Journal of Engineering Science and Technology
Vol. 10, No.10 (2015) 1326 - 1341

NATURAL CONVECTION IN A TRIANGULAR TOP WALL ENCLOSURE WITH A SOLID STRIP

AHMED KADHIM HUSSEIN[1], SACHINDRA KUMAR ROUT[2,]*, FARSHID FATHINIA[3], RAMESH CHAND[4,] H. A. MOHAMMED[5]

[1]Mechanical Engineering Department, College of Engineering, Babylon University, Babylon City, Iraq
[2]Sachindra Kumar Rout, Assistant Professor, Department of mechanical Engineering, CV Raman College of engineering, Bidyanagar, Mahura, Janla, Bhubaneshwar, Odisha 752054
[3]Department of Thermofluids, Faculty of Mechanical Engineering, University Teknologi Malaysia, 81310 UTM Skudai, Johor Bahru, Malaysia
[4]Department of Mathematics, Government P. G. College, Dhaliara 177103, Himachal Pradesh, India
[5]Department of Thermofluids, Faculty of Mechanical Engineering, University Teknologi Malaysia (UTM), 81310 UTM Skudai, Johor Bahru, Malaysia
*Corresponding Author: sachindra106@gmail.com

Abstract

Natural convection inside a two-dimensional rectangular cavity with a triangular roof having an adiabatic solid strip inserted at a middle of the cavity is studied numerically using a finite volume method. Both of the triangular roof and the bottom wall are considered adiabatic while the vertical left side wall is maintained at constant temperature higher than that of inside fluid temperature. The right side wall is considered differentially heated by supplying a constant heat flux. The working fluid is chosen for analysis is air. The Computational Fluid dynamics (CFD) solution commercial package ANSYS FLUENT 14.0 is used for the numerical simulation purpose. The results are presented in the form of isotherms, streamlines, velocity vector and average Nusselt number (Nu) for Rayleigh numbers in the range of 10^3 to 10^6.Throughout this study, the aspect ratio is kept equal to 0.5.It is found that the solid adiabatic strip inside the cavity has a significant effect on the flow and thermal performance. The results indicated the usual fact that when the Rayleigh number increases the average Nusselt number also increases.

Keywords: Natural convection, FVM, Cavity, Uniform heat flux, Triangular roof, Solid strip

Nomenclatures

E	Roof height, m
g	Acceleration due to gravity, m/sec^2
H	Height of the cavity, m
k	Fluid thermal conductivity, W / m. °C
L	Characteristic length, m
Nu	Average Nusselt number
Pr	Prandtl number
q	Heat flux per unit area, W/m^2
Ra	Rayleigh number
T	Fluid temperature, K
T_c	Temperature of the cold wall, K
T_o	Reference temperature, K
u,v	Dimensional velocity components in x and y directions respectively, m/s
x,y	Dimensional coordinates

Greek Symbols

α	Fluid thermal diffusivity, m^2/sec
β	Coefficient of volumetric expansion, K^{-1}
ρ	Fluid density , kg/m^3

Abbreviations

SAS	Solid adiabatic strip

1. Introduction

The convection due to buoyancy forces, i.e. natural convection in a rectangular enclosures is a fundamental problem due to the wide range of applications in thermal insulation, heating and cooling of buildings, energy drying processes, cooling of electronics, lakes and geothermal reservoirs, solidification of casting, underground water flow, solar collector, solar energy collection, nuclear energy, be the prevailing drag form at supersonic speeds, careful selection of the nose and tail shapes is mandatory to ensure performance and operation of the over-all system. Cooling of electronic components, micro-electromechanical systems (MEMS) and lubricating grooves. For the last 50 years, it was an interesting area of research on natural convection due to its application in various fields as described above and the coupling of fluid and energy flows. However, a less number of works has been reported on the more complex case of cooling from the one side wall, mainly with simultaneous heating of the other side wall. When the analysis is related for a cavity, if side walls are in different boundary conditions and the top and bottom walls are at an adiabatic boundary condition. Then the fluid flow and heat transfer occur due to the temperature difference between two vertical walls. Here, gravity force plays a crucial role and acted perpendicular downward. The fluid circulation direction inside the cavity is due to the gradient vector which acts perpendicular to each other.

