EFFECT OF INLET AND OUTLET POSITION IN A VENTILATED CAVITY WITH AN INSULATED SQUARE BLOCK

AHMED KHAFEEF OBAID ALBDOOR

Abstract— A numerical analysis is carried out to study the performance of mixed convection in a rectangular enclosure. Fifteen different placement configurations of the inlet and outlet openings were considered. The bottom wall at constant temperature and the fluid considered is air. The numerical scheme is based on the finite volume method adapted to square uniform mesh elements by a nonlinear parametric solution algorithm. Results are obtained for a range of Richardson number from 1 to 10 obtained by varying the Reynolds number only .at Pr = 0.71 with constant physical properties. At the outlet of the computational domain a convective boundary condition (CBC) is used. The results indicate that the average Nusselt number Nu and the dimensionless bulk average temperature (θav) of the fluid on the heated wall strongly depend on the positioning of the inlet and outlet. The basic nature of the resulting interaction between the forced external air stream and the buoyancy-driven flow by the heated wall is explained by the heat transfer coefficient and the patterns of the streamlines and isotherms.

Index Terms— Mixed convection , Ventilation , Vortex , Richardson number

1 INTRODUCTION

Buoyancy-driven flow and heat transfer in open-ended enclosures is receiving increasing attention by many researchers in recent years. Natural convection in open-ended cavities has been the subject of interest both experimentally and numerically. This is primarily because several applications of practical interest, such as nuclear reactors, fire research, thermal insulation, thermal storage systems, electric transmission cables, and brake housing of an aircraft, can be modeled by various extensions of this type of geometry [1].

Many numerical and experimental studies have been performed on the mixed convection in the cavity. A numerical investigation to the cooling performance of electronic devices with an emphasis on the effects of the arrangement and number of electronic components by Ghasemi and Aminossadati [2] . The analysis uses a two dimensional rectangular enclosure under combined natural and forced convection flow conditions and considers a range of Rayleigh numbers. The results show that increasing the Rayleigh number significantly improves the enclosure heat transfer process.A numerical analysis by Saha et al. [3] is carried out to 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 . 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.

• *AHMED KHAFEEF OBAID ALBDOOR , Southern Technical University , Al- Nassiriah Technical Institute, Department of Mechanical Techniques ,Thi Qar , Iraq , Ahmedalbadry82@yahoo.*

The development of magnetic field effect on mixed convective flow in a horizontal channel with a bottom heated open enclosure has been numerically studied by Rahman et al.[4]. The governing two-dimensional flow equations havebeen solved by using Galarkin weighted residual finite element technique. The results indicate that the mentioned parameters strongly affect the flow phenomenon and temperature field inside the cavity whereas in the channel these effects are less significant.Shrama [5] studied the 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. The results show that the aspect ratio and Ri are effect 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.

A numerical study has been performed on mixed convection in a vented enclosure by finite element method by Rahman et al. [6] . 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.From the analysis it is found that with the increase of Reynolds and Richardson numbers the convective heat transfer becomes predominant over the conduction heat transfer and the rate of heat transfer from the heated wall significantly depends on the position of the inlet port.Ahammad et al. [7].studied the performance of different inlet and outlet locations on the flow and heat transfer for the MHD mixed convection problem in a ventilated cavity containing a heat generating square block. A Galerkin weighted residual based finite element method is applied to obtain the numerical solutions of the governing equations. The result is observed that both flow and thermal fields are strongly influenced by the position of inlet and outlet openings of the cavity. A numerical investigation has been carried out by RahmanandAlim [8] for mixed convection flow in a rectangular ventilated cavity with a heat conducting solid circular cylinder at the center. Finite element method was used to carry out the investigation.. The computational results indicated that the flow and thermal fields as well as average Nusselt number at the hot wall, average temperature of the fluid and the temperature at the cylinder center strongly depended on the cavity aspect ratio and mixed convection parameter Ri .A square cavity with two ventilation ports in the presence of an adiabatic fin of different lengths placed on the walls of the cavity is numerically analyzed for the mixed convection case for a range of Richardson numbers was studied by Selimefendigiland Hakan[9] . The results are observed that for the convection dominated case, fin length and its position on the one of the four walls of the cavity do not alter the thermal performance whereas when the buoyancy effects become important thermal performance increases for high fin length.A numerical investigation by Ahammad et al. [10] has been carried out for an MHD mixed convection problem to realize the influence of solid fluid thermal conductivity ratio as well as diameter of the centered obstacle on the flow and thermal fields in a ventilated cavity.AGalerkin weighted residual finite element technique is adopted with the help of Newton–Raphson iterative algorithm. The result of analysis are displayed by the average Nusselt number on the heated surface and average fluid temperature in the cavity.

In the present paper a numerical investigation is accomplished on the mixed convection of air in the square cavitycontaining an insulated square solid block. The main objective of this work is to examine the effect of inlet and exit port locations and the Diameter of the Solid Block as well as the effect of Richardson numbers on the heat transfer characteristics..

2 Numerical Model

2.1 Physical Model

For papers accepted for publication, it is essential that the The geometry of the present study is illustrated in Fig. 1. in which a Cartesian coordinate system is used with origin at the lower left corner of the working domain. It consists of a square enclosure of length L having a centered insulation square solid block. The bottom wall of the cavity is subjected to hot with temperature T_h while the other sidewalls are kept insulated. The solid body with a size of diameter d (d = 0.2L).The opening is placed on the left side with three locations, one of them at the bottom corner of the cavity and the other at the top corner and the last in the middle of the cavity while the others opening located on the right side with three locations have same position of the opining in the left side as shown in the schematic and the size of each opening is w = 0.1L. The incoming flow through the inlet is assumed at a uniform velocity u_i, ambient temperature T_i and the outgoing flow by the exit port is assumed to have zero diffusion flux for all variables and all solid boundaries are supposed to be rigid no-slip walls. In each case the inlet flow is come from one of the ports on the left side and the outflow is leave through one of the other ports which located in the left or right sides while the other fourth ports is closed.

2.2 Governing Equations

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 to be constant except the density variations causing a body force term in the momentum equation. The Boussinesq approximation is invoked for the fluid properties to relative density changes to temperature changes, and to couple in this way the temperature field to the flow field[11]. The governing equations for steady mixed convection flow can be expressed in the dimensionless [12].

$$\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}=0 \qquad (1)$$

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

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

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

The dimensionless variables are defined as:

$$X=\frac{x}{L}\ ,\quad Y=\frac{y}{L}\ ,\quad U=\frac{u}{u_i}\ ,\quad V=\frac{v}{u_i}$$

$$P=\frac{p}{\rho u_i^2}\ ,\quad \theta=\frac{(T-T_i)}{(T_h-T_i)}$$

Where X and Y are dimensionless coordinates varying along horizontal and vertical directions respectively, U and V are dimensionless velocity components in the X and Y directions respectively, θ is the dimensionless temperature and P is the dimensionless pressure. The non-dimensional numbers seen in the above, Re, Riand Pr are the Reynolds number, Richardson number and Prandtl number respectively, and are defined as :

$$Re=\frac{u_i L}{\gamma} \qquad Ri=\frac{g\beta(T_h-T_i)L}{u_i^2}\ ,\quad Pr=\frac{\gamma}{\alpha}$$

2.3 Boundary Conditions

The boundary conditions for this analysis are:

- At the inlet: U=1; V=0; θ=0
- At the outlet: P=0
- At the heated walls: U=0; V=0; θ=1
- At the rest of the adiabatic walls:U=V=0

$$\frac{\partial\theta}{\partial x}=0\ ,\quad \frac{\partial\theta}{\partial y}=0$$

Heat Transfer Calculations

The local Nusselt number is evaluated [11]as:

$$Nu_L=\frac{\partial\theta}{\partial n}$$

wheren denotes the normal direction on a plane.

