Journal of Mechanical Engineering Research and Developments
ISSN: 1024-1752
CODEN: JERDFO
Vol. 44, No. 5, pp. 136-148
Published Year 2021

Numerical investigation for Natural Convection in a square Enclosure With partially active thermal sides' wall

Noor Raad Mohammed*, Luma F. Ali

Department of Mechanical Engineering / Engineering College/ University of Baghdad

*Corresponding Author Email: noor.raad1993.nr@gmail.com

ABSTRACT

Three-dimensional cavity was investigated numerical in the current study filled with porous medium from a saturated fluid. The problem configuration consists of two insulated bottom and right wall and left vertical wall maintained at constant temperatures at variable locations, using two discretized heaters. The porous cavity fluid motion was represented by the momentum equation generalized model. The present investigation thermophysical parameters included the local thermal equilibrium condition. The isotherms and streamlines was used to examine energy transport and momentum. The meaning of changing parameters on the established average Nusselt number, temperature and velocity distribution are highlighted and discussed.

KEYWORDS

Natural convection, two heaters, Porous medium, discretized, Brinkman_ Forchheimer_ extended Darcy

INTRODUCTION

The subject of heat transfer by convection through porous media represents an important development and an area of rapid growth in contemporary heat transfer research. Flow in porous media requires a description of both the media and the flow. A porous medium commonly points to a material that forms from a solid matrix and interconnected voids. Examples of natural porous media are Beach Sand, Sandstone, Limestone, Rye Bread, Wood, Human Lung. The most common problem of natural convection through porous media is the adiabatic behavior of porous media, where the particles of hot region do not reach easily to upper regions due to the flow resistance by porous media so it's considered great insulation [1]. In general, the heat may have transferred by natural convection through the fluid inside pores and by conduction through solid particles. Porous media promotes insulation and heat storage at laminar conditions. While turbulence eddies are developed at turbulent conditions[1], [2], and[3]. The largest resistance to heat transfer comes from the free convection; therefore it plays a major role for systems design and performance [4]. The numerical investigation of the problems of the free convection of porous media by using computational fluid dynamics (CFD) principles is a good option.

This method has many privileges because of the wide range of dimensions, boundary conditions, geometries could be covered and reduce the cost and time. Some existing studies of free convection in the cavity has assumed without two discrete heaters wall and with uniform heating boundary conditions [5].The free convection of porous media with discrete heat source is utilized in promised heat transfer applications. Numerical examination for natural convection inside various shapes of enclosure cavities with a discrete heaters sources fixed at different walls location with different conditions in term of cooling and insulation , were surveyed by [6][7]. Others researches included a discrete heater in the confined porous enclosures, some of these presented in [8][2][9] . These studies included one heater on the side of the square enclosure to make temperature difference inside the enclosure, only [10] used two discrete heaters enclosures.

Very few researches included two discrete heaters enclosures. Various techniques have been offered to study the performance enhancement of free convection heat transfers inside the enclosure. It could be done by adding a discrete heaters wall at a different position. Experimental and numerical investigation of heat transfer in Cavities was performed [11-19]. Some of the experimental work included the study of porous media effect on forced convection heat transfer [11]. It was found that the rate of average Nusselt number decreases with the increase of heat flux. The effect of adding a discrete heater in different positions, distance, and different side opening wall on free convection in the cavity porous confined and boundary conditions are studied in the present work. The motivation behind our adaptation to the problem of the present work may be referred to the using These technological applications in electronics cooling, building design, solar energy systems and cores of nuclear reactors …etc .The main objective of the present work is to investigate numerically the heat transfer inside a three-dimensional opened cubical cavity containing two heaters and also to estimate the best condition for enhancing heat transfer in the case of heaters position with fixed the distance (S) between them (top and bottom) and distance (s) between the heaters at constant Rayleigh number (Ra) $of\ (6.15 * 10^6)$ and constant values of porosity

MATHEMATICAL FORMULATION

The assumptions made during the present study [20] are; the flow inside the enclosure is steady and laminar. The convective fluid and the solid material are in local thermal equilibrium in all places. Air is the working fluid inside the enclosure with an impermeable boundary and no internal heat generation. The chemical reaction, thermal dispersion, radiation heat transfer, pressure work and viscous dissipation are negligible. By considering the assumptions mentioned above, the steady, three dimensional, free convection governing equations in Cartesian coordinate system, the heat transfer convection process in porous medium (Required parameters for porous media used in this study are listed in Table (1)) is governed by the basic conservation principles of mass, momentum (Forchheimer–Brinkman-extended Darcy model), and energy equations. Therefore, the governing equations were derived and developed in the non-dimensional form are given by[21].

The Mass Conservation (Continuity) Equation

$$\nabla \cdot (\vec{V}) = 0 \tag{1}$$

Where $\vec{V}$ is dimensionless velocity vector

Momentum Conservation Equations

$$\frac{1}{\varepsilon}\left[\frac{\partial(\vec{V})}{\partial\tau} + (\vec{V}\cdot\nabla)\vec{V}\right] = -\nabla(P)^f + \frac{1}{\varepsilon\sqrt{Gr}}\nabla^2(\vec{V}) - \frac{(\vec{v})}{\mathrm{Da}\sqrt{Gr}} - \frac{F\varepsilon}{\sqrt{Da}}[\vec{V}\cdot\vec{V}]J + \vartheta \tag{2}$$

Where;

ε is Porosity; τ is dimensionless time; P is dimensionless pressure; Gr is Grashof number, (Gr= Ra / Pr); Da is Darcy number, (Da= K / H2); ϑ is Kinematic Viscosity

Energy Conservation Equations

$$\sigma\frac{\partial\theta}{\partial\tau} + \vec{V}\cdot\nabla\vartheta = \frac{k_{eff}}{k_f}\frac{1}{Pr\sqrt{Gr}}\nabla^2\vartheta \tag{3}$$

$$k_{eff} = k_f + (1-\varepsilon)k_s \tag{4}$$

Where;

k_{eff} is thermal conductivity of effective; k_f is thermal conductivity of effective fluid; k_s is thermal conductivity of effective solid

Meanwhile, the transport process within the wall can be

represented as

$$\nabla^2\vartheta_w = 0 \tag{5}$$

p : dimensionless pressure
V : dimensionless velocity vector

Da: Darcy number, K/H2
Gr: Grashof number, $g\beta\Delta T\ H^3/\upsilon^2$
F Forchheimer constant
K thermal conductivity
K permeability of the porous medium
ε: porosity of the porous medium
σ: heat capacity ratio $\left[\varepsilon(\rho C_p)_f + (1-\varepsilon)(\rho C_p)_s\right]/(\rho C_p)_f$
τ dimensionless time
θ dimensionless temperature
Subscripts
eff effective
f fluid
porous porous medium
s solid
w wall

