SEMARAK ILMU PUBLISHING

Journal of Advanced Research in Numerical Heat Transfer

Journal homepage: https://semarakilmu.com.my/journals/index.php/arnht/index
ISSN: 2735-0142

Opposing Mixed Convection in an Open Parallelogram Cavity with the Horizontal Channel: Effects of the Heat Source Length and Location

Ahmed Kadhim Hussein[1], Amaal Abdul Razaq Abdul Hussein[2], Awatef Abidi[3], Ali Basem[4], Farhan Lafta Rashid[5,*], Mohamed Bechir BEN HAMIDA[6,7,8], Uddhaba Biswal[9,10], Bagh Ali[11], Sajjad Firas Abdulameer[12,13]

1 College of Engineering -Mechanical Engineering Department - University of Babylon - Babylon City – Hilla, Iraq
2 Babylon Governorate - Directorate of Babylon Sewerage- Hilla, Iraq
3 Physics Department, College of Sciences Abha, King Khalid University, Saudi Arabia
4 Air Conditioning Engineering Department, Faculty of Engineering, Warith Al-Anbiyaa University, Iraq
5 Petroleum Engineering Department, College of Engineering, University of Kerbala, Karbala, Iraq
6 College of Engineering, Department of Mechanical Engineering, Imam Mohammad Ibn Saud Islamic University (IMSIU), Riyadh, Saudi Arabia
7 Research Laboratory of Ionized Backgrounds and Reagents Studies (EMIR), Preparatory Institute for Engineering Studies of Monastir (IPEIM), University of Monastir, Monastir City, Tunisia
8 Higher School of Sciences and Technology of Hammam Sousse, Department of Physics, University of Sousse, Sousse, Tunisia
9 Department of Mathematics, National Institute of Technology Rourkela, Rourkela 769008, Odisha, India
10 Department of Mathematics, Laxminarayan College, Jharsuguda, 768201, Odisha, India
11 School of Mechanical Engineering and Automation, Harbin Institute of Technology, Shenzhen 518055, China
12 Scientific Research Center, Al-Ayen University, Thi-Qar, Iraq
13 Civil Engineering Department, College of Engineering, University of Kerbala, Karbala, Iraq

ARTICLE INFO

Article history:
Received 21 July 2023
Received in revised form 25 August 2023
Accepted 22 September 2023
Available online 6 January 2024

Keywords:
Opposing mixed convection; parallelogram cavity; horizontal channel

ABSTRACT

In this work, a numerical simulation was used to investigate the impact that the length and positioning of the heat source had on the mixed convection opposing flow that occurred within the horizontal channel that included an open parallelogram cavity. When the wall being heated is facing away from the direction of the incoming force. Different placements of the heat source along the cavity's sidewalls were explored, with the length of the heat source set at (ε) (0.25<ε<1). The cool, steady-speed air came in through the sides of the canal. All other walls are adiabatic, while the vertical walls on the inflow and outflow sides are isothermal. The governing equations were solved using the finite element technique. For several different values of the Richardson number (Ri=0.1-100), we estimated the flow and heat fields. While the Prandtl number is held at 0.71 and the Reynolds number is maintained at 100. The average Nusselt values, as well as the findings of the flow and temperature fields, were reported. The findings demonstrate that both the Richardson number (Ri) and the distance from the heat source (ε) positively affect the heat transfer rate. It was also determined that for all Richardson numbers, the highest average Nusselt number is attained at the higher portion of the right wall of the hollow.

* *Corresponding author.*
E-mail address: farhan.lefta@uokerbala.edu.iq (Farhan Lafta Rashid)

https://doi.org/10.37934/arnht.14.1.118135

1. Introduction

Because the mixed convection (also called combined convection) is used in so many different kinds of industrial and technical processes. Ingot quenching, energy extraction, polymer crystallization, and float glass processing are just some of the many applications of mixed convection in cavities that have been reviewed in the literature recently. Compact heat exchangers, furnaces, nuclear reactors, and the food sector are a few more places you could see heat exchangers in use. In order to improve heat transmission in cavities, researchers have conducted a number of numerical and experimental investigations, all of which demonstrate how to do so by various means (such as altering the cavities' form or tilt). Air mixed convection within a conduit with a U-shaped open cavity was investigated statistically by Mancaet *et al.,* [1]. All other walls were maintained adiabatically cold, and cold air was introduced into a horizontal channel with a constant heat flux at three distinct heating modes. Experiments were conducted with a range of Reynolds (Re=100 and 1000), Richardson ($0.01\leq$ Ri≤100), and inflow/outflow opening height to heat source length (H/D) ratios ($0.1\leq H/D\leq1.5$). The maximum temperature was found to be lower when (Re) and (Ri) rose.

Mixed convection of air that entered the duct from the left at a constant speed and a low temperature was later experimentally tested by Manca *et al.,* [2]. The left wall of the channel or hollow is assumed to be heated uniformly at heat flux q ($50\leq q\leq250 W/m^2$). The findings were shown for various values of the aspect ratio of the cavity ($0.5\leq AR\leq1.5$) and the ratio between the height of the channel and the height of the cavity ($0.5\leq H/D\leq1$). The range of Richardson numbers considered was (30-110) for (Re =1000) and (2800-8700) for (Re =100). It was discovered that, for all values of (Re) and (Ri) considered, the (Nu_{av}) number rises as the cavity's aspect ratio rises. Convection within the channel and convection in a cavity heated from below were both examined numerically by Leong *et al.,* [3]. For ($1\leq Re\leq2000$), ($0\leq Gr\leq10^6$), and ($0.5\leq AR\leq 4$), the findings were shown. They determined that the flow field was governed by the ratio of Reynolds to Gustafson numbers (Re/Gr). For ($0.01\leq Ri\leq100$) and ($1\leq AR\leq5$)at ($Gr=10^4$), Numerical findings for a rectangular hollow with heat sources on its left, right, and bottom sides were provided by Aminossadati and Ghasemi [4]. Increasing the cavity's aspect ratio was shown to boost heat transfer for all three heat source sites while maintaining a constant (Ri) value.

The coupled convection in a horizontal channel with a rectangular chamber heated from below was explored statistically by Rahman *et al.,* [5]. The channel-cavity assembly was exposed to a magnetic field from its right side. For ($0\leq Ha\leq20$), ($10^3\leq Ra\leq10^5$), ($100\leq Re\leq500$), and Pr=0.71, the corresponding numerical findings were shown. They determined that a larger value for (Ha) resulted in a smaller (Nu_{av}), whereas larger values for (Re) and (Ra) resulted in larger (Nu_{av}) values.

Mixed convection in an open rectangular cavity in a horizontal channel was studied numerically by Rahman *et al.,* [6]. The rectangular opening had a hollow cylinder at its core, which was heated uniformly from the inside. While the (Re) was held constant at 100, the findings were shown over a range of ($0.2\leq K\leq50$) and ($10^3\leq Ra\leq10^5$) using a variety of fluids with Pr values ($0.7\leq Pr\leq7$). A higher (Ra) number and higher thermal conductivities were found to raise the (Nuav) at the heated surface, but a higher (Pr) number had no effect.

Laminar and steady forced and natural convection in a 2-dimensional channel with an open triangular heated chamber below it was investigated numerically by Rahman *et al.,* [7]. Joule heating and the effects of a magnetic field were also studied. The study used the Joule parameter ($0\leq J\leq 5$) and the specific heat capacity ($10\leq Ha \leq100$) of a wide range of fluids ($1\leq Pr\leq10$), with illustrative findings shown for ($10^3\leq Ra\leq10^5$) and ($100\leq Re\leq2000$). Heat transmission was shown to improve when both (Re) and (Pr) were raised. While raising (Ha) and (J) lowered it. A numerical analysis of mixed convection in a square enclosure with a forced air flow entering from the channel above it was given by Carozza *et al.,* [8]. The cage was set up with hot and cold temperatures on the left and right walls. Numbers in the range (Ri=0.1-1.7x10^4) and (Re=10-1000) were used in the study. We looked at two scenarios (flow that helps and flow that hinders). They concluded that in both situations, increasing (Re) increased the (Nuav) value.

Selimefendigil [9] conducted a numerical investigation of laminar mixed convection in a two-dimensional air-filled channel connected to a square open cavity, heating the cavity from below and to the left. The bottom wall of the hollow was maintained at an isothermal high temperature while the other walls of the cavity and channel were presumed thermally insulated in order to get the findings shown here for Ri=0.01 to 20 and Re=400 to 800.At (Re=800), the researchers found that the (Nu_{av}) number was less for (Ri=10) than for (Ri=5). The numerical and experimental research of numerically mixed convection of water (Pr=7) in a cubical cavity at the base of a square channel was conducted by Abdelmassih *et al.,* [10]. Except for the bottom wall of the hollow, which was heated, the other walls of the cavity and the channel were considered to be adiabatic, and results were reported for (100≤Re≤1500) and (0.1≤ Ri ≤10). For (Re ≥ 500) and (Ri = 1), it was determined that the flow was unstable. However, at (Ri = 10), it became unstable for all (Re) values.