The average Nusselt number

$$Nu=\frac{1}{s}\int_0^s Nu_L\,ds$$

where (s) is the length of the heated walls

The bulk average temperature, defined [11] as;

$$\theta_{av}=\int \theta\ dV/V$$

whereV is the cavities volume.

ISSN 2229-5518

2.4 Numerical method

The governing equations were solved using FLUENT-CFD code; version 6.3 is employed for all numerical simulations . Gambit, Version 2.2.30, is used for the development of the computational grid. Fig.2 shows the computational grid. The continuity is satisfied using a semi-implicit method for pressure linked equations, which is referred to as the SIMPLE procedure. To reduce numerical errors, second order upwind discrimination schemes are used in the calculations. Each computational iteration is solved implicitly. The convergence of the computational solution is determined on scaled residuals for the continuity, energy equations and for many of the predicted variables [13].

3 RESULTS AND DISSCUSSION

Two-dimensional mixed convection is studied for a laminar flow in an air-cooled cavity with a Prandtl number of 0.71. The controlling parameter, for the fifteen configurations of the geometry as defined in the table 1 , is the Richardson number, Ri. The Grashof number ,Gr, is kept fixed . The range of Richardson number used for the simulations is $1 \le Ri \le 10$ and it is obtained by varying the Reynolds number only. The results of this present study are displayed in terms of streamlines and isotherms. Moreover, the effects of heat transfer of the enclosure are shown in terms of average Nusselt number Nu, and the dimensionless average temperature θav the Table 2 shows The Nomenclature.

3.1 Flow and Temperature Fields

The combined forced and buoyancy driven flow and temperature fields inside a cavity, with a centered insulation square solid block, and heated bottom wall , for different inlet and outlet configurations are illustrated by means of Streamlines and isotherms in Fig. 3-8, for Richardson numbers of 1, 3, 6 and 10.

First Group

Figures (3A-3E) shows the dynamics for the first group of configurations . The streamlines describe the interaction of forced and natural convection under various convection regimes . In the figure 3A At Ri = 1, for the higher Re values, forced convection dominates the major flow from the inlet to the exit without much penetrating into the cavity. With decreasing the velocity of the cold forced flow , it will moves towards the lower horizontal heated wall. The figures 3B and 3C shows that tow cells is formed , one on the right top corner and the other on the left bottom corner . When the Richardson number is increase , buoyancy effects become important,

and the cell grows in size and shape. It can be seen from figure 3D that vortexes is formed and moved towards the center of cavity with increase value of Richardson number . figure 3E is illustrate that the flow is separate on the top and bottom right corners clearly. While increasing the magnitude of velocity the vortexes on the right part of the cavity are move to occupy half of the cavity.

The corresponding isotherm plots for the above cases are presented in figure 4A to figure 4E . As shown, the thermal boundary layer decreases in thickness slowly as the Ri decreases (increase in Re) , This is revealed by the denser concentration of isotherms near the bottom hot wall. because the effect of gravitation force becomes negligible at the large value of Re. In regions where flow separation begins, the isotherms diverge reflecting a decrease in heat transfer associated with the onset of separation.

Second Group

Figures (5A-5E) shows the streamlines for the second group of configurations . It can be seen that the flow will separate into two parts because the square solid block is faced the flow . Because the impact the flow is loose a lot from it is inertia force , the lower part of flow is continued by the gravity effect. This lead to formed three cells , tow in the right side above and down the inlet port . Another rotating cell is noted at right side of the cavity behind the square solid block , whose grows in shape and size with increasing in Ri .

A close inspection on the isotherms for different cases in figures (6A-6E) reveal that maximum temperature gradients appear on the bottom near the hot wall . In figures (6A-6E) it can be seen that the isotherms on the top part of the cavity wall become less clustered and parallel and this shows that heat transfer is not effective in that region of the cavity.

Third Group

Figures (7A-7E) presents the streamlines for the third group of configurations . it is found that the fluid flow is characterized by the open lines about the whole domain and a vortex appears at the left side above the inlet port . As shown in Figures 7A,7B the flow separate on the right bottom corner due to inertia force . Figure 7C shows that A secondary vortex is also formed on the top-right corner of the cavity . The reason is due to the effect of buoyancy driven flow and convective currents (the fresh and colder fluids entering the cavity cannot come into intimate mixing with the hotter fluids). Similarly the isotherms inside the cavity are shown in the Figures (8A-8E). It is clear from these figures that the isotherms are almost same near the heated wall for the five different outlet port locations. Although a slightly thin thermal boundary is observed at figure 8C , the corresponding isotherm plots do not significantly propagates from the heated surface.

3.2 Heat Transfer Characteristics

The effect of Reynolds number on the average Nusselt number (Nu) at the hot wall and the bulk average temperature (θav) of the fluid are displayed as a function of Richardson number as shown in figure 9 and figure 10 respectively. It is apparent from Figure 9 that the highest value of Nu is found for DBB configuration (Case No. 13). Also Nu decreases generally with increasing Ri due to the reducing in the growth of the thermal boundary layer near the heated surface and the dominate of forced convection . In the other hand, the bulk average temperature of the fluid is increased with the increasing the Ri. This behavior is because the increasing of the effect of the buoyancy with the increasing of Ri . Also the bulk average fluid temperature (θav) in the cavity is lowest for DBB configuration (Case No. 13) for all values of Ri which can be observed from Figure 10 .

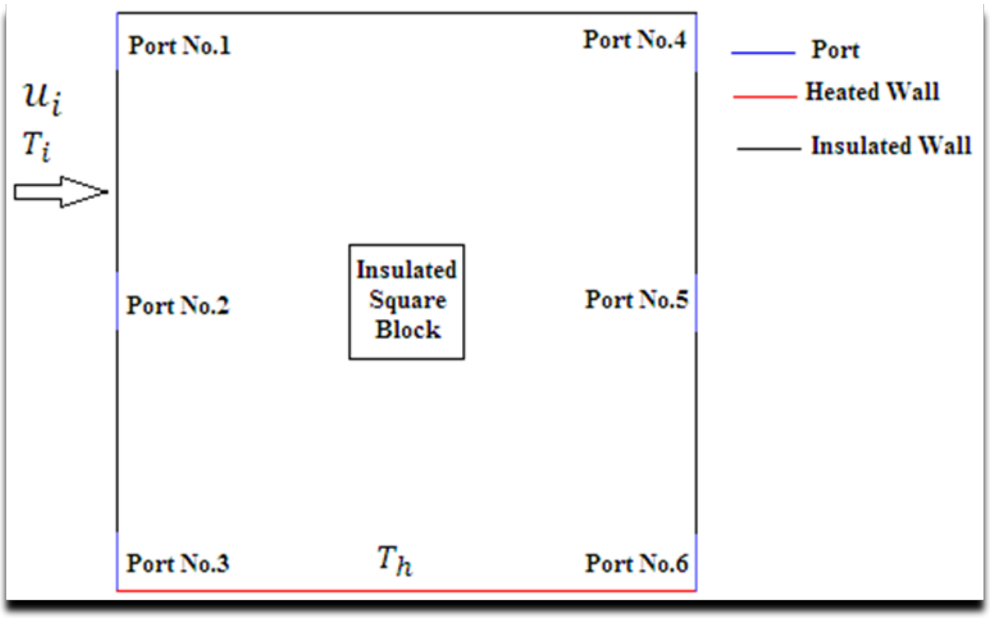


Fig. 1 The geometry of the study Cases .

