NUMERICAL ANALYSIS OF MIXED CONVECTION HEAT TRANSFER FOR LAMINAR FLOW IN A CHANNEL WITH AN OPEN CAVITY

Sana M. Shrama
Mechanical Engineering Department
College of Engineering - University of Basrah

ABSTRACT

Mixed convection heat transfer for laminar air flow in an open cavity is studied numerically by finite element method using software package (FlexPDF) to solve the conservation of governing equations. Three basic heating modes are considered: case 1, the heated wall is the horizontal surface of the cavity (heating from below) at uniform temperature, case 2, all the walls of the cavity are isothermal, and case 3, all walls of the cavity and the bottom wall of the channel are kept at constant temperature, for each case the boundary condition of the other walls are adiabatic. The results in terms of streamlines, isotherms, average temperature of the fluid and average Nusselt number of the heated wall are presented for Ri = 0 to 5, Re=100 , Pr = 0.71, and cavity aspect ratio (L/D) is in the range from(1–2). The present results show that the aspect ratio and Ri are affect on streamline and isotherm patterns for different heating configurations. In addition, the thermal performance in terms of both the overall heat transfer coefficient and bulk mean temperature of fluid is affected by the two parameters. The results of streamlines and isotherms are compared with available result of (Manca et al.,2003)[10] and a good agreement has been achieved.

KEYWORDS: Heat Transfer, Mixed Convection, Laminar Flow, Channel With An Open Cavity.

التحليل العددي لانتقال الحرارة بالحمل المختلط للجريان الطباقي في قناة بتجويف مفتوح

سناء مهدي شرامة
قسم الهندسة الميكانيكية- كلية الهندسة- جامعة البصرة

الخلاصـــــة

يقدم البحث دراسة عددية لانتقال الحرارة بالحمل المختلط لجريان الهواء الطباقي داخل قناة بتجويف مفتوح تم تحليلها عدديا بطريقة العناصر المحددة باستخدام (FlexPDF) لحل معادلات الحفظ الحاكمة. تم دراسة ثلاث حالات لبيان تأثير موقع التسخين (1) تسخين السطح الأفقي للفجوة بدرجة حرارة ثابتة ، (2) تسخين جدران الفجوة بدرجة حرارة ثابتة ،(3) تسخين الجدار السفلي للقناة بدرجة حرارة ثابتة ، ولكل الحالات الشروط الحدية للجدران المتبقية تكون معزولة .وعرض البحث أشكال توزيع خطوط الجريان ودرجات الحرارة و معدل درجة الحرارة ومعدل رقم نسلت لرقم ريكاردسون Ri يتراوح من (0-5) ولرقم رينولدز Re=100 ، رقم برانتل Pr

= 0.71 ولنسب مختلفة من الطول إلى العرض (L/D) تتراوح من (2-1). أظهرت النتائج الحالية بأن رقم ريكاردسون Ri ونسب الطول للعرض لها تأثير على أشكال خطوط الجريان وخطوط درجات الحرارة كذلك الأداء الحراري الممثل بمعامل انتقال الحرارة ودرجة حرارة المائع يتأثر بكلا العاملين. نتائج خطوط الجريان وخطوط درجات الحرارة قورنت مع ما منشورفي (Manca et al., 2003) وأظهرت توافقا جيدا.

NOMENCLATURE

- *AR* cavity aspect (L/D)
- D height of the cavity, m
- *g* gravitational acceleration (ms^{-2})
- *Gr* Grashof number
- *H* height of the cavity (m)
- *L* length of the cavity (m)
- L_h length of the heated wall (m)
- *Nu* average Nusselt number
- *N* normal direction on a plane
- *p* pressure (Nm^{-2})
- *P* non-dimensional pressure
- *Pr* Prandtl number
- *Re* Reynolds number
- *Ri* Richardson number
- s heated surface
- *T* temperature (K)
- θ non-dimensional temperature, $(T-T_i)/(T_h-T_i)$
- θ_{av} average non-dimensional temperature
- *u, v* velocity components (ms^{-1})
- *U, V* non-dimensional velocity components, u/u_i v/u_i
- V cavity volume (m^3)
- *x, y* Cartesian coordinates (m)
- *X, Y* non-dimensional Cartesian coordinates, *x/D,y/D*

Greek symbols

- α thermal diffusivity, k/ρCp (m^2s^{-1})
- β thermal expansion coefficient (K^{-1})
- ρ density of the fluid (kgm^{-3})
- υ kinematic viscosity of the fluid (m^2s^{-1})

Subscripts

- av average
- i inlet state
- h hot state
- s heated surface

INTRODUCTION

Efficient convection heat transfer is essential in modern technology and very important in many industrials areas. Hence, it is necessary to study and simulate these phenomena. Heat transfer in flows in which the influence of forced convection and natural convection are of comparable magnitude (mixed convection flows) occurs frequently in engineering situations. Recently great attention has been pointed on mixed convection in open-ended cavities for their wide use, such as thermal control of electronic equipments, chemical vapour deposition (CVD) of solid layer and solar collectors. Buoyancy force due to the heating of the lower cavity wall induces secondary flows hence the local heat transfer increases (**Buonomo et al., 2008**). The electronic components, which are considered as heat sources, are usually mounted on the vertical boards and the heat generated by these sources is removed by both natural convection and application of an externally induced flow of air

The studies on open top upright cavities (U-shaped open cavities) have been conducted for either pure natural convection or mixed convection. A numerical analysis of laminar mixed convection in an open cavity with a heated wall bounded by a horizontally insulated plate was presented in (**Manca et al., 2003**). Three heating modes were considered: assisting flow, opposing flow, and heating from below. Results for Richardson number equal to 0.1 and 100, Re =100 and 1000, and aspect ratio (H/D) in the range of 0.1-1.5 were reported. It was shown that the maximum temperature values decrease as the Reynolds and the Richardson numbers increase. The effect of the ratio of channel height to the cavity height was found to play a significant role on streamline and isotherm patterns for different heating configurations. The investigation showed that opposing forced flow configuration had the highest thermal performance, in terms of both maximum temperature and average Nusselt number. (**Chaves et al., 2008**) presented a numerical analysis of combining force and free convection heat transfer inside a semi porous two-dimensional rectangular open cavity. The open cavity consists of two vertical walls closed to the bottom by a uniform heat flux. One vertical wall is a porous wall and fluid inflows normal to it. The other wall transfers the same uniform heat flux to the cavity. The study showed how natural convection effects may improve the forced convection inside the open cavity. The main motivation for this research is its application for electronic equipment where the cooling devices used for the electronic equipment are frequently based on natural and forced convection and the equipment may reach dangerous limits of temperature reducing its efficiency. Results of the maximum temperature are presented for both Reynolds and Grashof numbers at the heated wall and in the bottom. (**Chan and Tien, 1985**) performed a numerical study of two-dimensional laminar natural convection in a square open cavity with a heated vertical wall and two insulated horizontal walls for Rayleigh numbers ranging from $10^3$to 10^9. Calculations were made in an extended computational domain beyond the aperture plane for a Prandtl number of 1. The obtained heat transfer results were found to approach those of natural convection over a vertical isothermal flat plate.

A numerical study of transient natural convection in a rectangular open cavity with an asymmetric thermal boundary condition was investigated by (**Jones and Cai, 1993**). In their investigation, one vertical wall of the cavity was either heated by a constant heat flux or cooled by convection to the surroundings. The top of the cavity was open to a large reservoir of fluid at a constant temperature. A numerically investigation of mixed convection in a rectangular enclosure with different numbers and arrangements of discrete heat sources was conducted by (**Ghasemi and Aminossadati, 2007**). The objective of their work was to identify the cooling performance of electronic devices with an emphasis on the effects of the arrangement and number of electronic components. The analysis uses a two dimensional rectangular enclosure under combined natural and forced convection flow conditions and considers a range of Rayleigh numbers. Their results showed that increasing the Rayleigh number significantly improves the enclosure heat transfer process. At low Rayleigh numbers, placing more heat sources within the enclosure reduces the heat transfer rate from the sources and consequently increases their overall maximum temperature. The arrangement and number of heat sources have a considerable contribution to the cooling performance.