Unsteady MHD mixed convection within a horizontal channel with an adiabatic square obstruction and an open cavity heated from below was numerically studied by Hussain *et al.,* [11]. Al2O3-Cu water hybrid nanofluid was used to fill the void. The chilly, uniformly moving fluid entered the tube from the left. The rest of the space within the hollow and along the channel sides was assumed to be thermally insulated. The barrier was positioned at three distinct heights. The results in terms of various values of the solid volume fraction (0.0≤Ø≤0.04), (0.01≤Ri≤20), (0≤Ha≤100), (1≤Re≤200) were considered. They deduced that, the increase in (Ri), (Re) and (Ø) increased the rate of heat transfer and entropy generation. Also, it was observed that the flow was deviated to the channel when (Ha) number increased.

Garca *et al.,* [12] quantitatively modelled the mixed convection of water in a slanted rectangular duct with two symmetric open cubic chambers on opposite sides. It was assumed that the walls of the cavities that faced the entrance were also isothermal. It was believed that the enclosing walls of the channel and cavities were not adiabatic. We looked at these characteristics for the ranges (100≤Re≤1000) and (0.01≤Ri≤100) and angles 0°≤γ≤ 90°). They came to the conclusion that the double cavity's flow and temperature fields, heat transfer characteristics, and so on all depend heavily on the cavity's acoustic resonant (AR) properties. Results for (Ri=0.1-100) and (ε =0.16-1) at Re=100 and Pr=0.71 from a numerical simulation by Laouira *et al.,* [13] show the effects of heat source length on combined convection in a channel with an open trapezoidal cavity heated at its base by a discrete heat source. They drew the conclusion that more Nu_{av} meant more Nuav.

In a horizontal channel with an open trapezoidal cavity located below it, Mebarek- Oudina *et al.,* [14] performed numerical simulations of 2D laminar mixed convection of air. A finitely long, locally heated source was applied to the cavity's insulated walls. At (Re=100) and (Pr=0.71), the results were shown for a range of (0.1≤Ri≤100) numbers and locations of heat sources (ε=0.75). It was discovered that the best heat transmission occurred when the heat source was positioned high on the left wall. The (Nuav) number was also raised in tandem with the (Ri) rise.

Two-dimensional laminar mixed convection in a hollow connected to a horizontal channel was investigated statistically by Ahmadi and Farsani [15]. A cavity-enclosed conduit was subjected to a two-phase flow of a non-Newtonian fluid. It was decided to use a polymer solution and water in a two-phase flow. All the walls of the hollow and channel were assumed to be thermally insulating except for the bottom wall, which was kept at an isothermal high temperature. Fluids in the channel and cavity were shown to be more sensitive to fluctuation in (Re) than in the rheological index (n), as measured by changes in velocity, pressure, and volume fraction. With an open complicated cavity and a discrete heating source at its bottom wall, Al-Farhany *et al.,* [16] examined numerically the coupled convection within the channel. The air flowed in from the left side of the canal. At the same time, the right side of the channel-cavity assembly was exposed to the a magnetic field. The data for (Ha=0-15), (Ri=0.1-10), (Re=100), and (Pr=0.707) were shown. They reasoned that when (Ri) rose, the (Nu_{av}) would rise, and as (Ha) fell, the (Nuav) would fall. Mixed convection in a channel coupled to an open cavity is discussed in more detail in Ref. [17–56]. Considering the research done, the aforesaid. To the best of our knowledge, there is a dearth of literature on the topic of mixed convection in a channel assembly with an open enclosure.

The purpose of this study is to investigate the impact that a discrete heat source with a variable length and position has on the predominant mixed convection that occurs in a channel that has an open parallelogram cavity. A tunnel was cut into the cavity's upper wall, and its walls were heated using a (0.25<ε<1) watt resistive heater. The channel is a horizontal entrance for cold air. Numerical research on the impact of heated wall position and length on air mixed convection throughout a broad Richardson number range was conducted. While the Prandtl and Reynolds numbers are held constant at (Pr = 0.71) and (Re = 100), respectively, the findings are given as contours of velocities, isotherms, and Nusselt numbers.

2. Geometry Description and the Governing Equations

The geometry under studied is shown in Figure 1. The channel of diameter (D) combined with an open cavity. Le, the extra-cavity channel length, equals 1H. The cavity's height (H) and length (L=2H) are shown. The cold air, travelling at a constant speed (u_{in}) and temperature (Tc), enters the duct horizontally from its left side. With discrete heat sources in two locations (the centre of the right wall and the upper region of the same wall with different lengths of heat source), the right sidewall of the cavity was heated to a high temperature (Th), while the other walls were assumed to be thermally inert. The oversimplifications made for the sake of this investigation:

i. Laminar, incompressible, steady, Newtonian, two-dimensional flow.
ii. The effects of radiation and heat production are disregarded.
iii. The air's thermo-physical properties remained unchanged. Additionally, the density-temperature dependence was resolved using the Boussinesq approximation.

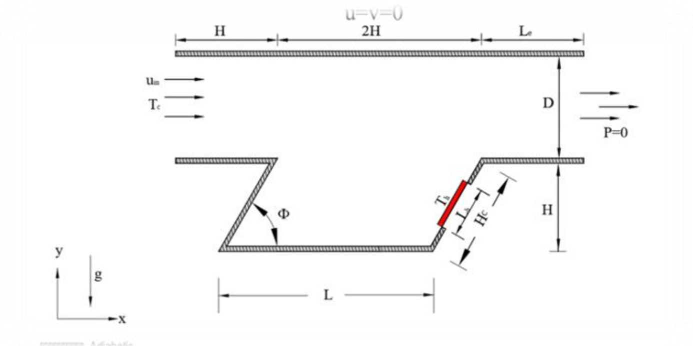


Fig. 1. Problem geometry

The non-dimensional version of the governing equations in this study are presented as Ref. [1,4] in Cartesian coordinates.

$$\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}=0 \tag{1}$$

$$U\frac{\partial U}{\partial X}+V\frac{\partial U}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{1}{Re_{in}}\left(\frac{\partial^2 U}{\partial X^2}+\frac{\partial^2 U}{\partial Y^2}\right) \quad (2)$$

$$U\frac{\partial V}{\partial X}+V\frac{\partial V}{\partial Y}=-\frac{\partial P}{\partial Y}+\frac{1}{Re_{in}}\left(\frac{\partial^2 V}{\partial X^2}+\frac{\partial^2 V}{\partial Y^2}\right)+Ri\theta \quad (3)$$

$$U\frac{\partial \theta}{\partial X}+V\frac{\partial \theta}{\partial Y}=\frac{1}{Re_{in}Pr}\left(\frac{\partial^2 \theta}{\partial X^2}+\frac{\partial^2 \theta}{\partial Y^2}\right) \quad (4)$$

The following non-dimensional variables were used to rewrite Eq. (1) through Eq. (4):

$$X,Y=\frac{x,y}{H} \quad , \ U,V=\frac{u,v}{u_{in}} \quad , \theta=\frac{T-T_c}{T_h-T_c} \quad , P=\frac{p}{\rho u_{in}} \quad , Pr=\frac{\vartheta}{\alpha} \quad , Re_{in}=\frac{\rho u_{in}H}{\mu}$$

$$\varepsilon=\frac{L_H}{H}, \ \ Ri=\frac{Gr}{Re_{in}^2}=\frac{gH\beta(T_h-T_c)}{u_{in}^2} \quad (5)$$

The Richardson number is the ratio of natural convection to forced convection, which is an important factor to keep in mind. The current research shows that buoyant force created by cold and hot temperature is responsible for the natural convection effect.The channel flow, however, is responsible for the forced convection term in the Richardson number.

