Journal Homepage: - www.journalijar.com

INTERNATIONAL JOURNAL OF ADVANCED RESEARCH (IJAR)

ISSN NO. 2320-5407

Article DOI: 10.21474/IJAR01/15970
DOI URL: http://dx.doi.org/10.21474/IJAR01/15970

RESEARCH ARTICLE

CFD MODELING OF THE OPTIMIZATION OF A VENTILATION SYSTEM IN ASQUARE ENCLOSURE WITH TWO INLETS

Omar Ngor Thiam[1,2], Oumar Drame[1,2], Aly Diouf[1,2], Cheikh Mbow[1,2] and Mamadou Lamine Sow[1,2]

1. Fluid Mechanics and Transfer Laboratory, Physics Department, Sciences and Technologies Faculty, Cheikh Anta DIOP University, Dakar-Fann, Senegal.
2. Research Group on Solar Energy and Transfers (GREST), Sciences and Technologies Faculty, Cheikh Anta DIOP University, Dakar-Fann, Senegal.

Manuscript Info

Manuscript History
Received: 31 October 2022
Final Accepted: 30 November 2022
Published: December 2022

Key words:-
Mixed Convection, Finite Volumes, Ventilated Cavity, Richardson Number

Abstract

In this article, we have numerically studied the phenomenon of laminar mixed convection in a ventilated square cavity. The vertical walls of the cavity are subjected to a temperature gradient while the horizontal walls are constrained as adiabatic. We used the finite volume method to discretize the system of non-dimensional equations and a purely implicit scheme for the temporal discretization. The results were presented in the form of hydrodynamic and thermal fields for different values of the Richardson number with the number of Reynolds constant. We found that the increase of the Richardson number with a fixed Reynolds number Re=100 allows us to visualize the evolution of the system and to appreciate the efficiency of the bottom ventilation.

Introduction:-

The heat convection induced by the temperature difference is an apparent phenomenon in several fields of industry such as the cooling of electronic components and the ventilation of premises... Mixed convection has therefore been the subject of numerous research works, both numerically and experimentally. A certain number of researchers have presented during the last two decades, work relating to the study of this phenomenon in the geometry mentioned above, among these works we can cite those of:

Khanafer et al [1] numerically investigated mixed convective heat transfer in open enclosures for three different flow angles of attack. Guanghong et al [2] made a numerical study of mixed convection in a two-dimensional rectangular cavity with constant heat flux from the partially heated bottom wall while the isothermal side walls move in the vertical direction. Zermane et al [3] presented a numerical study of the phenomenon of laminar mixed convection in a ventilated square cavity, one of the walls of which is subjected to a constant temperature, while the other walls are considered to be adiabatic.Tmartnhad et al [4] numerically studied mixed convection in a trapezoidal cavity whose lower horizontal wall is heated at a constant temperature, and the inclined upper wall is kept cold at a temperature. Omara et al [5] presented a study of the behavior of energetic (parietal flux) and dynamic (axial velocity and coefficient of friction) quantities in a transient laminar mixed convection problem descending in a thick cylindrical pipe. Benachour et al [6] have numerically studied laminar mixed convection in a ventilated rectangular cavity subjected to a constant heat source inside. Hicham et al [7] worked on a numerical study of heat transfer by mixed convection in an isothermal heating source in a three-dimensional ventilated cavity with different inlet and

Corresponding Author:- Omar Ngor Thiam
Address:- Fluid Mechanics and Transfer Laboratory, Physics Department, Sciences and Technologies Faculty, Cheikh Anta DIOP University, Dakar-Fann, Senegal.

outlet locations. ABBASSI et al [8] carried out a numerical study of transfers by mixed convection and radiation in a biomass pyrolysis fume combustion furnace.

In the present work, we highlight from a numerical study the importance of mixed convection to examine the thermal and dynamic field inside a ventilated, partially heated square cavity. Ventilation is ensured by the movement of air entering through three openings at the upper left corner, at the bottom and exiting through another at the lower right corner.

