CFD Letters www.cfdl.issres.net Vol. 5 (4) – December 2013

Numerical Study of Mixed Convective Cooling in a Ventilated Cavity Utilizing a Guide Baffle

Wael M. El-Maghlany[1*] Mohamed A. Teamah[2], Ahmed M. Hanafy[3], Akram Zeid[4]
[1]*Mechanical Engineering Dep. Faculty of Engineering*
Alexandria University, EGYPT
[2,3]*Mechanical Engineering Dep. College of Engineering and Technology*
Arab Academy for Science, Technology and Maritime Transport, EGYPT
[4]*College of Maritime Transport*
Arab Academy for Science, Technology and Maritime Transport, EGYPT

Received: 20/08/2013 – Revised 09/12/2013 – Accepted 10/12/2013

Abstract

Laminar mixed convective flow in ventilated square cavity with inlet and outlet ports has been executed. The natural convection effect is attained by heating from the constant flux heat source which is symmetrical located at the bottom wall and cooling from the injected flow. An external Newtonian fluid flow (Air, Pr =0.71) enters the cavity through an opening at the bottom of the left vertical wall and exits from another opening at the top of the right vertical wall, which is creating the forced convection flow conditions. Laminar regime is considered under steady state condition. The governing equations including continuity, momentum, and energy are transformed into non-dimensional form and the resulting partial differential equations are solved by the finite volume technique. In addition, horizontal guide baffle with different locations was attached from the left vertical wall to enhance the cooling effect of the source part. A parametric study was performed presenting the influence of Reynolds' number ($50 \le Re \le 500$), Richardson number ($0.01 \le Ri \le 10$), and hence the effect of heat flux represented in the Grashof number automatically generated from both values of Reynolds and Richardson numbers.

Key words: Mixed Convection; Ventilated Cavity; Numerical Study; Heat flux, Baffle

1. Introduction

Fluid flow and heat transfer in a ventilated cavity are present in many transport processes in nature and in the engineering devices. The most usage of the mixed convection flow can be found in atmospheric flows, solar energy storage, heat exchangers, lubrication technologies, drying technologies and cooling of the electronic devices. Due to practical importance, the mixed convection flow has been studied extensively for conventional fluids in literatures [1–8].

Also,the electronic devices are treated as the heat sources which are embedded on the surfaces [9]. Shahi et al. [10] executed a numerical investigation of mixed convection flows through a

[*] Corresponding Author: Wael M. El-Maghlany
Email: elmaghlany@yahoo.com Telephone: +201223741891 Fax: +2035921853
 PII: S2180-1363(13)54197-X

copper–water nanofluid in a square cavity with inlet and outlet ports. The natural convection effect is attained by heating from the constant flux heat source which is symmetrical located at the bottom wall and cooling from the injected flow. Results are presented in the form of streamlines, isotherms, average Nusselt number and average bulk temperature. In addition, the effects of solid volume fraction of nanofluids on the hydrodynamic and thermal characteristics have been investigated. The results indicate that increase in solid concentration leads to increase in the average Nusselt number at the heat source surface and decrease in the average bulk temperature. Paranthoën and Gonzalez [11] presented analysis, visualizations and differential pressure measurements of the flow within a confined ventilated space resulting from a buoyant turbulent 2D plume originating at the bottom of the enclosure with initial source volume flux and buoyancy flux. The enclosure is ventilated through lower and upper openings of heights and separated by a vertical distance. Mixed convection flow and temperature fields in a vented square cavity subjected to an external copper–water nanofluid are studied numerically by A. Mahmoudi et al [12], the natural convection effect is attained by heating from the constant flux heat source on the bottom wall and cooling from the injected flow, both the inlet and outlet ports are alternatively located either on the top or the bottom of the sides and external flow enters in to the cavity through an inlet opening in the left vertical wall and exits from another opening in the opposite wall. The remaining boundaries are considered adiabatic. The present study is focused on the laminar mixed convection flows in a two-dimensional square ventilated cavity with a heat source partially embedded on the bottom wall. A horizontal guide baffle with different locations was attached from the left vertical wall to enhance the cooling effect of the source part; indeed this geometry has the potential application in a cooling of electronic device. The consequence of varying the Reynolds number, Richardson number and the baffle location on the hydrodynamic and thermal characteristics have been investigated and discussed.

2. Mathematical Model

A two-dimensional square cavity considered for the present study is shown in Fig. 1. The cavity is filled with air. In order to induce the buoyancy effect, the bottom horizontal wall has an embedded constant heat flux source. The non-heated parts of the bottom wall and the remaining walls are insulated. Horizontal insulted guide baffle was attached from the left vertical wall at position of (W/L=0.25, 0.5, 0.75), respectively, while both the baffle and embedded constant heat flux source length ratios are (G/L=1/2 and D/L=1/3). The cavity is subjected to an external flow entering in to cavity from the bottom opening located in left vertical wall and leaving from the upper opening of the opposite vertical one. Except for the density the properties of the fluid are taken to be constant. It is further assumed that the Boussinesq approximation is valid for buoyancy force.