These are the boundary conditions:

- Channel inlet:

$X=0$ $\quad H\le Y\le H+D$ $\quad \Theta=1, \ U_{in}=1$ (6)

- Outlet of channel:

$X=4H$ $\quad H\le Y\le H+D$, $\quad \frac{\partial \theta}{\partial X}=0, \quad \frac{\partial U}{\partial X}=\frac{\partial V}{\partial Y}$, $P=0$ (7)

- On the heater: θ=1, otherwise, 0n $\left(\frac{\partial \theta}{\partial n}\right)=0$, where (n) is the normal vector.
- On the solid fixed walls: U = V = 0. (8)

The flow field within the horizontal channel with an open horizontal cavity may be characterised by the dimensionless stream function (Ψ), which is generated from the dimensionless velocity components (U and V).

$$U=\frac{\partial \Psi}{\partial Y} \quad and \quad V=-\frac{\partial \Psi}{\partial X} \quad (9)$$

Where $\Psi=\frac{\psi}{\alpha}$

Since this is the case, we may combine Eq. (9) with the continuity equation (Eq. (1)) to get Poisson's equation:

$$\frac{\partial^2 \Psi}{\partial X^2}+\frac{\partial^2 \Psi}{\partial Y^2}=\frac{\partial U}{\partial Y}-\frac{\partial V}{\partial X} \quad (10)$$

Finding (Ψ) requires first determining (U, V), and then solving Eq. (10) numerically with proper boundary conditions. In order to get the mean Nusselt number, we may integrate the local temperature gradient around the hot source, which gives us:

$$Nu_{av} = -\frac{1}{L_h}\int_0^{L_h}\frac{\partial\theta}{\partial X}\,dY \tag{11}$$

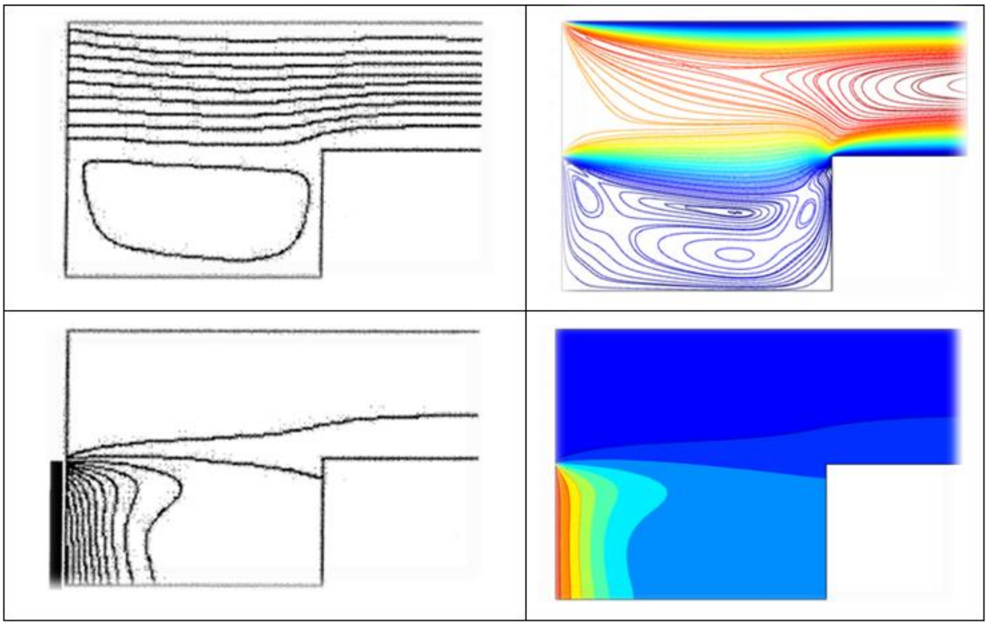
Where L_h is the hot source length (ε)

3. Validation and Grid Independency Analysis

Novel algorithms, calculation techniques, and user-friendly and powerful interfaces have all been developed as a consequence of the development of scientific computing. These improvements, in conjunction with the rapid development of technology, will lead to increased software efficiency and a decrease in the required time for numerical computations. The numerical findings for the present study were calculated using COMSOL. In order to solve the partial differential equations (i.e., Eqs. (1)-(4)) with boundary conditions, it is necessary to transform them into linear algebraic equations. Algebraic linear equations are used to describe the physical domain of the issue, which has been discretized into multiple parts connected by nodes. By plugging in the approximations into the Navier-Stokes equations, the residuals of each conservation equation may be calculated. The first step in choosing the optimal grid size for a calculation is verifying grid independence. The average Nusselt number for each heat source was used to choose which of five element numbers to use in the experiment. Table 1 shows the results of testing with a total of six grid sizes (Ri=0.1, 1, 10, 100, Pr =0.7, and Re = 100). The Nu_{av} values for G6 (44316 elements) grids were found to be within a little margin of error (< 0.9%) from one another. Due to its time-economy benefit and minimal variances in the average Nusselt number, a grid size of G6 (44316 elements) is used in the numerical solution. As shown in Figure 2, the current model was used to re-solve the examples reported by Manca *et al.,* [2], and a good agreement was found between the two sets of findings. Figure 3 shows the generated grid G6.

Table1
Variation of the average Nusselt number Nu_{avg} for various grids for heating from below at (Pr =0.7, ε = 1, Ri = 0.1, 1, 10,100and Re=100)

Grid	Ri							
	Ri=0.1	[Error%]	Ri=1	[Error%]	Ri=10	[Error%]	Ri=100	[Error%]
G1(1928)	2.4178	___	3.2635	___	5.1522	__	8.9287	___
G2(2988)	2.4279	0.416	3.2671	0.1102	5.0953	1.1044	8.4527	5.332
G3(4852)	2.4349	0.2875	3.2719	0.1467	5.0684	0.528	8.1844	3.1742
G4(11680)	2.4586	0.964	3.2981	0.7944	5.0845	0.31665	8.1302	0.663
G5(28634)	2.47207	0.5448	3.3119	0.41668	5.0955	0.21587	8.0517	0.965
G6(44316)	2.4702	0.0756	3.3113	0.01812	5.0916	0.0765	7.9805	0.884



(a) Manca *et al.,* [2] (b) Present study

Fig. 2. Streamlines and isotherms compared to Manca *et al.,* [2] 's numerical analysis

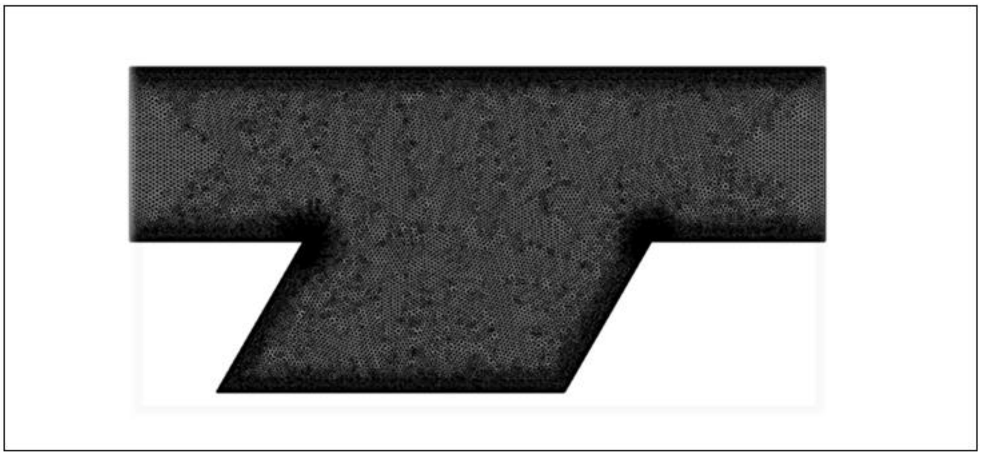


Fig. 3. The generated grid G6 (44316 elements)

4. Discussion of Results

Using a heat source with a length of ε ($0.25 \le \varepsilon \le 1$), an aspect ratio of (AR=2), and Reynolds and Prandtl numbers of (Re=100) and (Pr=0.71), we present and discuss results for 2D laminar combined convection heat transfer and air flow inside the channel with the cavity from streamline, isotherm contour, and (Nu_{av}).

4.1 Flow and Thermal Field Characteristic

4.1.1 Richardson Number (Ri) effect

The effect of (Ri) on streamline and isotherm contours for opposing flow case was explained in Figure 4 – 6. This effect was studied for ($0.25 \le \varepsilon \le 1.0$) and at (Re=100 and AR=2). As expected, the variation in (Ri) has a clear effect on both the flow and thermal fields inside the channel-enclosure assembly. Therefore, the increase of it from (Ri=0.1) to (Ri=100) causes to a dramatic increase in flow disturbance inside the assembly. This can be evident from the growth of the flow vortices and the increase in their numbers. This behaviour was repeated for all considered range in (ε). Also, the increase in (Ri) accelerates from the process of the thermal exchange inside the assembly and makes the air leaving process more fast. The same positive effect of the increase in (Ri) can be seen on isotherm contours. So, the increase in it leads to increase the cold regions inside the assembly as a signal a good flow mixing. Also, it increases the intensity of isotherms above the heat source compared with the case at low values of (Ri). Moreover, the isotherm contours are accumulated adjacent the heat source location in the middle of the right sidewall of the enclosure. This is due to the sever temperature gradient at this region.

4.1.2 Effect of the heat source length (ε)

Figure 4 – 6 illustrate also the effect of (ε) on the flow and thermal fields for opposing flow case. The result indicated that, the increase in (ε) from (ε=0.25) to (ε=1.0) does not effect on the flow field pattern when the forced convection is dominant (i.e., Ri=0.1). While, at (Ri=1) or when the natural convection becomes equivalent to the forced convection, the increase in (ε) leads to construct a minor vortices adjacent the heat source location in the right sidewall of the enclosure. Now, with the increase in (Ri), a clear change in the flow pattern in both the channel and enclosure can be noted, therefore, it can be concluded that the increase in (ε) has a positive contribution on the flow mixing between the channel and the enclosure especially at high value of (Ri).

