a numerical point of view, two main approaches can be proposed to solve the problem of natural convection flow in the channel: either a complete simulation of the channel and its external environment, or a truncated simulation considering the single channel limited to its geometric limitations. The second approach is problematic in that boundary conditions at bottom and top channel interfaces are a priori unknown since the driving flow is located within the computation domain (Garnier [21]). Desrayaud et al. [20] have carried out a numerical study to investigate the sensitivity and the influence of several boundary conditions on the heat transfer in an asymmetrically heated channel. The authors have shown that when the fluid domain is limited only to the channel geometry, the pressure boundary conditions at the inlet and the outlet of the channel are difficult to model. This non-exhaustive state of the art shows how this problem still questions.

The present work aims at evidencing the influence of the spatial arrangement of several heat sources placed on one of the channel side walls by performing 2D simulations. The influence of three different values of the total heat power (P/P<sub>2</sub>:P<sub>2</sub>:P<sub>2P</sub>) given to the channel flow and five heat source arrangements on the flow, the wall temperature distribution and the heat transfer inside a vertical channel are examined. To compare the influence of the geometrical configurations on the flow and temperature fields, the results analysis is done at constant heat power. The studied range of Ra* corresponds to the boundary layer regime in order to identify the overall response of the backflow at the top channel to the heat source location.

The computational domain includes a large tank surrounding the channel to model a channel with its environment, in order to avoid defining boundary conditions at the top and bottom ends of the channel. The strong coupling between the channel and its environment, which exists in all industrial applications as for the electric heaters, is then implicitly taken into account.

The outline of the paper is as follows. In Sections 2 and 3, we describe the geometrical and physical problem. Sections 4 and 5 details the numerical methods and the validation of the numerical model. Section 6 discusses the influence of the stratification establishing inside the tank on the flow channel. Section 7 analyses the effect of the spatial distribution of several heat sources. Concluding remarks are given in Section 7.
2. Geometry setup

In the present study, we consider the same channel and tank geometry as used in the experimental work of Ospir et al. [13] and Polidori et al. [22]. The channel consists of two parallel planar vertical walls of 376 mm high separated by a gap b (b = 36 mm; $R_f = 5.2$). One wall is composed of a heated central part (height $A = 188$ mm) surrounding by two unheated parts (length $A/2$) while opposite wall remains entirely unheated (Fig. 1). In addition, for a better control of the flow conditions at the entrance, a quarter circle ($R = 36$ mm) is added at the bottom of each wall. The channel is immersed in a vertical tank ($500 \times 500 \times 1000$ mm$^3$) made of 20 mm thick Plexiglas$^®$ plates and filled with water. Despite the high computational cost of this configuration, this configuration is retained for not imposing boundary conditions at inlet and outlet of the channel (Desrayaud et al. [20]). The choice of water as working fluid is considered to be newtonian, and the physical properties ($\rho$, $C_p$, $k$, $\mu$) of the fluid are temperature dependent.

3. Physical setup

The physical model of natural convection flow includes the transport equations of mass, momentum and energy. We use an unsteady approach to reach a quasi-steady-state inside the channel, while the flow inside the tank keeps a transient behavior: a stratification inside the tank appears and the buoyant jet at the top of the channel can oscillates. Consequently, we consider the flow as laminar, incompressible, unsteady and two-dimensional.

The governing equations are:

- **Continuity equation:**
  \[
  \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2}
  \]

- **Momentum equations:**
  \[
  \left( \frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} \right) = -\frac{\partial P}{\partial x} + \frac{\partial}{\partial x} \left( \mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial x} \left( \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial y} \left( \frac{\partial v}{\partial x} \right) - \rho g \tag{3}
  \]

- **Energy equation:**
  \[
  \rho C_p \left( \frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} \right) = \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) \tag{5}
  \]

The fluid is considered to be newtonian, and the physical properties ($\rho$, $C_p$, $k$, $\mu$) of the fluid are temperature dependent.