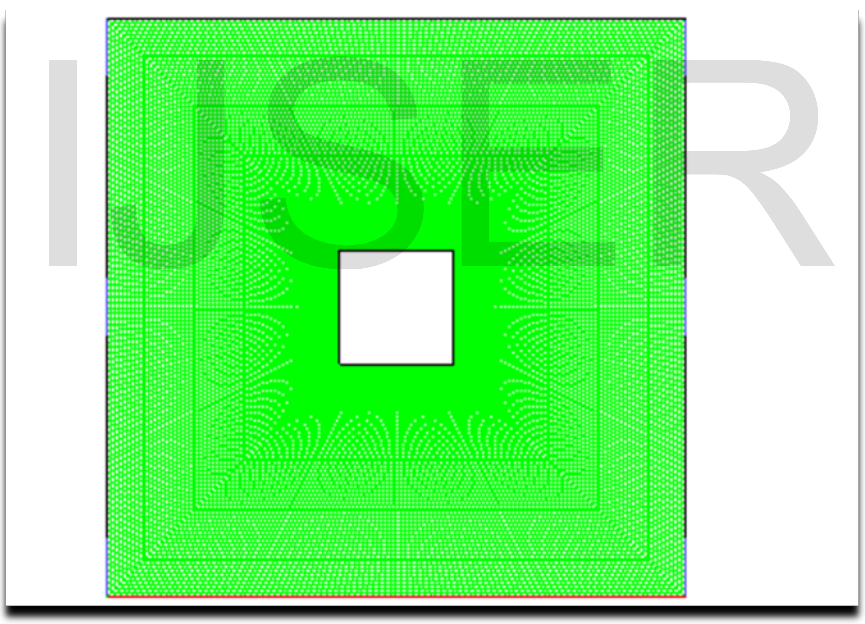


Fig . 2 The computational grid

ISSN 2229-5518

Table 1 The configurations of the geometry for Study Cases:

Group No.	Case No.	No. of Inlet Port	No. of Outlet Port	Case Symbol
First Group	1	1 (Top Left Corner)	4 (Top Right Corner)	Direct Top to Top (DTT)
	2	1 (Top Left Corner)	5 (Center of Right Side)	Direct Top to Center (DTC)
	3	1 (Top Left Corner)	6 (Bottom Right Side)	Direct Top to Bottom (DTB)
	4	1 (Top Left Corner)	2 (Center of Left Side)	Inverse Top to Center (ITC)
	5	1 (Top Left Corner)	3 (Bottom Left Corner)	Inverse Top to Bottom (ITB)
Second Group	6	2 (Center of Left Side)	4 (Top Right Corner)	Direct Center to Top (DCT)
	7	2 (Center of Left Side)	5 (Center of Right Side)	Direct Center to Center (DCC)
	8	2 (Center of Left Side)	6 (Bottom Right Side)	Direct Center to Bottom (DCB)
	9	2 (Center of Left Side)	1 (Top Left Corner)	Inverse Center to Top (ICT)
	10	2 (Center of Left Side)	3 (Bottom Left Corner)	Inverse Center to Bottom (ICB)
Third Group	11	3 (Bottom Left Corner)	4 (Top Right Corner)	Direct Bottom to Top (DBT)
	12	3 (Bottom Left Corner)	5 (Center of Right Side)	Direct Bottom to Center (DBC)
	13	3 (Bottom Left Corner)	6 (Bottom Right Side)	Direct Bottom to Bottom (DBB)
	14	3 (Bottom Left Corner)	1 (Top Left Corner)	Inverse Bottom to TOP (IBT)
	15	3 (Bottom Left Corner)	2 (Center of Left Side)	Inverse Bottom to Center (IBC)

Table 2 The Nomenclature :

d : The solid body diameter (m)	Greek symbols
g : gravitational acceleration (ms-2)	α:thermal diffusivity, k/ρCp (m2s-1)
Gr : Grashof number	β: thermal expansion coefficient (K-1)
L : length of the cavity	λ: penalty parameter
n : normal direction on a plane	θ: non-dimensional temperature
Nu : average Nusselt number	θav : average non-dimensional temperature
NuL : local Nusselt number	v : kinematic viscosity of the fluid (m2s-1)
P : pressure (Nm-2)	ρ: density of the fluid (kgm-3)
P : non-dimensional pressure	
Pr : Prandtl number	Subscripts
Re : Reynolds number	i Inlet
Ri : Richardson number	av Average
s : length of the heated walls	
T : temperature (K)	
Th : hot temperature (K)	
Ti : inlet temperature (K)	
ui : inlet velocity (ms-1)	
u, v : velocity components (ms-1)	
U, V : non-dimensional velocity components	
W : height of the inflow and outflow openings	
x, y : Cartesian coordinates (m)	
X, Y : non-dimensional Cartesian coordinates	

ISSN 2229-5518

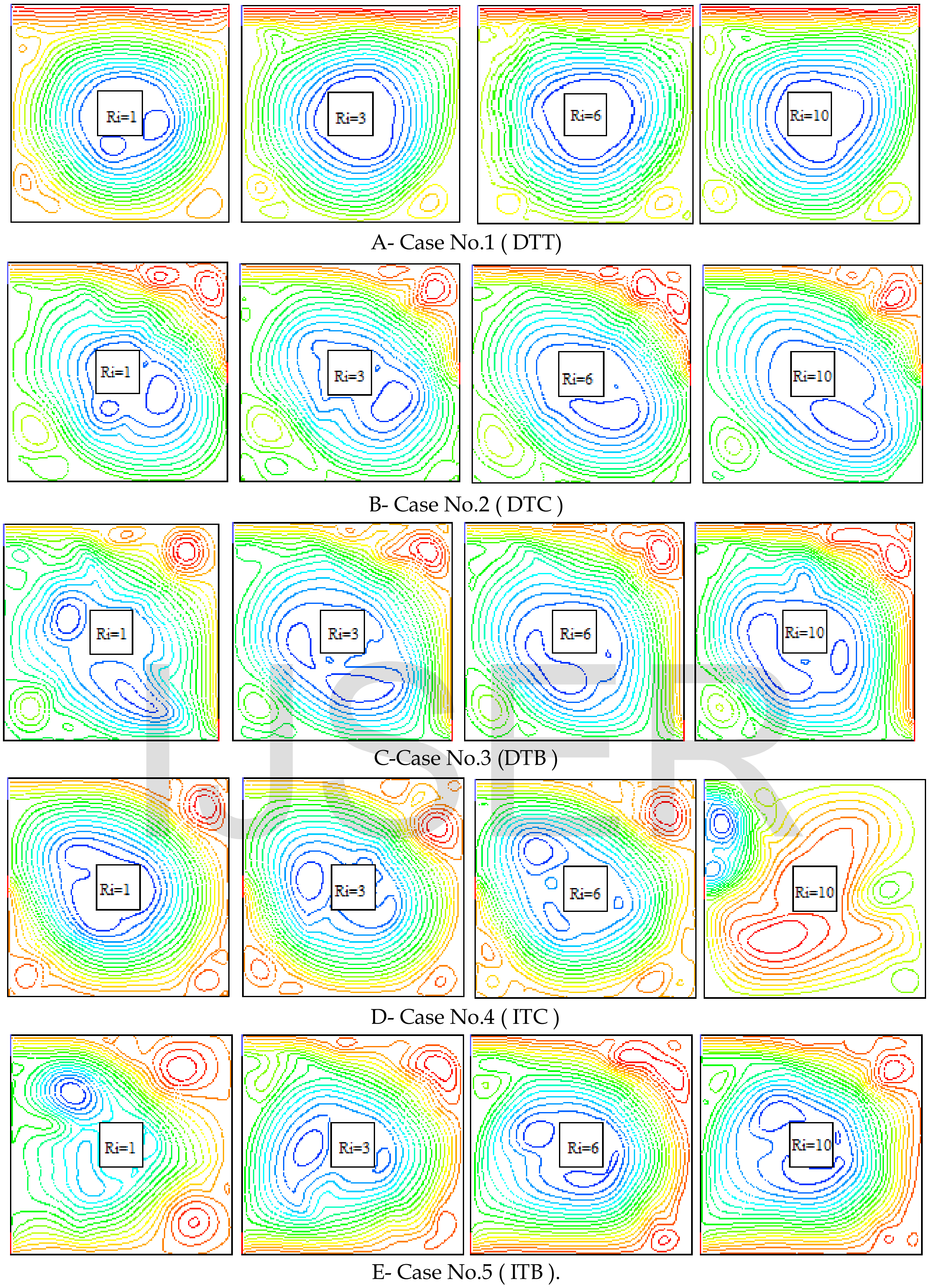
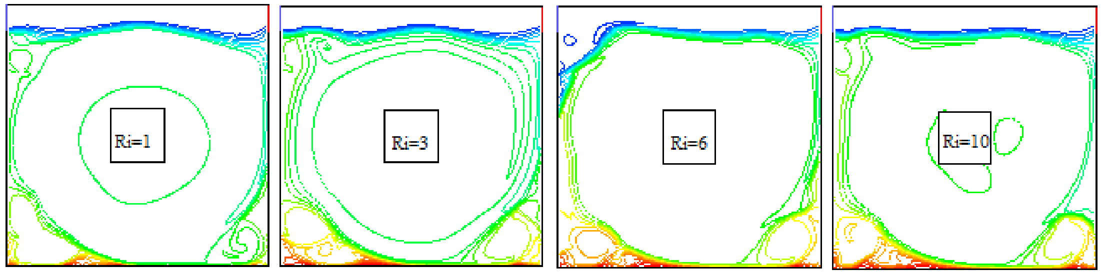
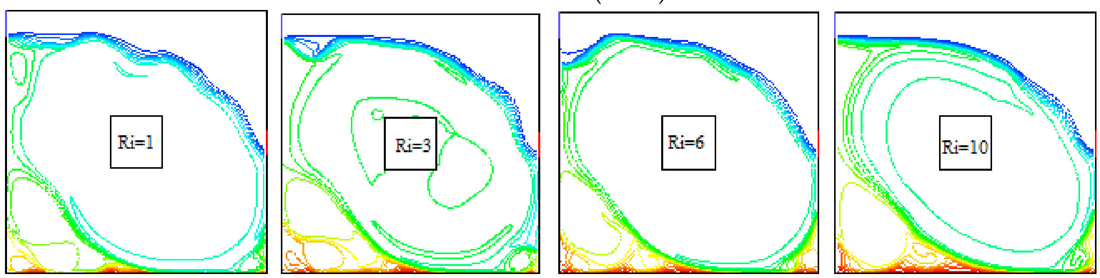


