Effect of Heated Wall Position on Magneto-Hydrodynamic Mixed Convection in A Channel with an Open Cavity.

Rowshon Ara Begum*

Abstract : The effect of heated wall position on magneto- hydrodynamic mixed convection in a channel with an open cavity has been investigated numerically. Magnetic field is acting countering the fluid flow, normal to the vertical wall of the cavity. Three different cases were considered based on heater position in the cavity as the left vertical side (Case 1), bottom side (Case 2) and right vertical side (Case 3). An external flow enters through an opening located at the left side of the channel, passes through the cavity and finally leaves the channel through an exit at the right side. The physical problems are represented mathematically by different sets of governing equations along with the corresponding boundary conditions. Using a class of appropriate transformations, the governing equations along with the boundary conditions are transformed into non-dimensional form, which are then solved by employing a finite-element scheme based on the Galerkin method of weighted residuals. Results are presented in terms of streamlines, isotherms, average Nusselt number along the hot wall, average fluid temperature at the exit port, pressure and temperature gradient in the domain for different combinations of the governing parameters namely Rayleigh number (Ra) at selected values of Hartmann numbers (Ha) and cavity aspect ratio AR.

Keywords: MHD, Heat transfer, Mixed convection, Cavity Aspect Ratio.

Introduction

Heat transfer through channel is an important development and an area of very rapid growth in contemporary trend of heat transfer research. The flow of energy carrying fluids through channel is a rapidly growing branch of fluid mechanics and heat transfer. Mixed convection heat transfer in a channel with an open cavity in the presence of magnetic field is a new branch of thermo-fluid mechanics. To describe the heat transport phenomenon, strong background of the hydrodynamics, the convective heat transfer mechanism and the electromagnetic field are prerequisite as they have a symbiotic relationship.

---

*Rowshon Ara Begum, Associate Professor, Department of Mathematics, Eden Mohila College, Dhaka

Application

Mixed convection in a channel with an open cavity plays a significant role in many practical applications. Simultaneous convection of buoyancy and forced convection is called as combined or mixed convection, which is of great interest in engineering applications such as nuclear reactors, lakes and reservoirs, cooling process of electronical devices, solar applications, combustion chambers, food processing and float glass production in industry.

Literature Review

Combined free and forced (mixed) convective flow in which neither the free convection nor the forced convection effects are dominant and both modes are in a comparable level arise in many natural and technological process. Various researchers investigated the effects of mixed convective flows in cavities, channels by using analytical, experimental and numerical methods. Several studies of mixed convection heat transfer in channels with open cavities have been reported in recent years. Leong et al[3] performed a numerical study on the mixed convection from an open cavity in a horizontal channel. Authors found that the heat transfer rate was reduced, and the flow became unstable in the mixed convection regime. Moreover, Raji and Hasnaoui[2] obtained numerical results by using a finite difference procedure for opposing flows mixed (forced and natural) convection flow in a rectangular cavity heated from the side with a constant heat flux and submitted to a laminar cold jet from the bottom of its heated wall. The fluid leaves the cavity via the top or the bottom of the opposite vertical wall. Later on, the same authors i.e. Raji and Hasnaoui investigated the mixed convection in ventilated cavities where the horizontal top wall and the vertical left wall were prescribed with equal heat fluxes. At the same time, Angirasa[1] numerically studied and explained the complex interaction between buoyancy and forced flow in a square enclosure with an inlet and a vent situated respectively, at the bottom and top edges of the vertical isothermal surface, where the other three walls are adiabatic. Also, Omri and Nasrallah performed numerical analysis by a control volume finite element method on mixed convection in a rectangular enclosure with differentially heated vertical sidewalls. Later on, Singh and Sharif extended their works by considering six placement configurations of the inlet and outlet of a differentially heated rectangular enclosure whereas the previous work was limited to only two different configurations of inlet and outlet. Hsu and Wang investigated the mixed convective heat transfer where the heat source was embedded on a board mounted vertically on the bottom wall at the middle in the enclosure. The cooling airflow enters and exits the enclosure through the openings near the top of the vertical sidewalls. Gau et al.[5] performed experiments on mixed convection in a horizontal rectangular channel with side heating. A numerical study of mixed convection heat transfer in two dimensional open-ended enclosures were investigated by Khanafer et al.[7] for three different forced flow angle of attack. Wang and Jaluria[4] numerically investigated the characteristics of the instability and the resulting effect on the heat transfer in mixed convection flow in a horizontal duct with discrete heat sources. The flow and temperature field for a two-dimensional confined slot jet impinging on an isothermal hot surface computed by Sahoo and Sharif . A finite-volume based computational study of steady laminar forced convection inside a square cavity with inlet and outlet ports was presented in Saeidi and Khodadadi[9]. Recently Rahman et al.[8] studied numerically the opposing mixed convection in a vented enclosure.

Material and Methods:-Mathematical model of physical phenomena may be ordinary or partial differential equations, which have been the subject of analytical and numerical investigations. The partial differential equations of fluid mechanics and heat transfer are solvable for only a limited number of flows. To obtain an approximate solution numerically, we have to use a discretization method, which approximated the differential equations by a system of algebraic equations, which can then be solved on a computer. The approximations are applied to small domains in space and /or time so the numerical solution provides results at discrete locations in space and time. Much as the accuracy of experimental data depends on the quality of the tools used, the accuracy of numerical solutions depend on the quality of discretizations used. Computational fluid dynamics (CFD) computation involves the formation of a set numbers that constitutes a practical approximation of a real life system. The outcome of computation process improves the understanding of the performance of a system. Thereby, engineers need CFD codes that can make physically realistic results with good quality accuracy in simulations with finite grids. Contained within the broad field of computational fluid dynamics are activities that cover the range from the automation of wellestablished engineering design methods to the use of detailed solutions of the Navier-Stokes equations as substitutes for experimental research into the nature of complex flows. CFD have been used for solving wide range of fluid dynamics problem. It is more frequently used in fields of engineering where the geometry is complicated or some important feature that cannot be dealt with standard methods.

Physical Model: Considered model is presented in Fig. 1. In these figures, channel includes a square cavity and magnetic field affects in –x direction and gravity acts in the vertical direction. Flow inlets to channel via inlet port at a uniform velocity, ui, temperature, Ti and exits the channel via outlet port. The length of channel is chosen as 3H, length and height of the cavity are defined by L and H respectively. In case 1, case 2 and case 3, left side, bottom side and right side are heated under constant temperature, T_h respectively. Remaining solid walls are adiabatic.