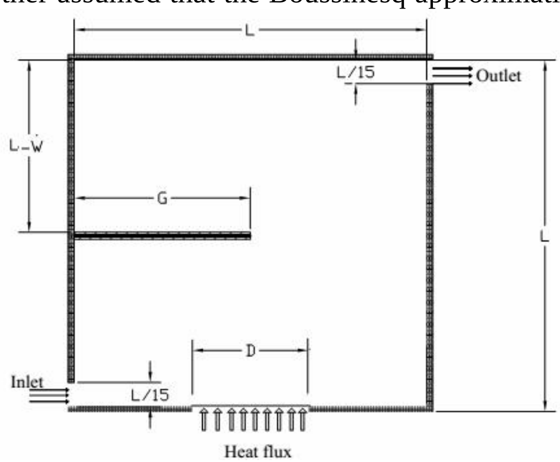

Figure 1. A schematic diagram for the problem with boundary conditions

With the above assumptions, the non-dimensional governing equations of continuity, momentum, and energy for are:

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

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

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

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

In the above equations, the following non-dimensional parameters are used.

$$\mathrm{X}=\frac{\mathrm{x}}{\mathrm{L}},\ \mathrm{Y}=\frac{\mathrm{y}}{\mathrm{L}},\ \mathrm{U}=\frac{\mathrm{u}}{\mathrm{U_o}},\ \mathrm{V}=\frac{\mathrm{v}}{\mathrm{U_o}},\ \mathrm{P}=\frac{\mathrm{p}}{\rho\,\mathrm{U_o}^2},\ \theta=\frac{\mathrm{T}-\mathrm{T_c}}{\mathrm{q\,L}}k,\ \mathrm{Re}=\frac{°\ \mathrm{U_o}\,L}{«}$$

$$Ri=\frac{Gr}{\mathrm{Re}^2},\ Gr=\frac{g\ \ L^4 q}{k\ ³\ ^2},\ \mathrm{Pr}=\frac{¬}{\sim} \tag{5}$$

The governing equations have elliptic nature therefore boundary conditions on the entire solution domain must be specified for all field variables. The non-dimensional boundary conditions are given by:

- *Left and right vertical walls*: $\begin{cases} U=V=0 \\ \dfrac{\partial\overline{}}{\partial X}=0 \end{cases}$

- *Inlet port*: $\begin{cases} U=1.0,\ V=0 \\ \overline{}=0 \end{cases}$

- *Outlet port*: $\begin{cases} \dfrac{\partial U}{\partial X}=0,\ V=0 \\ \dfrac{\partial\overline{}}{\partial X}=0 \end{cases}$

- *Bottom horizontal wall*: $\begin{cases} U=V=0 \\ \dfrac{\partial\overline{}}{\partial Y}=\begin{cases} 0 & at\ 0\le X\le\frac{1}{3},\ \frac{2}{3}\le X\le 1 \\ -1 & at\ \frac{1}{3}\le X\le\frac{2}{3} \end{cases} \end{cases}$

- *Top horizontal wall*: $\begin{cases} U=V=0 \\ \dfrac{\partial\overline{}}{\partial Y}=0 \end{cases}$

- *At baffle*: $U=V=0$

3. Nusselt Number

Equating the heat transfer by convection to the heat transfer by conduction at heat flux part led to local Nusselt number

$$Nu_L = \frac{1}{s} \tag{6}$$

The average Nusselt number is obtained by integrating the above local Nusselt number over heat flux part:

$$Nu = \int_{Heat\ Flux\ Part} Nu_L \, dX \tag{7}$$

4. Solution Procedure

The governing equations were solved using the finite volume technique developed by Patankar [13]. This technique was based on the discretization of the governing equations using the central difference in space. The discretization equations were solved by the Gauss–Seidel method. The iteration method used in this program is a line-by-line procedure, which is a combination of the direct method and the resulting Tri Diagonal Matrix Algorithm (TDMA). The convergence of the iteration is determined by the change in the average Nusselt as well as other dependent variables through one hundred iterations to be less than 0.01% from its initial value.

5. Program Validation and Comparison with Previous Research

In order to check on the accuracy of the numerical technique employed for the solution of the problem considered in the present study, it was validated with Shahi et al [10] by performing simulation for square cavity without nano particles (clear fluid) or baffle. The results showed good agreement between the present code and their results, Fig. 2.