Introduction

Mathematical approach

The geometry considered in this work is presented in figure 1. We consider a square cavity filled by a fluid of side L equipped with three small openings, one located at the upper left corner of height h, the second is located at the bottom and middle of distance h and the third is at the lower right corner of height h. The vertical walls of the cavity are maintained at the temperature T_c(left), except the base wall which is brought to the temperature T_h(right). The horizontal walls are adiabatic (q=0). We assume the hot air entering through the opening at the lower left corner with a temperature of $T_c$and that from below with a temperature of T_f and exiting through the other upper right corner as Newtonian fluid.

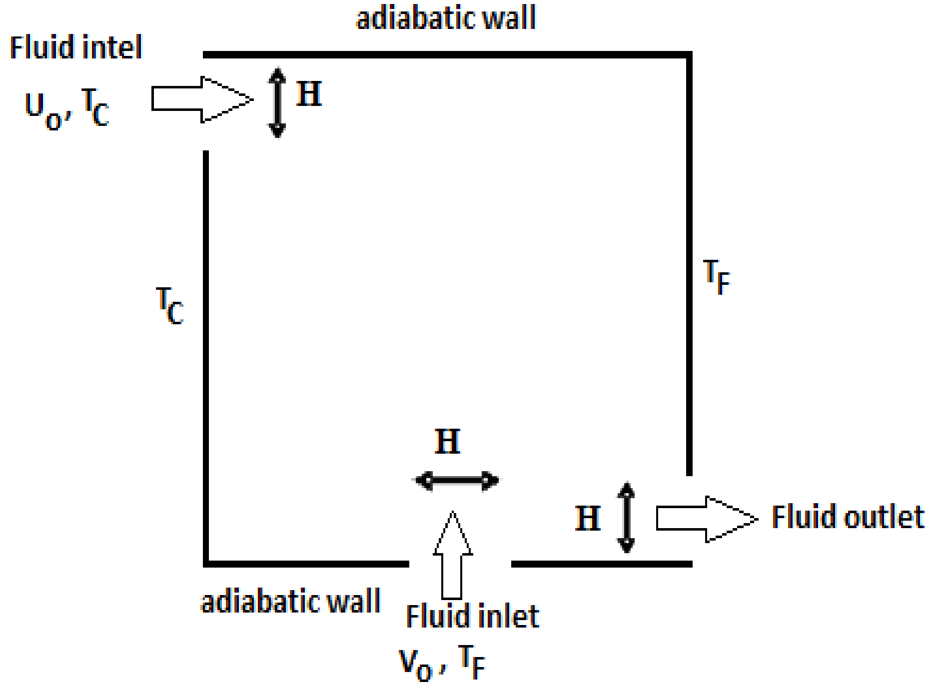


Figure 1:- Schematic representation of the physical model.

Modeling of the studied system isbased on the followingsimplifyingassumptions:

1. Fluid flow and heattransfer are incompressible, two-dimensional and the laminarregime.
2. The thermophysicalproperties of the fluid (μ, Cp, and k) are constant.
3. Viscous dissipation isnegligible. There is no heat source.
4. Boussinesq's approximation isvalid, itconsists in consideringthat the density variations are negligible at the level of all the terms of the momentumequations (ρ = ρ0),except at the level of the gravityterm. The variation of the densityρaccording to the temperatureisgiven by: $\rho - \rho_0 = \rho_0 \beta (T - T_0)$
   ρ_0 : the density of the fluid at the inlettemperatureT_0
   β: the volume expansion coefficient of the fluid

By introducing the currentfunction and the vorticity, the transferequations and the associatedboundary conditions are written in the followingdimensionlessforms:

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

$$\frac{\partial \omega}{\partial \tau}+U\frac{\partial \omega}{\partial x}+V\frac{\partial \omega}{\partial y}=\frac{1}{Re}\left(\frac{\partial^2 \omega}{\partial X^2}+\frac{\partial^2 \omega}{\partial Y^2}\right)+Ri\ \frac{\partial \theta}{\partial x}$$
$$\frac{\partial \theta}{\partial \tau}+U\frac{\partial \theta}{\partial X}+V\frac{\partial \theta}{\partial Y}=\frac{1}{Re*Pr}\left(\frac{\partial^2 \theta}{\partial X^2}+\frac{\partial^2 \theta}{\partial Y^2}\right)$$
$$-\omega=\frac{\partial^2 \psi}{\partial X^2}+\frac{\partial^2 \psi}{\partial Y^2}$$
$$U=\frac{\partial \psi}{\partial Y}\ et\ V=-\frac{\partial \psi}{\partial X}$$