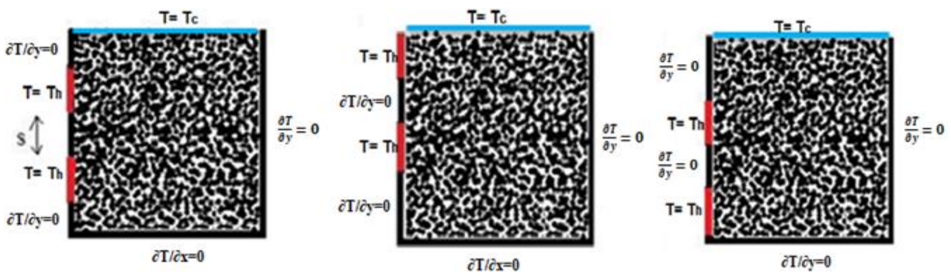
Table 1. Properties of Porous media

Parameters	Values (2.4-2.9) mm diameter
Porosity (ε)	0.418
Permeability (K)	1.61 x10^{-9} (m^2)
Density (ρ)	2515.91(kg/m^2)
Thermal Conductivity (k)	1.072054 (W/m.°C)
Specific Capacity (C_p)	1329 (J/kg.°C)

The governing Equations (1)–(4) subjected to the boundary conditions are solved numerically using the finite volume method [22]built in ANSYS CFD software[23]. This method is based on the discretization of the solution domain into finite number of control volumes (or cells) and integrates the governing equations over each control volume.

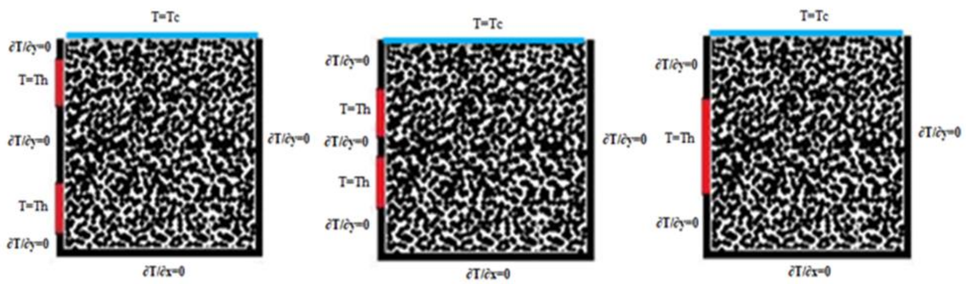
The associated boundary conditions for the problem under consideration can be expressed as

$V_x = V_y = 0$, ω = 0,and $\partial T/\partial y = 0$ at x = W, 0 ≤ y ≤ H
$V_x = V_y = 0$, ω = 0, and ∂T/∂x= 0 at y = 0, 0 ≤ x ≤ W
$V_x = V_y = 0$, ω = 0, and T=T_c at y = H, 0 ≤ x ≤ W
While at x = 0, 0 ≤ y ≤ H the velocity component are $V_x = V_y = 0$, ω = 0
and the thermal conditions are as depicted in figure 3.



Middle heaters (slandered) Top heater Bottom heaters

Effect of heater position with fixed distance (s) between them.



S*2 S\2 S=0

Figure 3. Shows the boundary conditions of enclosure in the present study.

Effect of the distance (s) between heaters

The Temperature at the interface can be sustained by employing the following condition:

$$K_{eff}\frac{\partial\vartheta}{\partial X})_{\text{porous}} = K_w\frac{\partial\vartheta}{\partial X})_{\text{solid}} \quad (5)$$

The physical quantities of interest in this investigation are the local Nusselt number and the average Nusselt number, which are, respectively, defined by

$$Nu = \frac{Q_{\text{wall}}}{Q_{\text{cond}}} = \frac{k_w}{k_f}\frac{\partial\vartheta}{\partial X})_{\text{solid}} \quad (6)$$

Where;

k_w isthermal conductivity of effective wall

$$Nu_{\text{ave}} = \int_0^1 Nu\, dY \quad (7)$$

ANSYS Fluent Grid Sensitivity Analysis

The dimensions of the proposed model are shown in Figure 2. Quadrilaterals ("quad") elements have been used in the meshing process. As an example, the square enclosure case is shown, i.e. aspect ratio, AR = 1 in Fig. 3. Quad elements are commonly used in simple geometries to reduce simulation times. Figure 4 illustrates the grid sensitivity analysis which shows that the simulations for calculated parameters attain mesh-independent convergence with approximately7038381elements, and 1074060 numbers of Nodes and number of iterations not excess than 40. There is no tangible modification in the calculated parameters which converges to a constant value. The program based on previous inputs will make calculation until reaching convergence with convergence factor of 0.0001. The discretized equations are solved based on the Pressure Implicit with splitting of Operators (PISO) method developed by Issa[24].The solver must perform enough iteration to achieve a converged solution as shown in figure 4. Computations are carried out for Ra ranging from 10^3 to 10^5, Pr=0.07–100.