(**Rahman et al., 2007**) investigated mixed convection in a vented enclosure using finite element method. An external fluid flow enters the enclosure through an opening in the left vertical wall and exits from another fixed opening in the right vertical wall. For mixed convection, the significant parameters are Grashof number (Gr), Richardson number (Ri) and Reynolds number (Re) by which different fluid and heat transfer characteristics inside the cavity are obtained. The results showed that with the increase of Re and Ri the convective heat transfer become dominant. Mixed convection heat transfer in open-ended enclosures has been studied numerically by (**Khalil Khanafer et al., 2002**) for three different flow angles of attack. Discretization of the governing equations is achieved using a finite element scheme based on the Galerkin method of weighted residuals. A wide range of pertinent parameters such as Grashof number, Reynolds number, and the aspect ratio are considered in their study. The results showed that thermal insulation of the cavity can be achieved through the use of high horizontal velocity flow. Various results for the streamlines, isotherms and the heat transfer rates in terms of the average Nusselt number are presented and discussed for different parametric values. (**Sumon Saha et al., 2008**) considered a numerical analysis of study the performance of mixed convection in a rectangular enclosure. Four different placement configurations of the inlet and outlet openings were considered, a constant flux heat source strip is flush-mounted on the vertical surface, modeling an integrated circuit board, and the fluid considered is air. The numerical scheme is based on the finite element method adapted to triangular non-uniform mesh elements by anon-linear parametric solution algorithm. Results are obtained for a range of Richardson number from 0 to 10 at Pr=0.71and Re=100 with constant physical properties. The results indicate that the average Nusselt number and the dimensionless surface temperature on the heat source strongly depend on the positioning of the inlet and outlet. (**Saldana et al., 2005**) studied laminar mixed convective flow over a three-dimensional horizontal backward-facing step heated from below at a constant temperature for air flowing through the channel. The flow at the duct entrance was considered to be hydro dynamically fully developed and isothermal. The bottom wall of the channel was subjected to a constant high temperature while the other walls were treated to be adiabatic. The numerical results show that the effect of increase in Richardson number(Ri) on the mixed convective flow not only decreased the size of the primary recirculation zone in both stream wise and transverse directions, but also moved the location of maximum in the averaged Nusselt number distribution and maximum in the averaged stream wise shear stress distribution further upstream. (**Rahman et al., 2009**) studied the mixed convection in cavity contains a heat conduction horizontal square block located inside the cavity. The investigations are conducted for various values of geometric size, location and thermal conductivity of the block under constant Re and Pr. The results indicated that the average Nusselt number and the temperature at the center of solid block are strong dependent on the system configurations studied under different geometrical and physical conditions.

(**Rahman et al., 2009**) performed computations on mixed convection heat transfer in a square cavity with a centered heat conducting horizontal square solid cylinder. The right vertical wall is kept at constant temperature and the remaining three walls are kept thermally insulated. An external flow enters the cavity through an opening in the left vertical wall and exits from another opening in the right vertical wall. The parameters, such as Reynolds number, Richardson number, Prandtl number and the inlet and exit port locations of the cavity are studied. The results indicated that Re has significant effect on the flow field in the pure forced convection and in the pure mixed convection region, but Pr has significant effect on the flow field in the pure mixed convection and in the free convection dominated region.

A study of mixed convection flow inside a rectangular ventilated cavity in the presence of a heat conducting square cylinder at the center has been carried out by (**Rahman et al., 2009**). An external fluid flow enters the cavity through an opening in the left vertical wall and exits from another opening in the right vertical wall. Results are presented in the form of average Nusselt number of the heated wall, average temperature of the fluid in the cavity and temperature at the cylinder center for the range of Richardson number (0-5) and cavity aspect ratio (0.5-2). The streamlines and isothermal lines are also presented. The relevant literature on mixed convection in in a lid-driven cavity has been reviewed by (**Prasad and Koseff, 1996**) and by (**Deshpande and Srinidhi, 2005**). Prasad and Koseff described the mixed convection heat transfer process in a deep lid-driven cavity flow. The mean heat flux values over the entire lower boundary were analyzed to produce Nusselt Number and Stanton number correlation which should be useful for design application and Deshpande and Srinidhi are reported some interesting results regarding mixed thermal convection in a rectangular parallelepiped .These results, apart from being of fundamental importance, have practical relevance in areas like material processing and other engineering applications.

The objective of the present study is to examine the effect of heated wall position, Ri and aspect ratio on mixed convection in a cavity with channel. Air flows through the channel. Three resulting cases are investigated (a) forced flow in the channel and natural convection due to a heat source over the cavity bottom wall;(b) forced flow in the channel and natural convection due to a heat source over all the cavity walls; and (c) forced flow and natural convection due to heating all cavity and the bottom wall of the channel isothermally. The results are shown in terms of parametric presentations of streamlines and isotherms. Also the effect of mixed convection parameter Ri and cavity aspect ratio (AR) on the heat transfer process are analyzed and the results are presented in terms of the average Nusselt number at the heated surface and average temperature of the fluid in the cavity. Finite element method has been used to solve the governing equations by using software package (FlexPDF).

PROBLEM DESCRIPTION AND GOVERNING EQUTIONS

The geometry under investigation is shown in **Figure.1**. Case 1, the bottom wall of the rectangular shaped cavity is heated by a uniform constant temperature T_h and the other walls are adiabatic. Flow enters through the left opening at a uniform velocity, u_i. It is assumed that the incoming flow is at an ambient temperature, T_i, and the outgoing flow is assumed to have zero diffusion flux for all variables (outflow boundary conditions). Case 2, the uniform constant temperature T_h, applied on the all cavity walls. Case 3, all walls of the cavity and the bottom wall of the channel are heated isothermally, all solid boundaries are to be rigid no- slip walls and adiabatic.

Mixed convection is governed by the differential equations expressing conservation of mass, momentum and energy. The present flow is considered steady, laminar, incompressible and two-dimensional. The viscous dissipation term in the energy equation is neglected. The physical properties of the fluid in the flow model are assumed constant except the density variations body force term in the momentum equation. The governing equations for steady mixed convection flow can be expressed in the dimensionless form as (**Rahman, 2009**):

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

$$U\frac{\partial U}{\partial X}+V\frac{\partial U}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{1}{\mathrm{Re}}\left(\frac{\partial^2 U}{\partial X^2}+\frac{\partial^2 U}{\partial Y^2}\right) \tag{2}$$

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

$$U\frac{\partial \theta}{\partial X}+V\frac{\partial \theta}{\partial Y}=\frac{1}{\mathrm{Re}\,\mathrm{Pr}}\left(\frac{\partial^2 \theta}{\partial X^2}+\frac{\partial^2 \theta}{\partial Y^2}\right) \tag{4}$$

Where X and Y are the coordinates varying along horizontal and vertical directions respectively, U and V are the velocity components in the X and Y directions respectively, θ is the dimensionless temperature and P is the dimensionless pressure.

The dimensionless parameters in the above equations can be given as:

$$X=\frac{x}{D},\quad Y=\frac{y}{D},\quad U=\frac{u}{u_i},\quad V=\frac{v}{u_i},\quad P=\frac{p}{\rho u_i^2},\quad \theta=\frac{T-T_i}{T_h-T_i}$$

$$\mathrm{Re}=\frac{Du_i}{\upsilon},\qquad Ri=\frac{g\beta(T-T_i)D}{u_i^2},\qquad \mathrm{Pr}=\frac{\upsilon}{\alpha}$$

Where ρ, β, ν, α and g are the fluid density, coefficient of volumetric expansion, kinematic viscosity, thermal diffusivity, and gravitational acceleration, respectively. The following boundary conditions are used:

a: case 1
Inlet: U = 1, V = 0, θ = 0
Exit: Convective boundary condition, P = 0
At the cavity walls (except the bottom wall): U=V=0, $\partial\theta/\partial N = 0$
At the heated bottom wall: U=V=0, θ=1

b: case 2
Inlet: U = 1, V = 0, θ = 0
Exit: Convective boundary condition, P = 0
At the heated cavity walls: U=V=0, θ=1
The other walls: U=V=0, $\partial\theta/\partial N = 0$

c: case 3
Inlet: U = 1, V = 0, θ = 0
Exit: Convective boundary condition, P = 0
At the heated bottom wall of the channel: U=V=0, θ=1
The upper wall of the channel: U=V=0, $\partial\theta/\partial N = 0$

The local Nusselt number at the hot wall is defined as:

$$Nu_L = \frac{\partial\theta}{\partial N} \tag{5}$$

The average Nusselt number at the hot wall is defined as

$$Nu = \frac{1}{L_h}\int_0^{L_h} Nu_L ds \tag{6}$$

And the bulk average temperature in cavity is defined as

$$\theta_{av} = \int \frac{1}{\overline{V}}\,\theta\, d\overline{V} \tag{7}$$

Where L_h is the length of the heat wall and $\overline{V}$ is the cavity volume.