The variables of the previous equations are dimensionless as follows:
$X=\frac{x}{L}, Y=\frac{y}{L}, U=\frac{u}{U_0}, V=\frac{v}{U_0}, \tau=\frac{U_0 t}{L}$ and $\theta=\frac{T-T_0}{T_p-T_0}$
Dimensionless numbers:
$Re=\frac{U_0 H}{\upsilon}=\frac{\rho\ U_0 H}{\mu}$: Reynolds number
$Pr=\frac{\mu C_p}{\lambda}$ Prandtl number
$Gr=\frac{g\beta H^3(T_p-T_0)}{\upsilon^2}=\frac{\rho^2 g\beta H^3(T_p-T_0)}{\mu^2}$thermal Grashof number
$Ri=\frac{Gr}{Re^2}$ Richardson number
Initial condition
At time t=0
$U=V=\Psi=\Omega=0$ and $T=0$
Boundary condition
Hot air inlet$(x=0, 0\le y\le L-h)$
$U=T=1$ et $V=\frac{\partial\Psi}{\partial Y}=0$, et $\Omega=-\frac{\partial^2\Psi}{\partial X^2}$
Outlet$(x=L, h\le y\le L)$

$$\frac{\partial U}{\partial X}=\frac{\partial V}{\partial X}=\frac{\partial \Psi}{\partial X}=\frac{\partial \Omega}{\partial X}=\frac{\partial T}{\partial X}=0$$

vertical walls
Left$(x=0, 0\le y\le L-h)$: U= $V=\frac{\partial\Psi}{\partial Y}=0, T=0$ et $\Omega=-\frac{\partial^2\Psi}{\partial X^2}$
Right$(x=L, h\le y\le L)$: $U=V=\frac{\partial\Psi}{\partial Y}=T=1$ et $\Omega=-\frac{\partial^2\Psi}{\partial X^2}$
Cold air inlet $(y=0, \frac{L}{2}-\frac{h}{2}\le x\le\frac{L}{2}+\frac{h}{2})$
$V=1$ et U= $\Psi=\Omega=T=0$
Lower horizontal walls
$(y=0, 0\le x\le\frac{L}{2}-\frac{h}{2}$ et $0\le x\le\frac{L}{2}-\frac{h}{2})$
$U=V=\frac{\partial\Psi}{\partial X}=\frac{\partial T}{\partial X}=0$ et $\Omega=-\frac{\partial^2\Psi}{\partial Y^2}$
Upper horizontal wall$(y=L, 0\le x\le L)$
$U=V=\frac{\partial\Psi}{\partial X}=\frac{\partial T}{\partial X}=0$ et $\Omega=-\frac{\partial^2\Psi}{\partial Y^2}$
The Nusselt Number provides a comparison between the flux transmitted by thermal convection and that transmitted by conduction. It is given in its dimensionless form by the following expression: $Nu_{local}=\frac{\partial\theta}{\partial k}$
with k is the direction normal to the heated plane

$$Nu=\frac{1}{S}\int Nu_{local}\,dS$$

Numerical Method:-

The transferequationsthatdescribeourproblem are nonlinear and coupled partial differentialequations. Due to theircomplexity, theseequations are solved by numerical techniques. The finite volume methodwasused for the discretization of the final system of non-dimensionalequationswith the boundary conditions while a purelyimplicitschemeisadopted for the temporal discretization [9]. For spatial discretization, using the power lawscheme, having a truncationerror of order one.

The meshadopted in thisstudyis a uniformmeshalong the two vertical and horizontal directions. It isthinnernear the wallswhere a significant change in physical variables isexpected. Practically, 15,876 nodes are located and 15,625 elements are found at the level of the enclosure. This wasthereforeused for all simulations in the presentwork as a compromise betweencomputationalcost, numericalstability, and good fieldresolution.