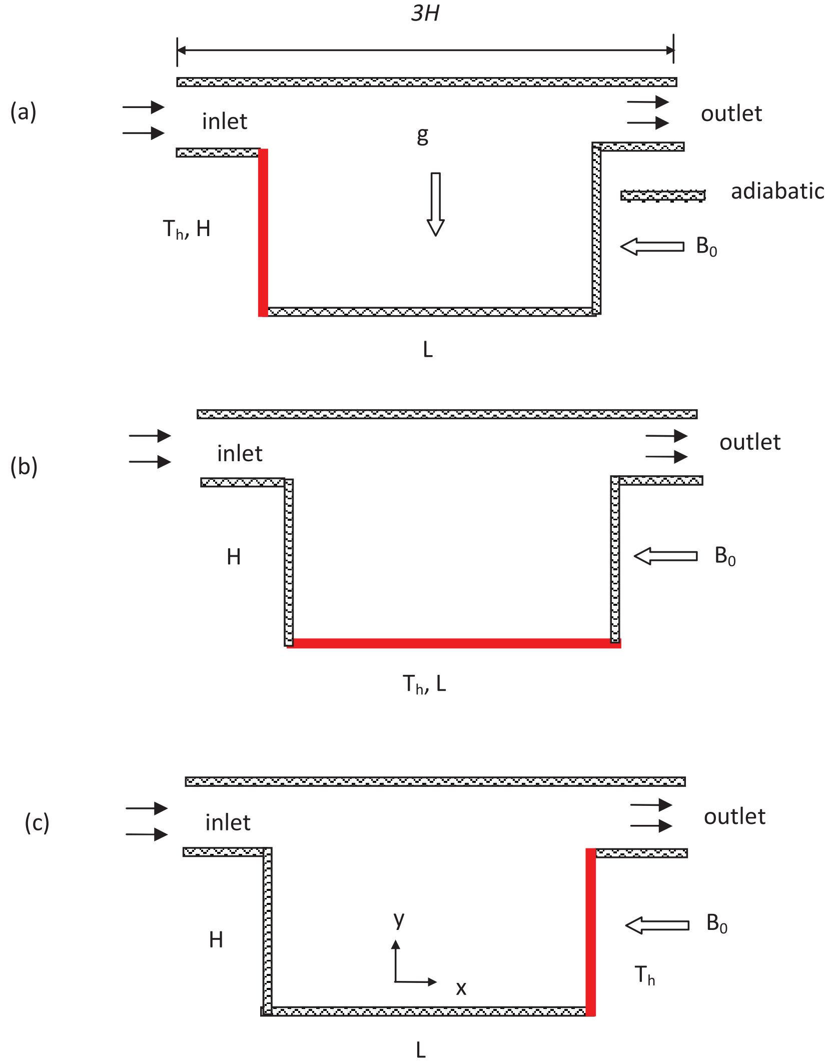


Fig. 1

Physical model under consideration: (a) heating from left, (b) heating

Governing Equations Along With Boundary Conditions: The electrically conducting fluids are assumed to be Newtonian fluids with constant fluid properties, except for the density in the buoyancy force term. Moreover, the fluid is considered to be laminar, incompressible, steady and two-dimensional. The electrically conducting fluids interact with an external horizontal uniform magnetic field of constant magnetic flux density B_0. Assuming that the flow-induced magnetic field is very small compared to B_0 and considering electrically insulated cavity walls. The electromagnetic force can be reduced to the damping factor–$B_{0\upsilon}^{9}$ where v is the vertical velocity component. The governing equations for the two-dimensional steady flow after invoking the Boussinesq approximation and neglecting radiation and viscous dissipation can be expressed as

Continuity Equation

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

Momentum Equations

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\upsilon\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\upsilon\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)+g\beta\left(T-T_i\right)-\frac{\sigma B_0^{2} v}{\rho} \tag{3}$$

Energy Equations

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\frac{k}{\rho c_p}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right) \tag{4}$$

where x and y are the distances measured along the horizontal and vertical directions respectively; u and v are the velocity components in the x and y directions respectively; T denote the fluid temperature, T*i* denotes the reference temperature for which buoyant force vanishes, p is the pressure and ρ is the fluid density, g is the gravitational constant, β is the volumetric coefficient of thermal expansion, c_p is the fluid specific heat, k is the thermal conductivity of fluid.

The boundary conditions for the present problem are specified as follows:

At the inlet: $u = u_i, v = 0,\ T = T_i$

At the outlet: $\frac{\partial u}{\partial x} = 0, v = 0, \frac{\partial T}{\partial x} = 0$

$$u = v = \frac{\partial T}{\partial n} = 0$$

At the heated wall: $u = v = 0, T = T_h$

where*n* is the non-dimensional distances either along *x* or *y* direction acting

to the surface and *k* is the thermal conductivity of the fluid.

Such local values have been further averaged over the entire heated surface to at all solid boundaries other than heated wall:
obtain the surface averaged or overall mean Nusselt number

$$Nu = -\frac{1}{L_s}\int_0^{L_s} \frac{\partial T}{\partial N} ds$$

Result and discussion

The effect of Hartmann number on the streamlines (on the left) and isotherms (on the right) at Ra = 10^3 and AR = 1. It is clearly seen from the figures, heating part of the cavity is not so effective on flow distribution and inlet flow goes through the channel from the top wall of the channel without any circulation inside the cavity except Ha = 0. This is because the lower value of Rayleigh number. One may notice that a circulating cell is formed in the clockwise direction and ψ_{min} = -0.001 at the bottom part of the cavity in absence of magnetic field. Isotherms are distributed from the left heated vertical wall into the cavity and hot fluid leave from the cavity from right top side. The effect of Hartmann number on the flow field and temperature fields has been depicted in Fig 2 at Ra = 104 and AR = 1. We observe that the flow and temperature fields are almost same as Fig. 3 for higher values Hartmann number Ha (= 50 and 100). But, an interesting result is found that a