![Fig. 1. Geometry setup: computational domain (left) and sketch of the channel (right).](image-url)
Indeed, the Boussinesq approximation is not valid for water with temperature variations higher than 3 K [23]. The following polynomial expansions are used [24]:

\[
\rho(T) = 5.3738 \times 10^{-10} \times T^5 - 9.59976 \times 10^{-7} \times T^4 + 6.93809 \\
\times 10^{-4} \times T^3 - 0.255822 \times T^2 + 47.8074 \times T - 2584.53
\]

(6)

\[
C_p(T) = -4.51782 \times 10^{-8} \times T^5 + 7.61613 \times 10^{-5} \times T^4 \\
- 5.12699 \times 10^{-2} \times T^3 + 17.2363 \times T^2 - 2894.85 \times T \\
+ 198532
\]

(7)

\[
k(T) = 5.15307 \times 10^{-11} \times T^5 - 8.15212 \times 10^{-8} \times T^4 + 5.138 \\
\times 10^{-5} \times T^3 - 1.61344 \times 10^{-2} \times T^2 + 2.52691 \times T \\
- 157.532
\]

(8)

\[
\mu(T) = -4.37087 \times 10^{-13} \times T^5 + 7.38482 \times 10^{-10} \times T^4 \\
- 4.99292 \times 10^{-7} \times T^3 + 1.68946 \times 10^{-4} \times T^2 \\
- 2.86313 \times 10^{-2} \times T + 1.94641
\]

(9)

The channel is initially immersed in a tank filled with water at \( T_0 = 289 \) K. Adiabatic and no-slip boundary conditions are applied on the tank walls and unheated walls of the channel. The problem depends on several physical parameters: the aspect ratio of the heated zone \( R_f \), the global Rayleigh number \( Ra \) and the modified Rayleigh number \( Ra^* \), as introduced by Elenbaas [6].

4. Numerical method

Transport equations of mass, momentum and energy (2)–(5) are solved numerically using the finite volume method [25]. This method is based on the spatial integration of transport equations on control volumes. The coupling between velocity and pressure is achieved with the Coupled algorithm that solves the equations of continuity and momentum simultaneously and gives an advantage to treat flows with a strong interdependence between dynamic and thermal fields. Central-Differenced Schemes and Second-Order Upwind Schemes are used for the spatial discretization for diffusive terms and convective terms respectively. The time advance uses the Second-Order Implicit formulation. The convergence criteria were based on the absolute residuals resulting from the integration of the conservation equations over finite control volumes. For all simulations performed in this study, converged solutions were achieved after a residuals decrease larger than \( 10^{-4} \) for all the governing Equations. 2d numerical simulations are performed with ANSYS Fluent® CFD commercial software.

A mesh independence study has been performed to ensure that the solution is independent on the grid size relative to the axial velocity profiles at the inlet and outlet of the channel. For example, in this study several mesh sizes inside the channel were tested (2100, 8400 and 36,600 cells). The optimal mesh between the computation time (CPU) and accuracy is then the mesh with 8400 cells inside the channel which corresponds to 22,000 cells in the whole calculation domain (Fig. 2). During the transient computation, the optimal time step determined with respect to the flow velocity and the mesh size is 0.01 s while the total CPU time is 92 h for each case. The retained mesh of the fluid domain is, on the one hand, structured inside the channel (\( \Delta x = 0.4 \) mm; \( \Delta y = 4 \) mm) and on the other hand, unstructured and progressive outward from the tank.

5. Validation

Before beginning any studies on the dynamic behavior of the flows, it is necessary to check that the numerical model developed

![Fig. 2. Detailed view of the mesh around the channel.](image)
is reliable. To perform this step, numerical results are compared with experimental data (Ospir et al. [13]). The dynamic flow structure was characterized experimentally by laser tomography utilizing discrete tracers, which are fine Rilsan® spherical particles, whose density is close to that of water ($\rho = 1060$ kg m$^{-3}$). This visualization technique allows giving the streamline patterns deduced from the flow velocity field materialized by the trajectories of tracers during a long exposure time of the camera.

In the Fig. 3(a) a comparison of the streamlines between experimentation and numerical simulation is made for the same geometrical and thermal configuration (case 1) at a modified Rayleigh number of $4.5 \times 10^6$ ($\phi = 510$ W/m$^2$) at $t = 30$ min which seems to be a sufficient time to reach the establishment of a steady-state flow inside the channel.

In this comparison study, one may observe that the dynamical flow structures are similar, namely composed by a boundary layer upward flow near the heated wall and a recirculation zone at the channel outlet.

Even if the present study aims at considering a sole channel aspect ratio ($R_f = 5.2$), a complementary quantitative comparison between experimental and numerical experiments on the recirculation lengths was conducted for $Ra^{*} = 4.5 \times 10^6$ and different $R_f$ (Fig. 3(b)). One can observe that both numerical and experimental recirculation sizes $L$ are very close with a maximum uncertainty of 6%.