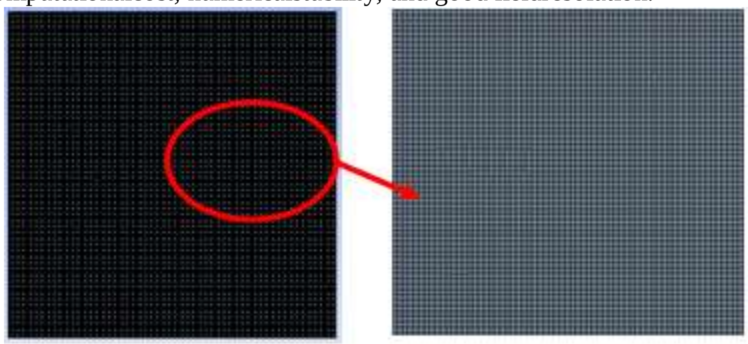


Figure 02:- mesh.

Further verification and validation of the current numerical results was performed by calculating the velocity and temperature fields of on the square cavity. Despite the scarcity of study simulating our problem in the literature, we were able to find the work of SumonSaha et al which deals with the mixed convection of the air in a rectangular enclosure with a single and different configuration between the inlet and the outlet. exit.So, in order to get closer to the geometry of his problem, we closed the bottom entry and performed the simulation for Rayleigh =1.0. Thus, we compared the streamlines and the board superimposing the speed and isothermal vectors.We can see a perfect similarity of the results between the work of SumonSaha et al for the TB configuration at the top and our work at the bottom. Despite the slight difference between the geometries, rectangular for SumonSaha et al and square in our case, we note the presence of two low intensity recirculation zones separated by a high intensity zone betwin the inlet and outlet for streamlines. Thus, given the similarity of the results with the work of SumonSaha et al, we draw a satisfactory conclusion from our numerical model, which pushes us to go even further by adding a second entry from below for the rest of the study.

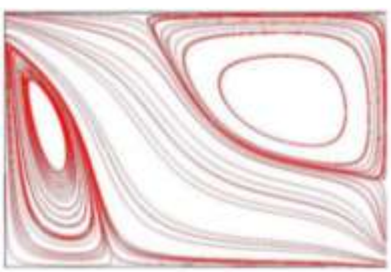
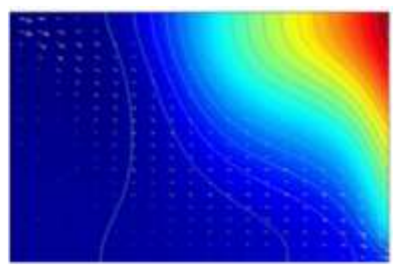
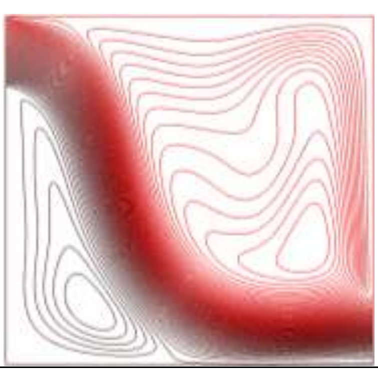
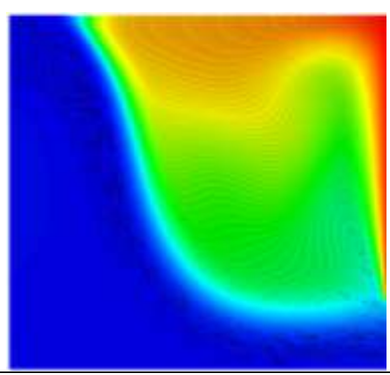
Streamlines	Velocity vectors and Isotherms
Work bySumon Saha et al	
	
Our work	
	

Figure 03:- mesh.

Results And Discussion:-

The flow structure, the dynamic field, as well as the heat transfer through the hot wall are examined in a chamber with two inlets and one outlet. Typical flow and temperature fields corresponding to the different steady-state solutions are presented.

The figures (Figure 04) show the evolution of isotherms and isocurrents respectively in the cavity with a flow in laminar regime for the Prandtl number Pr =0.72, for the Reynolds number Re=100 and a variation of the Richardson number of 0.1 to 100. The main parameter which governs the natural thermal flow is the number of Grashof thus giving to the various values of the number of Richardson which makes it possible to appreciate the nature of the dominant convection.

The isotherms are shown, by figure 4b, the distribution of the heat in the cavity is in conformity with the circulation of the fluid revealed by the isocurrents (figure 4a).