A detail study of literature and investigation inside a rectangular cavity was performed by Ostrach [1-3] and Hoogendoorn [4]. Moreover, Ganzarolli and Milanez [5], numerically investigated the natural convection inside a rectangular enclosure with a constant heat flux at the bottom wall and symmetrically cold from its vertical sidewalls while the upper wall was maintained at an adiabatic boundary condition. They presented the results in the form of Nusselt number (Nu) as a function of Rayleigh number (Ra) within a range of 103 to 107, whereas the aspect ratio was varied from 1 to 9. They also reported that at a prescribed heat flux, the isotherms and streamlines occupied more uniformly inside the whole cavity at a low Rayleigh number value. An another numerical study was conducted by Aydin et al. [6], to investigate the natural convection inside a rectangular enclosure subjected to a boundary condition, heating from left sidewall and cooling from the top ceiling , while right sidewall and bottom wall were maintained at an adiabatic case. They reported that the Nusselt number was dependent upon the Rayleigh number and also analyzed the effect of the aspect ratio.

A detailed numerical analysis of three-dimensional natural convection from a horizontal discrete flush-mounted heater on the bottom of different aspect ratios had been analyzed by Sezai and Mohamad [7]. They reported that the variation of the Nusselt number depended upon the Rayleigh number and the aspect ratio of the heat source.The effect of variation of thermal boundary conditions at the sidewalls for the steady laminar natural convection in air-filled, two-dimensional rectangular enclosures was studied by Corcione [8]. They reported the results in some dimensionless correlation-equations. Aydin and Yang [9], numerically investigated the laminar natural convection of air in a square cavity with a localized heating from the bottom surface and symmetric cooling from the sidewalls for Reyleigh numbers in the range of 103 to 106. They reported that the increase in heating length or Reyleigh numbers caused an increase of the heat transfer rate and also mentioned that both flow and temperature fields were symmetrical about the mid-width of the cavity, due to the symmetrical boundary conditions. Sharif and Mohammad [10], used the same configuration as Aydin and Yang [9] to study the natural convection using a finite volume based computational method. They replaced a constant heat flux heating in the place of localized isothermal heat source at the bottom wall. They reported that for Grashof number below 104 , the heat transfer was dominated by diffusion, but for Grashof number greater than 105, buoyancy driven flow was dominant. It was also reported that the average Nusselt number was a function of the aspect ratio and the length of heat source. Türkoglu and Yücel [11], numerically investigated the natural convection inside a partially cooled and a partially heated square cavity.

Lakal [12], numerically studied the transient laminar natural convection in a square cavity heated from below periodically using the finite difference scheme. They reported that the heat transfer rate depended upon the period of the variable temperature and the amplitude. A numerical investigation had been carried out by Sarris et al. [13], to study the natural convection in an industrial glass melting tank which are heated locally from below. They concluded from the investigation that for low Rayleigh numbers, the rate of the heat transfer was dominated by the conduction, while at high Rayleigh numbers, it was dominated by the convection. It was also found that the flow circulation intensity and the temperature of the fluid increased with an increase in the tank aspect ratio and the heated strip width intensified. Lage and Bejan [14] investigated theoretically as well as numerically the natural convection inside an enclosures with pulsating heat flux boundary condition to one sidewall while the other sidewall was maintained at a constant lower temperature. They reported that at higher Reyleigh numbers, the flow due to the buoyancy had the tendency to resonant to the pulsating heat input. A numerical investigation had been carried out by Sarris et al. [15], to analyze the natural convection inside a 2-D enclosure with a sinusoidal temperature on the upper wall and an adiabatic boundary condition on the bottom and sidewalls. They showed that the local Nusselt number at the heated wall depended on the Rayleigh number, while the circulation strength in the cavity depended on the aspect ratio. Roy and Basak [16], studied numerically by using the Galerkin finite element method, the influence of non-uniformly heating of the walls on the natural convection flow in a square cavity. They observed that the rate of heat transfer was higher at the right edge of the bottom wall, uniform at the remaining part of the bottom wall and the same for the hot vertical wall for uniform heating case. They also stated that the rate of heat transfer was lower at the edges of the heated walls and higher at the center of both heated walls for non-uniform heating condition. Bilgen and Yedder [17] performed a numerical analysis to investigate the natural convection in a rectangular cavity with sinusoidal temperature profiles in a wall. They stated that the heat transfer rate was high when the heating source in the lower half of the enclosure. Cheikh et al. [18] had considered the natural convection numerically in air-filled 2D square, rectangular cavity heated partially from below and cooled from the top for a variety of thermal boundary conditions at the top and sidewalls. They showed their results in the form of streamline and isotherm plots.