6. Numerical stratification

In this kind of channel/environment geometry, the outgoing thermal plume necessarily impacts the free surface of the tank. Consequently, a thermal stratification is therefore established in the tank. It seems necessary to have numerically a precise idea of how this thermal stratification establishes in the tank and how it can interact with the channel flow as well.

The thermal stratification has been therefore numerically highlighted at $t = 30$ min within the tank for $Ra^{*} = 4.5 \times 10^6$ in case 1. Figs. 4 and 5 respectively show the velocity profiles measured along a vertical axis positioned between the unheated wall of the channel and the tank edge (Fig. 4) and the thermal field in the tank (Fig. 5).

The temperature increase between $t = 30$ min and the initial temperature field $T_0$ is shown at the Fig. 4. It can be seen from Fig. 4 that no temperature difference occurs in the tank bottom, as illustrated by a constant temperature profile. Rising in to the tank, one can observe a quasi-linear temperature difference evolution in the middle part of the tank reaching a constant temperature difference in the upper part of the tank.

The first conclusion is that the stratification inside the tank is not dependent on the studied thermal cases. This makes sense in that, whatever the case studied, the same thermal power is injected into the same area of the tank.

The second conclusion is that the maximum gap temperature observed is of the order of 0.25 K, which represents 6% of the maximum temperature increase on the heating zone (Table 2), whatever the considered studied case. It is therefore estimated that this bias has a too weak influence to affect both the dynamics and heat flow. As a complement, we present in Fig. 5 the thermal stratification field in the tank. One can observe that the outgoing thermal plume seems to have an oscillating behavior.

7. Results and discussion

The flow dynamics and heat transfer in the asymmetrically heated vertical channel are investigated numerically in the range of the modified Rayleigh number between $2.25 \times 10^6$ and $9 \times 10^6$ (Table 1 for test cases definitions). Only the convective aspects related to natural convection are studied by carrying out numerical simulation in a tank filled with water, which allows neglecting the radiation. The study focuses specifically on the influence heat source distribution at heated wall for three dissipated powers ($P/2; P; 2P$). Numerical transient results are presented after 30 min of heating which is a sufficient time to reach the establishment of a steady-state flow for such a configuration inside the channel [13].
7.1. Flow structure at Ra* = 4.5 \cdot 10^6

Fig. 6 presents the streamline patterns for the five geometrical configurations. As expected, one may observe a boundary layer-type flow which develops near the heated wall due to the fluid alimentation from the bottom of the channel. In the same time, cooler ambient fluid enters the channel from the upper side of the unheated wall. This leads to the formation of a recirculation zone, which can be observed at the channel outlet, near the unheated wall. In some cases, the thermosyphon loop gives rise to a vortex structure characterized by two large recirculation cells, connected by a neck and forming an elongated eight-shaped structure (cases 5). This thermosyphon loop extends over a distance of up to more than half of the channel height (case 1). It is noticed that the eight-shaped structure observed in case 5 has already been highlighted experimentally [13].

To analyze the flow in the channel, different horizontal sections (S1, S2, S3, S4, S5) have been introduced (Fig. 6). Sections S1, S3 and S5 correspond respectively to the inlet (y = 0), middle (y = A) and outlet (y = 2A) of the channel. Sections S2 and S4 correspond to heights of (y = A/2) and (y = 3A/2) respectively.

7.2. Velocity profiles and mass flow rate

In Figs. 7 to 11, the axial velocity profiles for each studied case for Ra* = 4.5 \cdot 10^6 have been plotted in the different sections Si. Fig. 7 presents the axial velocity (Vy) profiles at the channel entry (section S1) for the five cases. One may observe a parabolic profile for the axial velocity whatever the heating distribution is. In cases 2, 4 and 3, one may note that the more the beginning of the heated zone is close to the channel inlet, the more velocities are high [17,18]. Fig. 7 shows also that when the heating source is divided (passage of case 4 in case 5) for a same flux density and with a same power, the velocities increase in accordance with [17]. At this section one may observe parabolic laminar Poiseuille-type profiles, which tend to flatten in the channel center when the heat source is located toward the channel bottom (cases 2 and 5). Nevertheless, it appears significant disparities in the value of the maximum velocity. Moreover, one can see that the more the incident flux density is high and concentrated in the lower part of the channel, higher the maximum velocity is.