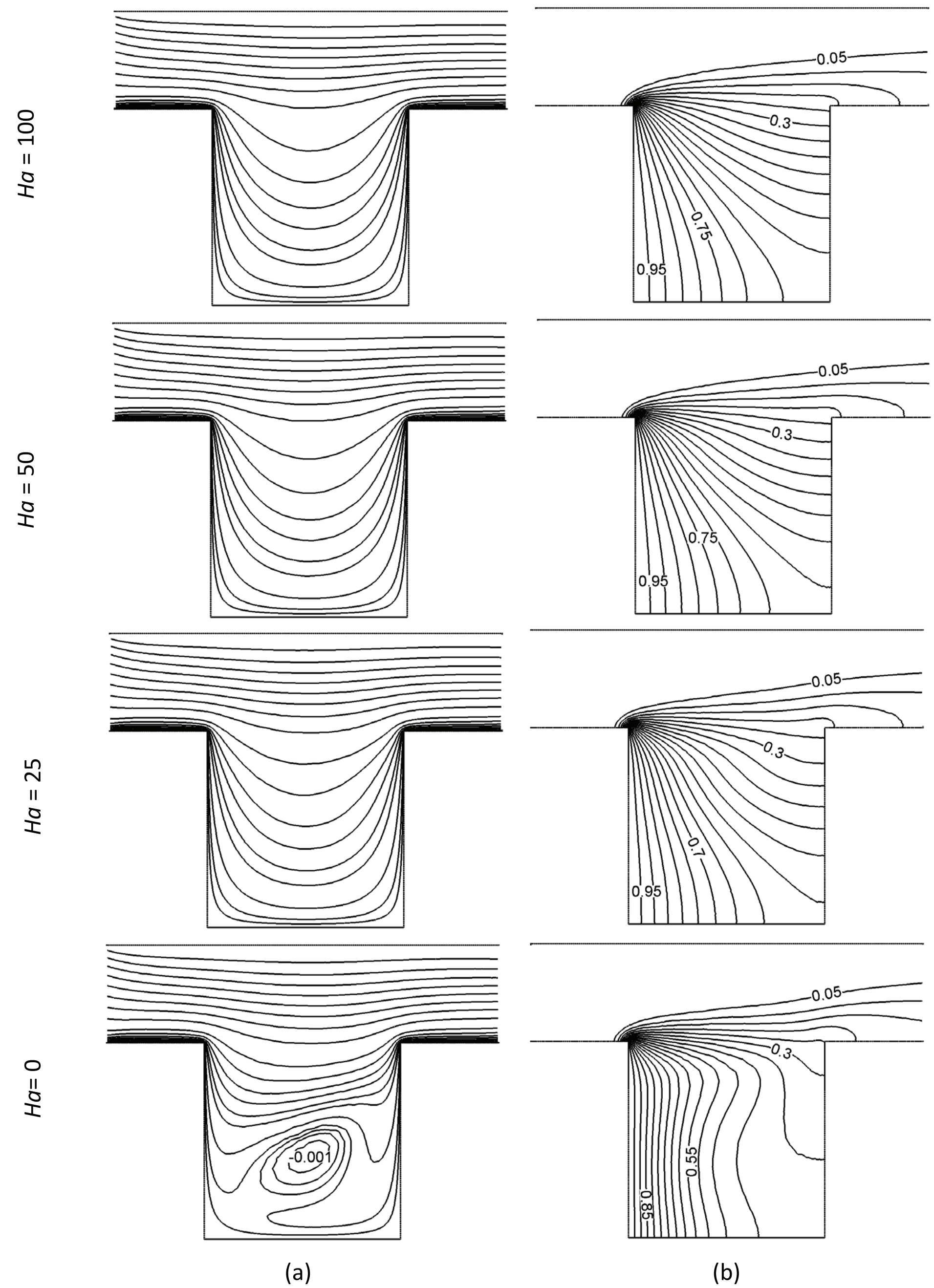


Fig. 1

Physical model under consideration: (a) heating from left, (b) heating from below, and (c) heating from right.

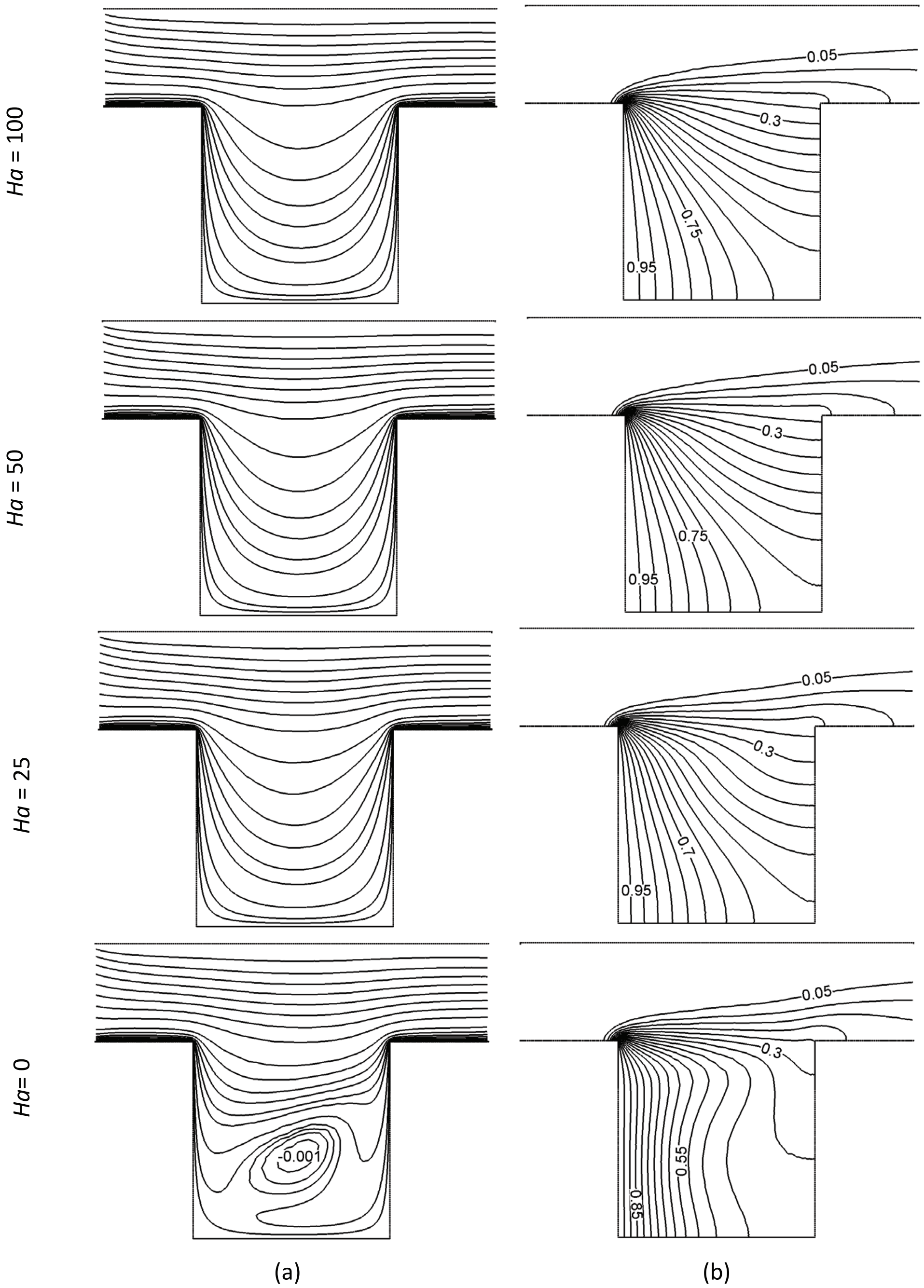


Fig. 2
(a) Streamlines and (b) Isotherms for the case 1 at $Ra = 10^3$, $AR = 1$ and selected values of Hartmann number Ha.