The isocurrents are shown in figure (4a).

The flow is greater at the inlet levels and at the level of the hot wall up to the outlet. The large part of the cavity is occupied by a recirculation zone with low intensity. We notice by increasing the speed we reduce the recirculation zone. The flow crosses the cavity along the diagonal by reducing the size of the convective cell and by favoring the shear effect at the exit with a high intensity of exit velocity of the fluid. By increasing the Ri, natural convection predominates until an instability is obtained with the formation of small convective cells for Ri = 100.

The isotherms are shown in figure (4b).

The distribution of heat in the cavity conforms to the circulation of the fluid revealed by the isocurrents (figure 4a). We note a heating of the fluid starting from the entry in top, all along the right wall (heated) and the upper wall (the hot fluid rises towards this wall) until the exit.

We observe the competition between the two upper right and lower left corner cells which is more intense with low Richardson numbers. By increasing the Richardson number, the upper right cell gains more and more intensity in favor of the lower right one.

Indeed, if the wall is heated, it can be seen that the latter tend to enlarge the size of the secondary recirculation located at the bottom left. If the left wall is cooled then the opposite effect is observed. This corner of less speed intensity has more temperature than the other corner with low speeds but this air recirculation zone increases more and more with the increase in the Richardson number. This phenomenon is to be related to the intensity of the main recirculation which favors the transport of heat on the right part of the lower wall. The main recirculation intensifies the heat transfers, which results in a greater thermal gradient.

We note that for low Richardson numbers, the isotherms are parallel and this representation characterizes the heat transfer which is dominated by conduction, as the Richardson number increases, the isotherms become more and more wavy and heat transfer increases, so flow intensifies and natural convection increases and predominates over conduction (natural convection is predominant). If the number of Richardson increases, Nusselt increases which justify the heat transfer which becomes accentuated.