Fig . 3 Streamlines for the First Group of Cases study to the Range of Richardson number.

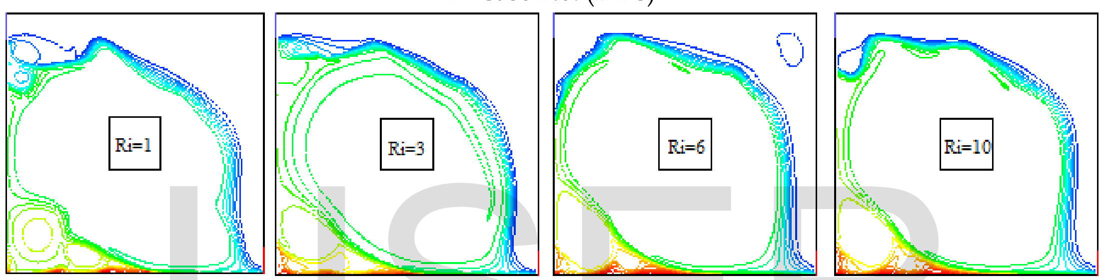
ISSN 2229-5518



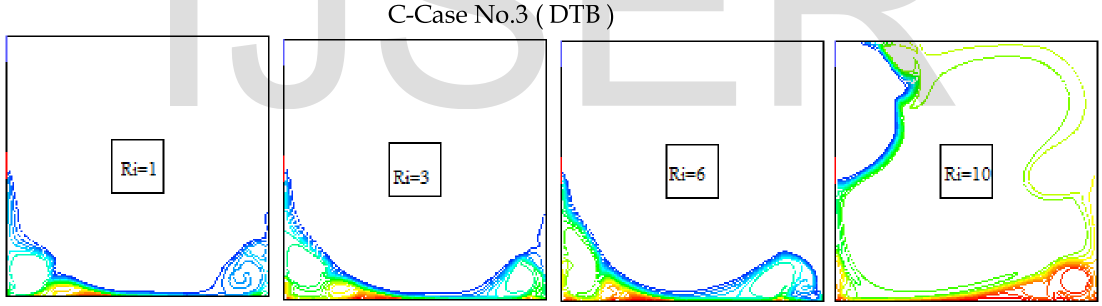
A- Case No.1 (DTT)



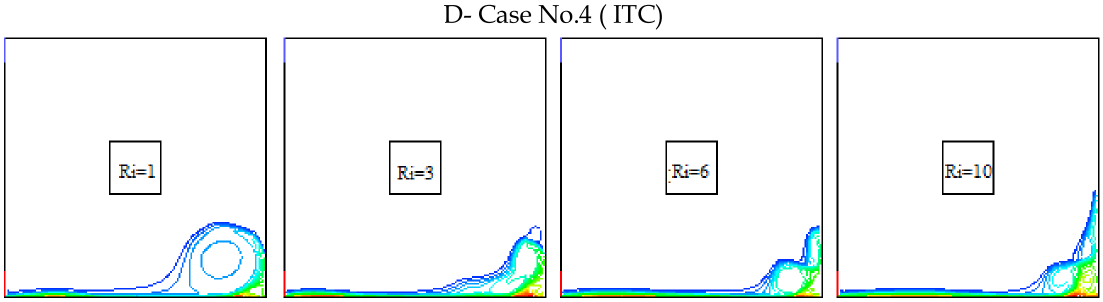
B- Case No. (DTC)



C-Case No.3 (DTB)



D- Case No.4 (ITC)



E- Case No.5 (ITB).

Fig . 4 Isotherms for the First Group of Cases study to the Range of Richardson number.