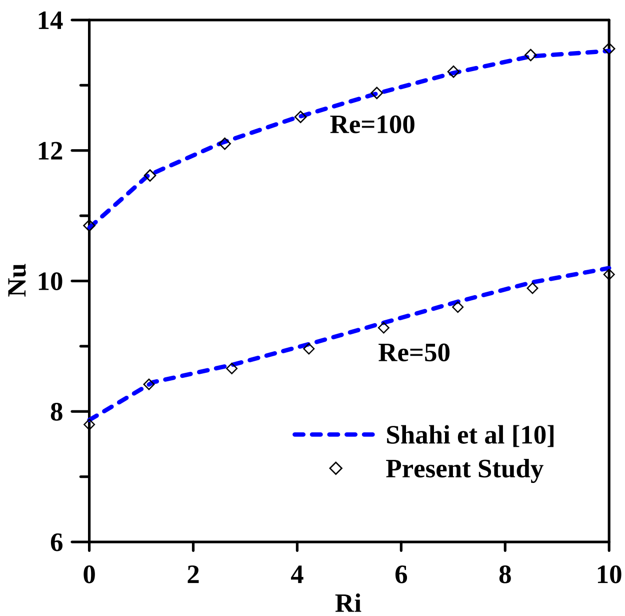

Figure 2. Average Nusselt number for different Re and Ri (No baffle and clear fluid)

6. Results and discussion

In this paper, the mixed convection flow inside ventilated cavity for cooling of a constant heat flux element in the bottom is numerically investigated; the baffle location has a great effect for enhancement of the heat transfer from the heat flux source part. The numerical study were performed for a range of Re, Ri and baffle location (Re = 50 to 500, Ri = 0.01 to 10 and W/L = 0.25 , 0.5 , 0.75). Before the discussion it is necessary to state that the effect of the heat flux variation appears in the Grashof number which is generated from both values of Re and Ri ($Gr = Ri\ Re^2$), in addition the local Nusselt number is very important on constant heat flux heat transfer mode due to the variation of the surface temperatures and so higher local Nusselt number led to lower cavity bulk temperature (enhanced cooling). Fig.3. represents the stream lines of Ri =1 (mixed convection domain $Gr = Re^2$) and different Reynolds number. At Re = 50 (lower Reynolds number) the induced flow has a little energy and expands suddenly due to high volume zone inside the cavity and so no vortices are formed within the cavity and the baffle location has no effect on the strength of the stream function ($\psi_{max} = 0.01728$) but only the baffle squeezes the flow at the tip of the baffle. As the Reynolds number increased to 200, the capability to absorb heat from the heat flux part increase and so the effect of the baffle location will affect, at W/L = 0.25 the strength of the stream function elevates ($\psi_{max} = 0.01879$) due to small area above the heat flux part and so enhancement of cooling will be established, the value of the stream function will decrease gradually as baffle location go away from the hot part from ($\psi_{max} = 0.01833$) to ($\psi_{max} = 0.01741$). It was also denoted that as the baffle location from hot source increases the expansion zone volume will also increase and so small vortices in corners will appear at W/L= 0.5 and W/L= 0.75. The same phenomena was found in case of Re = 500 but with high stream line strength. The effect of Reynolds number on the isotherms is represented in Fig. 4. It is denoted that the isotherms are bunched near the heat source depend on the value of Reynolds number. It was found that as the Reynolds number increases the energy absorption from the heat source increases, but as the baffle location increase the intensity of the isotherms near the heat source decrease due to the low strength of the stream function. Local Nusselt number is a very important number related to isotherms in case of heat flux heat transfer, the higher the local Nusselt number, the lower the temperature at this location, in Fig.5 to Fig. 7 as the local Nusselt number increase at certain location it means an effective cooling at this location ($Nu_L = 1/\theta_s$), the figures show that as Re number increases, the local Nusselt number increases for all baffle locations with different increment level according to the baffle location. For high Reynolds number (Re = 200-500) the leading edge of the heater near the inlet port has maximum local Nusselt number (low temperature) due to the low temperature of the incoming fluid through the inlet port and decrease gradually due to increase of the air temperature due to absorbed heat from the heater led to the decreasing in the absorption ability and increasing in the tail part of heater due to natural convection effect. It is denoted that, for low values of Reynolds number (Re=50), the effect of both natural and forced convection is equivalent and so the variation of surface temperature is approximately symmetrical along the heater length. As mentioned earlier, in order to get effective cooling, the average Nusselt number should be higher. It is observed that for a given Reynolds number the Nusselt number increase with an increase in Richardson number due to natural convection effect ($Gr = Ri\ Re^2$) for all cases (with baffle or without baffles), Fig. 8 to Fig. 11. This increase is remarkable for high Richardson number, also for a given Richardson number the increase in Reynolds number lead to increase in average Nusselt number. Comparison among baffle locations on the average Nusselt number values is represented in Fig. 12 to Fig. 15 for different Richardson number. It must be denoted that, as the baffle location from the heat flux part decreases it assists the forced convection mode and oppose the natural convection mode, so for lower value of Richardson number (Ri=0.01 to 1); the natural convection is very small with respect to forced convection and so the lower values of W/L given high heat transfer coefficient and so effective cooling, but for high values of Richardson number (Ri=10 natural convection domain) at low values of Reynolds number the addition of baffle led to decrease in heat transfer.