Basak et al. [19] numerically studied the steady laminar natural convection in a square enclosure with uniformly and non-uniformly heated bottom wall and an adiabatic top wall. They reported that the sinusoidal non-uniform heating of the bottom wall produced greater heat transfer rates at the center of the bottom wall than the uniform heating case for all Rayleigh numbers. Numerical study had been carried out by Sathiyamoorthy et al. [20], to analyze the natural convection flow in a closed square cavity when an isothermal heat source on the bottom wall and linearly varying isothermal heated sidewalls. They found that a couple of stronger secondary circulations were formed at the lower portion of the cavity for high value of Rayleigh number. Schmidt et al. [21], compared the experimental and predicted results for laminar natural convection in a water filled enclosure. Zhong et al. [22], numerically investigated the laminar natural convection in a square enclosure using finite-difference method to determine the effects of variable properties on the temperature, velocity fields and the heat transfer rate in a differentially heated case.

Nithyadevi et al. [23], numerically investigated the effect of aspect ratio on the natural convection in a rectangular cavity with partially heating and cooling mountain to sidewalls. They found that, the heat transfer rate was high when the hot source was present in the bottom and the cold one was present in the top of the cavity. They also reported that the heat transfer rate depended upon the aspect ratio. Mahmud et al. [24], studied the natural convection in an enclosure with wavy sidewalls for a fluid with Prandalt number equals to 0.7. They showed their result in the form of streamlines and isothermal lines to present the flow pattern and thermal field inside the cavity and also reported that for a constant Grashof number and aspect ratio, the average heat transfer decreased with an increase of surface waviness up to a certain value of surface waviness, above which the average heat transfer increased again. Sharma et al. [25] numerically analyzed the two-dimensional conjugate turbulent natural convection and surface radiation in a rectangular cavity heated from below and symmetrical cooled from vertical sidewalls. They derived a correlation for the average Nusselt number in terms of Rayleigh number and aspect ratio for heating. Also, they reported the influence of the wall emissivity and the external heat transfer coefficient on the heat transfer from the cavity. Rout et al. [26] numerically investigated the performance of pulse tube refrigerator. Henkes et al.[27], investigated numerically the laminar and turbulent natural-convection flow in a 2-D enclosure heated from the vertical sidewalls for two different working fluids air and water. They used three different turbulence models and compared their results with an experimental result. It was found that the standard k-ε model gave a high prediction, whereas the low-Reynolds-number models were approximately close to the experimental result. Hsieh and Lien [28] investigated numerically the buoyancy-driven turbulent flows in enclosures using four different models and compared the average Nusselt number with experimental results. Markatos and Pericleous [29], performed a computational method to obtain solutions of the buoyancy-driven laminar and turbulent flows and heat transfer in a square enclosure with differentially heated sidewalls. They used (k ~ ε) two-equation turbulent model for Rayleigh numbers greater than 106.The results were presented in a tabular and graphical forms, and in a correlations of the Nusselt and Rayleigh numbers. Another useful reference can be found in [30-34].

Our review of literature has indicated that most of the published papers on this subject had not considered the problem of the natural convection in a complex cavity with an inclined triangular roof subjected to a uniform heat flux from its left sidewall and having an adiabatic solid strip. This paper tries to investigate this problem for the first time in much detail.