With respect to the effect of (ε) on the thermal fields. This results show that at (Ri=0.1), the increase in (ε) leads to extend the thermal plume far away from the heat source location towards the left sidewall leading to increase the hot regions inside the enclosure. But, at (Ri=0.1) the thermal plume begins to retard towards the heat source location and this retardation becomes more slow with the increase in (ε). Now, further increase in (Ri) causes the effect of (ε) restricts with the region above the location of the heat source. So, it can be noted from Figure 4 – 6**,** that the intensity of the thermal plume at this region increases with the increase in (ε) from (ε=0.25) to (ε=1.0). Therefore, it can be deduced that the increase in (ε) helps the thermal plume to leave more fast through the channel exit especially in the natural convection domain. Also, the increase in (ε) leads to increase the rate of the heat generation and increases the activity of the heat transfer by the natural convection due to the increase in the buoyancy force.

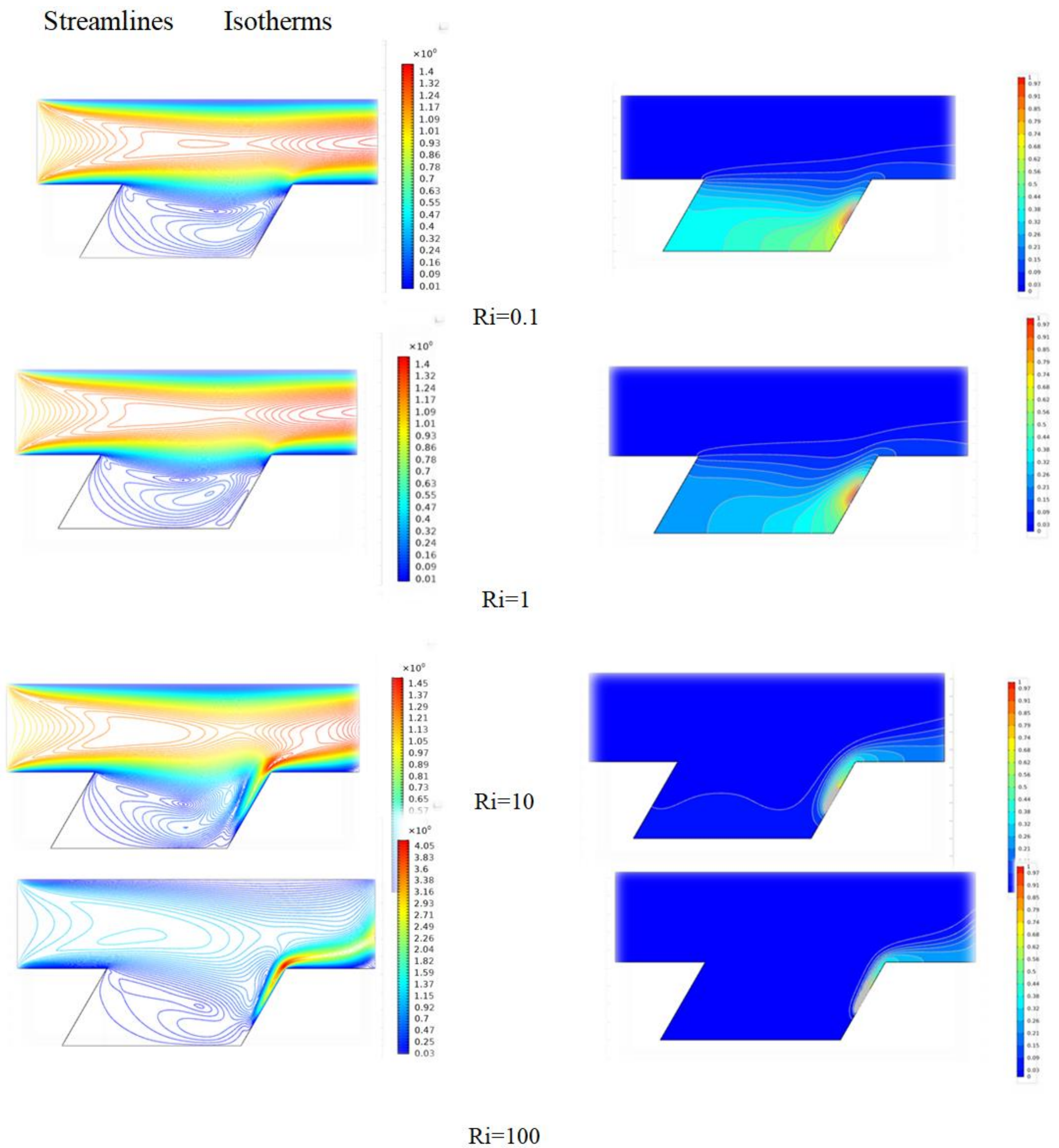


Fig. 4. Streamlines and isotherms for different Richardson numbers of opposing flow at 0.25 (centre of right wall)

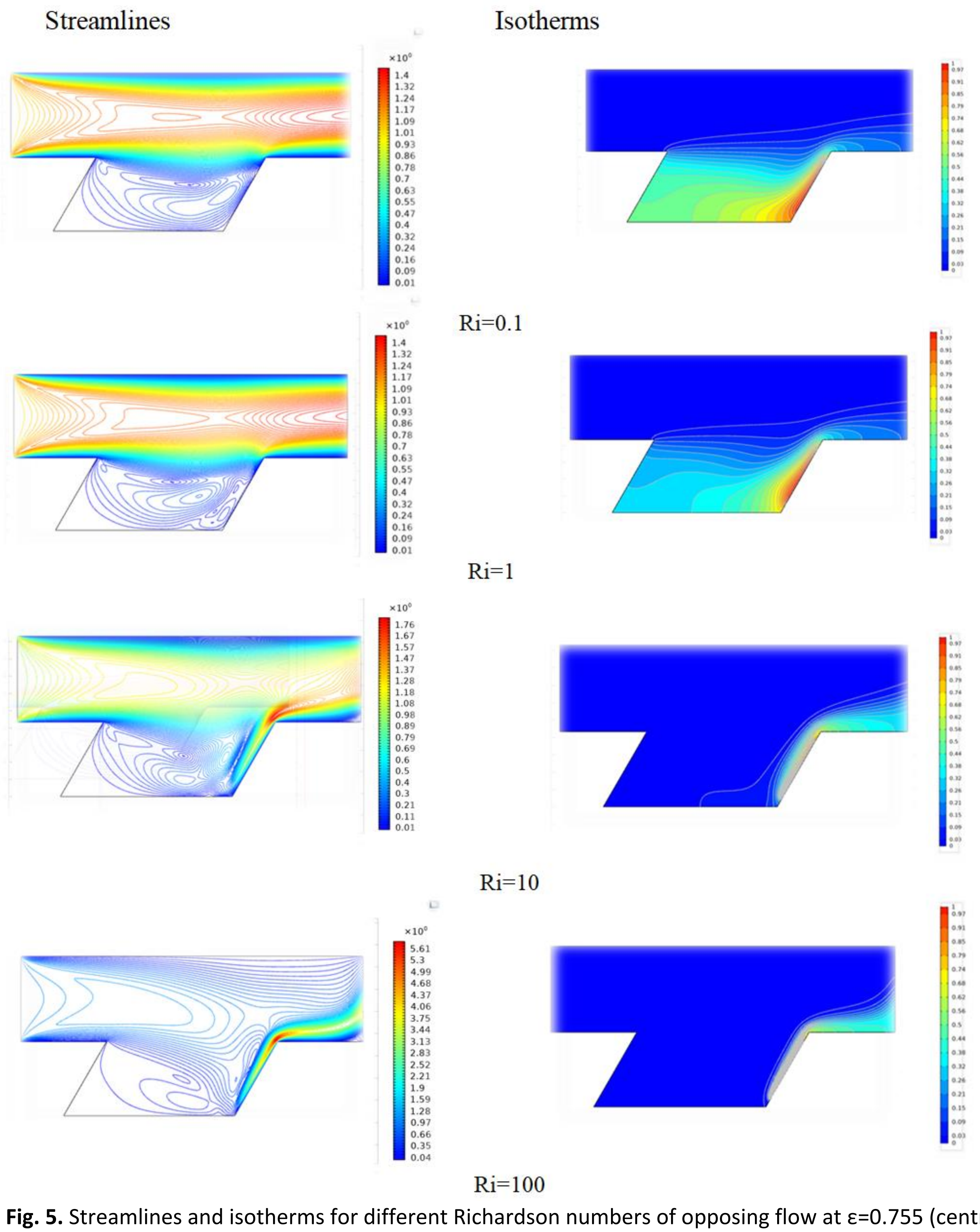


Fig. 5. Streamlines and isotherms for different Richardson numbers of opposing flow at ε=0.755 (centre of right wall)