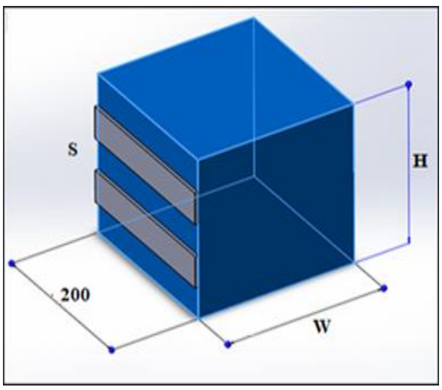


Figure 2. the model dimensions

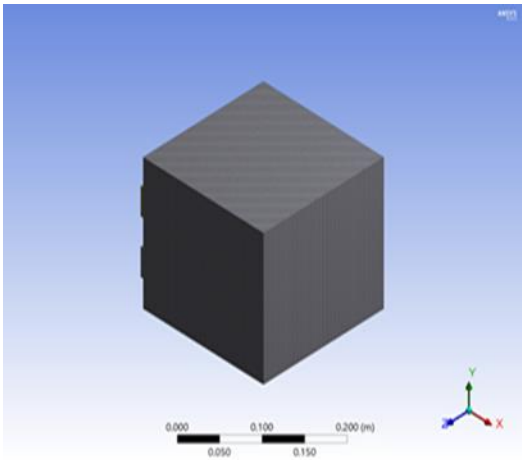


Figure 3. The mesh

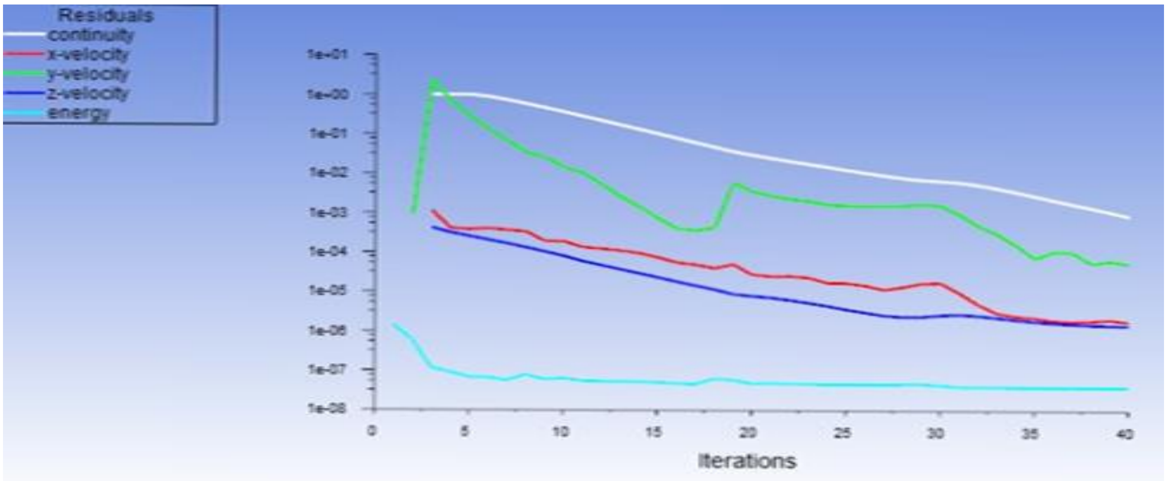


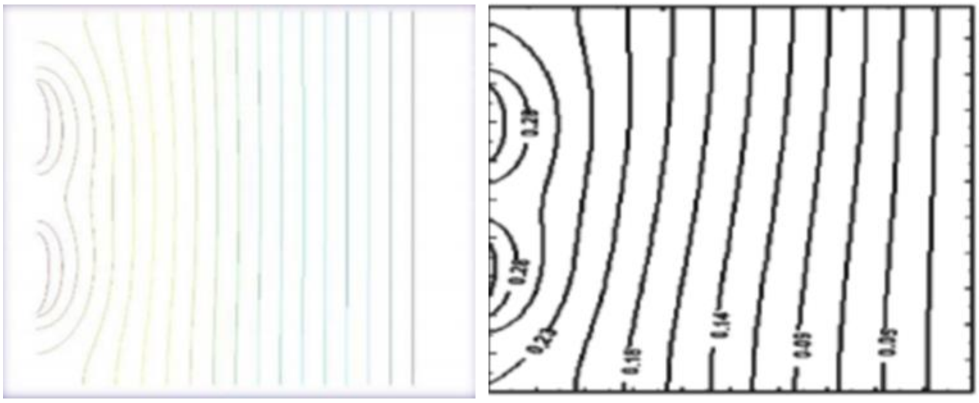
Figure 4. Solution Convergence (Scaled Residuals)

RESULTS AND DISCUSSION

A three-dimensional numerical modeling for different parameters was performed to illustrate the thermal and the flow behaviors for different models. The standard model consists of the two discrete heaters placed on the middle of left wall of the cubical cavity with top wall opened, while the right and down are adiabatic. The cavity is filled with air saturated porous media. The effects of the heaters position with fixed distance (s) between them and the changes of distance (s) between two heaters are discussed and compared with the standard model.

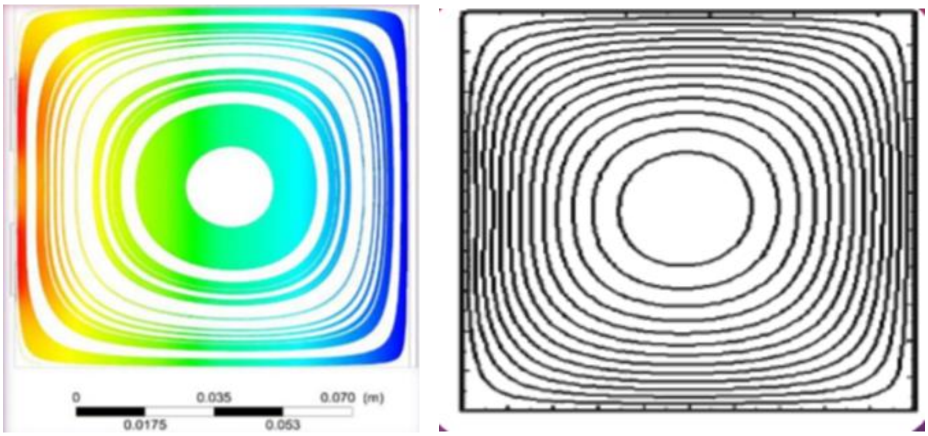
Verification of the Results

The results of the present work are compared in the results of previous study[10], in terms of the thermal and flow behavior in order to verify the numerical simulation. The results of velocity and temperature distribution at middle plane of the cavity were shown in figures (5) and (6) respectively. The results show a same behavior for flow structure and temperature gradient.



(a) (b)

Figure 5. Streamlines behaviors verification (a) present study and (b) previous work [10]



(a) (b)

Figure 6. Thermal behaviors verification (a) present study and (b) previous work[10]

The effect of the heaters position

Figure (7a) shows the temperature contours for the standard case and figures (7b and 7c) illustrate the effect of heaters positions at top and bottom location respectively, at constant Rayleigh number (Ra= $6.15 * 10^6$). Generally, the temperature counters show the temperature gradient growth from the left side of the wall (heaters sources) to right side. Moreover, the cold temperatures appear near the upper right side that exhibits the entrance boundaries of the cold air region to the cavity. For heaters at the top model (figure 7b), the temperature gradient is higher than others models (the middle and lower locations), that indicate the heat transfer near the cavity opening (at the upper location) is high. For heaters at the bottom model (figure 7c), the temperature gradient is lower and especially at the lower heaters which the temperature is highest among others heaters positions. At the models S/2 and S=0 as in the figure (8b) and (8c) respectively, the temperature magnitudes are higher as compared with the standard case.