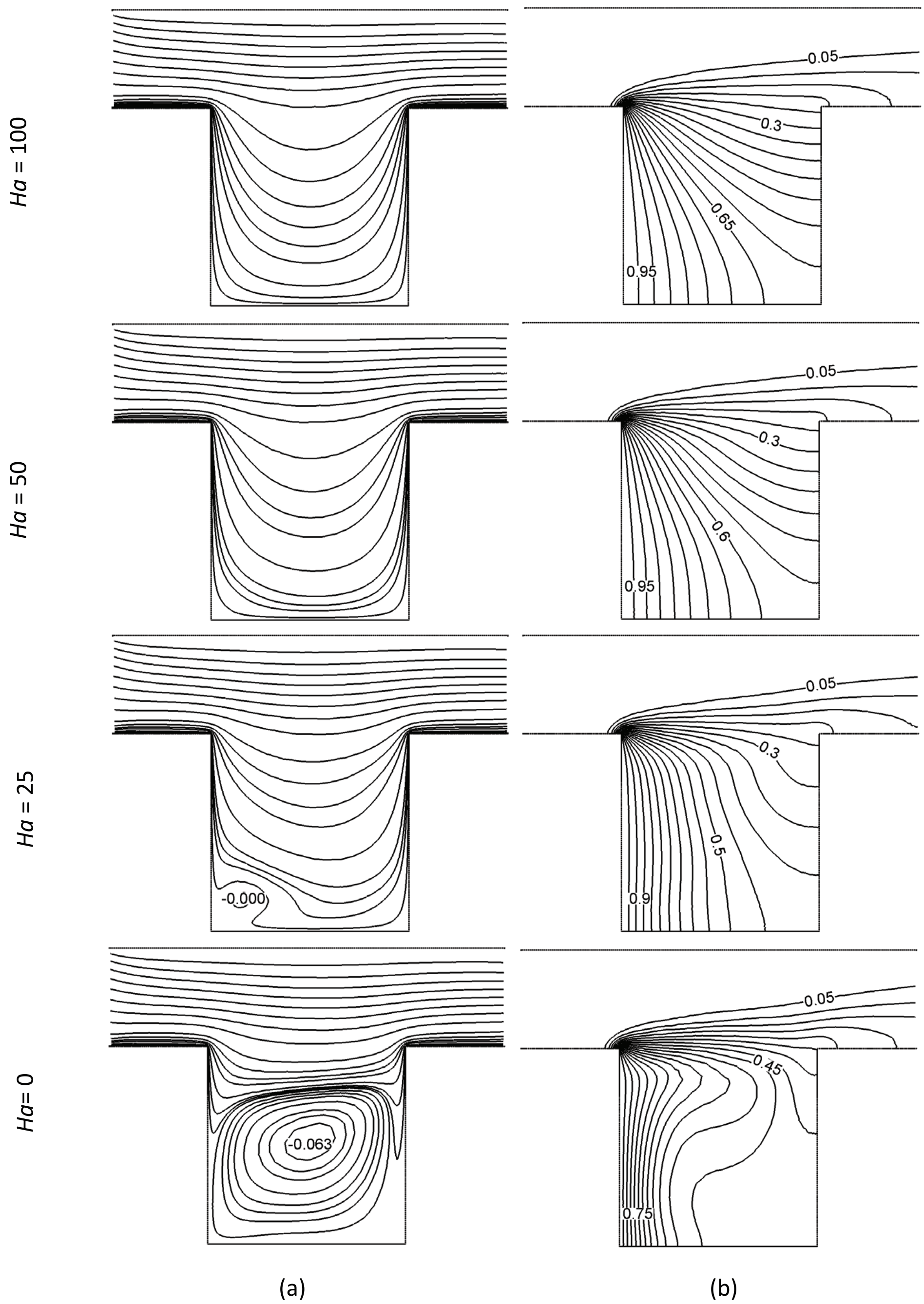


Fig. 3
(a) Streamlines and (b) Isotherms for the case 1 at $Ra = 10^4$, $AR = 1$ and selected values of Hartmann number Ha.

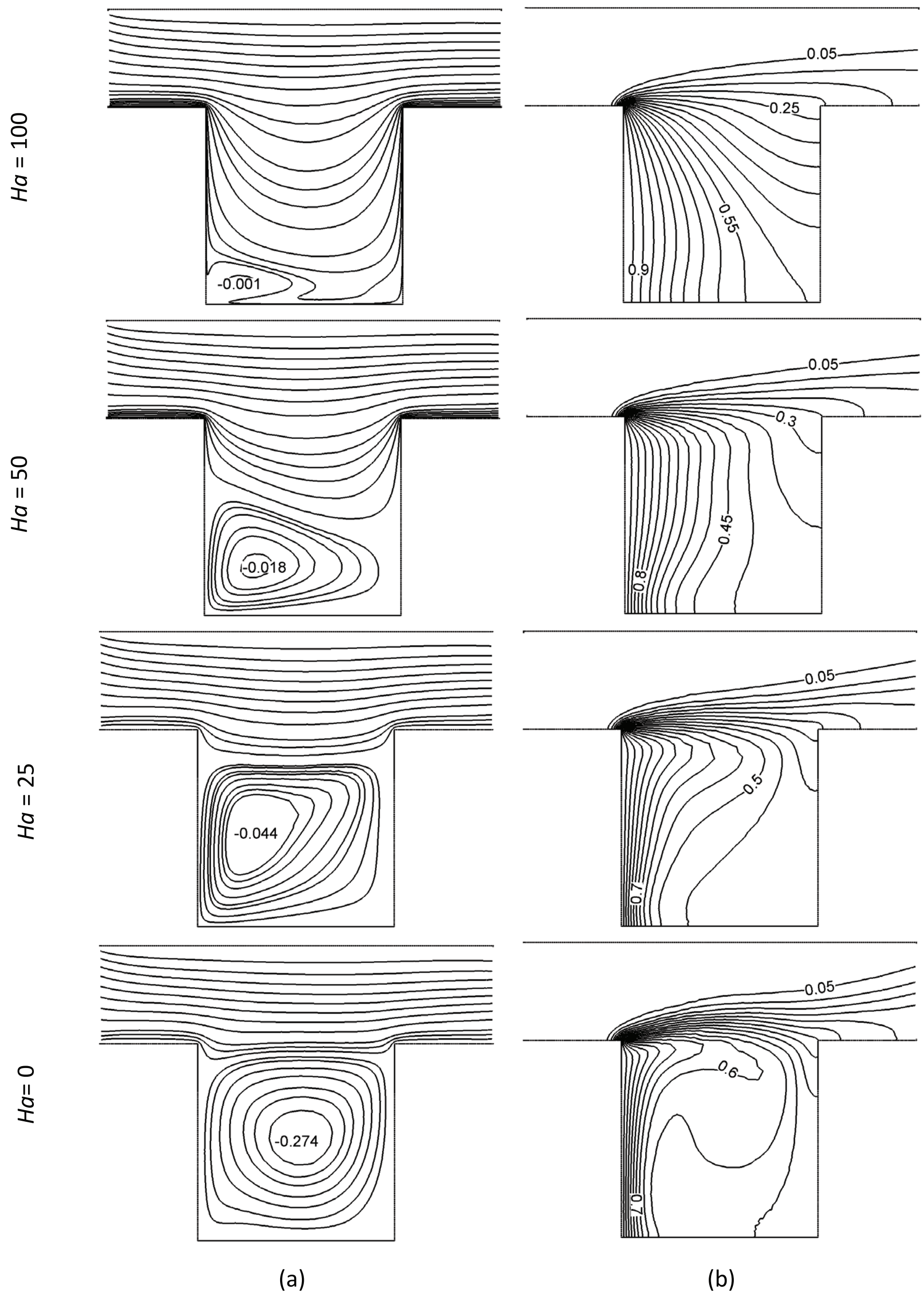


Fig. 4
(a) Streamlines and (b) Isotherms for the case 1 at $Ra = 10^5$, $AR = 1$ and selected values of Hartmann number Ha.