NUMERICAL METHOD

The code FlexPDE (**Backstrom, 2005**) is used to perform finite element method to analyze the laminar mixed convection heat transfer and fluid flow in a channel with cavity. It is well known in the numerical solution field that the set of equations above (1-4) may be highly oscillatory or even sometimes undetermined because of inclusion of the pressure term in the momentum equations. In finite element method there is a derived approach with purpose of stabilizing pressure oscillations and allowing standard grids and elements. This approach enforces the continuity equation and the pressure to give, which called penalty approach as follows (**Langtangen, 2002**).

$$\nabla^2 P = \gamma\left(\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y}\right) \tag{8}$$

Where γ is a parameter that should be chosen either from physical knowledge or by other means (**Langtangen, 2002**). A most convenient value for γ was attained in this study to be 1E5μ/L^2.
Hence, the continuity eq.(1) is excluded from solution system and replaced by eq.(8).

VALIDATION

Software Validation

The grid dependency is checked together with continuity equation and obtained results showed an exact validation of the velocity distribution for a grid size obtained by imposing an accuracy of 10^{-4}. This accuracy is a compromised value between the result accuracy and the time consumed in each run. The girded domain for Ri =1, Re=100 is shown in **Figure 2(a)** and the distribution of $\left(\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}\right)$ over the domain is presented in **Figure 2(b)**.

Numerical results Validation

The present code was extensively validated based on the problem of (**Manca et al., 2003**). The results obtained by our code in comparison with those reported in (**Manca et al.,2003**). For Ri = 0.1 and Re=100, Pr=0.71. The physical problem studied by (**Manca et al., 2003**) was an open cavity with a heated wall is the horizontal surface of the cavity (heating from below), and the ratio between the channel and cavity heights (H/D=1). The present results have an excellent agreement with the results obtained by (**Manca et al., 2003**). **Figure 3** shows the comparison of the flow and thermal fields between the present investigation and (**Manca et al., 2003**). From these comparisons, it can be concluded that the current code can be used to predict the flow and thermal field for the present problem.

RESULT AND DISCUSSION

Results are presented for laminar mixed convection heat transfer and air flow inside a channel with cavity, where the Ri has been varied from 0 to 5, Re=100 and Pr = 0.71. The aspect ratio (AR=L/D) considered in the range of 1.0, 1.5 and 2.0. Computation was carried out for the three cases; case 1, the cavity wall heated from below isothermally, case 2, all the walls of the cavity heated at uniform temperature and case 3, the bottom wall of the channel is heated with constant temperature. For all cases the others walls are considered insulated.

Flow and Temperature Fields

Case 1: heating from blow

The effect of the cavity aspect ratio on the flow structure and temperature distribution is shown in **Figure 4(a to c)**, for steady state flows obtained for Ri=1, Re =100 and the values of AR between 1.0 and 2.0. For AR=1, the streamlines are shown in **Figure 4**(left) **(a)**. It can be seen that the streamlines are parallel to each other in channel part indicating a sign of supremacy of forced convection in this part and a small recalculating cell is formed in the center of the cavity. This is because of enhancement of the natural convection. The size of the cell and the intensity of the streamlines are increased with increasing of AR but the open lines are still parallel to each other in channel part. The corresponding isotherms plots are presented in **Figure 4**(right) **(a through c)**. The thermal boundary increases in thickness as AR increases due to the reduced fluid density and make a denser of isotherms close to the heated wall.

The effect of Richardson numbers (Ri) on the flow structure and temperature distribution for Re=100, AR=1.5 and Ri=0, 1, and 5 is illustrated in **Figure 5 (a to c)**. For Ri=0, it can be seen that the conduction and forced convection effects are dominant, since the open lines are still parallel to each other in channel part and vortex is presented only in center of the cavity. On other hand, for higher values of Ri it clearly seen that a double vortex cell is developed in cavity center. This is due to the natural convection effect. The corresponding isotherms are shown in **Figure-5**(right) **(a to c)**, the isotherm lines are parallel to the hot wall when Ri $\leq$ 1, then the parallel lines are destroyed with the increase Ri. This is because of the dominant of the secondary flow as Ri increases.

Case 2: heating all cavity walls

Figure-6 shows the streamlines and isotherms for heating all cavity walls at Re=100 and Ri=1.0 with a range of aspect ratios from 1.0 to 2.0. It is noticed in **Figure-6** (left). The induced flow enters into the channel part through inlet area and sudden expansion of the bulk fluid in the cavity occurred. Thus, the bulk fluid occupies most channel part and a single cell is formed in the cavity space. It can be observed that the intensity of the streamlines increases with increasing aspect ratio. This is because of the heat source is quite big for high aspect ratio. The isotherms show that the temperature distribution is nearly uniform in channel while they are represented as curves parallel to each other and close to heated enclosure walls as depicted in **Figure-6** (right)(a to c).

Figure-7 represents streamlines and isotherms at AR=1.5 and Re=100 for different Richardson numbers. **Figure-7** (left) (a-c) shows two different regions within the enclosure. The upper part of the enclosure is almost stratified streamlines. The streamlines within the remaining region are circulation. These increases in their size and intensity as Ri increases indicates that significant effect of the buoyancy force. Furthermore, the isotherms are accumulated in the regions close to the heat source and the heat diffusion is predominant for small Richardson number (Ri =0) as shown in **Figure-7** (right) (a). For a higher Richardson number (Ri =5), the buoyancy effects are much stronger and causes a more vigorous loop in the enclosure as shown in **Figure-7** (right) (c).

Case 3: heating the bottom wall for channel

The effect of various aspect ratio (AR) on streamlines and isotherms are studied while the values of Re and Pr are fixed at 100 and 0.71, and Ri=1, also the effect of Richardson number is studied and considered from 0 to 5, as shown in the **Figures 8** and **9** respectively. In **Figure-8** (left) the shape of the streamlines is very similar to those reported in Figure-6 (left), and this is due to the fact that the buoyancy effect is overwhelmed by the effect of the external airflow. In this figure a small vortex appears just at the cavity center then it is spreads as AR increased because of the wide heated region. Also it can be seen for **Figure-8** (right)(a-c), that the isotherms become clustered near the heat surface, and thermal boundary layer starts growth at entrance flow. Next, as AR increases the heating region increases, and the isothermal lines fill the most space of the channel and cavity due to the growth of the thermal boundary layer.

Effect of Richardson number on streamlines and isotherms have presented in **Figure-9**. The flow structure in the absence of free convection effect (i.e. Ri =0) has shown in **Figure-9**(a), a small recirculation appears in cavity center. Further at Ri = 1.0, the existence of large cell indicates that natural convection effect is comparable with forced convection effect. Finally, at Ri = 5.0, the cell becomes largest and spreads at mid section in the cavity. This indicates that the forced convection is overwhelmed by the natural convective current. The isotherms are accumulated in the regions close to the heat source and the heat diffusion is predominant for small Richardson number (Ri =0) as shown in **Figure-9**(right) (a). As the Richardson number increases (Ri =5), the heat removal improves because the convective effects are stronger as shown in **Figure-9**(right) (c).

Heat transfer characteristics

Figure-10 illustrates the average Nusselt number at hot wall as a function of Richardson number for three modes at Re=100 and AR=1.5. Average Nusselt number (Nu) at hot wall increases gradually with increasing Richardson number (Ri). It can be concluded that more heat transfer from the heat source is expected. The minimum average Nusselt number is obtained for case 2. This is because the heated two vertical walls lead to reduce the convection heat transfer and the conduction become predominant. Next, the maximum Nu is noticed for case 3. This is due to the induced cold fluid flows over the heated surface. On the other hand, the average temperature of the fluid (θ_{av}) for three cases at Re=100 increases due to changing from dominant forced convection to dominant natural convection with increasing Ri as shown in **Figure-11**. From this Figure, it is clear that the maximum values of θ_{av} are presented in case 3. This points out to the increasing of heating region.

The effect of varying AR on average Nusselt number along the hot wall for different cases is illustrated in **Figure-12**. It can be seen from this Figure that for case 1, the maximum value of Nu at AR=1.5 then decreases as AR increases. This is because of the varying of the growth of thermal boundary layer along the hot surface with aspect ratio. In case 2, the overall heat convection enhancement with increasing of AR due to dominant convection heat transfer, while case 3, as AR increases Nu decreases due to the thermal boundary layer near the hot wall is thicker for higher AR than the lower AR. Furthermore, it is seen that the average temperature of the fluid in the cavity increases gradually with AR for three cases as reported in **Figure-13**, which indicated that a large area of the cavity remains at higher temperature.

CONCLUSIONS

A numerical investigation on mixed convection in a channel with cavity was carried out using a finite element method. The present study examined and explained the complex interaction between buoyancy and force flow in a channel with cavity.