2. Mathematical formation

In the present investigation, the left sidewall of the cavity is supplied with a uniform heat flux (q) while the right sidewall is maintained at a constant cold temperature (T_c). Some assumptions are taken into account during the simulation as follows:-

a. Fluid is assumed to be incompressible and Newtonian in nature.
b. Two- dimensional flows inside the cavity and radiation is neglected.
c. Boussinesq approximation has been employed and the density in the buoyancy term, is assumed to vary with the temperature according to the following condition:

$$\rho = \rho_0 \left(1 - \beta \left(T - T_0\right)\right) \tag{1}$$

where $\beta = \left[\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right) \right]$

2.1. Problem definition

A two-dimensional rectangular cavity with vertical wall height (H), roof height (E) and bottom length (L) having a solid strip inside it at a distance of (L/2) as shown in Fig.1 is considered for simulation purpose.

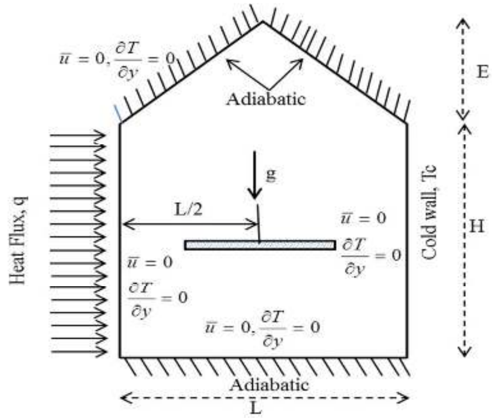


Fig. 1. Schematic physical configuration and boundary conditions of the present problem.

2.2. Governing equations

The governing equations solved during the simulation for the natural convection flow inside the domain are, conservation of mass, momentum and energy can be written as:

Continuity equation:

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial x}(\rho v)=0 \tag{2}$$

Momentum equation in x direction:

$$\rho\left[\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right]=-\frac{\partial \rho}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{3}$$

Momentum equation in y direction:

$$\rho\left[\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right]=-\frac{\partial \rho}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{4}$$

Energy equation:

$$\frac{\partial T}{\partial t}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\alpha\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right) \tag{5}$$

The dimensionless number affecting the heat transfer and fluid flow are, the Rayleigh number

$$Ra=\frac{g\beta qL^4}{k\nu\alpha} \tag{6}$$

While the Prandtl number is defined by:-

$$\Pr = \frac{\nu}{\alpha} \tag{7}$$

The average Nusselt number for the domain is given by [37]:-

$$Nu = 0.18\left[\frac{\Pr Ra}{0.2+\Pr}\right]^{0.29} \tag{8}$$

2.3.. Boundary condition

The simulation domain with boundary conditions is shown in Fig.1. The left sidewall of the domain is supplied with uniform heat flux boundary condition while the right sidewall is maintained at constant wall temperature and the rest top and bottom walls are considered adiabatic. The internal solid strip is also considered adiabatic in nature. The gravitational acceleration (g), which acts vertically downwards. Near vertical and horizontal walls for velocity components, no slip boundary condition is applied. The relevant non-dimensional boundary conditions used to solve the governing equations (Eqs. (2) - (5)) are given by:-

I. No-slip condition are applied at all cavity boundaries, i.e., u = v = 0

II. The right vertical sidewall (at x = 1) is subjected to isothermal cold temperature (T_c).

III. The left vertical side wall (at x = 0) is subjected to a uniform heat flux (q)

IV. The rest of the cavity walls and the internal solid strip are taken thermally insulated everywhere, i.e., dT/dy=0

3. Numerical Method

The commercial software package ANSYS 14 is used for the Computational Fluid Dynamics (CFD) solution approach. It includes the control volume generation to solve the governing equations with the corresponding boundary conditions as expressed in equations [1-4]. The coupling between pressure and velocity has been achieved using the SIMPLE algorithm by Patankar [35] with the PRESTO scheme for the pressure correction equation and the cell-face values of pressure could be obtained from simple arithmetic averaging of centroid values. Due to the presence of source term in momentum equation the pressure variation from cell to cell will not be smooth as a result SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm for pressure velocity coupling will not work properly. To avoid this phenomenon new pressure interpolation, PRESTO (Pressure Staggered Option) scheme technique is used. The PRESTO (Pressure Staggering Option) scheme uses the discrete continuity balance for a “staggered” control volume about the face to compute the “staggered” pressure. For triangular, tetrahedral, hybrid, and polyhedral meshes, a good accuracy is obtained using a similar algorithm. The advantages of the PRESTO scheme for pressure interpolation are available for all meshes in Fluent. Under relaxation factors of 0.3 for pressure, 0.8 for momentum and 1.0 for energy were used for convergence of all the considered variables. Triangular cells were used for the entire computational domain. The diffusion terms are discretized by the second order differencing scheme and convective term is discretized with the second order upwind scheme. The solution convergence criteria were set to 10^{-6}.