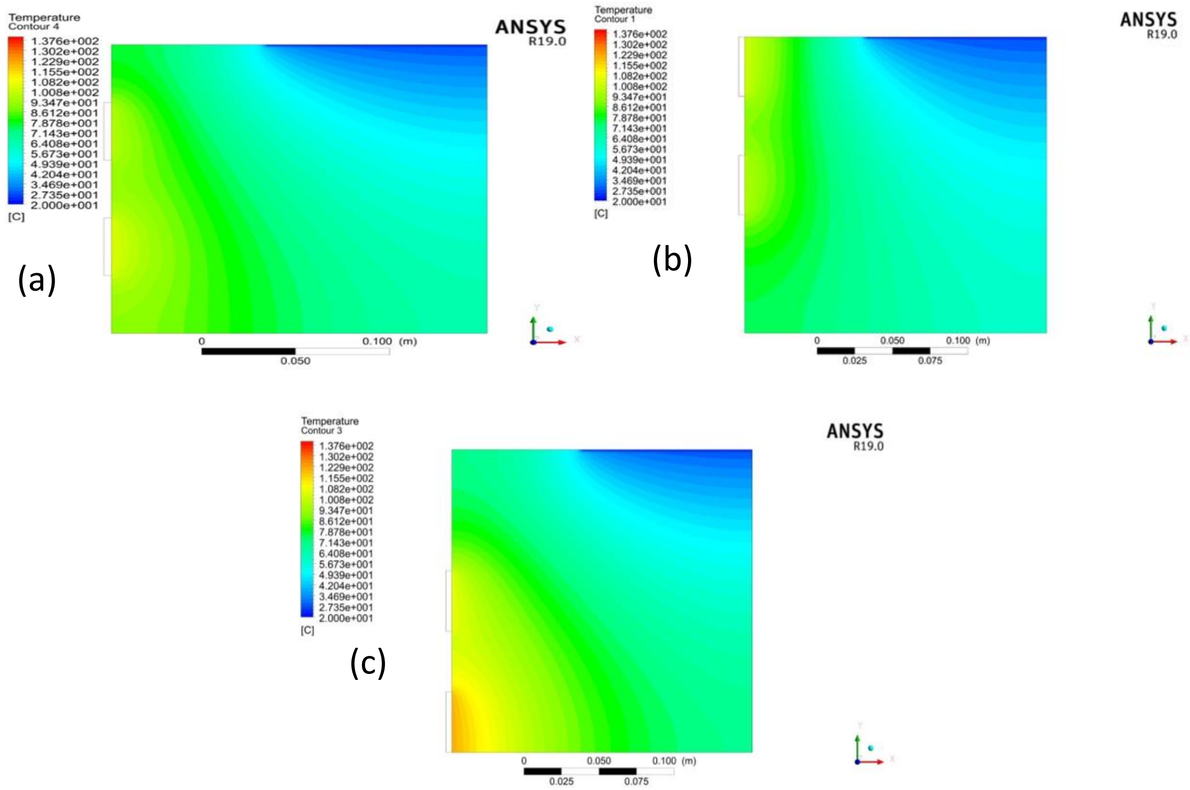


Figure 7. Temperature contours at the middle plane (Z=0.1 m) of the cavity for different heater positions models at Ra= $6.15 * 10^6$ (a) heaters middle (standard) (b) heaters at the top and (c) heaters at the bottom

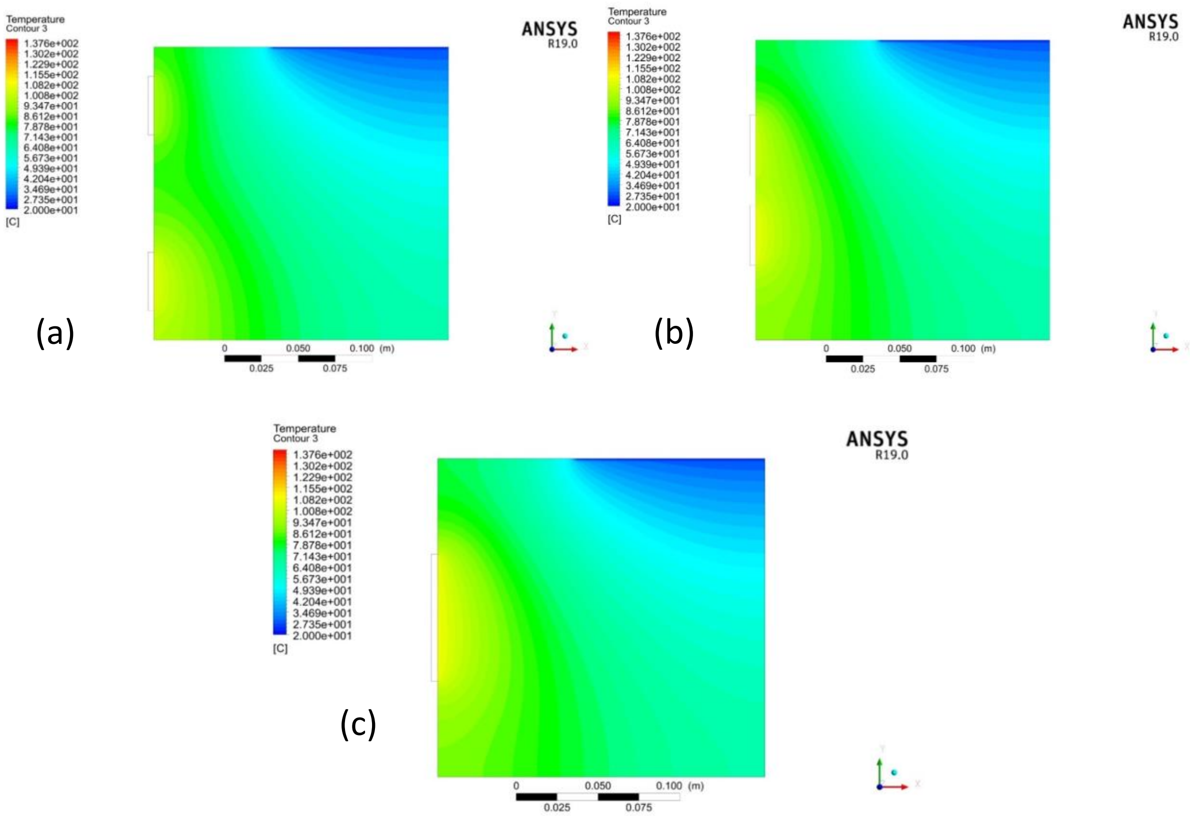


Figure 8. Temperature contours at the middle plane (Z=0.1 m) of the cavity for different heaters distance S models at constant Ra= $6.15 * 10^6$ (a) Sx2 (b) S/2 and (c) S=0

The velocity contours with streamlines are shown in figures (9) and (10), which represent the flow patterns and velocity magnitude. For the streamlines presented, the shape of the circulating cells is the same for three models. The inlet velocity magnitudes give high values near the upper right side and then decreases with length (H), which indicate the porous resistance, obstructed the flow to be induced at the bottom. The outlet velocity magnitude at the left wall side is high especially at the upper heater for all models that expected the buoyancy effect is strengthened near the opening. The model (upper heaters position) as in the figure (9B) gives highest velocity magnitude among others. For the three planes velocity contours at z direction, the velocity profile is constant for the whole planes. This exhibit declares that the velocity magnitude is constant for the z direction.