Table 2

<table>
<thead>
<tr>
<th>Numerical cases</th>
<th>L, separation length (m)</th>
<th>L/L_{ref} [%]</th>
<th>N_{ref}/N_1 [%]</th>
<th>Mass flow rate [g/s]</th>
<th>T_{Max} Heating wall [K]</th>
<th>T_{Average} Heating zone [K]</th>
<th>T_{Average} Adiabatic wall [K]</th>
<th>T_{Max} Flow outlet [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (Ref)</td>
<td>0.229</td>
<td>100%</td>
<td>100%</td>
<td>28.2</td>
<td>291.7</td>
<td>291.2</td>
<td>289.0</td>
<td>290.1</td>
</tr>
<tr>
<td>2</td>
<td>0.093</td>
<td>41%</td>
<td>60%</td>
<td>42.4</td>
<td>293.0</td>
<td>291.6</td>
<td>289.1</td>
<td>290.0</td>
</tr>
<tr>
<td>3</td>
<td>0.143</td>
<td>62%</td>
<td>65%</td>
<td>23.0</td>
<td>293.1</td>
<td>291.2</td>
<td>289.1</td>
<td>290.3</td>
</tr>
<tr>
<td>4</td>
<td>0.165</td>
<td>72%</td>
<td>65%</td>
<td>30.5</td>
<td>293.0</td>
<td>291.2</td>
<td>289.1</td>
<td>290.1</td>
</tr>
<tr>
<td>5</td>
<td>0.179</td>
<td>78%</td>
<td>70%</td>
<td>33.8</td>
<td>292.7</td>
<td>291.1</td>
<td>289.1</td>
<td>290.2</td>
</tr>
</tbody>
</table>
Fig. 8 shows the velocity profiles in Section 2. In this figure one can see parabolic velocity profiles for cases 3 and 4. However, for cases 1, 2 and 5 where the heating process occurs at this location, the maximum velocity is shifted to the hot wall of the channel. Consequently, the observed change in the slope at the origin is certainly due to the initiation of the thermal boundary layer having the effect of accelerating the flow in the vicinity of the heated wall.

One may also note in Section 2, an identical distribution of the velocity profiles as in Section 1, namely greater velocities for case 2 and lower velocities for case 3. For the cases were the heating zone starts in section S2 (cases 1, 2 and 5) the inlet parabolic velocity profiles are modified due to the development of a thermal boundary layer.

All profiles of Fig. 9 illustrate a fully developed boundary layer type flow with the exception of case 3 where one may find the previously discussed situation with emergence of a thermal boundary layer in this study section (parabolic profile shifted to the hot wall).
Indeed, case 2 where the whole thermal power has been supplied related to the thermal power injected upstream of the study area.

For the cases 2, 3 and 4 velocity values are always positives traducing the presence of a return flow.

One may note that the maximum velocity values are closely related to the thermal power injected upstream of the study area. Indeed, case 2 where the whole thermal power has been supplied to the flow gives the higher maximum velocity value.

For cases 1, 4 and 5 where half of the thermal power is injected to the flow at the S3 abscissa maximum velocity values are observed to be less those for case 2.

Comparing cases 1, 4 and 5 leads to the observation that case 5 gives the highest velocity due to the fact that its thermal power is separated into bottom and top of the heated area. A contrario, case 4 that corresponds to the case where the heat flux is concentrated on the upper part of the heating area, gives the lowest maximum velocity. For case 1, which represents a more homogeneous distribution throughout the first part of the heating area one observes an intermediary situation. This demonstrates that the way the thermal power is injected to the flow highly influences maximum velocities within the channel.

In Fig. 10, one may note a similar behavior in the velocity profiles between cases 1 and 5 presenting a symmetrical heating along the channel height.

Overall, one may find that in Figs. 10 and 11 corresponding to study areas approaching the exit of the channel, the convection flow acts like a fully developed boundary layer one whatever the case studied. The position and layout of heat sources then tend to not have any influence on the profiles of the vertical velocities near the heated wall. In addition, the same width of the recirculation zone is observed for cases 1, 3, 4 and 5.

Comparing the velocity profiles in sections S4 and S5 allows highlighting the influence of the adiabatic upper portion of the heated wall. Indeed, one may observe that this upper adiabatic zone tends to erase differences on the maximum velocities values while on the contrary the presence of an adiabatic zone at the channel inlet tends to accentuate them (zones S1 and S2).