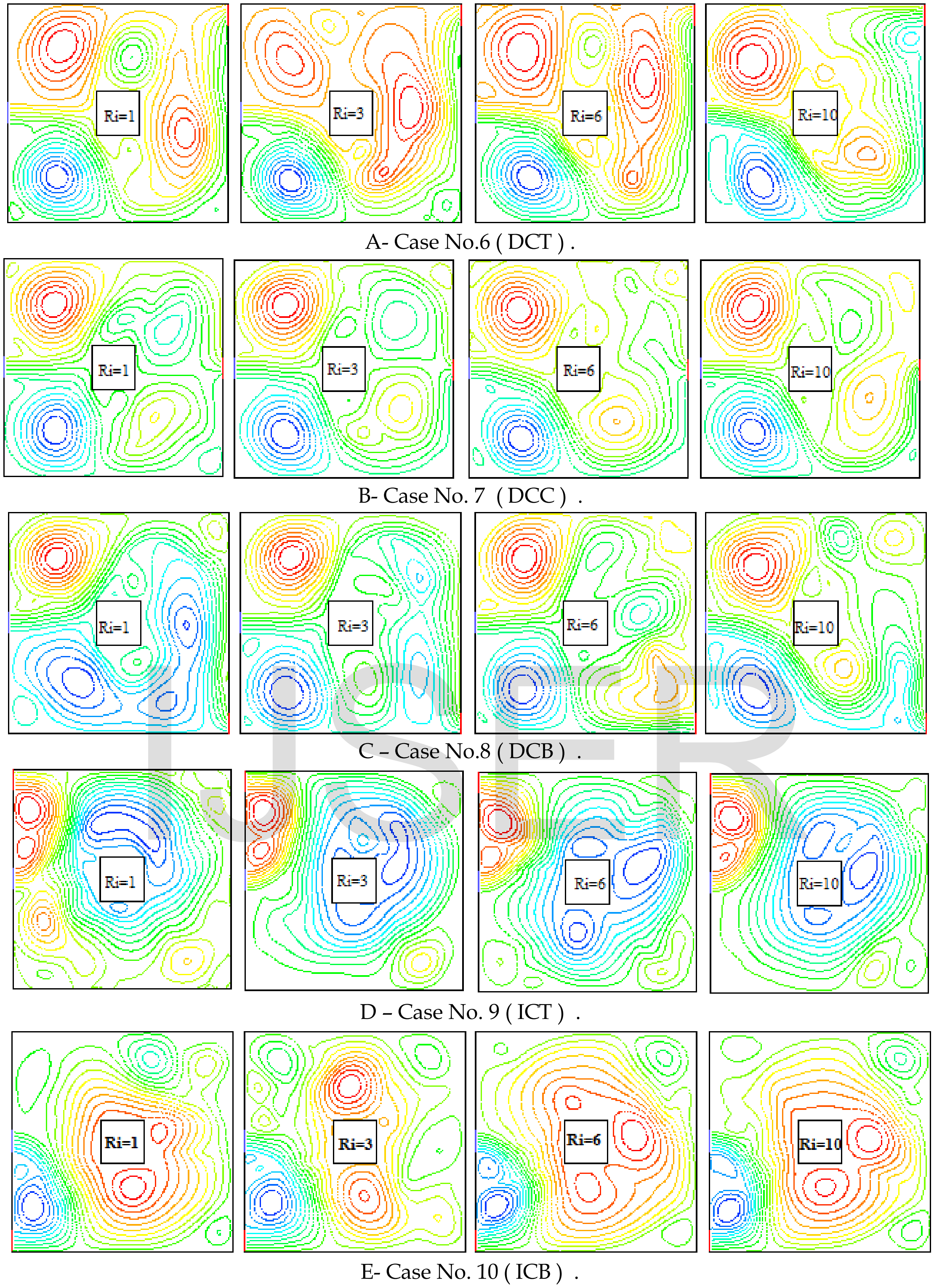
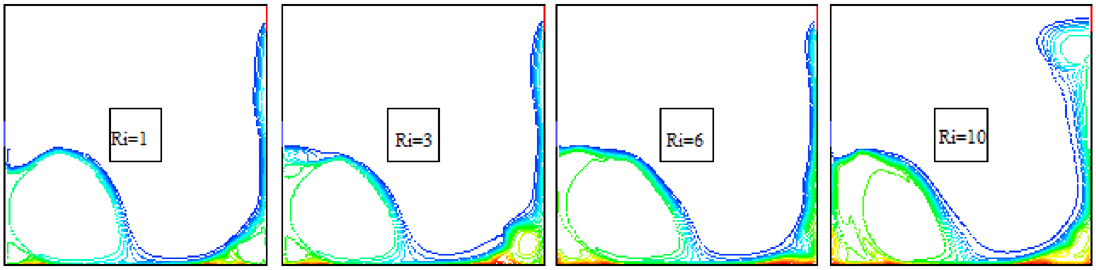
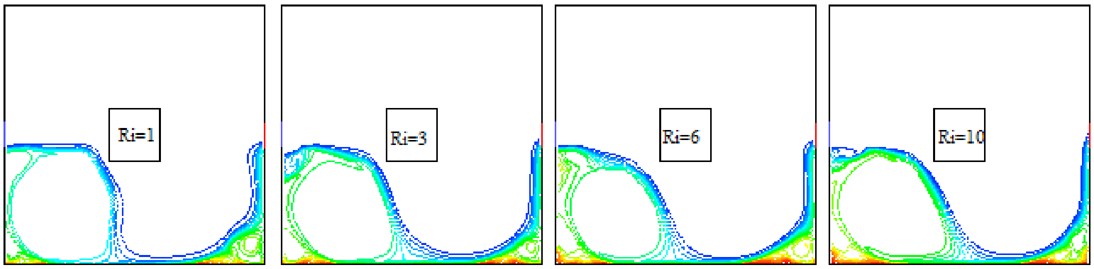


Fig . 5 Streamlines for the Second Group of Cases study to the Range of Richardson number.

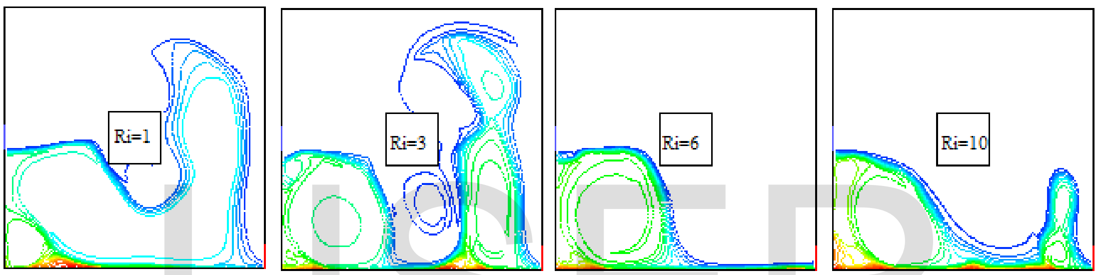
ISSN 2229-5518



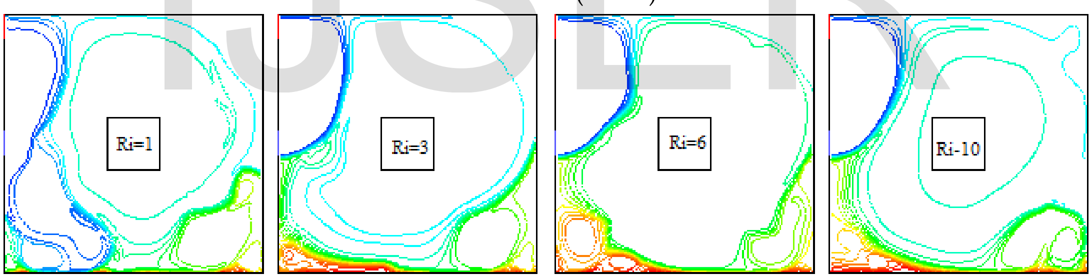
A- Case No.6 (DCT) .



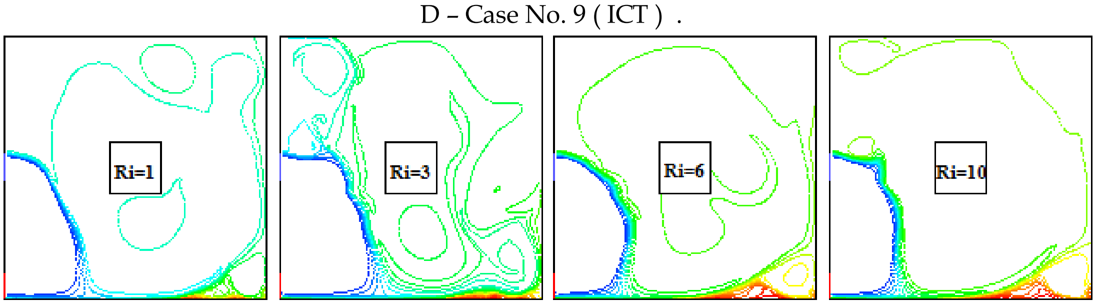
B- Case No. 7 (DCC) .



C – Case No.8 (DCB) .