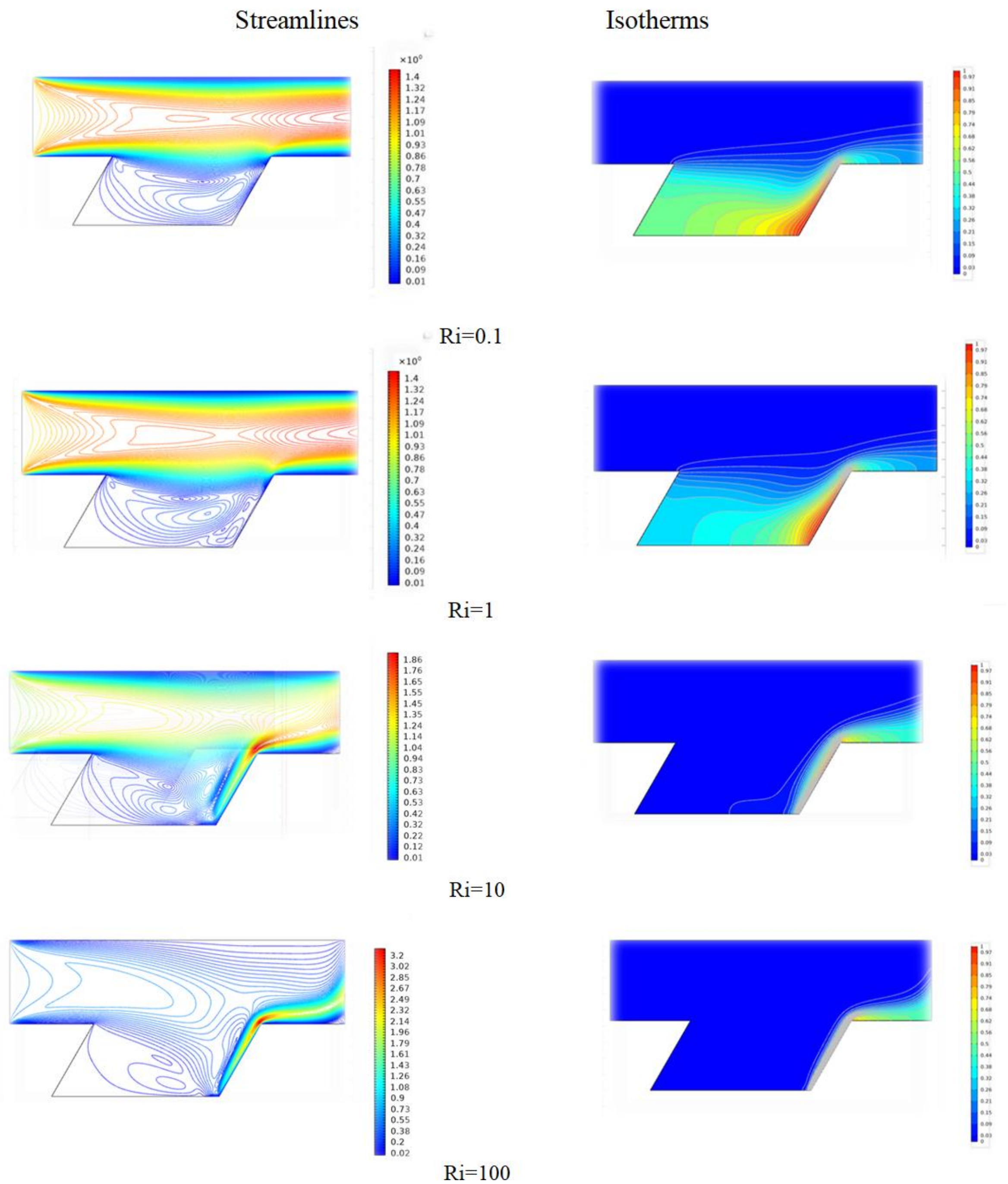


Fig. 6. Streamlines and isotherms with different Richardson numbers of opposing flow at ε=1 (centre right)

4.1.3 Effect of the heat source location

Figure 7 and 8, and Figure 4 and 5 explain the effect of the heat source location on streamline and isotherm contours when it location at an opposite direction to the flow enters the channel. These figures are drawn at (0.1≤Ri≤100, Re=100, AR=2 and ε=0.25 and ε=0.75). To illustrate this effect, two different locations were considered. The first location, at the top corner of the right sidewall of the enclosure. While, the second location was assumed at the center of the same wall.

With respect to the effect of the heat source location on the flow pattern, it can be seen that the change in it does not have a significant influence on the flow pattern when (0.1 ≤Ri≤1). This can be approved from the similar flow pattern in Figure 7 and 4 for (ε=0.25), Figure 5 and 8 for (ε=0.75) for this range of (Ri). While, this difference between the flow fields becomes more clear for (10≤ Ri≤100) especially at the heat source location in the right sidewall. Therefore, it can be concluded that the effect of the heat source location is more pronounced for the natural convection domain than the forced convection one. For the thermal field, the results show that the isotherms are clustered near the heat source location which indicates a high temperature gradient in this place. Also, it can be noted that the thermal plume can be seen when the heat source located at the center of the right wall of the enclosure especially at (0.1≤Ri ≤1). While, it was absent for another considered location. Now, with the increase in (Ri) to (10≤Ri ≤100), the effect of the heat source location begins to diminish gradually except in the region above the heat source. This can be indicated from the high similarity of the thermal fields in Figure 4, 7, 5, and 8 at (10≤Ri ≤100).

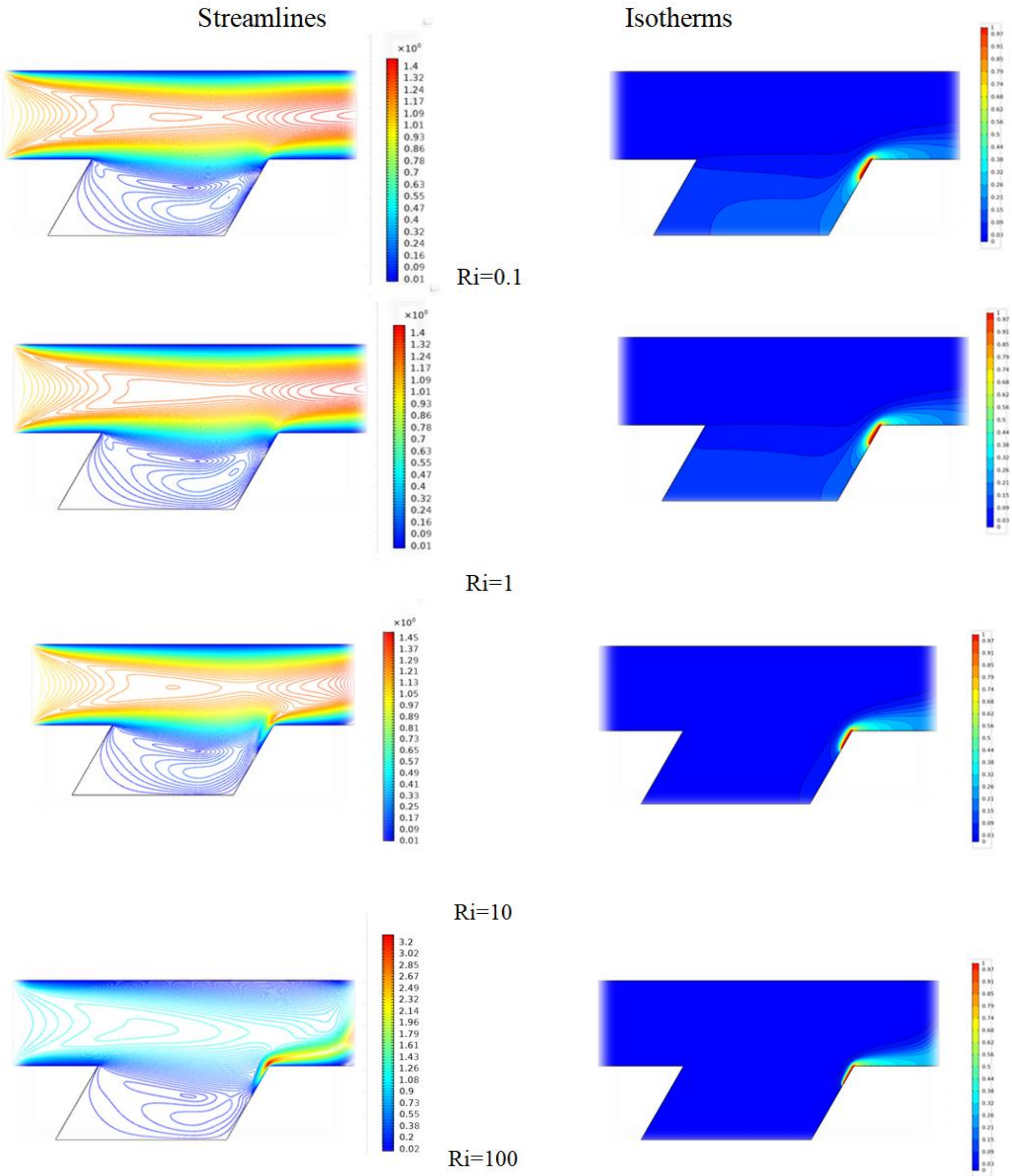


Fig. 7. Streamlines and isotherm contours for different Richardson numbers of opposing flow at ε=0.25 (upper right wall)