The following obtained results have been summarized:

- Cavity aspect ratio (AR) has a great influence on the streamlines and isotherms distributions for three cases. Results show that the intensity of the flow increased and recirculation cell spread within the cavity as aspect ratio increased.
- Richardson number (Ri) has a significant effect on the streamlines and isotherms, high intensity of the vortex are found for the highest Ri =5. The thermal performance in terms of both average Nusselt numbers and the average fluid temperature are affected by Ri, they are increases with increasing Ri for all considered cases.
- Position of the heated wall has a significant effect on the streamlines and isotherms. Case 3 has higher Nusselt numbers and the average fluid temperature θ_{av} than the other cases.
- The results showed that there were marked differences among the three considered heating modes. For case 1at Ri =5 nonlinearity in the isotherms are found. For case 2 and case 3 at Ri =5 nonlinearity of the isotherms becomes higher and loop formation is presented.

REFRENCES

[1]Backstrom Gunor," Field of physics by finite element analysis using FlexPDE," GB publishing and Gunner Backstrom, Malmo, Sweedn, 2005.

[2]Buonomo B., Foglia G., Manca O., Nardini S.," Experimental investigation on the effect of longitudinal aspect ratio in mixed convection in horizontal channels heated below", 5th European Thermal-Sciences Conference, 2008.

[3]Carlos Alberto Chaves, José Rui Camargo and Valesca Alves Correa," Combined forced and free convection heat transfer in a semiporous open cavity, "Scientific Research and Essay Vol.3 (8), pp. 333-337, 2008.

[4]Chan Y.L.and Tien C.L.," A numerical study of two-dimensional natural convection in square open cavities", Numer.Heat Transfer 8, pp. 65–80, 1985.

[5]Deshpande M. D. and Srinidhi B. G.," Mixed convection in a lid-driven cavity: Appearance of bifurcation, periodicity and Hysteresis", Current Science, VOL. 89, No. 10, 2005.

[6]Ghasemi Behzad and Saiied Mostafa Aminossadati,"Numerical Simulation of Mixed Convection in a Recctangular Enclosure with Different Numbers and Arrangements of Discrete Heat Sources," The Arabian Journal for Science and Engineering, Volume 33, Number 1B, 2007.

[7]Jones G. F. and Cai J.," Analysis of a Transient Asymmetrically Heated/Cooled Open Thermosyphon," J. Heat Transfer, vol. 115, pp. 621–630, 1993.

[8]Khalil Khanafer, Kambiz Vafai, Marilyn Lightstone "Mixed convection heat transfer in two-dimensional open-ended enclosures," International Journal of Heat and Mass Transfer 45 (2002) 5171–5190, 2002.

[9]Langtangen, H.P., Mardal K.A. and Winther R.,"Numerical methods for incompressible viscous flow,"Advances in water Resources 25 1125-1146, 2002.

[10]Manca, O., Nardin, S.,Khanafer, K.,Vafai K.,"Effect of Heated Wall Position on Mixed Convection in a Channel With an Open Cavity," Numerical Heated Transfer ,Part A, 43: 259-282, 2003.

[11]Rahman M. M., Alim M. A., Mamun M. A. H., M. K., and Chowdhury A. K. M. S. Islam,"Numerical Study of Opposing Mixed Convection in a Vented Enclosure," ARPN Journal of Engineering and Applied Sciences, VOL. 2, No. 2, 2007.

[12]Rahman Md.M. , Alim M., Saha S., Chowdhury M.K.,"Effect of the Presence of Heat Conducting Horizontal Square Block on Mixed Convection inside a Vented Square Cavity," Nonlinear Analysis Modeling and Control ,Vol.14,No.4,531-548, 2009.

[13]Rahman Md.M. , Alim M., Saha S.,"Mixed Convection in a Square Cavity with a Heated-Conducting Horizontal Square Cylinder" Suranaree J. Sci. Technol. 17(2):139-153, 2009.

[14]Rahman Md. Mustafizur, Elias Md. and Alim M. A.,"Mixed Convection flow in a Rectangular Ventilated Cavity with a Heated Conducting Square Cylinder at the Center," ARPN Journal of Engineering and Applied Sciences, VOL. 4, No. 5, ISSN 1819-6608, 2009.

[15]Saldana J. G. Barbosa, N. K. Anand and V. Sarin,"Numerical Simulation of Mixed Convective Flow Over a Three-Dimensional Horizontal Backward Facing Step," Journal of Heat Transfer, Vol. 127, SEPTEMBER 2005.

[16]Sumon Saha,Md. Arif Hasan Mamun,M. Zakir Hossain and A.K.M. Sadrul Islam,"Mixed Convection in an Enclosure with Different Inlet and Exit Configurations," Journal of Applied Fluid Mechanics, Vol.1, No.1,pp.78-93, 2008.

[17] Prasad Ajay K. and Koseff Jeffrey R.," Combined forced and natural convection heat transfer in a deep lid-driven cavity flow", Int. J. Heat and Fluid Flow 17: 460-467, 1996.

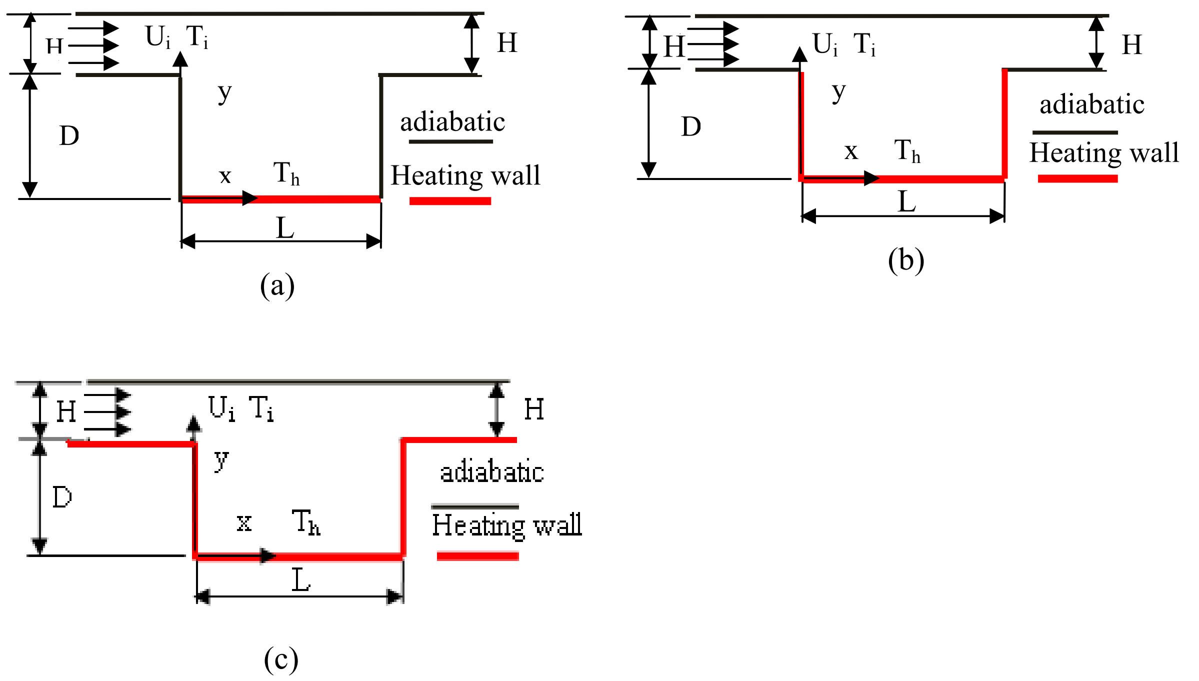

Figure 1 Geometry under consideration: (a) heating from below, (b) heating all cavity walls and (c) heating bottom wall of channel and walls of the cavity.

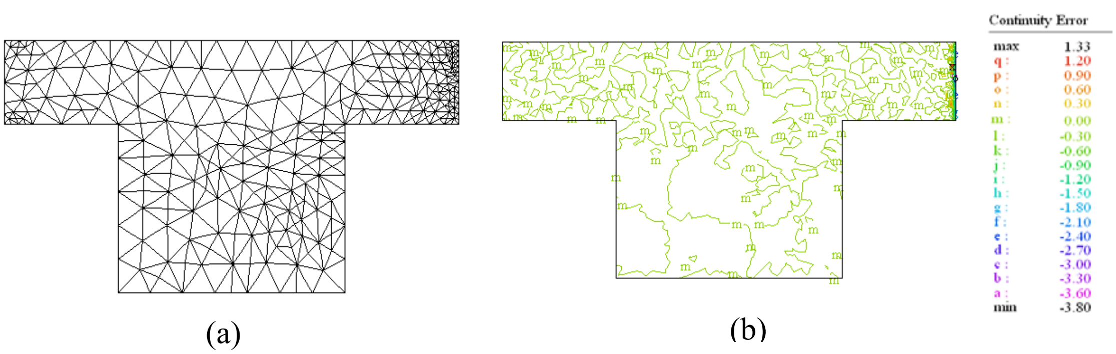

Figure 2(a) Grid distribution over the domain (b) Validation of continuity equation