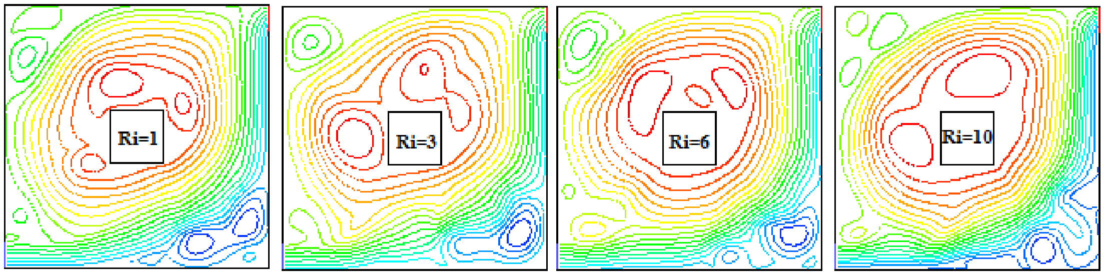
D – Case No. 9 (ICT) .



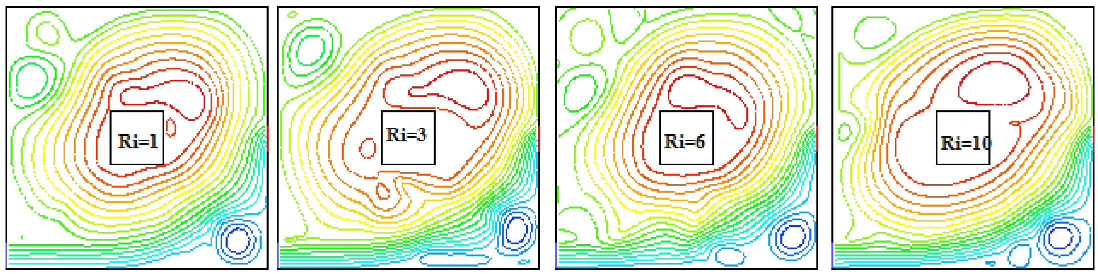
E- Case No. 10 (ICB) .

Fig . 6 Isotherms for the Second Group of Cases study to the Range of Richardson number.

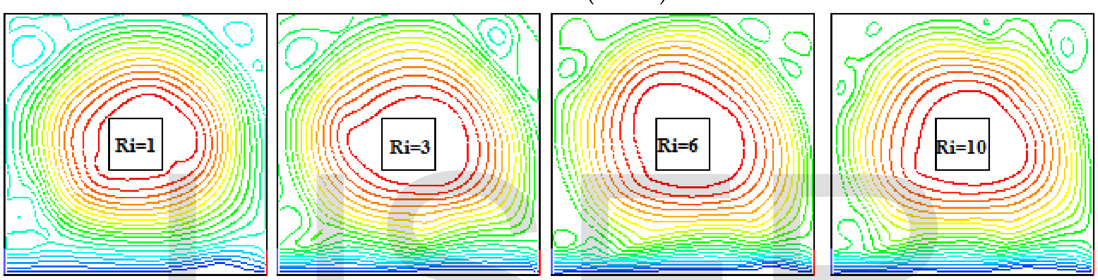
ISSN 2229-5518



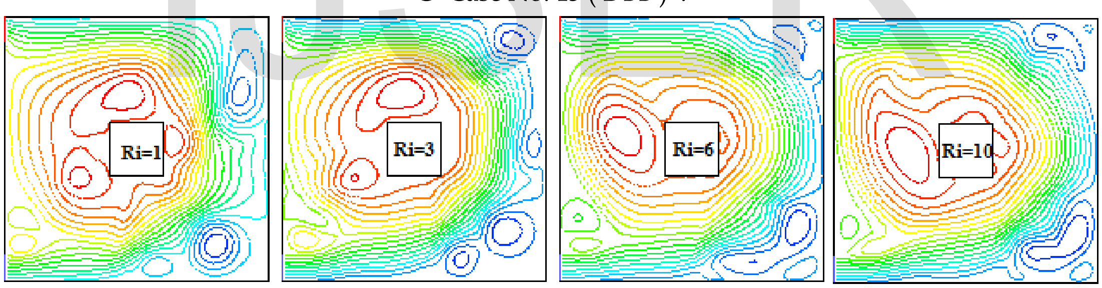
A – Case No.11 (DBT) .



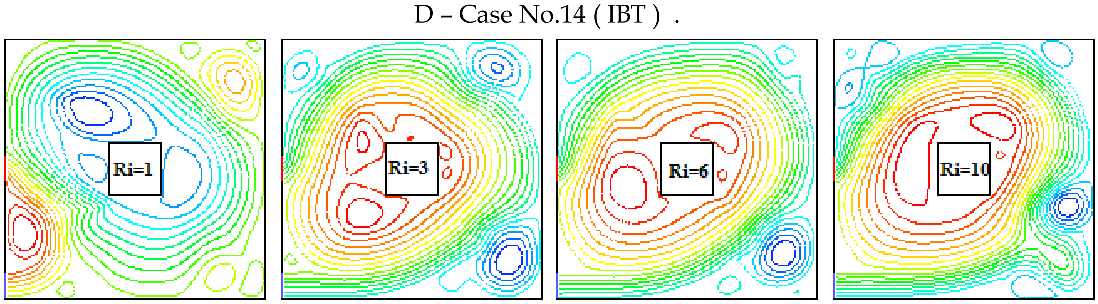
B- Case No. 12 (DBC) .



C- Case No. 13 (DBB) .



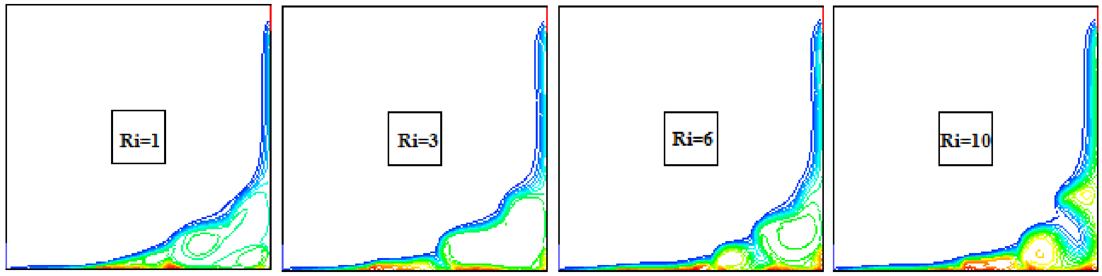
D – Case No.14 (IBT) .



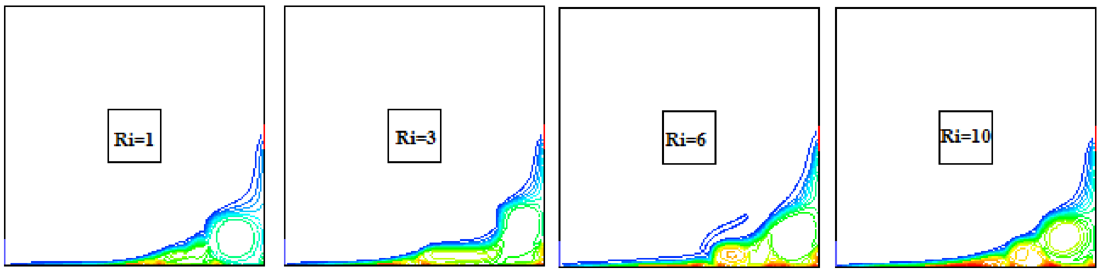
E – Case No.15 (IBC) .

Fig . 7 Streamlines for the Third Group of Cases study to the Range of Richardson number.