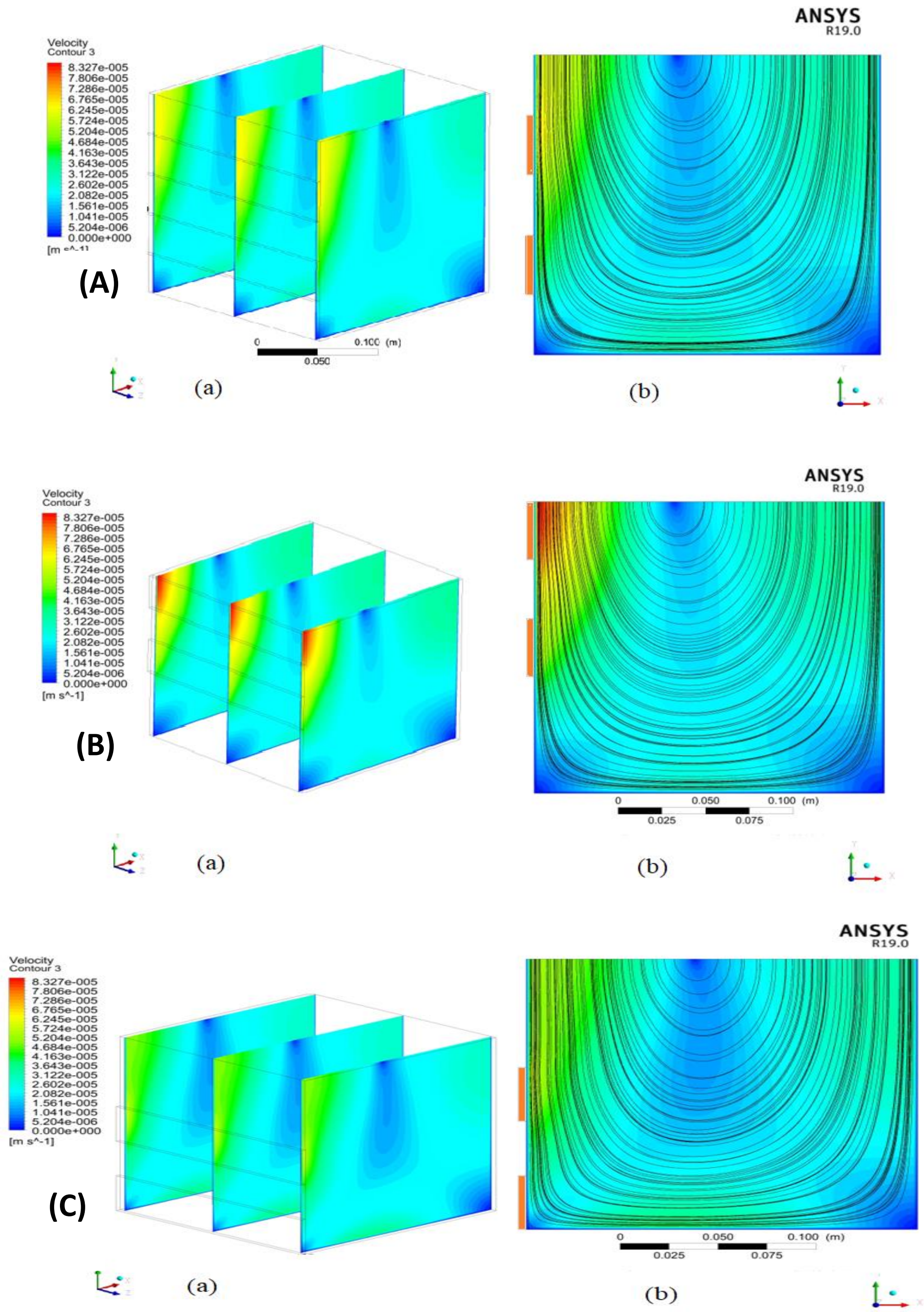


Figure 9. Velocity contours with streamlines for different heater positions models at constant Ra= $6.15 * 10^6$
(A) heaters at the middel standard (B) heaters at the top and (C) heaters at the bottom