Finally, it must be denoted that the selection of the inlet port at lower part of the cavity was a good decision for augmentation in cooling of the heating part due to the assisting between both natural and forced convection.

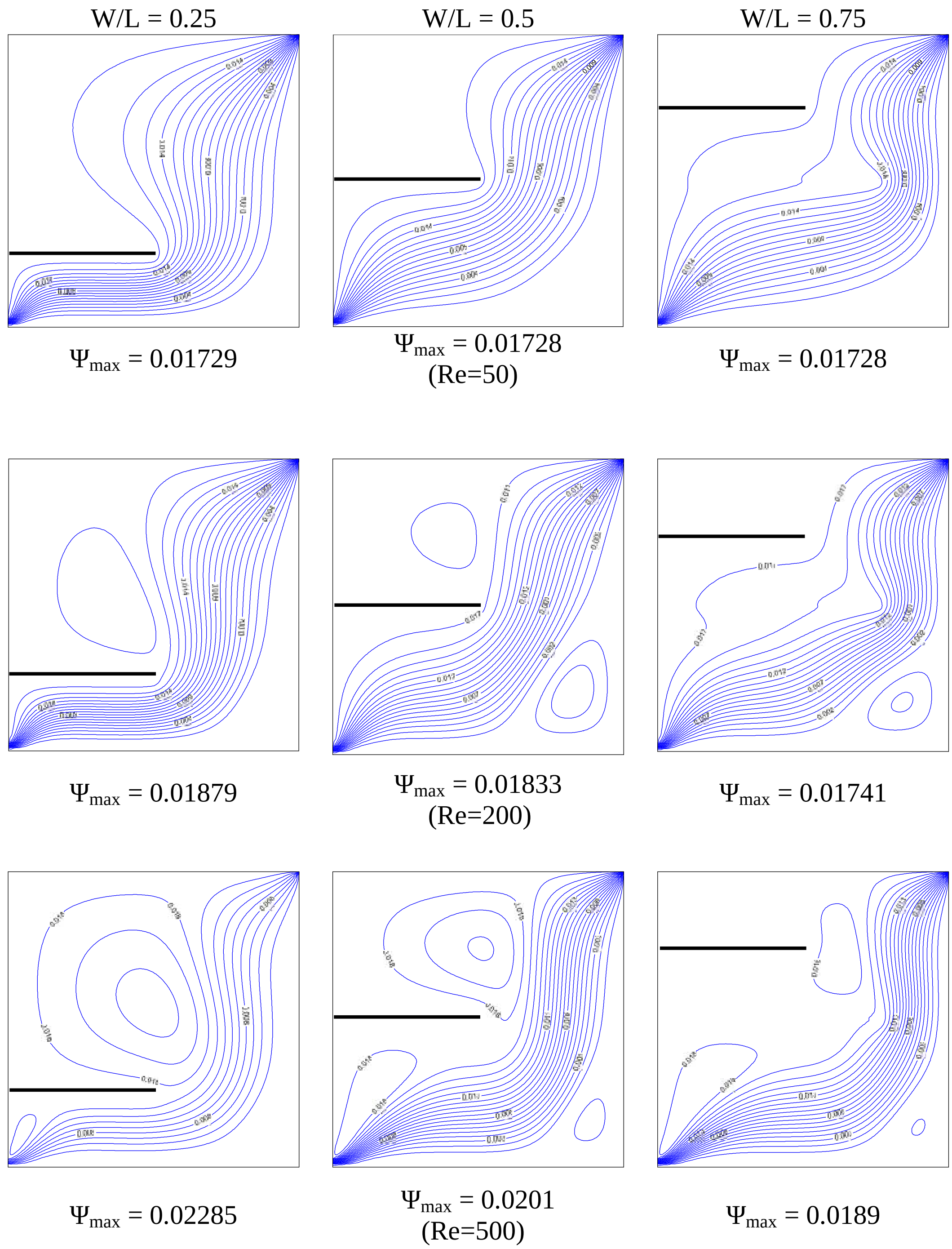

Figure 3. Streamlines for different Re at Ri=1

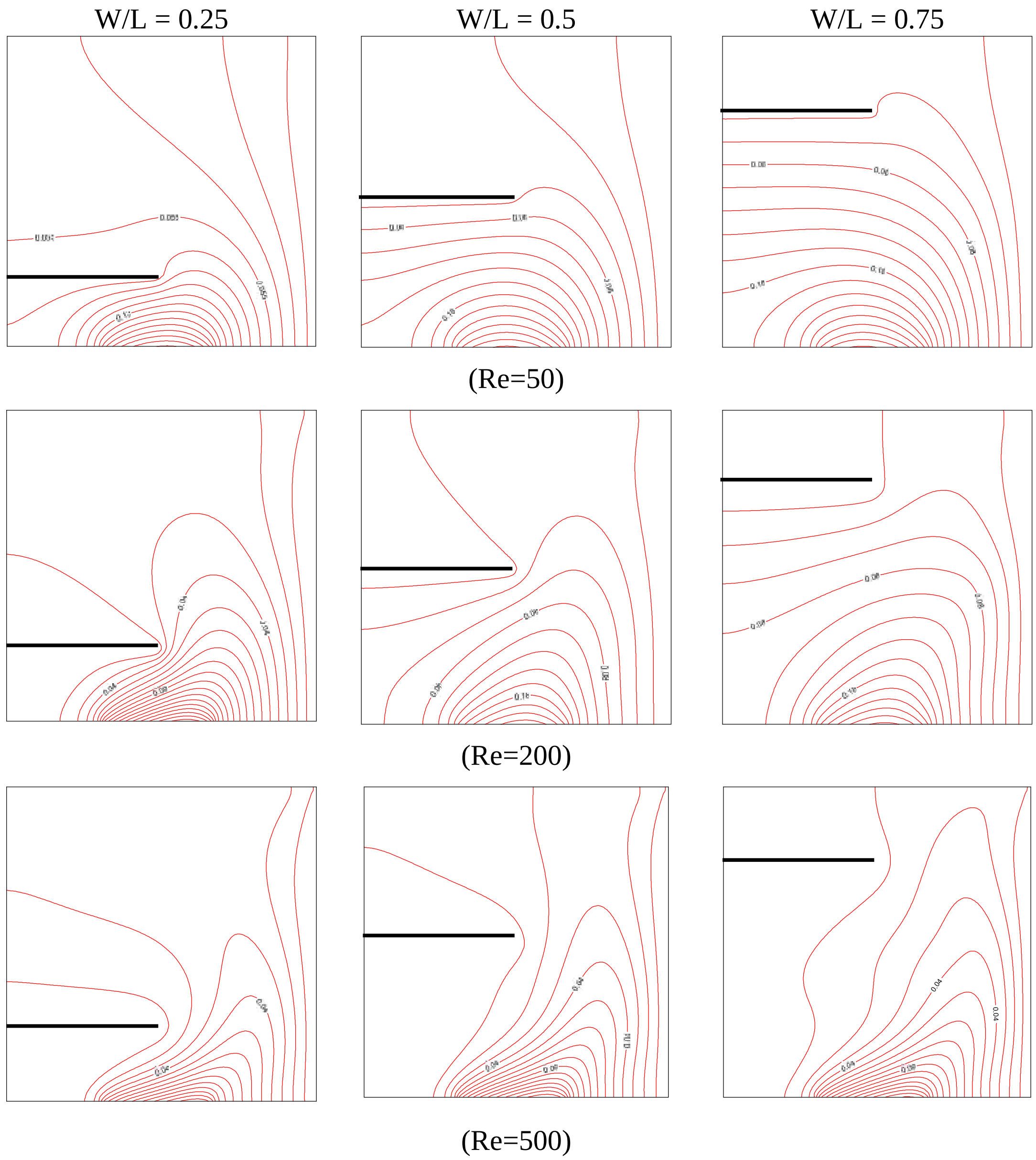