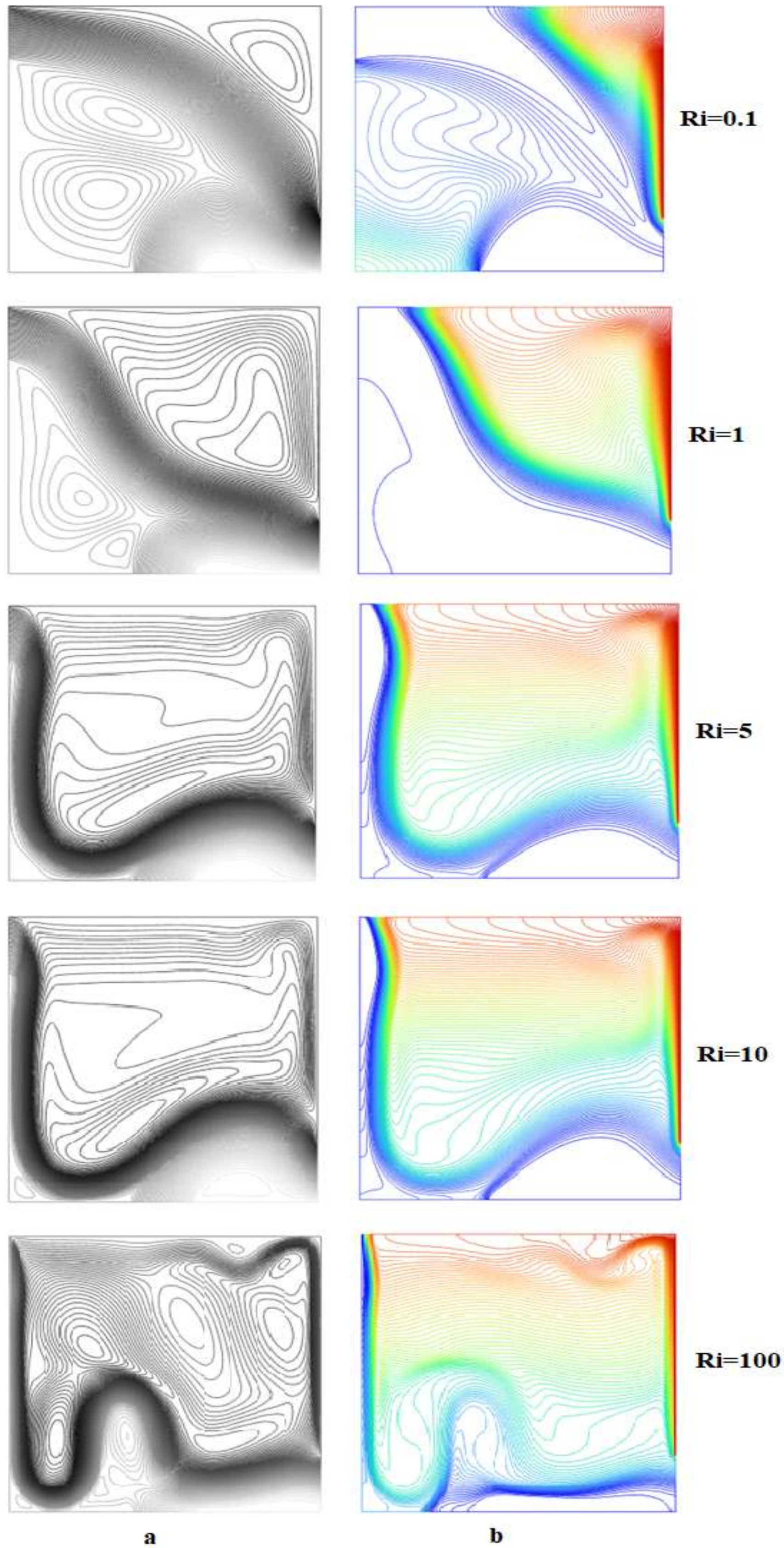


Figure 04:- Evolution of isocurrents and isotherms as a function of Richardson number.

The figure 05 shows us the evolution of the temperature at the outlet in the position with different Richardson numbers. We find that the outlet temperature increases with increasing Richardson's number. This is due to the strong thermal gradient at the level of the right side wall and the entry speeds. The temperature gradient is 2°C for the low Richardson number and 10°C for the large Richardson number. This puts in the difference between natural convection and forced convection for the values of high Richardson numbers. When the Richardson number is increased, the profile of the evolution of the temperature increases with deformations which is the flow of the fluid turbulence at the outlet level upwards. The evolution of the output speed of the increased fluid as a function of Ri confirms the temperature profile.