In Table 2, the mass flow rate has been calculated at the channel inlet for Ra* = 4.5 × 10^6. For this dissipated power (P), case 3 is the case where the mass flow rate is the lowest. It corresponds to the case where the heating is focused at the upper half part of the channel while case 2, where the heating is focused on the downer half part of the channel, gives a higher mass flow rate. This confirms the observed trends in the velocity profiles obtained in sections S1, S2 and S4. Moreover, if cases 1, 4 and 5 are compared, one may note that the spacing between the heating sources influences the mass flow rate through the channel. For equal thermal power incident with a heating zone located in the middle of the channel (cases 1 et 4), decreasing the length of the heated zone leads to increase the mass flow rate up to 7%.

Comparing cases 2, 3 and 4 which present an equivalent heating length it is found that the more the heating area is located close to the channel inlet, higher the mass flow rate is, in accordance with results obtained by Menezo et al. [19]. For example, there is an increase of the mass flow rate between cases 2 and 3 in the order of 46%.

Comparing cases 4 and 5 which have a symmetrical heating of the same length, it is found that spreading heat sources (ε = 0 for case 4 and ε = A/2 for case 5) allows to increase the flow rate up to 10%. Similar trend has already been mentioned by Dehghan et al. [17].

In Table 3 one finds the values of the mass flow rate for all studied cases in the range of the modified Rayleigh number between 2.25 × 10^6 and 9 × 10^6. First of all, it can be noted that, whatever the power dissipated, the greater mass flow rate is found in the case 2, wherein the heating source is located in the lower part of the channel. At the same dissipated power, this is due to the application of the highest heat flux density in the lower part of the canal, which leads to a pressure drop in the channel. Furthermore, separating the heat sources (case 1 to case 5) for the same dissipated power, the mass flow rate increases.

7.3. Heat transfer

Fig. 12 represents the evolution of the wall temperature (T-T0)

<table>
<thead>
<tr>
<th>Table 3</th>
<th>Mass flow rate for three dissipated powers.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dissipated power</td>
<td>P/2</td>
</tr>
<tr>
<td>Case 1</td>
<td>20.3</td>
</tr>
<tr>
<td>Case 2</td>
<td>33.6</td>
</tr>
<tr>
<td>Case 3</td>
<td>35.5</td>
</tr>
<tr>
<td>Case 4</td>
<td>36.3</td>
</tr>
<tr>
<td>Case 5</td>
<td>27.4</td>
</tr>
</tbody>
</table>
along the left heated wall for $Ra^* = 4.5 \times 10^6$. One may note that for all studied cases the evolution of the parietal temperature along the heat sources scales as $y^{1/4}$ (Fig. 13) as found for uniform heat flux (UHF) in natural convection on a vertical plate in an infinite medium, followed by a thermal relaxation zone. In order to determine this power law, we make dimensionless the temperature increase ($T^+$) and the height ($H^+$) from the heated wall, for three typical cases, which each have a different heating typology.

$$T^+ = \frac{T - T_{\min}}{T_{\max} - T_{\min}}$$

$$H^+ = \frac{y}{H_{Hz}}$$

![Fig. 13. Influence of the spatial arrangement of the heat sources on the dimensionless temperature profiles along the heated zone for $Ra^* = 4.5 \times 10^6$.](image)

As example, the case 1 comprises a uniform heating over the entire heated area, case 4 is made of a uniform heating about half of the heated zone and the case 5 consists of heating with a maximum distance of the source. One notices a very similar behavior for cases 2, 3 and 4 having the same heating length. However, it is noted that the maximum temperature value increases when the heat source is close to the channel outlet. The observed difference in temperature is then about for $4\%$.

When the length of the heated zone is doubled with a lower flux density as in case 1, there is a situation where the temperatures are lower. Moreover, for case 5 with a heat source-spreading situation one can observe a sawtooth-shaped temperature profile having the effect of decreasing the maximum temperature on the heated wall.

Table 2 provides thermal and dynamic calculation data for $Ra^* = 4.5 \times 10^6$. To appreciate the heat transfer evolution versus the different thermal cases in the channel, Table 2 shows the values of $Nu$ (Eq. (15)) calculated on the central part of the heated left wall. Also have been added the values of the recirculation zone lengths. All these data have been normalized by taking as reference case 1 which is the symmetric case already studied experimentally by Ospir et al. [13] and Polidori et al. [22].