Figure 4. Isotherms for different Re at Ri=1

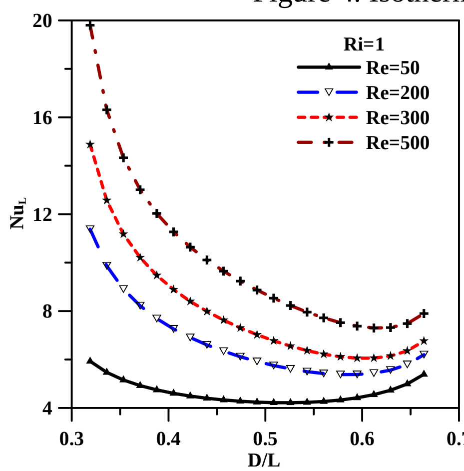

Figure 5. Local Nusselt number for different Re at W/L=0.25

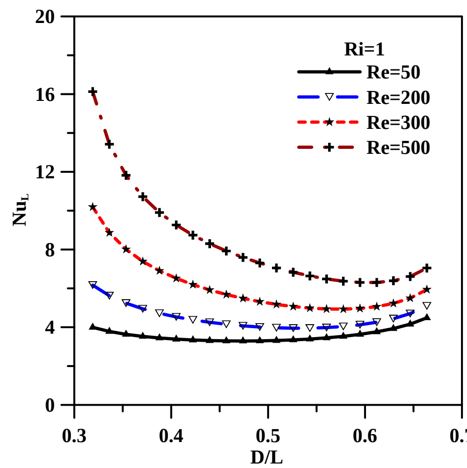

Figure 6. Local Nusselt number for different Re at W/L=0.5

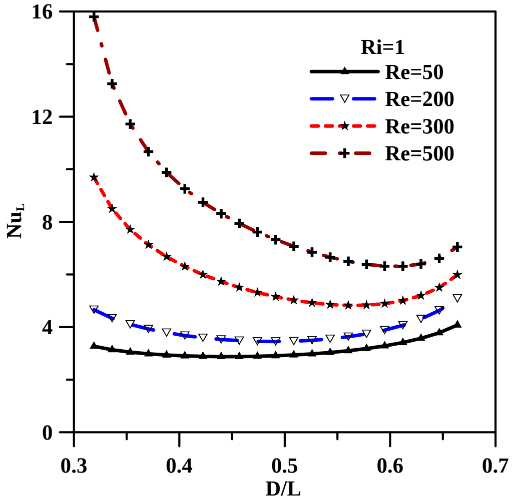

Figure 7. Local Nusselt number for different Re at W/L=0.75