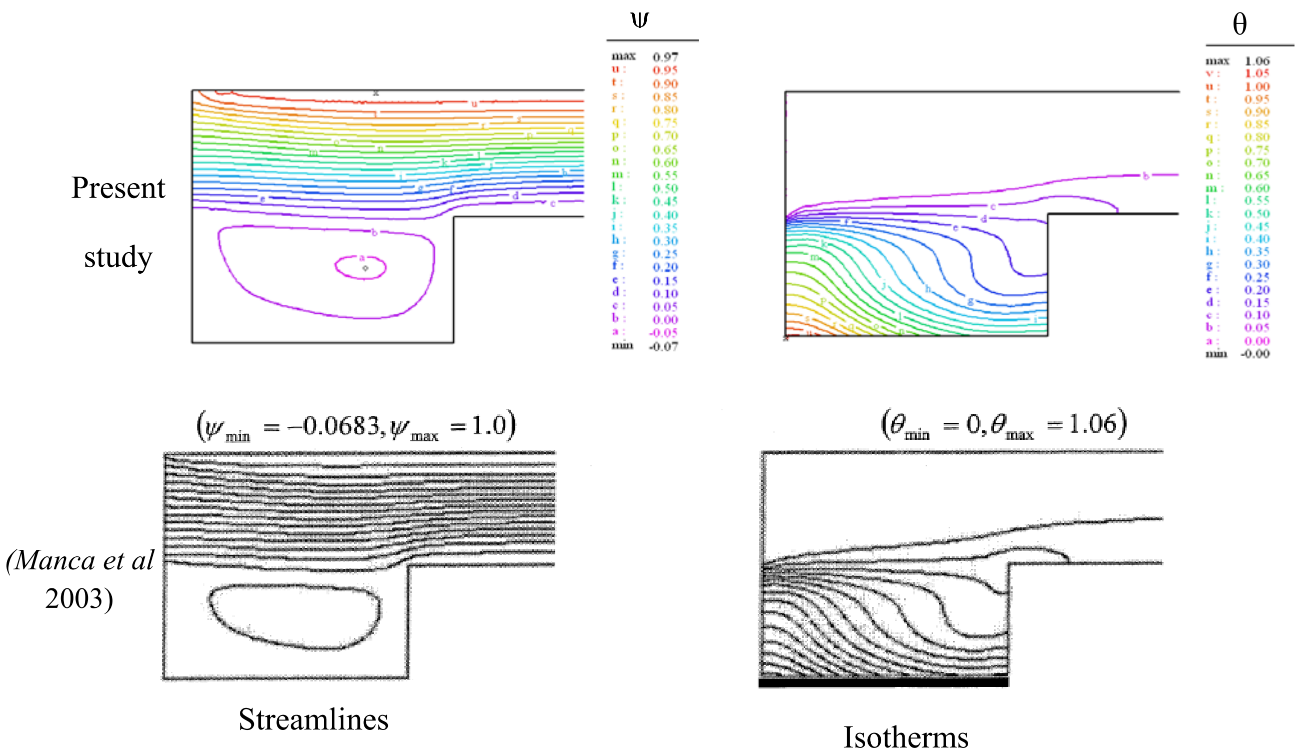

Figure 3 Comparison of streamlines and isotherms for validation at Pr = 0.71, Re = 100, Ri = 0.1 with the results of (**Manca et al., 2003**).

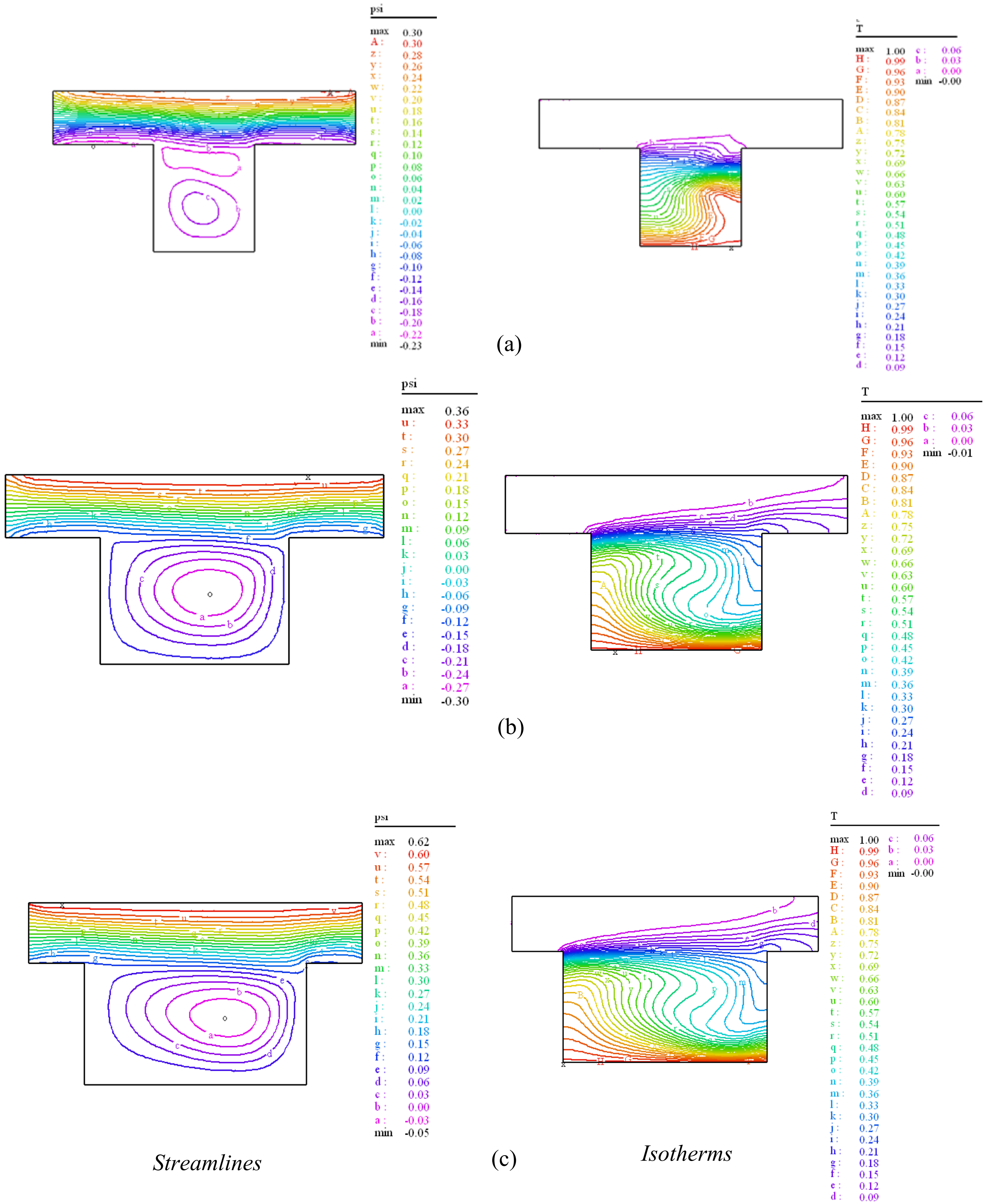

Figure 4 Streamlines (left) and isotherms (right) for heating from below (a) AR=1, (b) AR=1.5, (c) AR=2 at Ri=1.0,Re=100