The average Nusselt number ($\overline{Nu}$) is defined as the ratio of convective to conductive heat transfers, which characterizes the heat transfer.

$$\overline{Nu} = \frac{1}{A} \int_0^L \frac{\varphi}{k(T_w - T_0)} dy$$

According to Table 2, there is a strong relationship between the recirculation length $L$ and the average Nusselt number $\overline{Nu}$. It is observed that the heat transfer is maximal for the case 1 which has the longest recirculation zone. Moreover, case 1 gives the most homogeneous temperature at wall. Indeed, in case 1, the maximum temperature of the heated zone is very close to the average temperature of the heated zone. Case 3 is the less homogeneous case and leads to the greater difference between the maximum temperature and the average temperature at wall.

In conclusion, the heating sources distribution allowing the increase of the mass flow rate through the channel corresponds to case 2 (Table 2), which presents a heating zone at the bottom of the channel. This case leads the lowest calculated average Nusselt number. A contrario, case 3 corresponding to the heating zone in the upper part of the channel, gives the lowest mass flow rate and thus the largest maximum temperature, at the channel output.

In Table 4 one finds the values of the average Nusselt number for three different dissipated powers ($P/2, P$ and $2P$) in the range of the modified Rayleigh number between $2.25 \times 10^6$ and $9 \times 10^6$. One can see that the convective heat transfer increases with the dissipated power. Moreover, one may note that, for the great dissipated power ($P$ and $2P$) and lowest dissipated power ($P/2$) the convective heat transfer is maximal respectively minimum for the case 1. Also, for cases with the lowest dissipated power ($P/2$), one can observe a maximal convective heat transfer for case 5 with a separate heat source.

8. Conclusions

The design of electric heaters made with respect to the European standards for an industrial application imposes strong industrial constraints related to different heater components to reduce the temperature on accessible surfaces of the heater. In order to simplify the design of electric heaters, this paper has analyzed the natural convection flow in a simplified electric heater with different heating configurations with varying the location of electrical resistances. This numerical study focuses on both heat transfer and flow dynamics resulting for three different dissipated powers.

### Table 4

<table>
<thead>
<tr>
<th>Dissipated power</th>
<th>$P/2$</th>
<th>$P$</th>
<th>$2P$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>5.6</td>
<td>20.0</td>
<td>33.8</td>
</tr>
<tr>
<td>Case 2</td>
<td>11.2</td>
<td>13.0</td>
<td>15.2</td>
</tr>
<tr>
<td>Case 3</td>
<td>11.3</td>
<td>12.9</td>
<td>15.1</td>
</tr>
<tr>
<td>Case 4</td>
<td>11.3</td>
<td>13.0</td>
<td>20.6</td>
</tr>
<tr>
<td>Case 5</td>
<td>12.5</td>
<td>14.4</td>
<td>16.8</td>
</tr>
</tbody>
</table>

![Fig. 12. Temperature profiles (T-T₀) along the channel heating wall for $Ra^* = 4.5 \times 10^6$.](image)
powers \((P; 2P\text{ and } 2P)\) in the range of the modified Rayleigh number between \(2.25\times10^{8}\) and \(9\times10^{8}\) in an asymmetrically heated vertical channel. In order to neglect radiation, only the convective aspects related to natural convection are studied by carrying out simulations in water.

In this study, regardless of the dissipated power, was highlighted the fact that the heat distribution has a strong impact on the flow structure and convective heat transfer in an asymmetrically heated vertical channel. The conclusions are as follows:

1. Separating the heat sources for the same dissipated power, the mass flow rate increases.
2. Spreading heat sources allows to increase the convective heat transfer for low dissipated power.
3. The maximum temperature value increases when the heat source is close to the channel outlet.
4. The convective heat transfer increases with the dissipated power and is maximal for the case 1.
5. Upper adiabatic zone tends to erase differences on the maximum velocities values while on the contrary the presence of an adiabatic zone at the channel inlet tends to accentuate them.
6. The more the position of the heating source is lower in the channel, the more mass flow rate coming in through the bottom of the channel is important whatever is dissipated power.

Acknowledgments

This work is financially supported by the European Union (EU) through the European Regional Development Fund (FEDER), the Champagne-Ardenne region and the company CAMPA as part of Effi-Siemce project.

References