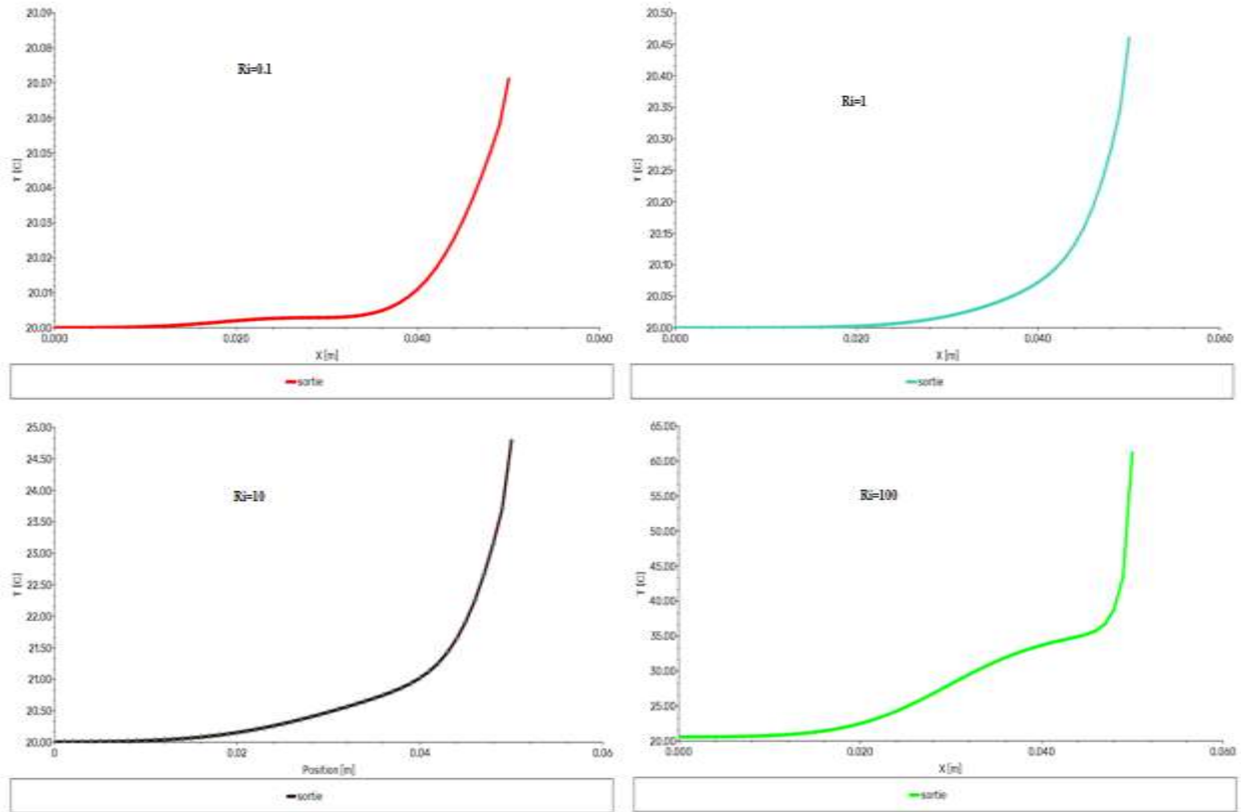


Figure 05:- Evolution of the temperature at the exit of the cavity.

Conclusion:-

The study presented in this work concerns laminar mixed convection in ventilated cavities with two entrances and one exit. The left and right vertical side walls of the cavity are subjected to a constant temperature higher than the ambient temperature (T_g greater than T_d), while the other horizontal walls are considered to be adiabatic. Based on the finite volume method to discretize the equations governing this phenomenon, it was possible to determine the isocurrents, the isotherms as well as the variations of the Nusselt number for different values of the Richardson number 0.1 to 100 and of the Reynolds number from 100. The variation of the heat transfer expressed by the mean Nusselt number, as a function of the Richardson number shows a significant increase when forced convection is dominant. We were able to demonstrate the existence of an influence on convection when the Richardson number is increased.

References:-

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

[2] Guanghong Guo, Muhammad 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 Sciences, 43 (2004) 465–475.

[3] Zermane S., Boudebous S., Boulkroune N, Etude numérique de la convection mixte laminaire dans des cavités ventilées, Sciences &Technologie B–N°23, Juin (2005), pp.34-44.
[4] Ilham TMARTNHAD, Mostafa NAJAM, Mustapha EL ALAMI, Rachid SEHAQUI, François PENOT, Convection mixte dans une cavité trapézoïdale chauffée par le bas, 12èmes Journées Internationales de Thermique, Tanger, Maroc du 15 au 17 Novembre 2005,363-366.
[5] Abdeslam OMARA, Said ABBOUDI, Etude de la convection mixte transitoire descendante dans une conduite verticale épaisse, 12èmes Journées Internationales de Thermique, Tanger, Maroc du 15 au 17 Novembre 2005,323-326.
[6] Benachour Elhadj, Draoui Belkacem, Rahmani Lakhdar, Mebarki Brahim, K. Asnoune, Imine Bachir, Simulation Numérique de la Convection Mixte d'une pièce (type d'habitat) avec la présence d'un Corps de Chauffe, Journal of Scientific Research N° 0 Vol. 1 (2010),19-23.
[7] Hicham Doghmi, BtissamAbourida, LahoucinBelarche, Mohamed Sannad, MeriemOuzaouit**,** Numerical study of mixed convection inside a three-dimensional ventilated cavity in the presence of an isothermal heating block, International Journal of Heat and Technology,Vol. 36, No. 2, June, 2018, pp. 447-456.
[8] Mohamed Ammar ABBASSI, Kamel HALOUANI, Xavier CHESNEAU and Belkacem ZEGHMATI, Modélisation des transferts par convection mixte et par rayonnement à l'intérieur d'un four de combustion des fumées de pyrolyse de la biomasse,13èmes Journées Internationales de Thermique, Albi, France du 28 au 30 Août 2007.
[9] S.V. Patankar, Numerical Heat Transfer and Fluid Flow, Mac Graw Hill,New York, 1980.