4. Grid independency Test

The simulation process was started from coarse mesh and subsequent increase of cell size produces the result as shown in Fig. 2. The domain is discretized by the use of unstructured triangular cells to deal with the triangular roof. The number of meshes for the domain is varied a total cell of 1700–148722. The air velocity along the vertical direction at the center position of horizontal direction is shown in Fig. 3 versus the number of cells. It is clearly visible that there is no change in vertical direction, velocity when the cell size changes from 6756 to 148722.

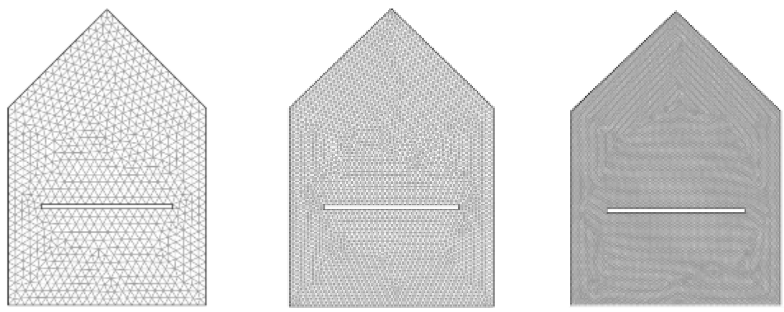


Fig. 2. Schematic of the different grids considered for grid sensitivity: (a)1700 (b) 6756 (c) 148722

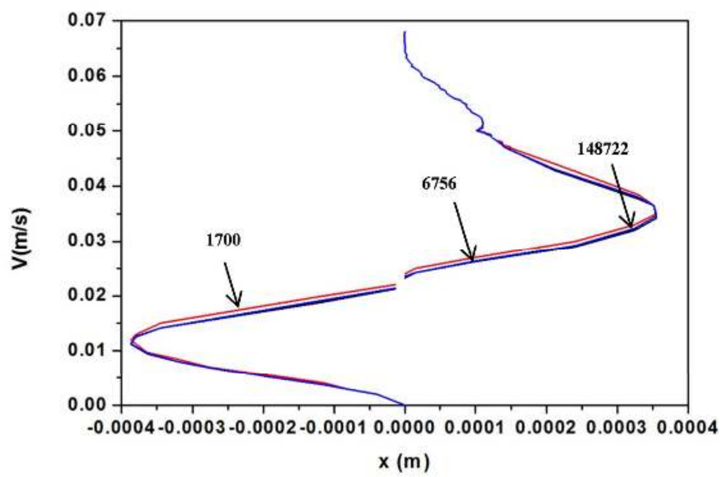


Fig. 3. Variation of y-velocity along the vertical distance.

5. Bench mark Comparison

The present numerical model is validated against the experimental data reported by Wu et al. [36] in terms of non-dimensional temperature distribution along different height of the enclosure. For this purpose, the present model is modified to a rectangular enclosure of (305mm x 305mm) dimension and constant temperature boundary condition as reported in Wu et al. [36]. The results of the validation analysis presented in Fig. 4 shows a good agreement between the two cases.