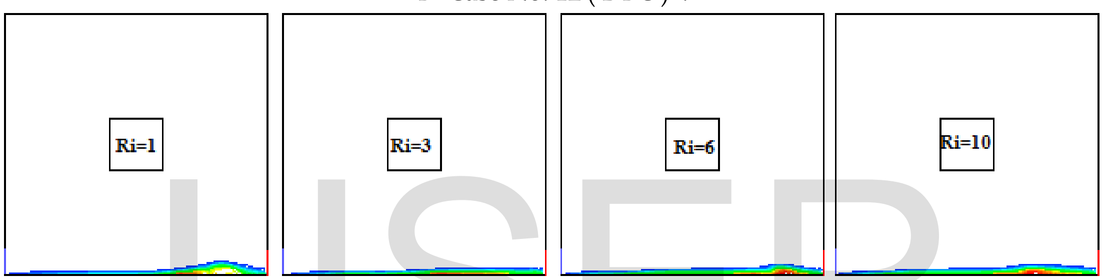
ISSN 2229-5518



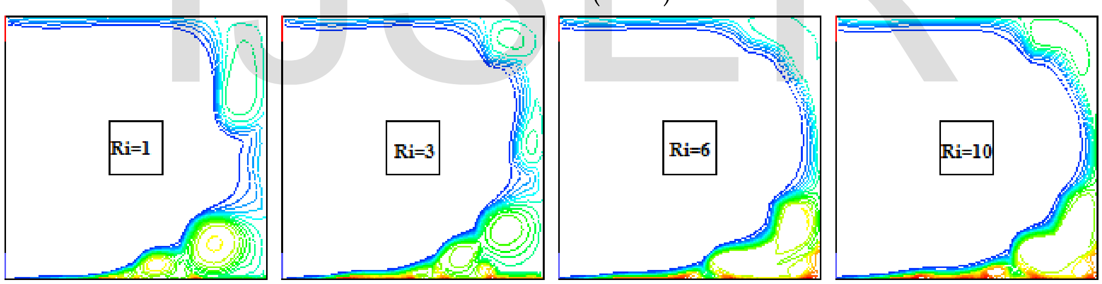
A – Case No.11 (DBT) .



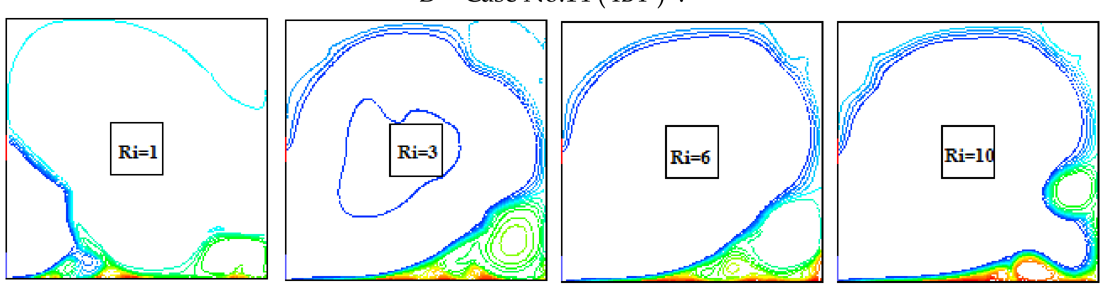
B- Case No. 12 (DBC) .



C- Case No. 13 (DBB) .



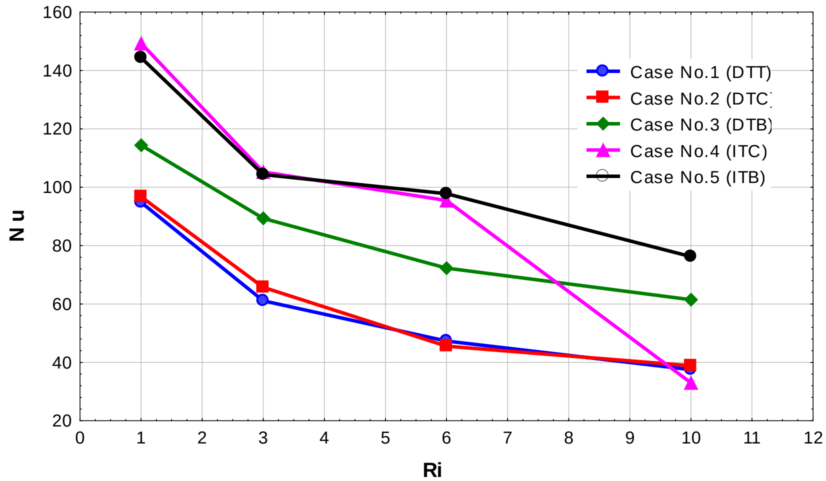
D – Case No.14 (IBT) .



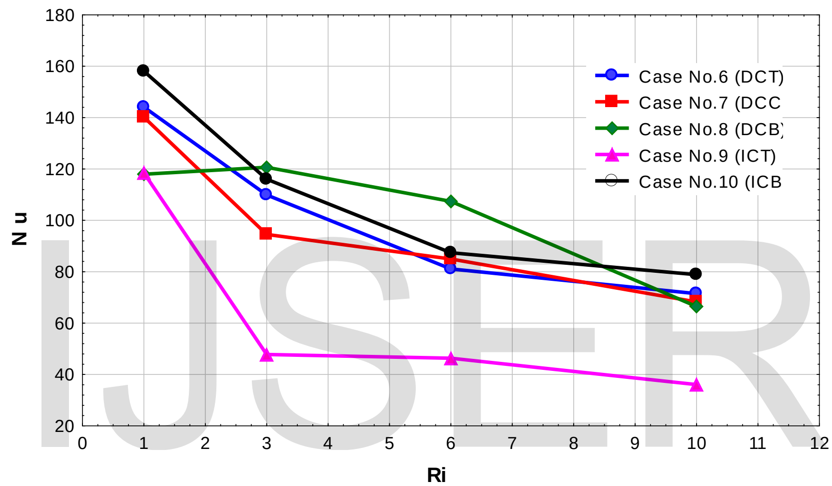
E – Case No.15 (IBC) .

Fig . 8 Isotherms for the Third Group of Cases study to the Range of Richardson number.

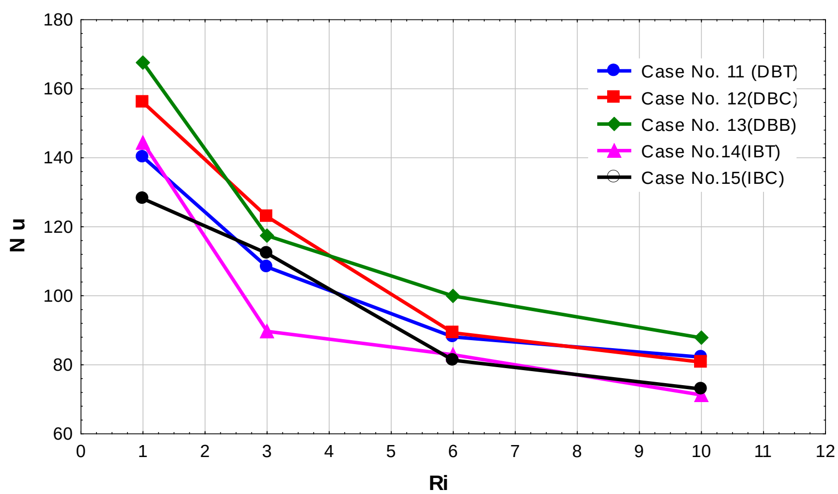
ISSN 2229-5518



A- First Group



B- Second Group



C- Third Group

Figure 9 Variation of the average Nusselt number with Richardson number .