circulating cell is formed in the clockwise direction and ψ_{min} = -0.063 in absence of magnetic field. It is also noticed that the clockwise rotating cell covered most of the part of the cavity. As the cavity is heated from the left wall, the cavity behaves like differentially heated cavity from left and right. Thus, isotherms are parallel to left wall and wavy distribution is formed. The influence of Hartmann number on the streamlines and isotherms has been displayed in Fig 4 at Ra = 10^5 and AR = 1. One may notice that both the flow field as well as the thermal field strongly influenced for higher values of Rayleigh number. It can easily be seen that the circulating cell is formed in the clockwise direction and ψ_{min} = -0.274 in absence of magnetic field. If this figure is compared with Fig. 2, the circulation cell becomes stronger with ψ_{min} = -0.274 at Ha = 0. It is also noticed that the clockwise rotating cell occupies almost whole of the part of the cavity. This clockwise circulating cell decreases with the increasing values of Hartmann number. The location of the main center is changed a little bit to right side. The circulating cell center move to vertical heated. It is also clearly seen that the shape and size of the eddy changes while magnetic force changes. For Ra = 10^5, thermal boundary layer becomes thinner due to higher values of Rayleigh number as seen from Fig. 4. Plume like temperature distribution is seen for Ha = 0. Isotherms are parallel to the heater for Ha = 100 due to low flow velocity. In this case, isotherms are clustered around the heater and fluid flows directly over the cavity. Because domination of buoyancy effective flow is increased. From the figure, thermal boundary layer becomes thicker with decreasing of Hartmann number.

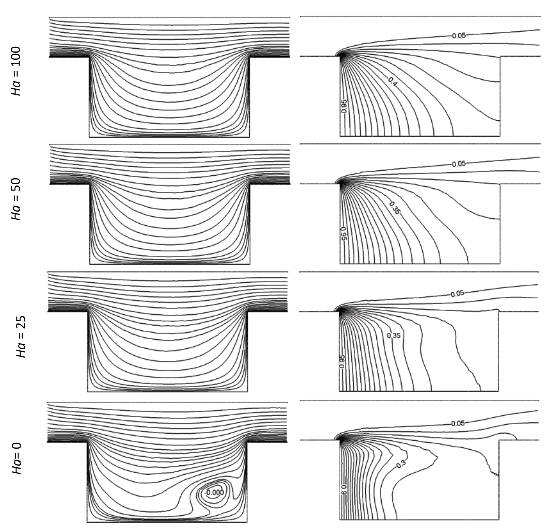


Fig. 5
(a) Streamlines and (b) Isotherms for the case 1 at $Ra = 10^{3}$, $AR = 2$ and selected values of Hartmann number Ha.

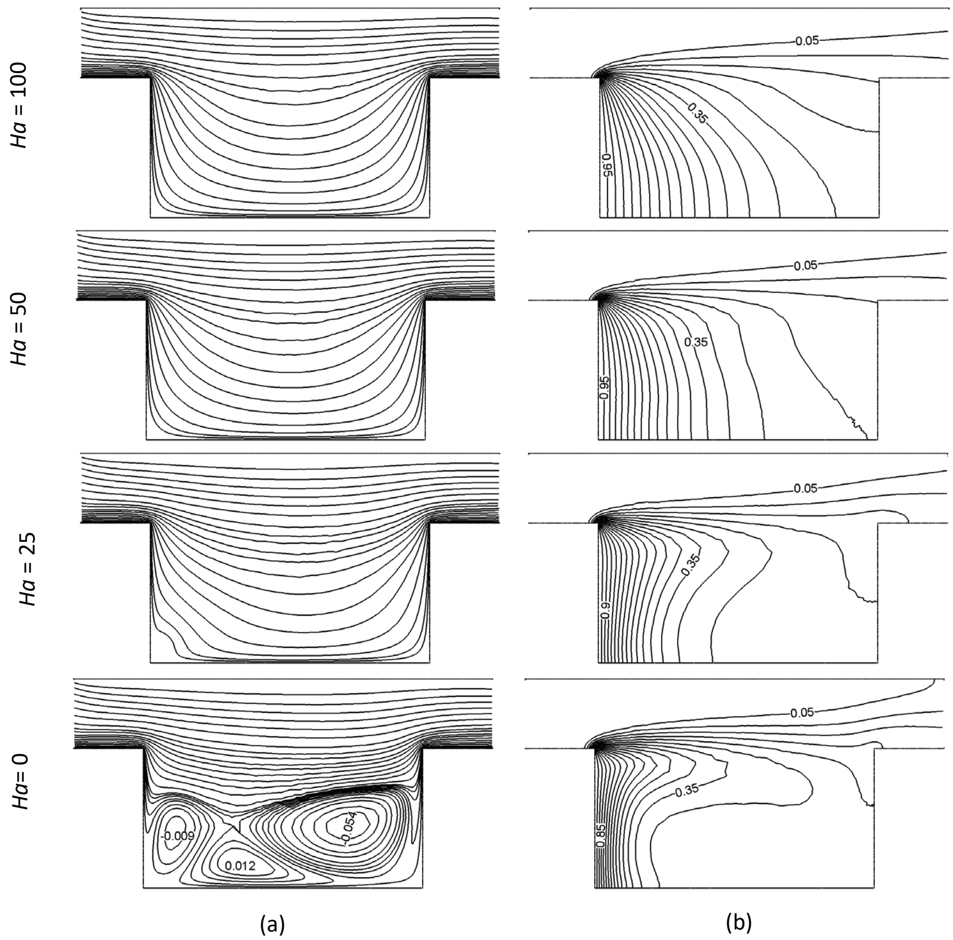


Fig. 6
(a) Streamlines and (b) Isotherms for the case 1 at $Ra = 10^4$, $AR = 2$ and selected values of Hartmann number Ha.