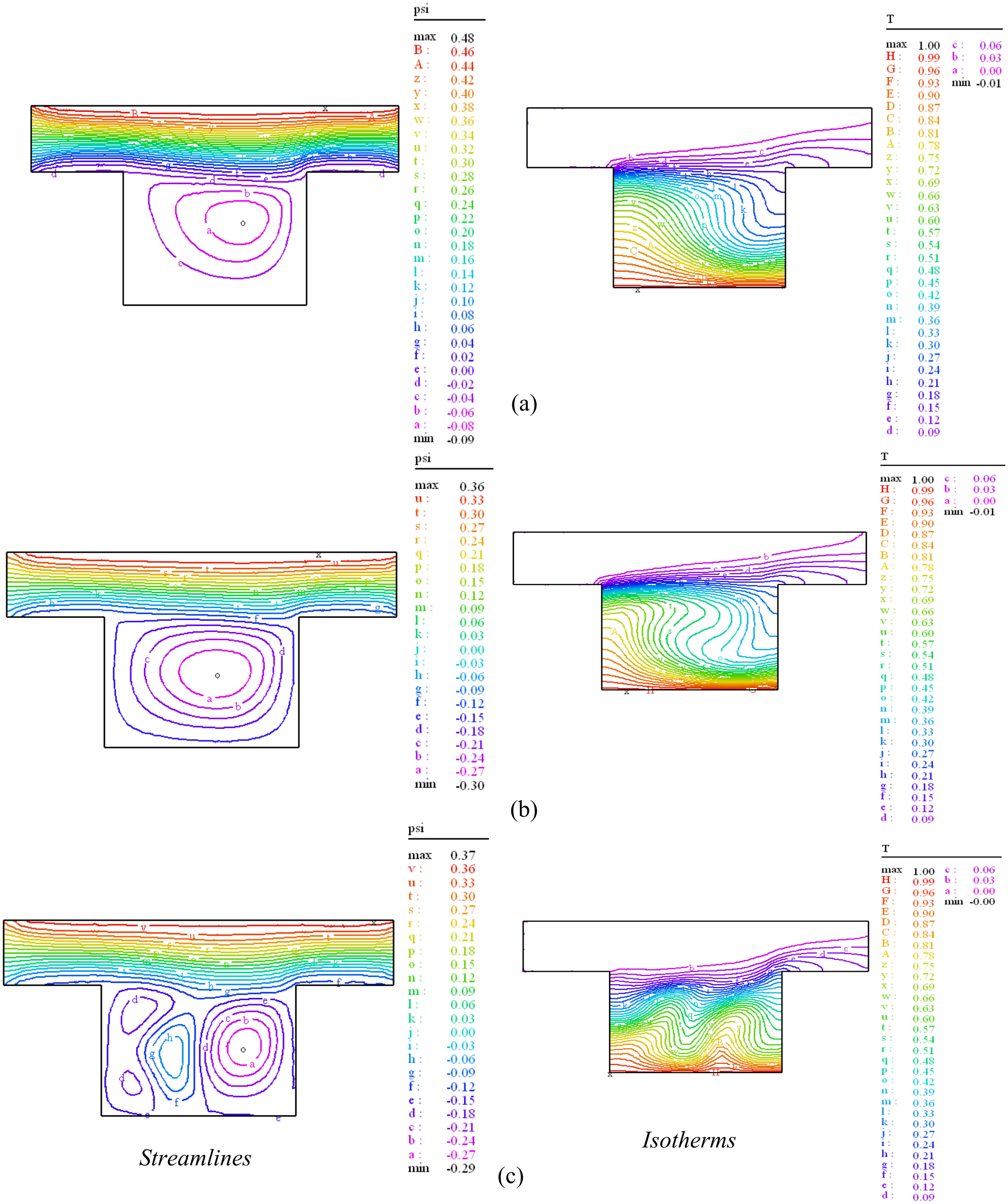

Figure 5 Streamlines (left) and isotherms (right) for heating from below (a) Ri=0, (b) Ri=1, (c) Ri=5 at AR=1.5 and Re=100

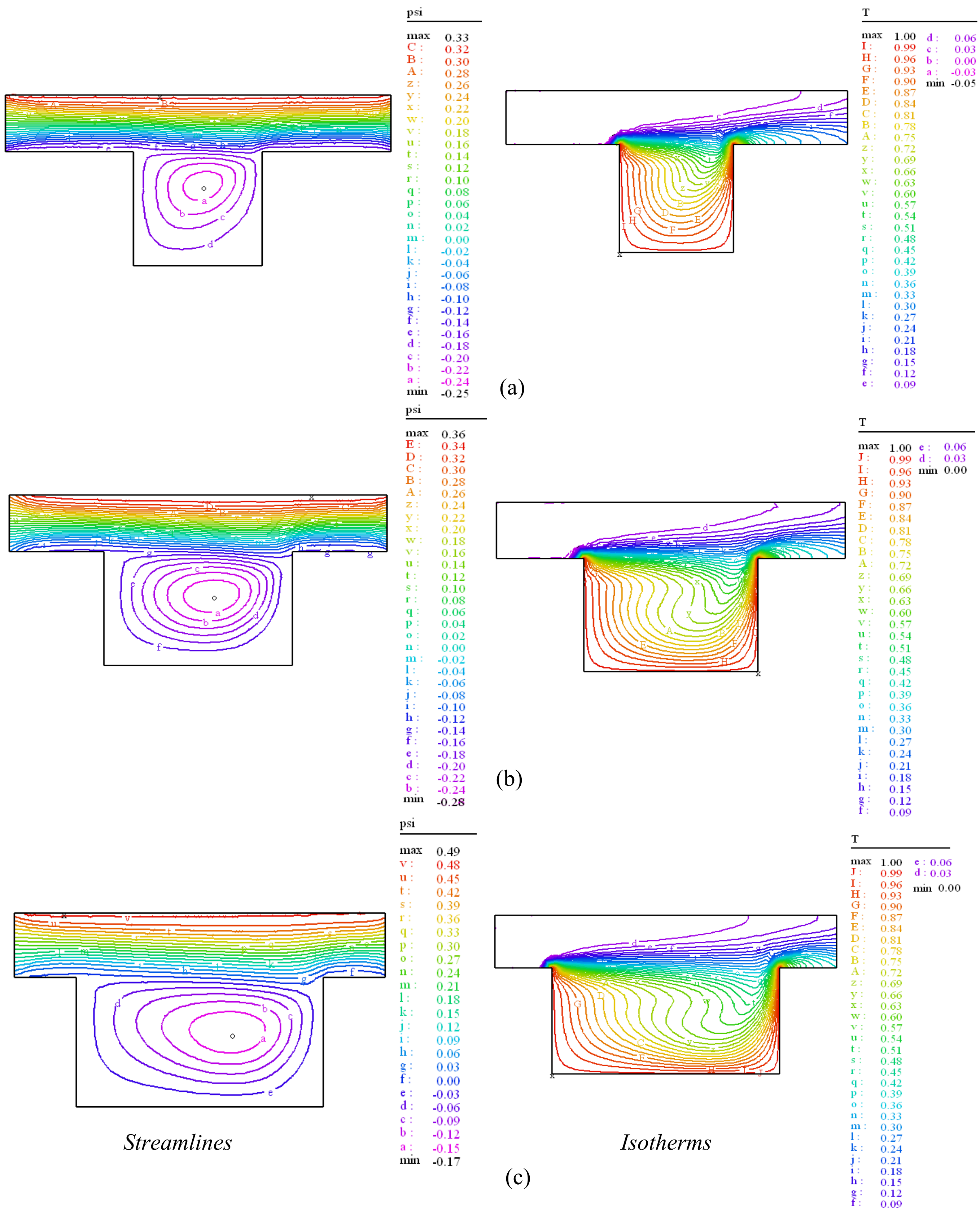

Figure 6 Streamlines (left) and isotherms (right) for heating all cavity walls (a) AR=1, (b) AR=1.5, (c) AR=2 at Ri=1.0,Re=100