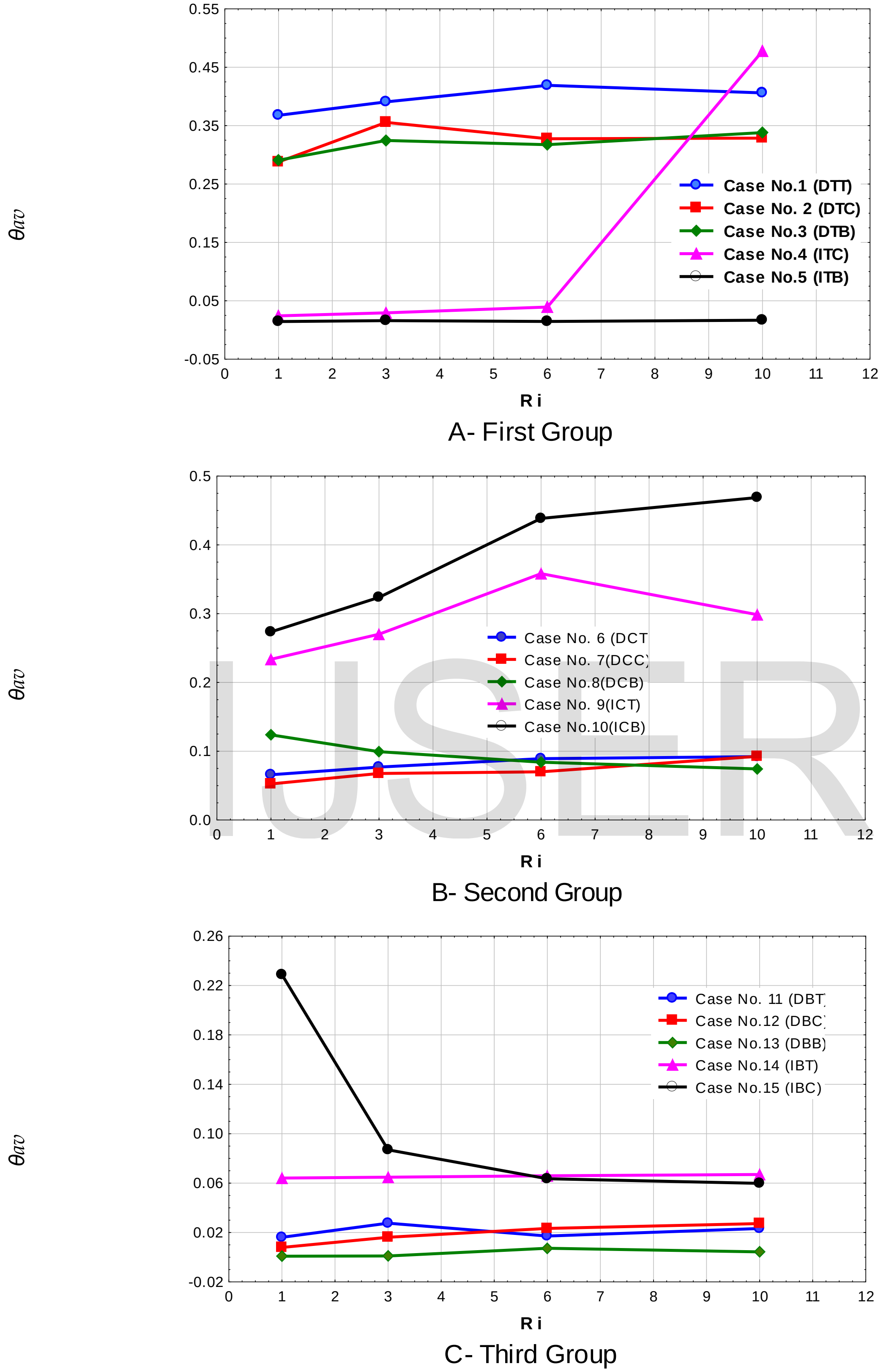


Figure 10 Variation of the bulk average temperature with Richardson number .

5 CONCLUSIONS

A numerical investigation of laminar mixed-convective cooling of a rectangular cavity with a centered insulation square solid block has been conducted to identify the optimum placement of inlet and exit for the best cooling effectiveness .The following major conclusions may be drawn from the present investigations:
the flow and thermal fields have strong dependence on the position of inlet and outlet openings.
The results of the study show that the flow structure and temperature distribution are considerably influenced by the interaction between natural convection and forced convection in the cavity.

REFERENCES

[1] *Kh. Khanafer , K. Vafai , Marilyn Lightstone " Mixed convection heat transfer in two-dimensional open-ended enclosures" International Journal of Heat and Mass Transfer ,Vol. 45 , P. 5171–5190 ,2002 .*

[2] *B. Ghasemi and S. M. Aminossadati "NUMERICAL SIMULATION OF MIXED CONVECTION IN A RECTANGULAR ENCLOSURE WITH DIFFERENT NUMBERS AND ARRANGEMENTS OF DISCRETE HEAT SOURCES " The Arabian Journal for Science and Engineering, Volume 33, Number 1B , P. 189-207 , 2007.*

[3] *S. Saha, Md. A.H. Mamun , M. Z. Hossain and A.K.M. S. 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.*

[4] *M.M. Rahman , S. Parvin , R. Saidur , N.A. Rahim " Magnetohydrodynamic mixed convection in a horizontal channel with an open cavity" International Communications in Heat and Mass Transfer ,Vol.38 ,P. 184–193, 2011.*

[5] *S. M. Shrama "NUMERICAL ANALYSIS OF MIXED CONVECTION HEAT TRANSFER FOR LAMINAR FLOW IN A CHANNEL WITH AN OPEN CAVITY" Al-Qadisiya Journal For Engineering Sciences, Vol. 4 , No. 4 , P. 420-438 , 2011 .*

[6] *M. M. Rahman , M. A. Alim , M. A. H. Mamun , M. K. Chowdhury and 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,P. 26-36 , APRIL 2007.*

[7] *M.U. Ahammad, M.M. Rahman and M.L. Rahman " EFFECT OF INLET AND OUTLET POSITION IN A VENTILATED CAVITY WITH A HEAT GENERATING SQUARE BLOCK " Engineering e-Transaction (ISSN 1823-6379) ,Vol. 7, No.2 , p.107-115, December 2012.*

[8] *Md. Mustafizur Rahman , Md. Elias and Md.A. Alim " MIXED CONVECTION FLOW IN A RECTANGULAR VENTILATED CAVITY WITH A HEAT CONDUCTING SOLID CIRCULAR CYLINDER AT THE CENTER" IJE Transactions B: Applications , Vol. 24, No. 1, P. 93-105, February 2011.*

[9] *F. SELIMEFENDIGIL and H. F. OZTOP "EFFECTS OF AN ADIABATIC FIN ON THE MIXED CONVECTION HEAT TRANSFER IN A SQUARE CAVITY WITH TWO VENTILATION PORTS " Preprint submitted to Thermal Science , P. 1-16 , April 4, 2013.*

[10] *M. U. Ahammad , M. M. Rahman , and M. L. Rahman "Mixed Convection Flow and Heat Transfer Behavior inside a Vented Enclosure in the Presence of Heat Generating Obstacle " International Journal of Innovation and Applied Studies, Vol. 3 , No. 4 , pp. 967-978, Aug. 2013.*

[11] *Z. K. Radhi " NUMERICAL STUDY OF MIXED CONVECTION HEAT TRANSFER THROUGH DOUBLE SQUARE CAVITY CONNECTED WITH EACH OTHER " Al-Qadisiya Journal For Engineering Sciences Vol. 4 No. 4 , P. 456-474 , 2011 .*

[12] *J.M. Jalil , K.A. Esmael and S. J. Habeeb " MIXED CONVECTION EROM ELECTRONIC EQUIPMENT COMPONDNI AT DIFFERENT POSTTION AN ENCLOUSERBY PRIMTTIVE VARABILS METHOD" Journal of Engineering , Vol.12, No. 1 , P. 37-52 , March 2006.*

[13] *W. M. Abed , A. J. Shareef and A. A. Najeeb " Natural Convection Heat Transfer in Horizontal Concentric Annulus between Outer Cylinder and Inner Flat Tube" Anbar Journal for Engineering Sciences, Vol.3, No.2 , P. 31- 45, AJES-2010 .*