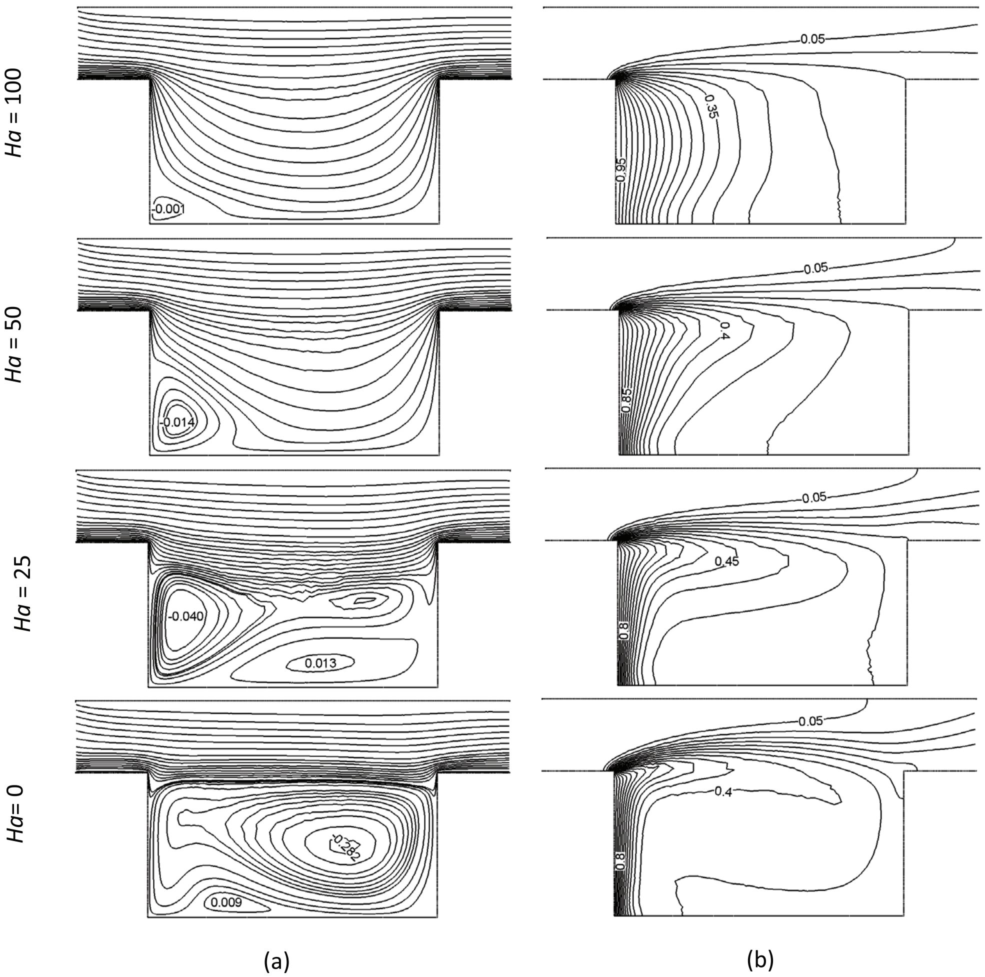


Fig. 7
(a) Streamlines and (b) Isotherms for the case 1 at $Ra = 10^5$, $AR = 2$ and selected values of Hartmann number Ha.

The effect of Hartmann number on the flow field and temperature fields has been shown in Fig 6 at Ra = 10^4 and AR = 2. It can be seen that the flow and temperature fields are almost identical as Fig. 5 for higher values Hartmann number Ha (= 25, 50 and 100). But, an interesting result is found that three circulating cells is formed in absence of magnetic field. As seen from the figure, thermal layer becomes thicker with decreasing of Hartmann number. For Ha = 0, plume like distribution is formed. Fig. 7 is plotted streamlines and isotherms for different values of Hartmann number *Ha* = 0, 25, 50 and 100 at Ra = 10^5. As seen from the left column of this figure, an amount of fluid near the heating wall of the cavity is activated so as to create a buoyancy-induced clockwise rotating cell for the lowest value of *Ha* = 0. As the Hartmann number increases the strength of the rotating cell is reduced and pushed to the left bottom corner of the cavity and then through flow in the channel gains its strength and occupies the whole of the cavity as well as the channel indicating the establishment of conduction mode of heat transfer. A higher value of Hartmann number, which is a measure of magnetic field, retards the flow velocity. Thus, this recirculation cell becomes smaller at Ha = 50, and 100 and it disappeared for further values of magnetic field. The corresponding isotherms for the lowest value of *Ha* = 10 shows the usual convective twist inside the cavity. The distortion of isothermal lines appears due to the high convective current inside the cavity. Distortions of isothermal lines start to disappear with increasing Hartmann number. As Hartmann number increases, isothermal lines inside the cavity as well as the channel approaches more and more towards the conduction-like distribution pattern of isothermal lines. For large Hartmann number *Ha* = 50 and 100, the convection is almost suppressed, and the isotherms are almost parallel to the horizontal wall, indicating that a quasiconduction regime is reached. Plume like temperature distribution is seen for Ha = 0 and 25.

Fig. 8 (a) and (b) illustrate the average Nusselt number and average fluid temperature at the exit port, respectively while AR = 1. The figures are given for different Rayleigh numbers at selected values of Hartmann numbers. Both Nusselt number and average fluid temperature at the exit port exhibit similar trends. In addition, both heat transfer and average fluid temperature are decreased with increasing of Hartmann numbers.