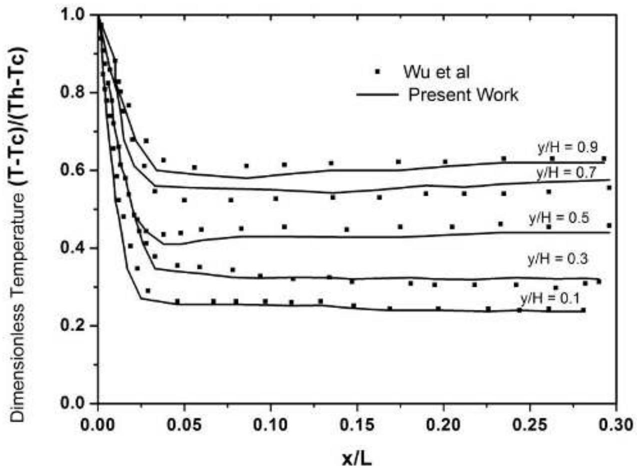


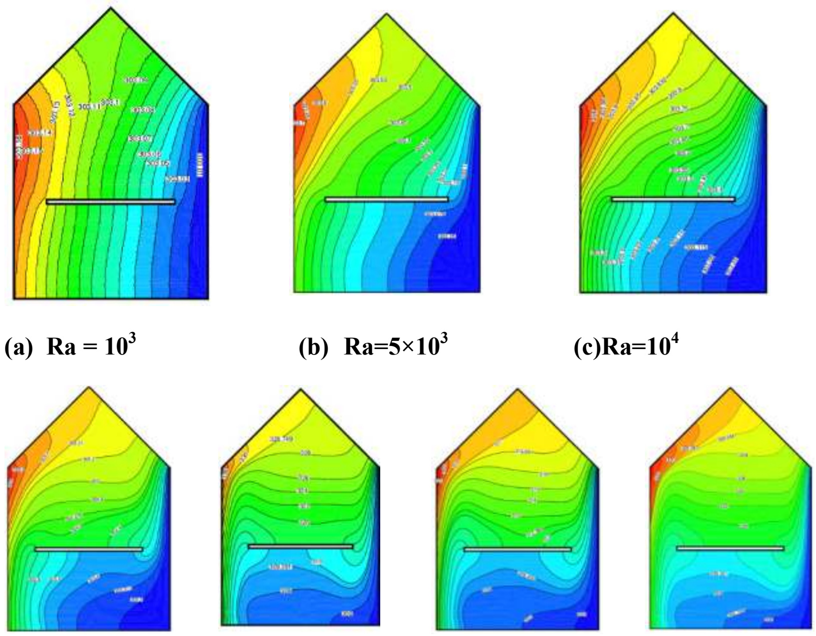
Fig. 4. Dimensionless temperature profile in cavity for validation with Wu et al.[36].

6. Results and Discussion

The control volume method used to solve the governing equations in the present work which is well described in **[35]**. It illustrates all discretized method and algorithm used in this investigation. A non-uniform unstructured triangular 6756 grid size arrangement is chosen in this work. The simulation has been carried out for different values of Rayleigh number from (10^3 to 10^6) at a constant aspect ratio for a triangular roof cavity. The entire simulation is performed for a fixed Prandtl number at 0.7 by taking air as a working fluid. The main emphasis is given on the influence of a constant heat flux as one boundary condition while the one adiabatic solid strip is present at middle portion of the cavity to affect the buoyancy driven flow pattern inside it. The validation of the present numerical model is shown in Fig. 4, which is compared with Wu et al. [36]. This comparison shows a good validation of our model. Therefore, a comparative numerical study was carried out to investigate the flow and thermal fields together with the heat transfer rate in the form of Nusselt numbers.

6.1. Variation of temperature inside the domain at various Rayleigh number.

Temperature fields are presented with the variation of Rayleigh number due to the presence of a solid adiabatic strip (SAS) by means of isotherms as shown in Fig. 5. The flow is characterized by Rayleigh number which is a measure of the convection intensity. It is found that the temperature distributions are strongly dependent on the Rayleigh number. When the Rayleigh number is low (i.e., Ra = 10^3) , the isotherms are parallel to the vertical sidewalls below the SAS .While , above the SAS isotherms are gradually becoming diverge towards the top portion. The reason of this behaviour is due to the presence of SAS. From the other hand, the bottom region remains at low temperature comparison to the upper one. Again, the reason of this behaviour is due to the existence of SAS. At high Rayleigh number (i.e., Ra = 10^6) the isotherms are parallel to each other in the horizontal direction above the SAS but not parallel below the SAS. The reason of this behaviour is due to the strong effect of convection when the Rayleigh number is high. The heat transfer rate is higher above the SAS due to the high temperature difference. Moreover, during high Rayleigh number the temperature inside the cavity increases but due to the presence of SAS, it can be clear from the analysis that the upper portion has a higher temperature than the lower one for the same reason explained above.