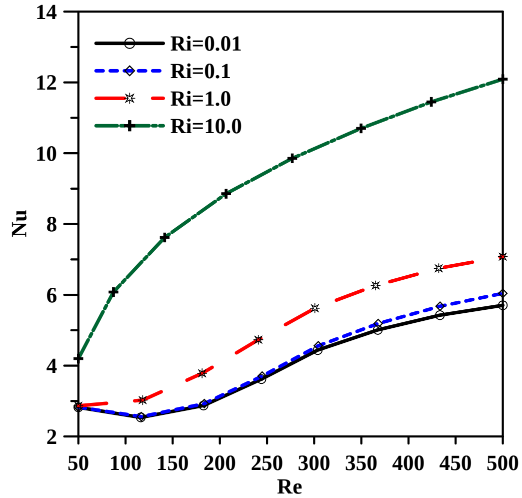

Figure 8. Average Nusselt number for different Re and Ri (No Baffle)

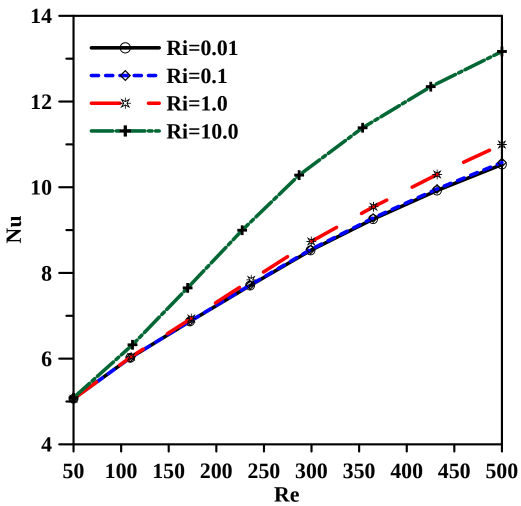

Figure 9. Average Nusselt number for different Re and Ri (W/L=0.25)

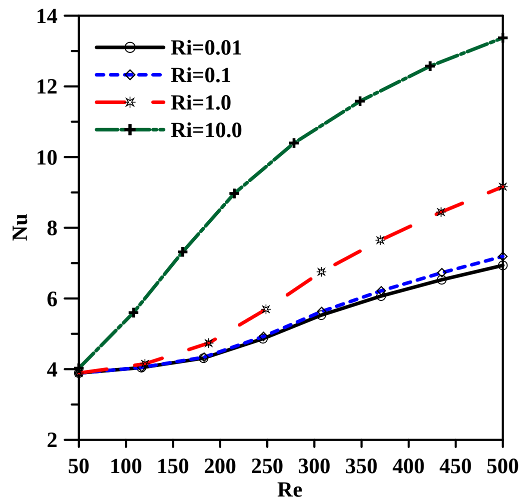

Figure 10. Average Nusselt number for different Re and Ri (W/L=0.5)