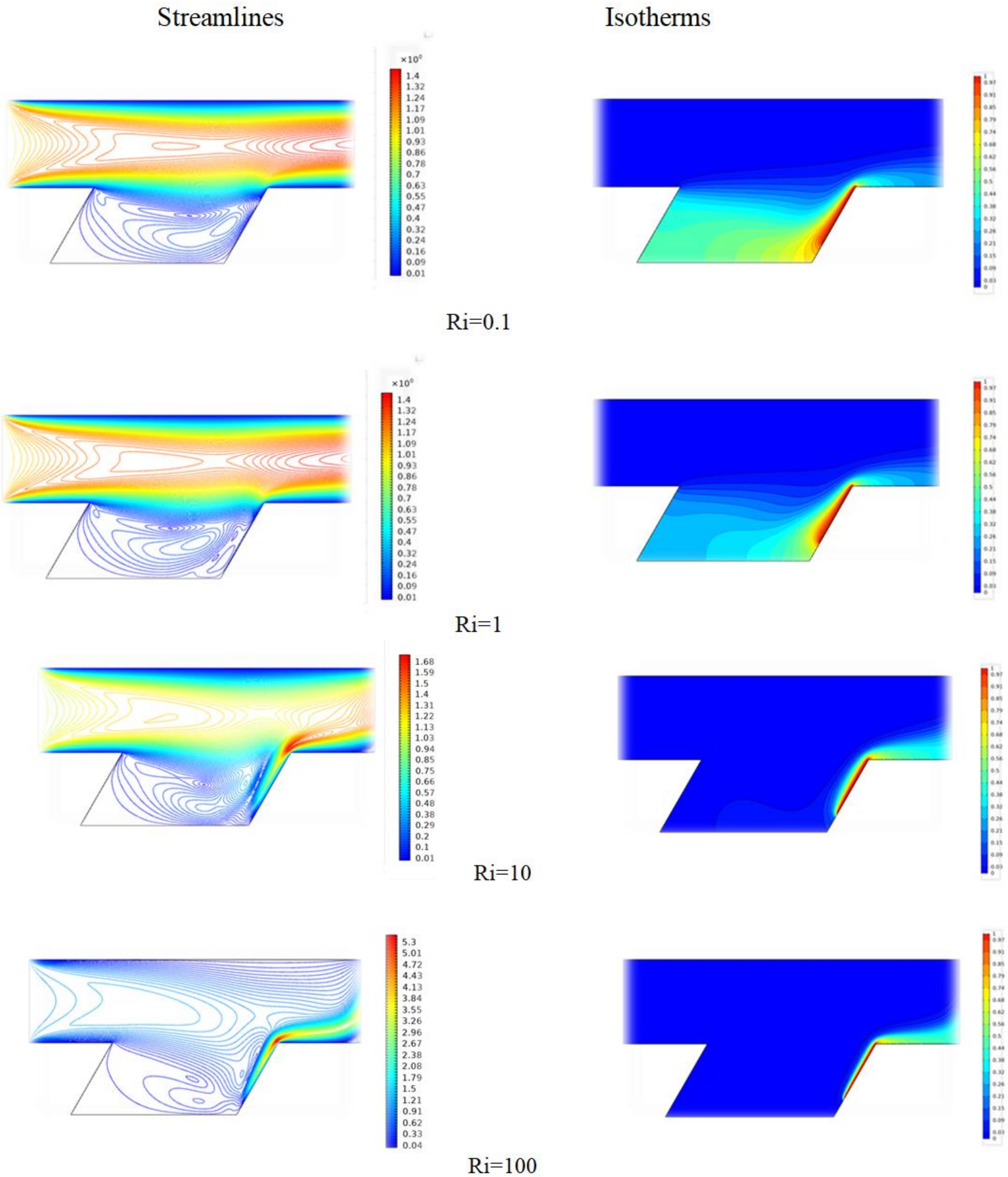


Fig. 8. Streamlines and isotherm contours for different Richardson numbers of opposing flow at ε=0.75 (upper right wall)

4.2 Heat Transfer Performance

4.2.1 Heat source length and average Nusselt number

The variation in the average Nusselt number with Richardson number for various value of (ε) at (Re=100 and AR=2) was described in Figure 9 and 10. In this case, the heat source was located opposite to the direction of the flow enters the channel. In Figure 9, the heat source was located in the center of the bottom wall of the enclosure, whereas in Figure 10, it walls located in left region of the same wall. It can be seen for both figures that, the values increase in (ε) and (Ri) increase the

(Nu_{av}). For ($0.1 \le Ri \le 1.0$). The (Nu_{av}) varies linearly with (Ri), while a clear increase in it can be observed beyond this range of (Ri). This logical result confirms the dominance of the natural convection for high values on (Ri). This behaviour can be seen for all values of (ε) and both location of the heat source.

4.2.2 Heat source position affects average Nusselt number

The shift in the position of the heat source is also shown in Figures 9 and 10, which illustrate how (Nu_{av}) values are affected. It is clear from a comparison of the findings shown in these figures that the values of (Nu_{av}) started rising when the heat source was situated on the top part of the right wall. This can be noticed when looking at the figures. This upward trend is discernible for any and all of the aforementioned values of (ε). Because this is where the heat source is placed, the cold fluid that travels from the channel to the heat source much more quickly than it did when the heat source was positioned in the middle of the right wall. Therefore, it can be determined that the optimal position of the heat source for Case four was in the top part of the right wall of the enclosure. This can be drawn from the discussion that has taken place so far.

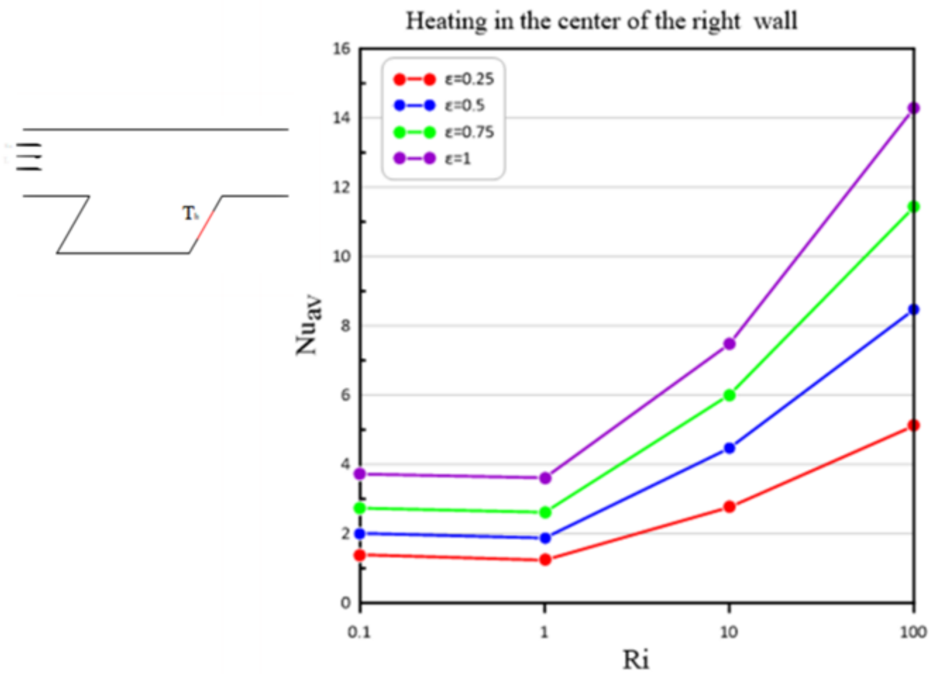


Fig. 9. Average Nusselt number variation with Richardson number of opposing flow at (Re=100 and AR=2)