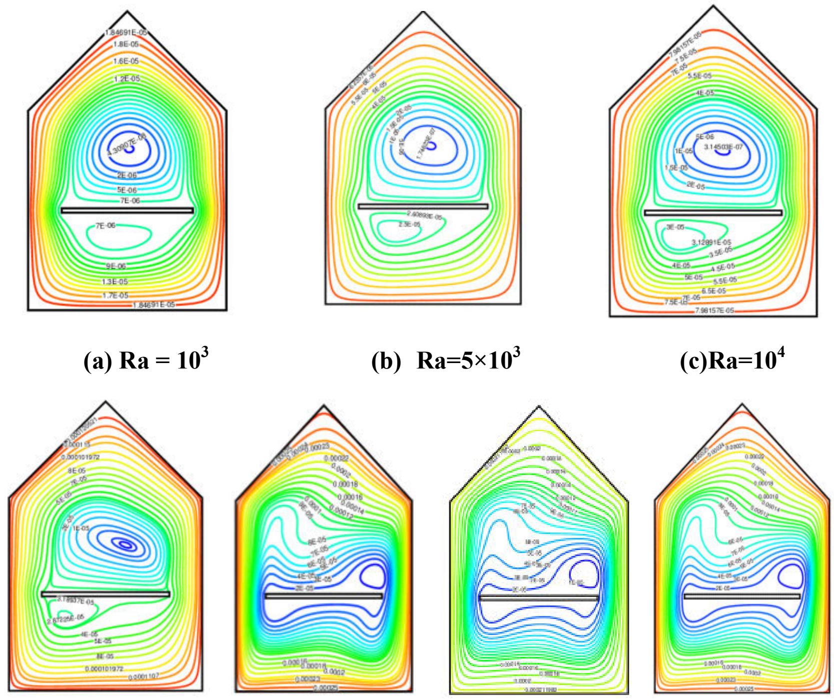
(a) Ra = 10^3 **(b) Ra=5×10^3** **(c)Ra=10^4**

(d) Ra=5×10^4 **(e) Ra=10^5** **(f) Ra=$5\text{x}10^5$** **(g) Ra=10^6**

Fig. 5. Temperature profiles in the cavity for different Rayleigh number.

6.2. Variation of stream function inside the domain due to Rayleigh number effect

The stream function vortices are shown in Fig 6. From this figure, multiple vortices can be noticed inside the cavity. As expected, because the heat flux is subjected to the left sidewall, the air adjacent to this wall will be heated and then begins to move upward due to the decreasing in its density. From the other hand, the air adjacent to the cold right sidewall which has a large density begins to move in the downward direction. This repeated opposite movement of air creates a column of hot and cold air that move upward and downward inside the cavity leading to produce these multiple vortices. From this figure, it is clearly observing that the intensity of circulation is weak for a small Rayleigh number (Ra = 10^3).This is due to the great role of viscous force and the weak role of buoyancy force.



(a) Ra = 10^3 **(b) Ra=5×10^3** **(c)Ra=10^4**

(d) Ra=5×10^4 **(e) Ra=10^5** **(f) Ra=$5\text{x}10^5$** **(g) Ra=10^6**

Fig. 6. Stream functions in the cavity for different Rayleigh number.

With the growth of the Rayleigh number (Ra = 10^6), there is a clear increase in the circulation intensity due to high buoyancy effect. In this case, the viscous force becomes slight. Also, a large temperature gradient can be noticed near the hot left sidewall due to the strong fluid motion coming from the higher temperatures near the hot left sidewall where a uniform heat flux exists which causing a large contribution of convection heat transfer. Moreover, it can be seen from this figure that the circulating vortices are compressed near the solid adiabatic strip SAS making a clear confusion in the flow vortices and then moves away from it causing a fluid to be compressed significantly. Therefore, a SAS can be used to damp the strong flow circulation. It can be seen also from the results that at low Rayleigh number (i.e., Ra = 10^3), circular vortices are generated away from the solid adiabatic strip SAS, both below and above of it. Gradually, flow vortices move nearer to the SAS as the Rayleigh number increases. The stream function value is higher nearer to the wall of the cavity and lower nearer the SAS. This behaviour is due to the adiabatic boundary condition of the SAS which leads to reduce the values of the stream function at this place.