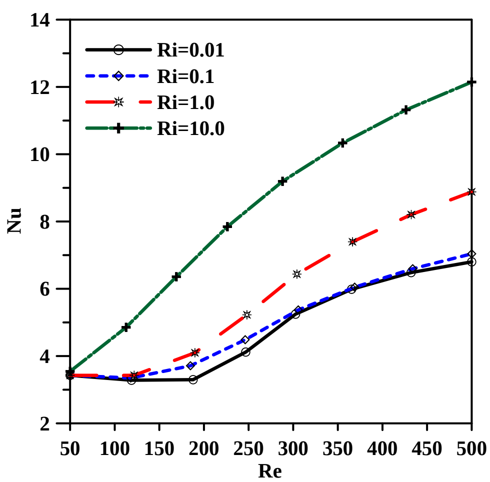

Figure 11. Average Nusselt number for different Re and Ri (W/L=0.75)

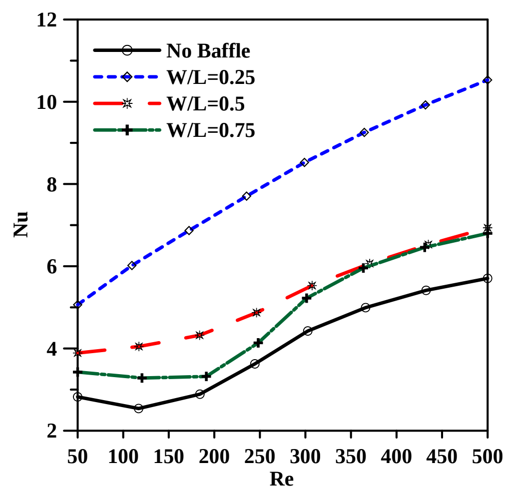

Figure 12. Effect of baffle position on average Nusselt for different Re(Ri =0.01)

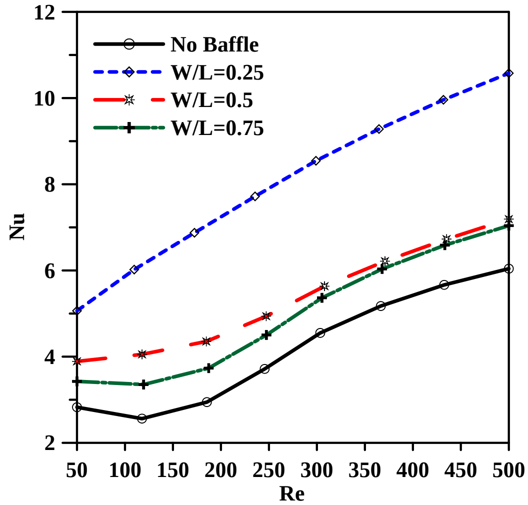

Figure 13. Effect of baffle position on average Nusselt for different Re (Ri =0.1)

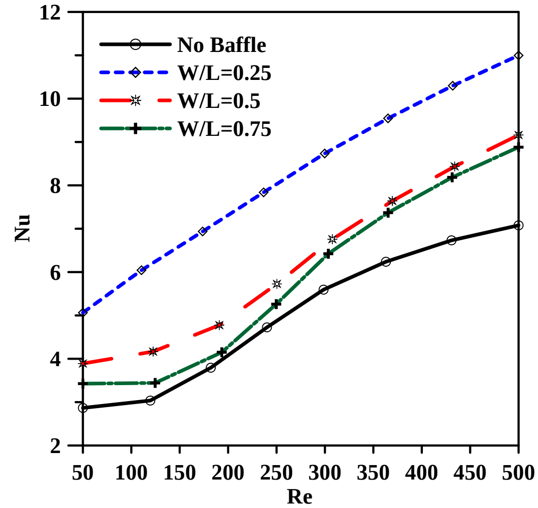

Figure 14. Effect of baffle position on average Nusselt for different Re (Ri =1.0)

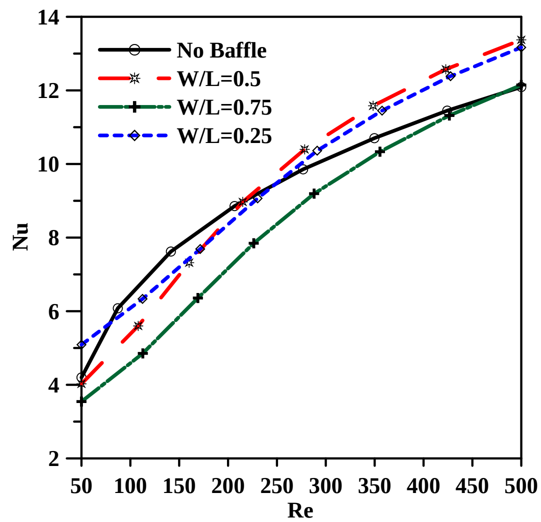

Figure 15. Effect of baffle position on average Nusselt for different Re (Ri =10.0)