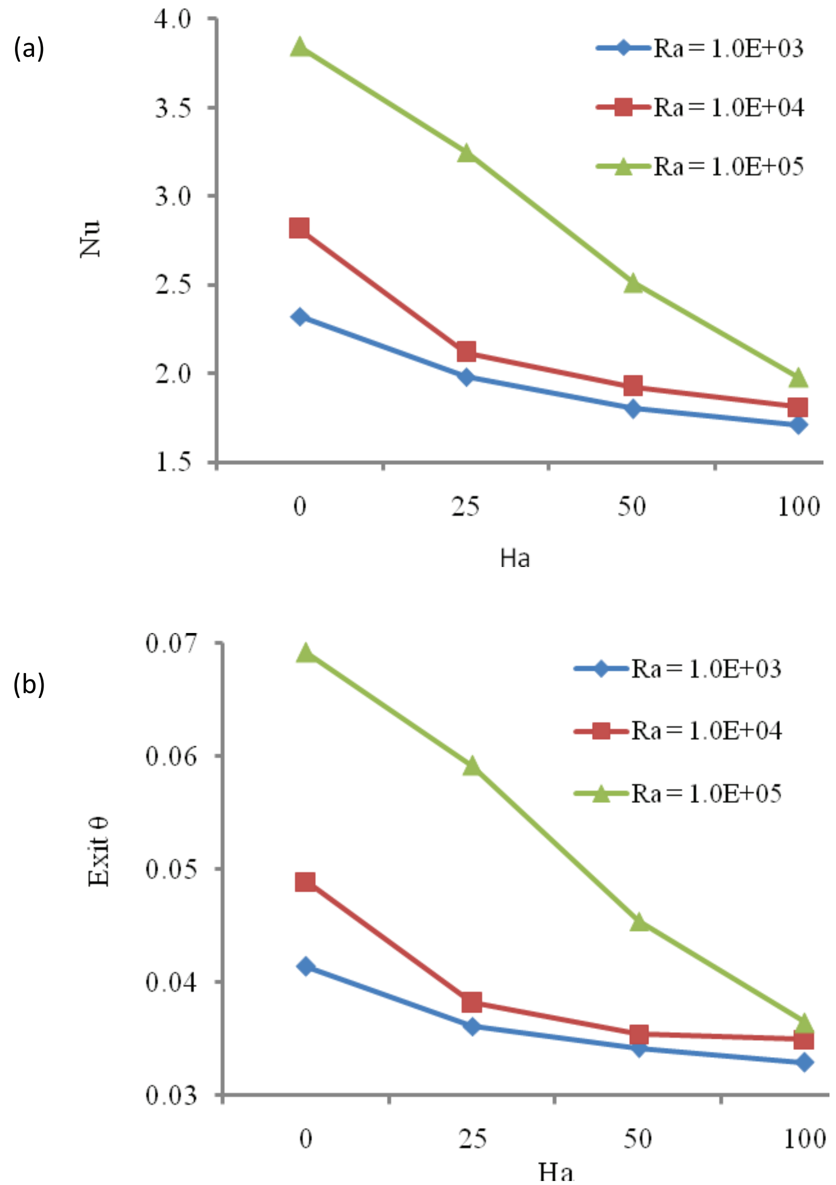


Fig. 8

(a) Average Nusselt number and (b) average fluid temperature at the exit port versus Hartmann number *Ha* for the case 1, at *AR* = 1 and selected values of Rayleigh number *Ra*.

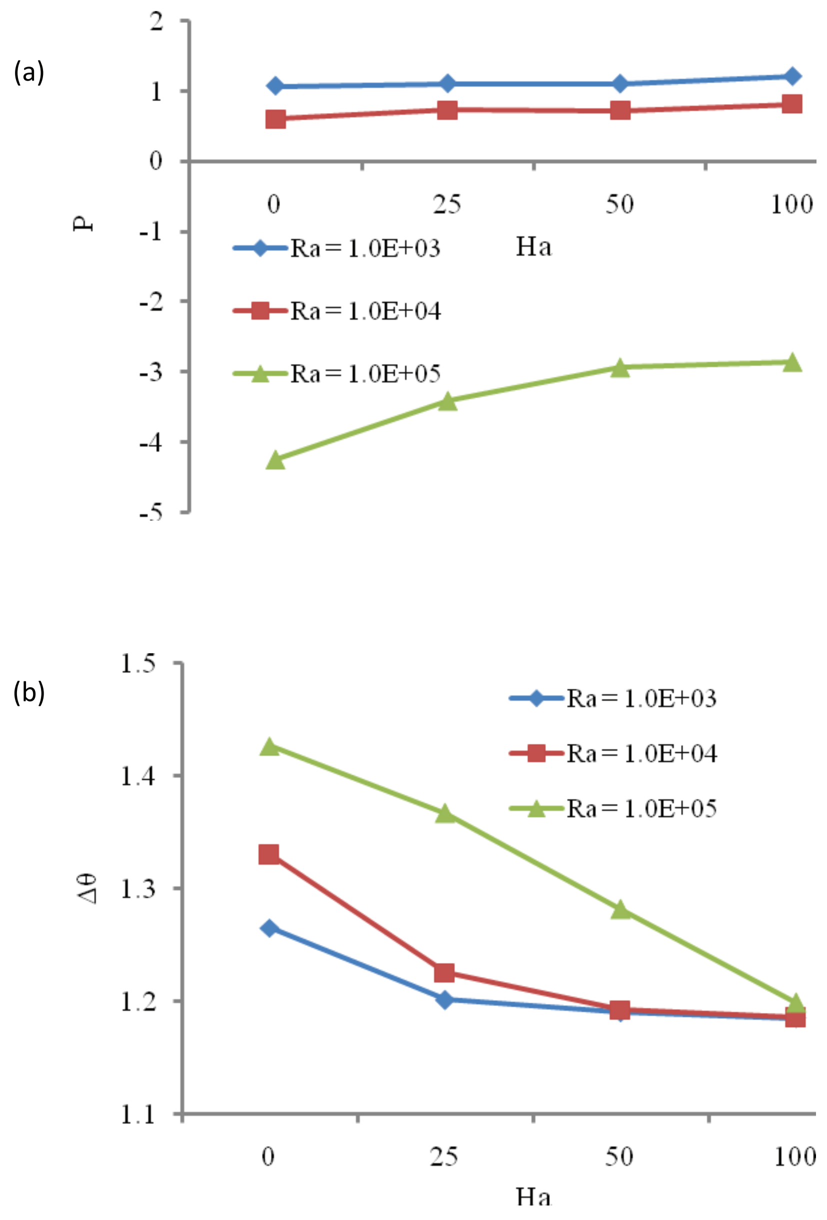
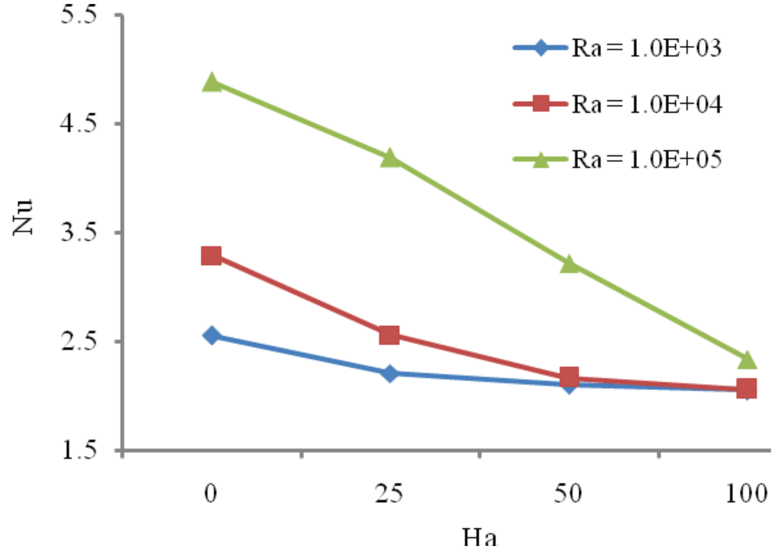


Fig. 9

(a) Pressure and (b) temperature gradient in the domain versus Hartmann number *Ha* for the case 1, at *AR* = 1 and selected values of Rayleigh number *Ra*.

(a)



(b)

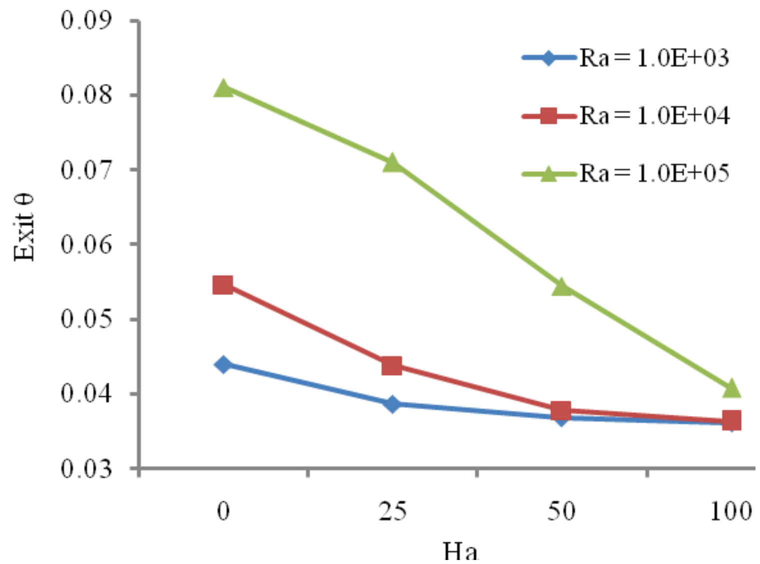


Fig. 10

(a) Average Nusselt number and (b) average fluid temperature at the exit port versus Hartmann number *Ha* for the case 1, at *AR* = 2 and selected values of Rayleigh number *Ra*.