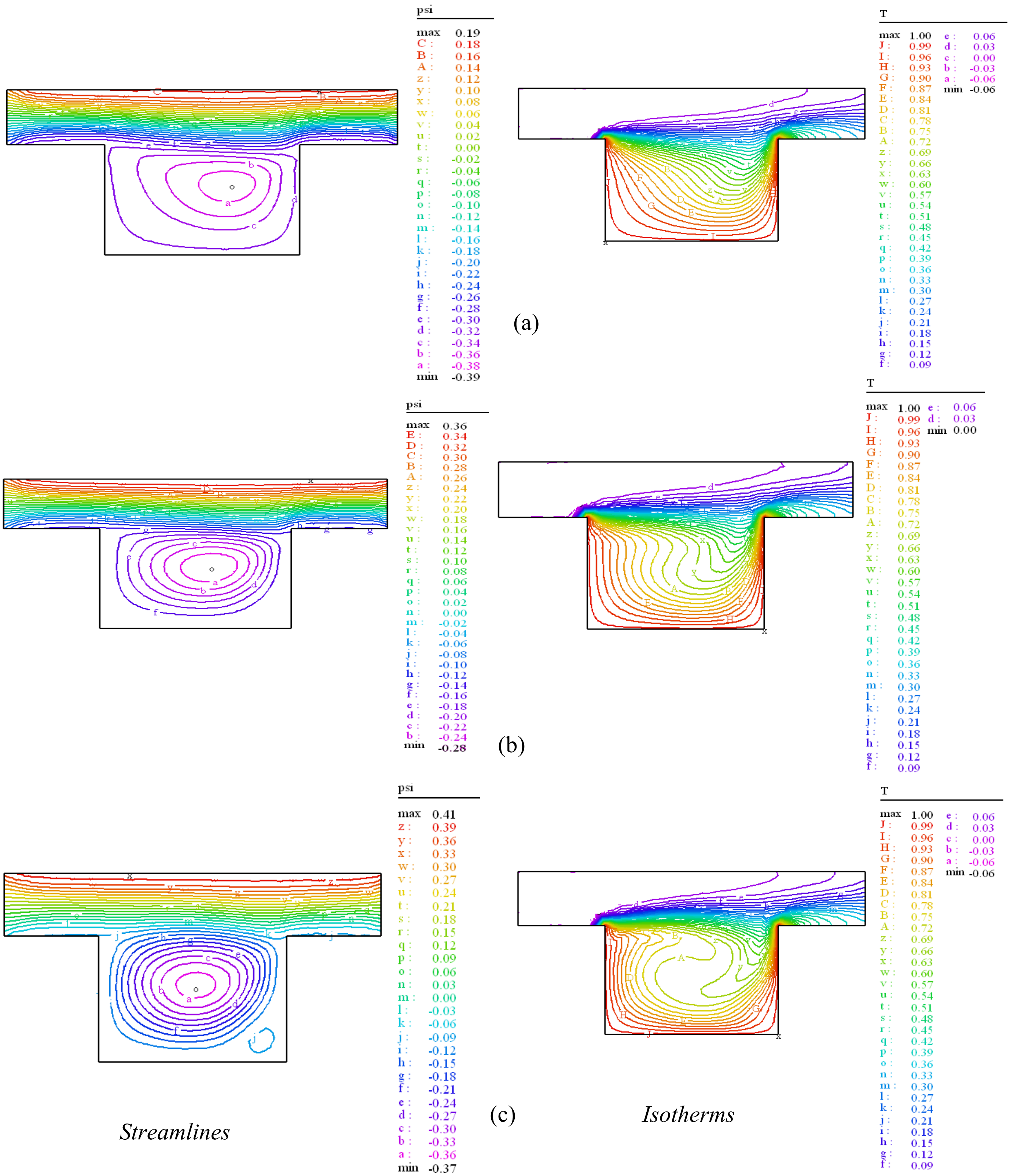

Figure 7 Streamlines (left) and isotherms (right) for heating all cavity walls (a) Ri=0, (b) Ri=1, (c) Ri=5 at AR=1.5 and Re=100

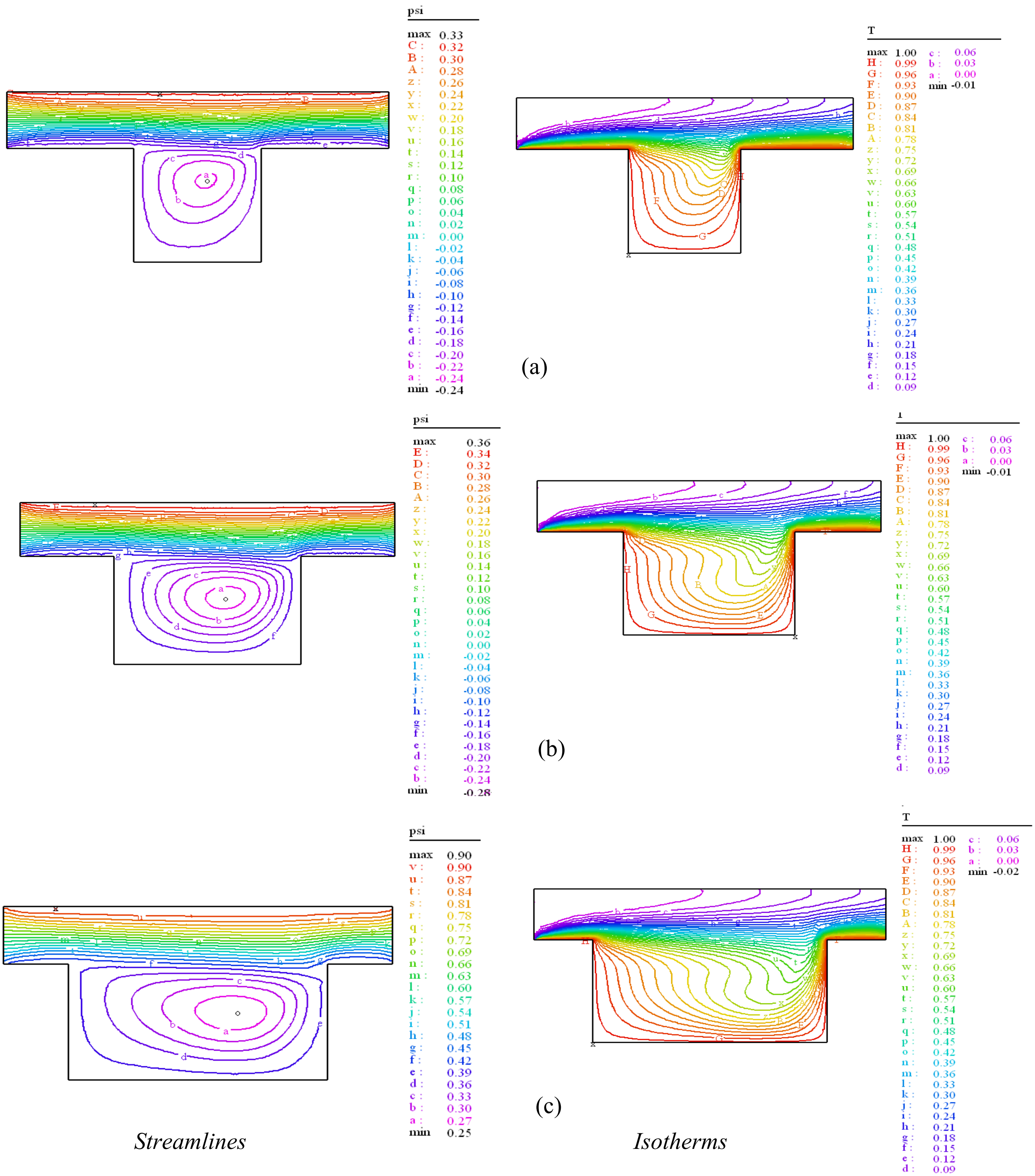

Figure 8 Streamlines (left) and isotherms (right) for heating bottom wall (a) AR=1, (b) AR=1.5, (c) AR=2 at Ri=1.0,Re=100

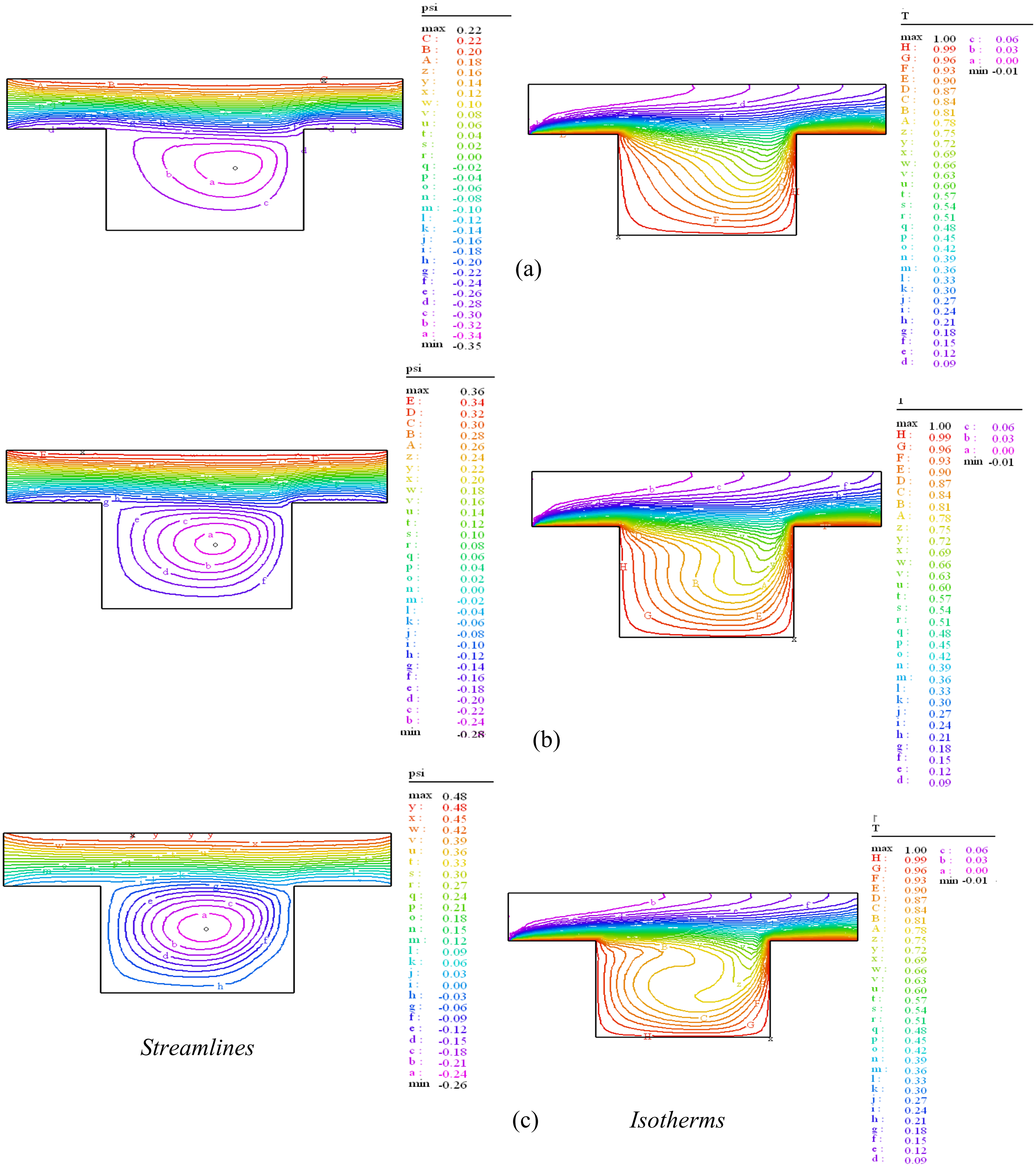

Figure 9 Streamlines (left) and isotherms (right) for heating bottom wall (a) Ri=0, (b) Ri=1, (c) Ri=5 at AR=1.5 and Re=100