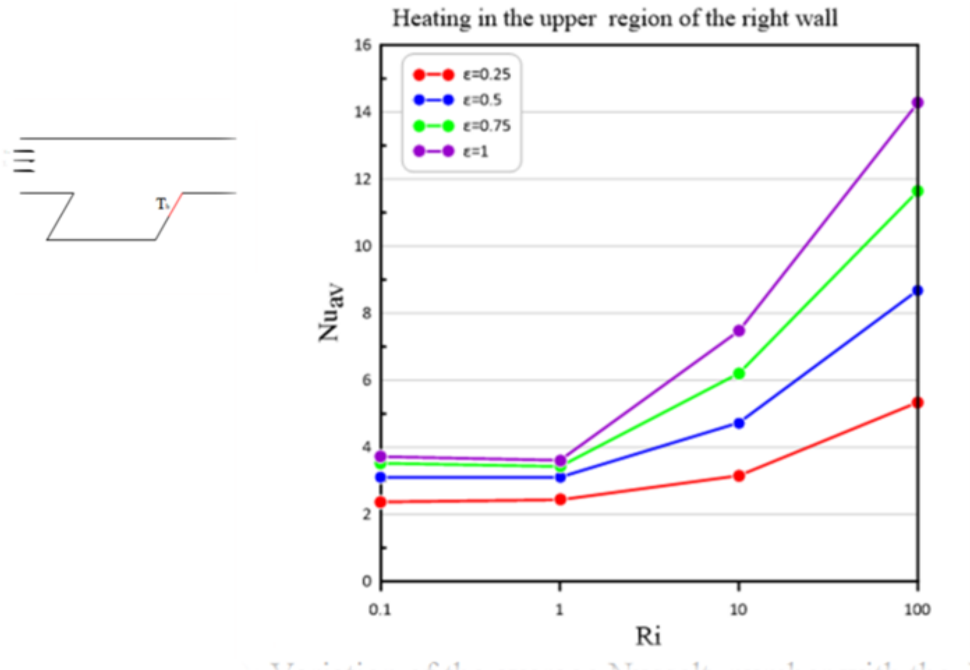


Fig. 10. Average Nusselt number variation with Richardson number of opposing flow at (Re=100 and AR=2)

5. Conclusion

In a parallelogram open cavity that was totally and partially heated from the right side wall of the enclosure, numerical research was conducted to investigate the opposing mixed convection of heat transfer. The following inferences are tenable in light of the discoveries that were made over the course of this work:

i. The increasing in the Richardson number and the heat source length enhance the flow circulation and the heat transfer. This behaviour was observed for all considered cases.
ii. The streamline and isothermal contours were significantly affect by the variation of (Ri), the length of the heat source and its location.
iii. The average Nusslte number was increased with increasing the (Ri) and (ε).
iv. The findings indicate that the largest possible average Nusselt number may be attained by increasing the length of the heat source, and that the highest Richardson number can be found in the top section of the right wall of the enclosure.

References

[1] Manca, O., Nardini, S., Khanafer, K. and Vafai, K. "Effect of heated wall position on mixed convection in a channel with an open cavity." *Numerical Heat Transfer - Part A* 43 (2003): 259-282. https://doi.org/10.1080/10407780307310
[2] Manca, O., Nardini, S. and Vafai, K. "Experimental investigation of mixed convection in a channel with an opencavity." *Experimental Heat Transfer* 19 (2006): 53-62. https://doi.org/10.1080/08916150500318380
[3] Leong, J., Brown, N. and Lai, F. "Mixed convection from an open cavity in a horizontal channel." *International Communications in Heat and Mass Transfer* 32 (2005): 583-592. https://doi.org/10.1016/j.icheatmasstransfer.2004.10.018
[4] Aminossadati, S. and Ghasemi, B. "A numerical study of mixed convection in a horizontal channel with a discrete heat source in an open cavity." *European Journal of Mechanics B/Fluids* 28 (2009): 590-598. https://doi.org/10.1016/j.euromechflu.2009.01.001

[5] Rahman, M., Parvin, S., Saidur, R. and Rahim, N. "Magnetohydrodynamic mixed convection in a horizontal channel with an open cavity." *International Communications in Heat and Mass Transfer* 38 (2011): 184-193. https://doi.org/10.1016/j.icheatmasstransfer.2010.12.005

[6] Rahman, M., Parvin S., Rahim, N., Hasanuzzaman, M. and Saidur, R. "Simulation of mixed convection heat transfer in a horizontal channel with an open cavity containing a heated hollow cylinder." *Heat transfer – Asian research* 41 (2012): 339-353. https://doi.org/10.1002/htj.21002

[7] Rahman, M., Oztop, H., Ahsan, A., Kalam, M. and Billah, M. "MHD mixed convection in a channel with a triangular cavity." *Numerical Heat Transfer - Part A* 61 (2012): 268-282. https://doi.org/10.1080/10407782.2012.648025

[8] Carozza ,A., Manca, O. and Nardini , S. "Numerical Investigation on Heat Transfer Enhancement due to Assisting and Opposing Mixed Convection in an Open Ended Cavity." *2nd International Conference on Emerging Trends in Engineering and Technology London (UK)* (2014) (ICETET'2014): 85-90.

[9] Selimefendigil F. "Numerical analysis and POD based interpolation of mixed convection heat transfer in horizontal channel with cavity heated from below." *Engineering Applications of Computational Fluid Mechanics* 7 (2013): 261-271. https://doi.org/10.1080/19942060.2013.11015469

[10] Abdelmassih, G., Vernet, A. and Pallares, J. "Steady and unsteady mixed convection flow in a cubical open cavity with the bottom wall heated." *International Journal of Heat and Mass Transfer* 101 (2016): 682-691. https://doi.org/10.1016/j.ijheatmasstransfer.2016.05.074

[11] Hussain S., Ahmed S., and Akbar, T. "Entropy generation analysis in MHD mixed convection of hybrid nanofluid in an open cavity with a horizontal channel containing an adiabatic obstacle." *International Journal of Heat and Mass Transfer* 114 (2017): 1054–1066. https://doi.org/10.1016/j.ijheatmasstransfer.2017.06.135

[12] Garcia, F., Trevino, C., Lizardi, J. and Martinez-Suastegui, L. "Numerical study of buoyancy and inclination effects on transient mixed convection in a channel with two facing cavities with discrete heating." *International Journal of Mechanical Sciences* 155 (2019): 295-314. https://doi.org/10.1016/j.ijmecsci.2019.03.001

[13] Laouira, H., Mebarek-Oudina, F., Hussein, A.K., Kolsi, L., Merah, A. and Younis, O. "Heat transfer inside a horizontal channel with an open trapezoidal enclosure subjected to a heat source of different lengths." *Heat Transfer-Asian Research* 49 (2020): 406-423. https://doi.org/10.1002/htj.21618

[14] Mebarek- Oudina, F., Laouira, H., Hussein, A.K. and El Ganaoui, M. "Convection Heat Transfer Analysis in a Channel with an Open Trapezoidal Cavity: Heat Source Locations effect." *MATEC Web of Conferences* 330, 01006 (2020): 1-5. https://doi.org/10.1051/matecconf/202033001006

[15] Ahmadi, M. and Farsani, A.K. " Computational Fluid Dynamic Simulation of Non-Newtonian two –phase fluid flow through a channel with a cavity." *Thermal Science* 24 (2020): 1045-1054. https://doi.org/10.2298/TSCI180102151A

[16] AL-Farhany, K., Azeez Alomari, M., and Faisa, A. "Magnetohydrodynamics Mixed Convection Effects on the open enclosure in a horizontal channel Heated Partially from the Bottom." *IOP Conf. Series: Materials Science and Engineering* 870, 012174 (2020):1-7. https://doi.org/10.1088/1757-899X/870/1/012174

[17] Manca ,O., Nardini ,S. and Vafai ,K. "Experimental investigation of opposing mixed convection in a channel with an open cavity below." *Experimental Heat Transfer* 21 (2008): 99-114. https://doi.org/10.1080/08916150701815820

[18] Stiriba, Y." Analysis of the flow and heat transfer characteristics for assisting incompressible laminar flow past an open cavity." *International Communications in Heat and Mass Transfer* 35 (2008): 901-907. https://doi.org/10.1016/j.icheatmasstransfer.2008.04.004

[19] Zamzari, F., Mehrez, Z., El- Cafsi, A. and Belghith, A. "Entropy generation and mixed convection in a horizontal channel with an open cavity." *International Journal of Exergy* 17 (2015): 219-239. https://doi.org/10.1504/IJEX.2015.069993

[20] Burgos, J., Cuesta, I. and Saluena, C. "Numerical study of laminar mixed convection in a square open cavity." *International Journal of Heat and Mass Transfer* 99 (2016): 599-612. https://doi.org/10.1016/j.ijheatmasstransfer.2016.04.010

[21] Selimefendigi, F. , Öztop, H., and Abu-Hamdeh, N. "Natural convection and entropy generation in nanofluid filled entrapped trapezoidal cavities under the influence of magnetic field." *Entropy* 18 (2016). https://doi.org/10.3390/e18020043

[22] Sabbar, W.A., Ismael, M.A., Almudhaffar, M. "Fluid-structure interaction of mixed convection in a cavity-channel assembly of flexible wall." *International Journal of Mechanical Sciences* 149 (2018):73–83. https://doi.org/10.1016/j.ijmecsci.2018.09.041

[23] Yasin, N., Jehhef, K. and Shaker, A. "Assessment of the baffle effects on the mixed convection in open cavity." *International Journal of Mechanical and Mechatronics Engineering* 18 (2018): 1-14.