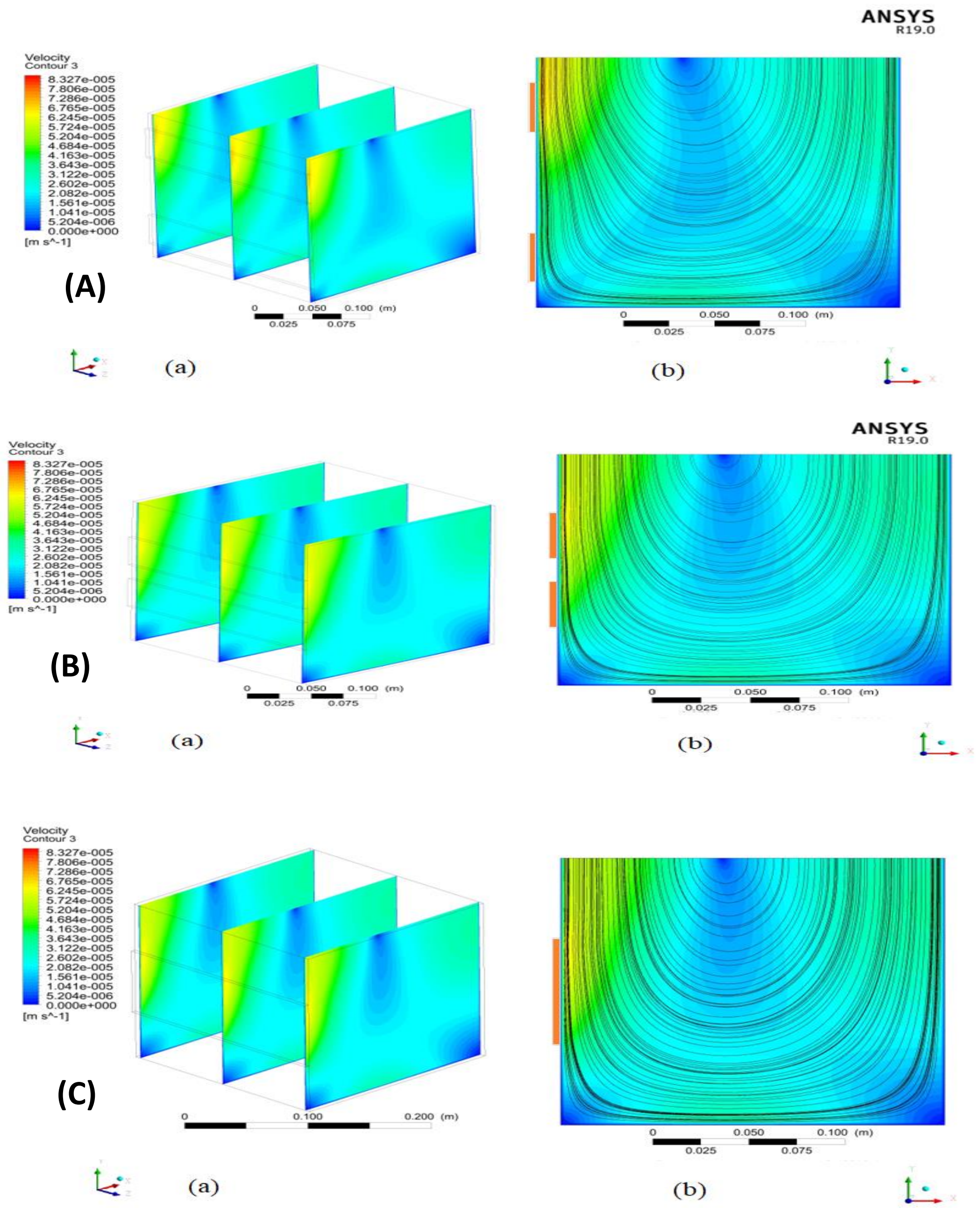


Figure 10. Velocity contours with streamlines for different heaters distance S models at constant Ra$= 6.15 * 10^6$ (a) Sx2 (b) S/2 and(c) S=0

The figures (11) and (12) depict the vertical temperature distribution inside cavity at different dimensionless x positions at constant Ra$= 6.15 * 10^6$. Generally, the isotherms become closed and nearly parallel in the bottom side of the cavity. The temperatures lines diverge towards the other sides with cavity length which the temperatures exposed the cold air entrance to the cavity. In the curve plot at dimensionless x=0, a two separated straight lines are appearing, that represent the constant surfaces temperatures in each heater. The temperature difference between the lower and the upper heaters at (heaters at the top mode figure (11a)) is very small as compared to the (heaters at the bottom figure (11b)) model. it's worth nothing, the lower heater temperature for the (Sx2 figure(12a)) model is slightly cooler than the lower heater for (S/2 figure(11b)) model although the latter one is located in upper position than (SX2). This indicates that as the distance between the heaters decreases, the surface temperatures will increase at each heater. From these figures, the surfaces temperature at the (heaters at the top) model gives a lowest one among others models.

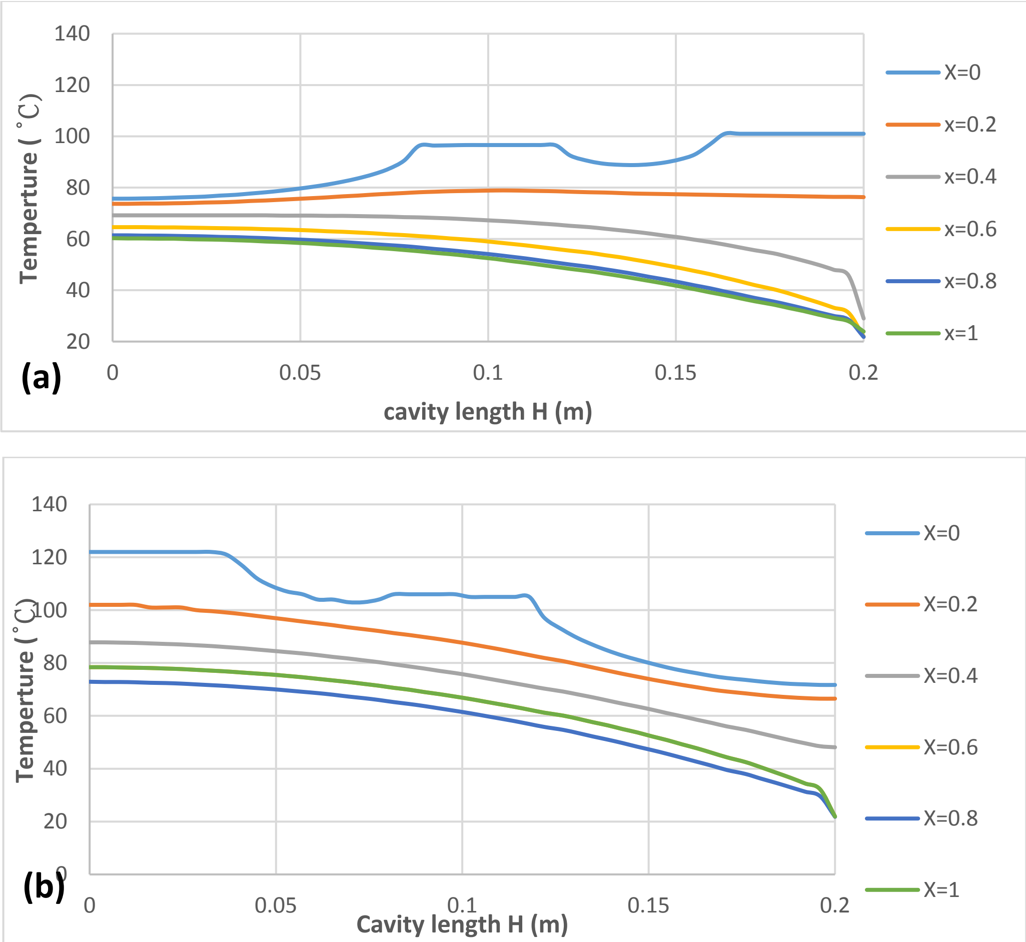
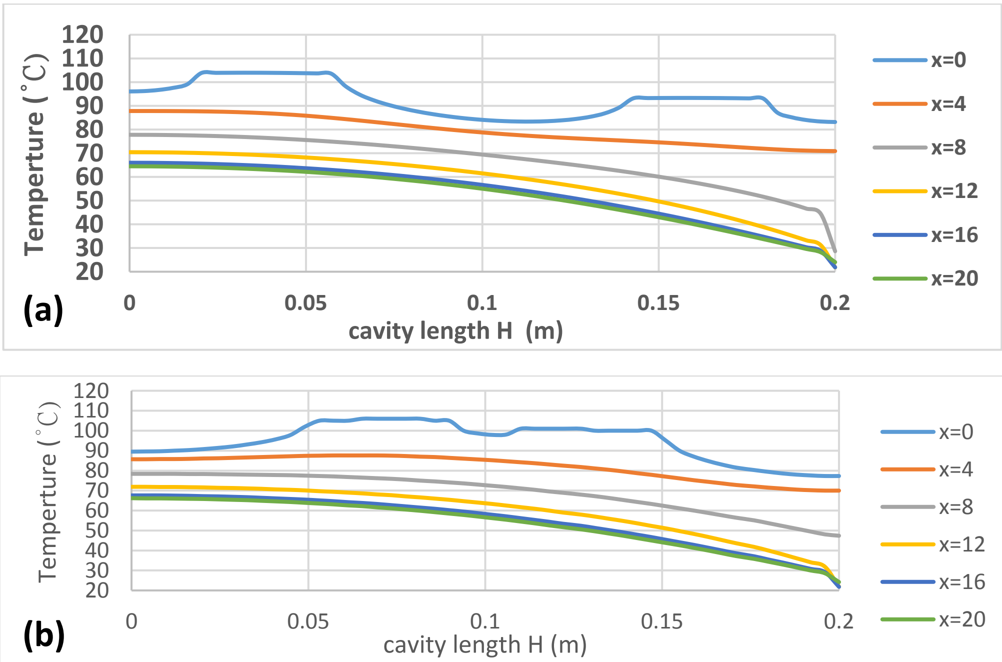