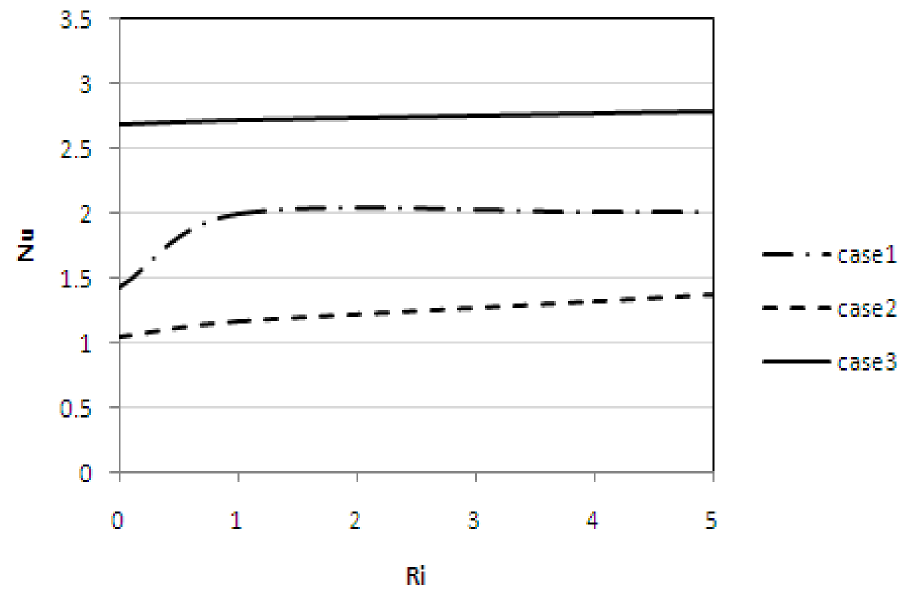

Figure 10 Variation of the average Nusselt number with Richardson number at AR =1.5 and Re =100.

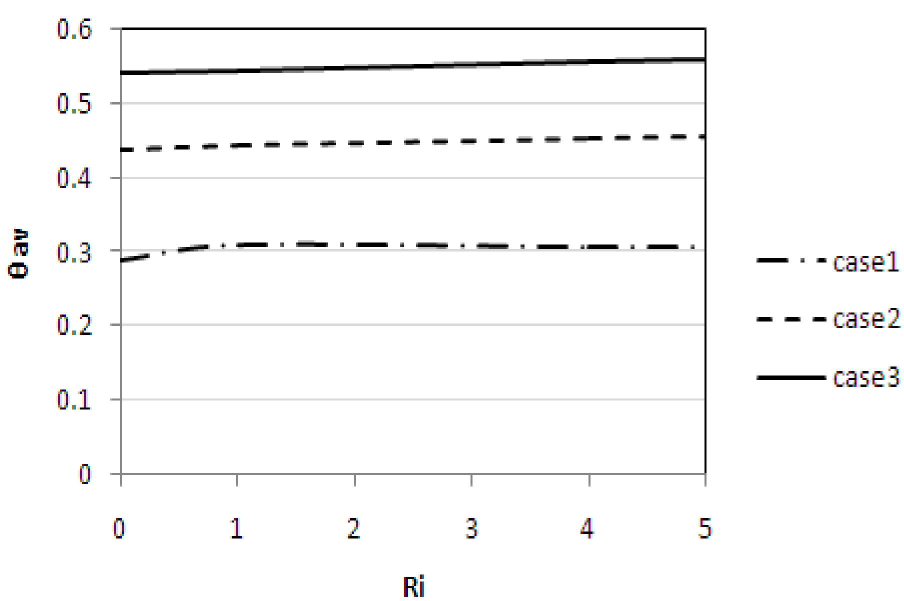

Figure 11 Variation of the average temperature fluid with Richardson number at AR =1.5 and Re =100

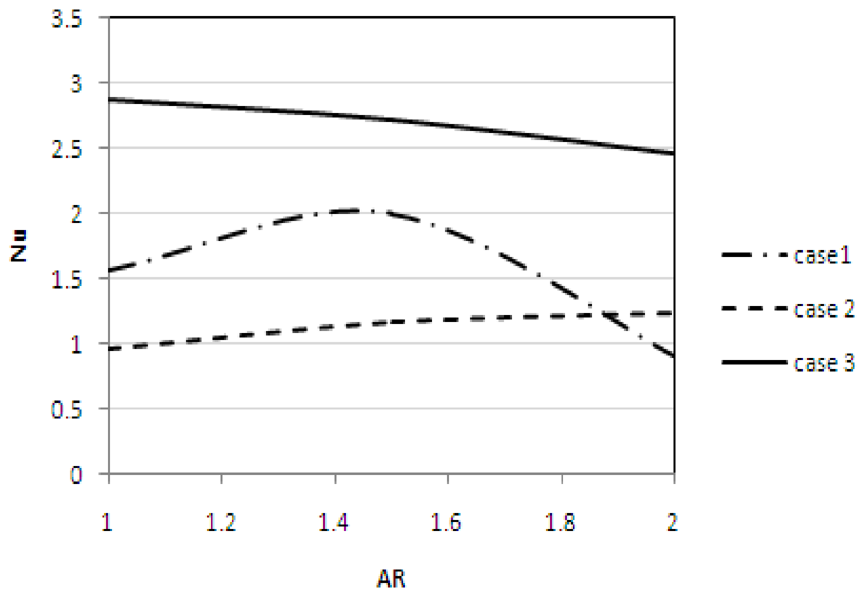

Figure 12 Variation of the average Nusselt number with aspect ratio at Ri =1 and Re=100.

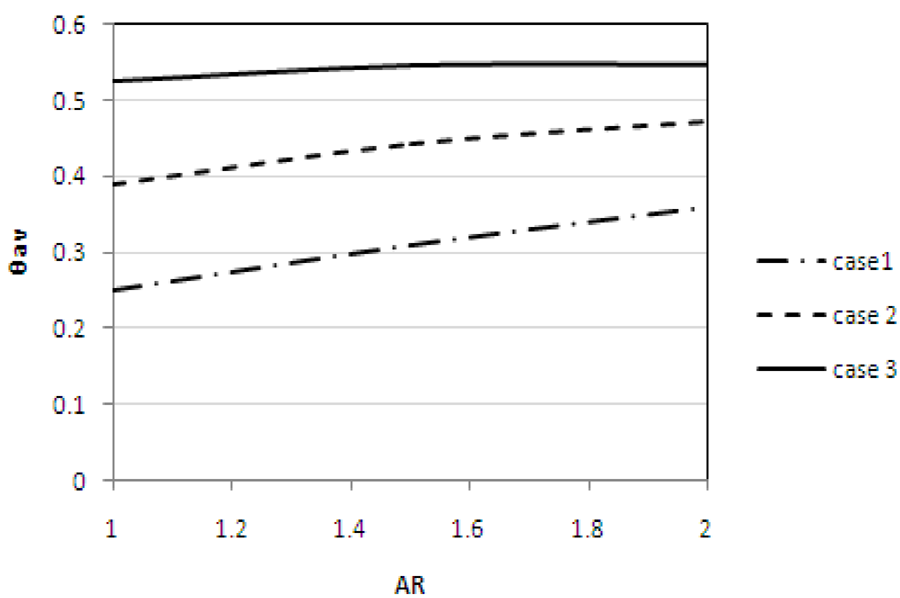

Figure 13 Variation of the average temperature fluid with aspect ratio at Ri =1 and Re=100.