[24] Cardenas, V., Trevino, C., Rosas, I. and Martinez-Suastegui, L. "Experimental study of buoyancy and inclination effects on transient mixed convection heat transfer in a channel with two symmetric open cubic cavities with

prescribed heat flux." *International Journal of Thermal Sciences* 140 (2019): 71-86. https://doi.org/10.1016/j.ijthermalsci.2019.02.024

[25] Contreras, H., Trevino, C., Lizardi, J. and Martinez-Suastegui, L. "Stereoscopic TR-PIV measurements of mixed convection flow in a vertical channel with an open cavity with discrete heating." *International Journal of Mechanical Sciences* 150 (2019): 427-444. https://doi.org/10.1016/j.ijmecsci.2018.10.049

[26] Hussain, S., Mehmooda, K, Sagheera, M., Farooqa, A. "Entropy generation analysis of mixed convective flow in an inclined channel with cavity with Al2O3-water nanofluid in porous medium." *International Communications in Heat and Mass Transfer* 89 (2017): 198–210. https://doi.org/10.1016/j.icheatmasstransfer.2017.10.009

[27] ALtaca Z and Timuralp C . Investigation of fluid flow and heat transfer in a channel with an open cavity heated from bottom side .Mugla Journal of Science and Technology, Vol 2, No .1, 2016, pp:55-59

[28] Rahman, M., Oztop, H., Rahim, N., Saidur, R., Al-Salem, K., Amin, N., Mamun, M. and Ahsan, A. "Computational analysis of mixed convection in a channel with a cavity heated from different sides." *International Communications in Heat and Mass Transfer* 39 (2012): 78-84. https://doi.org/10.1016/j.icheatmasstransfer.2011.09.006

[29] Rahman, M., Oztop, H., Saidur, R., Mekhilef, S. and Al-Salem, K. "Finite element solution of MHD mixed convection in a channel with a fully or partially heated cavity." *Computers and Fluids* 79 (2013): 53-64. https://doi.org/10.1016/j.compfluid.2013.03.003

[30] Rahman, M., Saidur, R. and Rahim, N. "Conjugated effect of joule heating and magneto-hydrodynamic on double-diffusive mixed convection in a horizontal channel with an open cavity," *International Journal of Heat and Mass Transfer* 54 (2011): 3201-3213. https://doi.org/10.1016/j.ijheatmasstransfer.2011.04.010

[31] Buonomo, B., Cresci, G., Manca, O., Mesolella, P. and Nardini, S. "Transient mixed convection in a channel with an open cavity filled with porous media." *Journal of Physics : Conference Series* 395 (2012): 1-8. https://doi.org/10.1088/1742-6596/395/1/012149

[32] Andreozzi, A., Manca, O. "Mixed convection in air in an open ended cavity with a moving plate parallel to the cavity open surface , ASME Summer Heat Transfer Conference." *San Francisco, California USA* (2005) HT2005-72511:1-13. https://doi.org/10.1115/HT2005-72511

[33] Fusegi, T. "Numerical study of convective heat transfer from periodic open cavities in a channel with oscillatory through flow." *International Journal of Heat and Fluid Flow* 18 (1997): 376-383. https://doi.org/10.1016/S0142-727X(97)00014-3

[34] Yee, H.C., Jing, K.T., and Yik, K.Z. "Current status of green building development in Malaysia." *Progress in Energy and Environment* 25, no. 1 (2023): 1-9. https://doi.org/10.37934/progee.25.1.19

[35] Gudekote, M., and Choudhari, R. "Slip Effects on Peristaltic Transport of Casson Fluid in an Inclined Elastic Tube with Porous Walls." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 43, no. 1 (2018): 67-80.

[36] Hamrelaine, S., Mebarek-Oudina, F., and RafikSari, M., "Analysisof MHD Jeffery Hamel Flow with Suction/Injection by Homotopy Analysis Method." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 58, no. 2 (2019): 173-186.

[37] Rashid, F.L., Khalaf, A.F., Hussein, A.K., Hamida, M.B., Ali, B., and Younis, O. "Thermal-hydraulic analysis of transient conjugate heating between hemi-spherical body and air." *Frontiers in Heat and Mass Transfer (FHMT)* 19 (2022). https://doi.org/10.5098/hmt.19.21

[38] Rashid, F.L., Fakhrulddin, S.K., Eleiwi, M.A., Hussein, A.K., Tahseen, T.A., Younis, O., and Ahmed, M.I. "CFD simulation in thermal-hydraulic analysis of air flow on different attack angles of row flat tube." *Frontiers in Heat and Mass Transfer (FHMT)* 19 (2022). https://doi.org/10.5098/hmt.19.6

[39] Rashid, F.L., Hussein, A.K., Malekshah, E.H., Abderrahmane, A., Guedri, K., and Younis, O. "Review of heat transfer analysis in different cavity geometries with and without nanofluids." *Nanomaterials* 12 (2022): 2481. https://doi.org/10.3390/nano12142481

[40] Eleiwi, M.A., Rashid, F.L., Khalaf, A.F., and Tuama, S.A. "Numerical investigation of conjugate heat transfer between spherical solid body and fluid." *Mathematical Modelling of Engineering Problems* 9 (2022): 491-497. https://doi.org/10.18280/mmep.090227

[41] Altaie, A., Hasan, M.R., and Rashid, F.L. "Numerical heat transfer and turbulent flow in a circular tube fitted with opened rings having square cross section." *Journal of Basic and Applied Scientific Research* 4 (2014): 28-36. https://doi.org/10.11591/eei.v4i1.331

[42] Rashid, F.L., Altaie, A., and Hasan, M. R. "Numerical investigation of heat transfer enhancement in a circular tube using ribs of separated ports assembly." *European Scientific Journal* 2 (2014): 172-183.

[43] Altaie, A., Hasan, M.R., and Rashid, F.L. "Numerical investigation of heat transfer enhancement in a circular tube with rectangular opened rings." *Bulletin of Electrical Engineering and Informatics* 4 (2015): 18-25. https://doi.org/10.11591/eei.v4i1.331

[44] Altaie, A., Hasan, M.R., and Rashid, F.L. "Heat transfer enhancement in a circular tube using ribs with middle arm." *Elixir International Journal* (2015).

[45] Altaie, A., Hasan, M.R., Rashid, F.L. "Numerical investigation in a circular tube to enhance turbulent heat transfer using opened rings-triangular cross section." *Journal of Babylon University/ Engineering Sciences* 23 (2015).
[46] Rashid, F.L., Al-Jibory, M.W., and Hussein, H.Q. "Cooling enhancement in gas turbine blade using coated circular ribs with a new nanocomposite material." Patent (5092) (2017).
[47] Al-Jibory, M.W., Rashid, F.L., and Hussein, H.Q. "Heat transfer augmentation in gas turbine blade rectangular passages using circular ribs with fins." *Journal of University of Babylon, Engineering Sciences*, 26 (2018): 247-258.
[48] Al-Jibory, M.W., Rashid, F.L., and Talib, Sh.M. "Numerical investigation of heat transfer enhancement in ribbed elliptical passage." *Journal of Engineering and Applied Sciences* 13 (2018): 7223-7234.
[49] Rashid, F.L., Azziz, H.N., and Hussein, E.Q. "Heat transfer enhancement in air cooled gas turbine blade using corrugated passages." *Journal of Petroleum Research & Studies* 20 (2018): 52-69. https://doi.org/10.52716/jprs.v8i3.230
[50] Azziz, H.N., Shareef, A.S., and Rashid, F.L. "Experimental investigation of the heat transfer for the effect of nanoparticles with different base fluid and solar collector tilt angle." *Journal of Engineering and Applied Sciences* 13(2018): 10614-10620.
[51] Aljibory, M.W., Rashid, F.L., and Alais, S.M. "An Experimental and numerical investigation of heat transfer enhancement using annular ribs in a tube." *IOP Conference Series: Materials Science and Engineering* 433(2018): 012057. https://doi.org/10.1088/1757-899X/433/1/012057
[52] Rashid, F.L., Al-Jibory, M.W., and Talib, Sh.M. "Numerical investigation of heat transfer augmentation in elliptical passage with different rib geometries and aspect ratios." *International Journal of Mechanical Engineering and Technology* 9 (2018): 1390-1409.
[53] Al-Jibory, M.W., Rashid, F.L., and Talib, Sh.M. "An experimental investigation of heat transfer enhancement in elliptical passage fitted with different rib geometries." *International Journal of Mechanical Engineering and Technology* 9 (2018): 1033-1048.
[54] AL-Jibory, M.W., Rashid, F.L., and Talib, Sh.M. "Review on cooling enhancement of different shape gas turbine ribbed blade with thermal barrier coating." *International Journal of Scientific Research and Engineering Development* 3 (2020): 313-329.
[55] AL-Jibory, M.W., Rashid, F.L., and Hussein, H.Q. "Review of heat transfer enhancement in air-cooled turbine blades." *International Journal of Scientific & Technology Research* 9(2020): 3123-3130.
[56] Hussein, H.Q., Al-Jibory, M.W., and Rashid, F.L. "Heat transfer enhancement of gas turbine blades using coated ribs with nanocomposite materials." *Journal of Mechanical Engineering Research and Developments*, 43(2020): 9-22.