Figure 11. The vertical tempertures distrbution with cavity length at different dimentionless x postions for different models (a) heaters at the top and (b) heaters at the bottom.



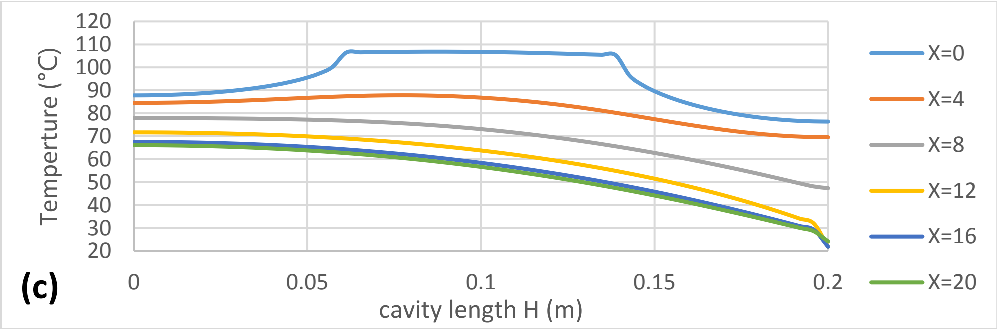


Figure 12. The vertical tempertures distrbution with cavity length at different dimentionless x postions for different models (a) Sx2 ,(b) S/2 and (c) S=0

To investigate the flow patterns and strength, the transversal velocity component v versus X abscissa, in upper side of the cavity (at y=0.2 m) for different models are shown in the figures (13) and (14). Generally, for all the models, the velocity behaviors are the same just like as a sinusoidal wave. The curves explain that the induced velocity magnitude at the entrance on the right side of the cavity is less than the outlet (left side), since the buoyancy force near the heaters sides assisted to increase the outlet velocity. As can see in the figure (13), the velocity at the (heaters at the top) model is higher than the standard model and the velocity at the (heaters at the bottom) is lower than the standard model. Figure (14) clearly shows that the difference in velocity values is a little bit and the velocity at (Sx2) model is higher than other models. From these mentioned velocity figures, the circulation intensity at the upper and lower side positions is more sensitive than the effect of S.

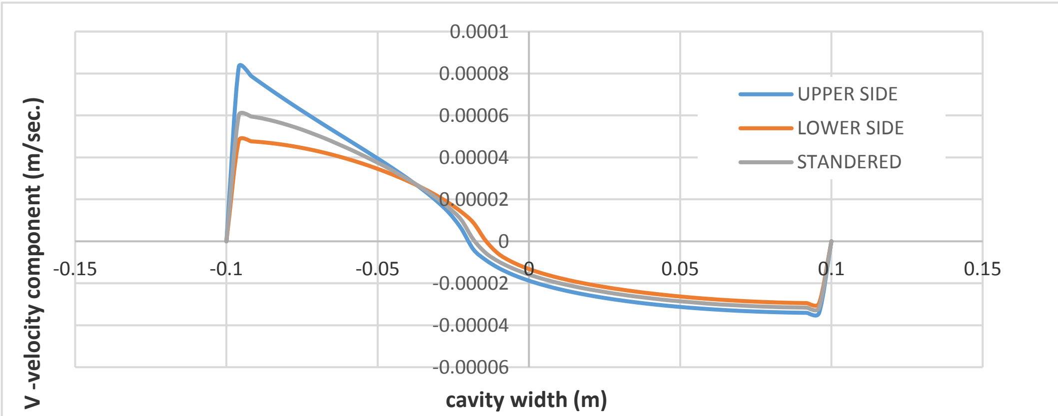


Figure 13. The effect of heaters postions with fixed distanse S on v-velocity profiles in the X-abscissa at the top line of the cavity (at y=0.2 m, z=0.1m)

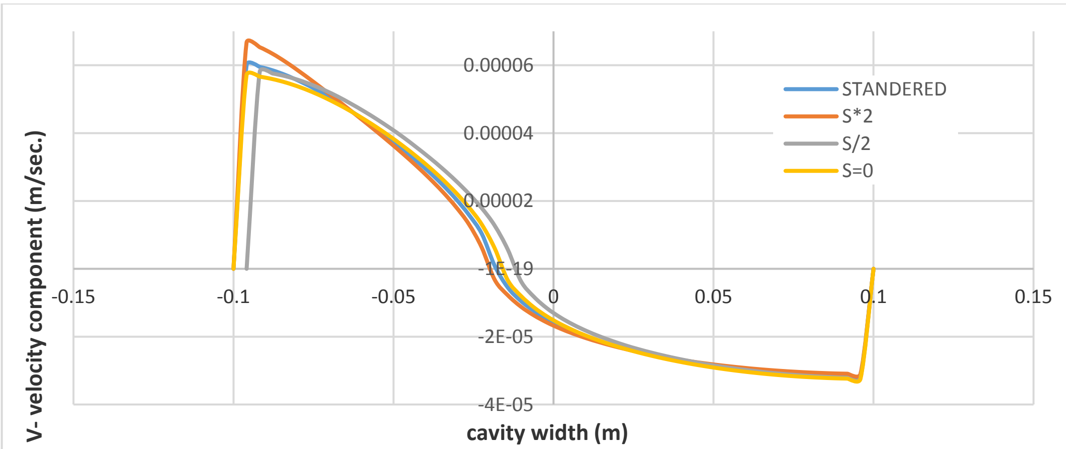


Figure 14. The effect of distance between heaters (S) on v-velocity profiles in the X-abscissa at the top line of the cavity (at y=0.2 m, z=0.1m)

CONCLUSION

The finite element method was utilized to perform numerical investigation for porous cavity with saturated fluid and natural convection. In this investigation, the porous medium general formulation was used to calculate the motion of fluid, which interpretations for solid viscous and inertial influence. Moreover, the isotherms and streamlines established from energy transport and momentum. average Nusselt numbers and temperature of interface wall are calculated for a high range of dimensionless parameters. These parameters include two heaters position with fixed the distance (S) between them (top and bottom) and distance (s) between the heaters at constant Rayleigh number (Ra) of $(6.15 * 10^6)$ with constant values of porosity. The results showed that the variation in heaters position has a little influence on enhancing the natural convection. The average Nu number for (Top position) model at the lower heater is enhanced by 9% whereas the upper heater is decreased by 5.6% as compared to the lower and upper heaters for the standard model in respectively. For all the models as the average Nu magnitude is high, the variation of local Nu number appears clearly along the heaters length and width. The local Nu is highest in magnitude at the middle length of cavity.