7. Conclusion

In this paper, the mixed convection flow inside ventilated cavity for cooling of a constant heat flux element in the bottom is numerically investigated. The numerical study was performed for a range of Re, Ri and baffle location, it was observed that as the Reynolds number increased, the capability to absorb heat from the heat flux part increase and so enhancement of cooling will be established. If the baffle location from the heat flux part decreases it assists the forced convection mode and oppose the natural convection mode, so for lower value of Richardson number (Ri=0.01 to 1); the natural convection is very small with respect to forced convection and so, the lower values of W/L gave high heat transfer coefficient and so effective cooling. Finally, for high values of Ri and lower values of Reynolds number the existence of baffle decreased the heat transfer rate.

NOMENCLATURE

D	Heat part length, m
G	The baffle length,m.
H	Cavity height, m.
k	Thermal conductivity, W/m K.
L	Cavity width, m.
Nu	Average Nusselt number.
Nu_L	Local Nusselt number.
p	Pressure, N/m^2.
P	Dimensionless pressure.
q	Heat flux, W/m^2.
T	Local Temperature, C.
T_i	Temperature at the inlet port of the cavity, C.
U_o	Fluid Velocity at the inlet port, m/s.
	Velocity components in x-direct on, m/s.
	Dimensionless velocity component in X-direction.
	Velocity components in Y-direction, m/s.
	Dimensionless velocity component in Y-direction
W	Baffle location at the left vertical wall, m.
X,Y	Dimensionless coordinates.
x.v	Dimensional coordinates.

Greek symbols

α	Thermal diffusivity, m^2/s.
	Coefficient of thermal expansion, K^{-1}.
	Stream function.
	Dimensionless temperature.
	Dimensionless surface temperature at heat flux part.
	Dynamic viscosity, kg/m.s.
	Kinematic viscosity.
	Local fluid density, kg/m^3.

References

[1] T. Basak, S. Roy, P.K. Sharma, I. Pop, *Analysis of mixed convection flows within a square cavity with uniform and non-uniform heating of bottom wall*, International Journal of thermal science **48** (2009) 891–912.

[2] G. Guo, M.A.R. Sharif, *Mixed convection in rectangular cavities at various aspect ratios with moving isothermal sidewalls and constant flux heat source on the bottom wall*, International Journal of Thermal Science **43** (2004) 465–475.

[3] K.M. Khanafer, A.M. Al-Amiri, I. Pop, *Numerical simulation of unsteady mixed convection in a driven cavity using an externally excited sliding lid*, European Journal of Mechanics B/Fluids **26** (2007) 669–687.

[4] S. Saha, G. Saha, M. Ali, Md.Q. Islam, *Combined free and forced convection inside a two-dimensional multiple ventilated rectangular enclosure*, ARPN Journal of Engineering and Applied Sciences **1**(3) (October 2006) 23–35.

[5] Md.M. Rahman, M.A. Alim, S. Saha, M.K. Chowdhury, *A numerical study of mixed convection in a square cavity with a heat conducting square cylinder at different locations*, Journal of Mechanical Engineering ME39 (No. 2) (December 2008) 78–85.

[6] S.M. Saeidi, J.M. Khodadadi, *Forced convection in a square cavity with inlet and outlet,* International Journal of Heat and Mass Transfer **49** (2006) 1896–1906.

[7] S. Saha, Md.A.H. Mamun, M.Z. Hossain, A.K.M.S. Islam, *Mixed convection in an enclosure with different inlet and exit configurations*, Journal of Applied Fluid Mechanics **1**(1) (2008) 78–93.

[8] B. Ghasemi, 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 **33**(Number 1B) (2008) 189–207.

[9] S. Saha, G. Saha, M. Ali, Md.Q. Islam, *Combined free and forced convection inside a two-dimensional multiple ventilated rectangular enclosure*, *ARPN* Journal of Engineering and Applied Sciences **2**(No. 2) (2007) 25–36.

[10] M. Shahi, A. Mahmoudi, F. Talebi, *Numerical study of mixed convective cooling in a square cavity ventilated and partially heated from the below utilizing nanofluid*, International Communications in Heat and Mass Transfer **37**(2010) 201–213.

[11] P. Paranthoën, M. Gonzalez, *Mixed convection in a ventilated enclosure,* International Journal of Heat and Fluid Flow **31**(2010) 172–178.

[12] A. Mahmoudi, M. Shahi , F. Talebi, *Effect of inlet and outlet location on the mixed convective cooling inside the ventilated cavity subjected to an external nanofluid*, International Communications in Heat and Mass Transfer **37** (2010) 1158–1173.

[13] S.V. Patankar, *Numerical Heat Transfer and Fluid Flow*, Hemisphere, New York, 1980.