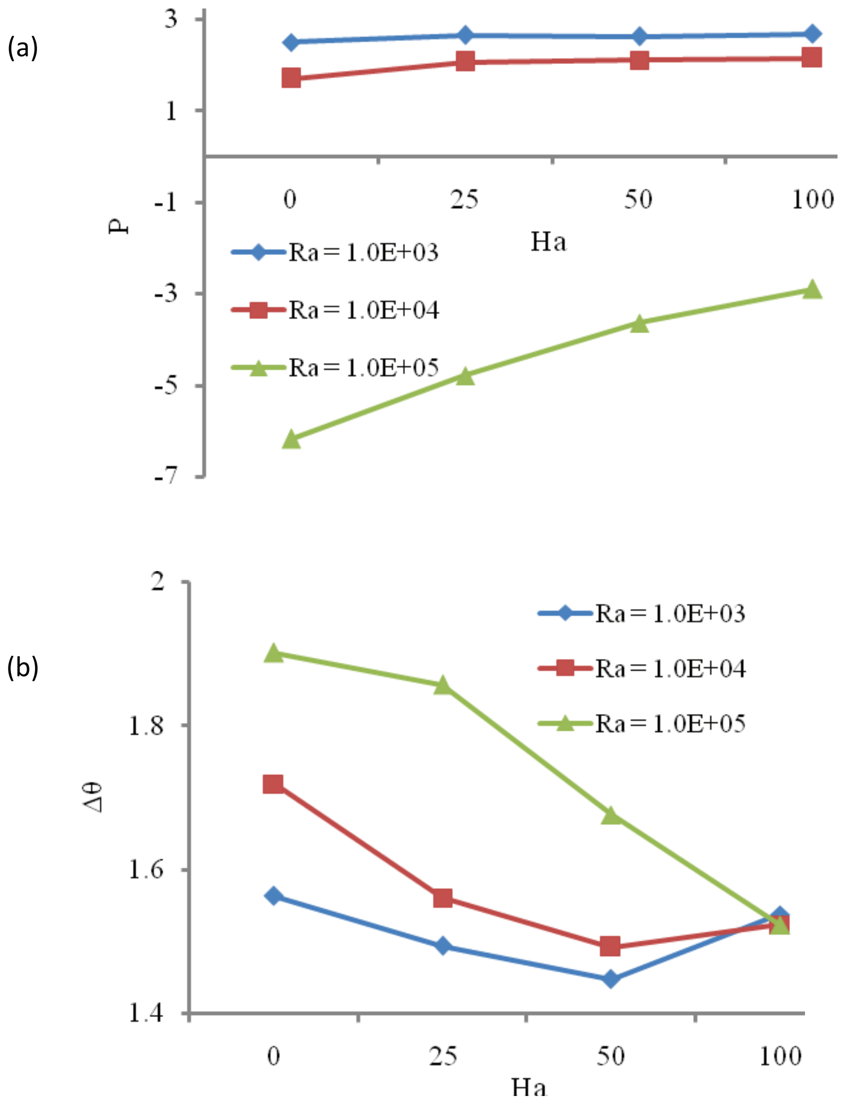


Fig. 11

(a) Pressure and (b) temperature gradient in the domain versus Hartmann number *Ha* for the case 2, at *AR* = 2 and selected values of Rayleigh number *Ra*.

Conclusion

Mixed convection in a channel with a cavity heated from different sides under the influence of the applied magnetic force has been investigated numerically. The results are presented for flow and thermal fields as well as heat transfer for the channel with an enclosure subjected to constant hot temperature at a wall of the cavity while the remaining sidewalls are kept adiabatic. Finite element method is used to solve governing equations. Comparisons with the beforehand published work are performed and found to be in excellent agreement. The influences of Rayleigh number, the Hartmann number and the cavity aspect ratio have been reported. The various ideas and results have been discussed in detail at the relevant chapters of the thesis. The figures are given for different Rayleigh numbers at selected values of Hartmann numbers. Both Nusselt number and average fluid temperature at the exit port exhibit similar trends. In addition, both heat transfer and average fluid temperature are decreased with increasing of Hartmann numbers.

Refferences

[1] Angirasa, D., “Mixed convection in a vented enclosure with isothermal vertical surface”, Fluid Dyn. Res. Vol. 26, pp. 219-233, 2000.

[2] Aminossadati, S. M., and Ghasemib, B., “A numerical study of mixed convection in a horizontal channel with a discrete heat source in an open cavity”, European J. Mechanics B/Fluids, Vol. 28, pp. 590-598, 2009.

[3] Bhuvaneswari, M., Sivasankaran, S., and Kim, Y.J., Magneto convection in a square enclosure with sinusoidal temperature distributions on both side walls, Numer. Heat Transfer A, Vol. 59(3), pp 167-184, 2011.

[4] Chamkha, A. J., “Hydro magnetic combined convection flow in a vertical lid-driven cavity with internal heat generation or absorption”, Numer. Heat Transfer, Part A, Vol. 41, pp. 529-546, 2002.

[5] Gau, C., Jeng, Y.C., and Liu, C.G., “An experimental study on mixed convection in a horizontal rectangular channel heated from a side”, ASME J. Heat Transfer, Vol. 122, pp. 701–707, 2000.

[6] Hsu, T.H., and Wang, S. G., “Mixed convection in a rectangular enclosure with discrete heat sources”, Numer. Heat Transfer, Part A, Vol. 38, pp. 627-652, 2000.

[7] Khanafer, K., Vafai, K., and Light stone, M., “Mixed convection heat transfer in two-dimensional open-ended enclosure”, Int. J. of Heat and Mass Transfer, Vol. 45, pp. 5171-5190, 2002.

[8] Rahman, M. M., Alim, M. A., Mamun, M. A. H., Chowdhury, M.K. and Islam, A.K.M.S., "Numerical Study of Opposing Mixed Convection in a Vented Enclosure", ARPN J. of Engineering and Applied Sciences, Vol. 2, No. 2, pp 25-36, 2007.

[9] Rahman, M. M., Alim, Khodadadi M. A., and Chowdhury, M. K., Magneto hydrodynamic mixed convection around a heat conducting horizontal circular cylinder in a rectangular lid-driven cavity with joule heating", J. Sci. Res. Vol. 1(3), pp 461-472, 2009.