REFERENCES

[1] R. Chowdhury, "Natural Convection in Porous Triangular Enclosure with a Circular Obstacle in Presence of Heat Generation," Am. J. Appl. Math., Vol. 3, No. 2, Pp. 51, 2015. doi: 10.11648/j.ajam.20150302.14.

[2] A.K. Hussein and S.H. Hussain, "Natural Convection in a Square Enclosure Filled with a Saturated Porous Matrix Under Different Discrete Heat Sources Locations," Pp. 1896–1902, 2013.

[3] A. Durmuş and A. Dalǒlu, "Numerical and experimental study of air flow by natural convection in a rectangular open cavity: Application in a top refrigerator," Exp. Heat Transf., Vol. 21, No. 4, Pp. 281–295, 2008. doi: 10.1080/08916150802291095.

[4] E. Hakymez, "Conjugate Natural Convection Heat Transfer in a Cavity With Finite Wall Thickness," Thesis Sumitted, no. May, 2009.

[5] Y. Varol, H.F. Oztop, and A. Varol, "Free convection in porous media filled right-angle triangular enclosures," Int. Commun. Heat Mass Transf., Vol. 33, No. 10, Pp. 1190–1197, 2006. doi: 10.1016/j.icheatmasstransfer.2006.08.008.

[6] C. Devaraj, E. Muthuswamy, and S. Kandasamy, "Numerical Investigation of Laminar Natural Convection in Inclined Square Enclosure with the Influence of Discrete Heat Source," J. Appl. Math., 2015. doi: 10.1155/2015/985218.

[7] M. Saglam, B. Sarper, and O. Aydin, "Natural convection in an enclosure with a discretely heated sidewall: Heatlines and flow visualization," J. Appl. Fluid Mech., Vol. 11, No. 1, Pp. 271–284, 2018, doi: 10.29252/jafm.11.01.28167.

[8] M.E. Ismaeel and D. Amir, S. Dawood, "Effect of Partial Heating on Natural Convective Heat Transfer in an Inclined Porous Cavity," AL-Rafdain Eng. J., Vol. 20, No. 1, Pp. 49–60, 2012, doi: 10.33899/rengj.2012.47156.

[9] S.W.W. Kalaoka, "Natural Convection in Porous Square Cavities with Discrete Heat Sources on Bottom and Side Walls," Thai J. Math., 2014.

[10] S. Sivasankaran, Y. Do, and M. Sankar, "Effect of Discrete Heating on Natural Convection in a Rectangular Porous Enclosure," Transp. Porous Media, Vol. 86, No. 1, Pp. 261–281, 2011, doi: 10.1007/s11242-010-9620-x.

[11] M.A. Mashkour, L. Habeeb, and H.J. Jaber, "Heat Transfer in a partially Opened Cavity Filled with Porous Media", 3rd Scientific International Conference / Najaf, pp. 601-614, 2013.

[12] H.J. Jaber, A.A.A.N. Abaas, F.A.M.A. Ali, and L.J. Habeeb, "Cooling of a Vertically Oriented Air-Ventilated Square Cavity", Journal of Mechanical Engineering Research and Developments, Vol. 43, No. 6, Pp. 23-38, 2020.

[13] M.M. Al-Azzawi, A.K. Hassan, H. Kareem, and L. Habeeb, "Numerical Study of Lid Driven Mixed Convection in Inclined Wavy Cavity", Journal of Mechanical Engineering Research and Developments, Vol. 43, No. 6, Pp. 184-196, 2020.

[14] R.C. Al-Zuhairy, M.H. Alturaihi, F.A.M.A. Ali, and L.J. Habeeb, "Numerical Investigation of Heat Transfer in Enclosed Square Cavity", Journal of Mechanical Engineering Research and Developments, Vol. 43, No. 6, Pp. 388-403, 2020.

[15] M.A.S. Mustafa, H.M. Hussain, A.A. Abtan, and L.J. Habeeb, "Review on Mixed Convective Heat Transfer in Different Geometries of Cavity with Lid Driven", Journal of Mechanical Engineering Research and Developments, Vol. 43, Special Issue: Mechanics and Energy, Pp. 12-25, 2020.

[16] A.A. Mohammed, S.T.M. Al- Musawi, S.K. Ayed, A. Alkhatat, and L.J. Habeeb, "Natural Convection Heat Transfer in Horizontal Elliptic Cavity with Eccentric Circular Inner Cylinder", Journal of Mechanical Engineering Research and Developments, Vol. 43, Special Issue: Mechanics and Energy, Pp. 340-355, 2020.

[17] A.N.A. Saieed, M.A.S. Mustafa, S.K. Ayed, and L.J. Habeeb, "Review on Heat Transfer Enhancement in Cavity with Lid Driven", Journal of Mechanical Engineering Research and Developments, Vol. 43, Special Issue: Mechanics and Energy, Pp. 356-373, 2020.

[18] H.S. Majdi, A.M. Abed, L.J. Habeeb, "Nanofluid Predication of Lid Driven Mixed Convection in Square Cavity with Adiabatic Elliptic Body", Journal of Mechanical Engineering Research and Developments, Vol. 44, No. 1, Pp. 151-163, 2021.

[19] H.S. Majdi, A.M. Abed, and L.J. Habeeb, "Mixed Convection Heat Transfer of Cuo-H2o Nanofluid in a Triangular Lid-Driven Cavity with Circular Inner Body", Journal of Mechanical Engineering Research and Developments, Vol. 44, No. 1, Pp. 164-175, 2021.

[20] A. Nield, Bejan, D.A., Convection in Porous Medium, 5th ed. 2017.

[21] I. Al-Amiri, A., Khanafer, and K. Pop, "Steady-State Conjugate Natural Convection in a Fluid-Saturated Porous Cavity," Int. J. Heat Mass Transf., 2008.

[22] V. Patankar, "Numerical Heat Transfer and Fluid Flow," 1980.

[23] "ANSYS FLUENT User's Guide," 2019.

[24] R.I. Issa, "Solution of the implicitly discretized fluid flow equations by operator splitting," 1986.