6.3. Variation of vertical velocity inside the domain due to effect of Rayleigh number

The results obtained in the form of velocity magnitude are shown in Fig. 7. It shows that as the Rayleigh number increases the velocity magnitude increases inside the cavity. This is due to the increase in the flow circulation as the Rayleigh number increases. Also, it can be observed from Fig. 7, that due to the presence of solid adiabatic strip SAS, the velocity of fluid inside the cavity is low compared to the sidewalls region. This behavior is due to the adiabatic boundary condition of SAS which leads to reduce the values of the stream function at this place. It also concluded that the fluid has higher velocity magnitude near the cold sidewall compared to the hot one.

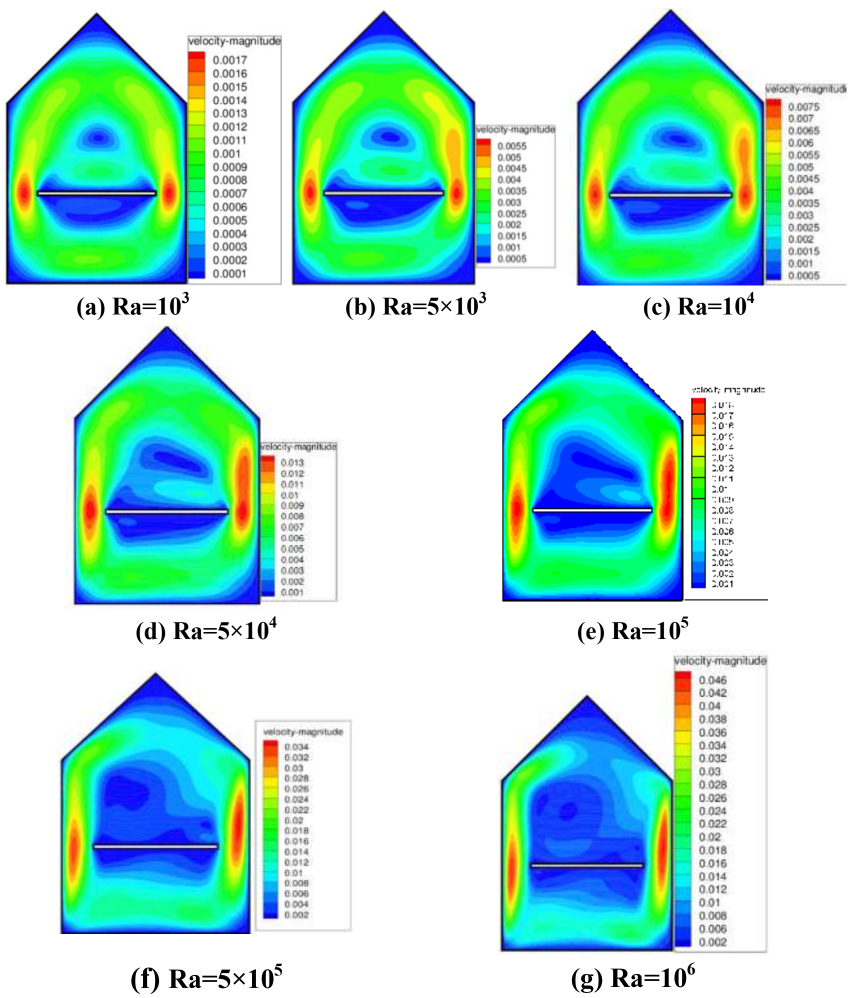


Fig. 7. The velocity magnitude contours for different Rayleigh number.

6.4. Variation of average Nusselt number inside the domain due to effect of Rayleigh number

Figure 8 shows the average Nusselt number variation as a function of the Rayleigh number with the presence of a solid adiabatic strip SAS in the middle of the cavity. It shows that when the Rayleigh number increases the average Nusselt number also increases. Moreover, the average Nusselt number is small for the low Rayleigh number (i.e., Ra = 10^3), referring that the heat transfer occurs mainly due to conduction and begins to increase as the Rayleigh number increases. This is due to the fact that the convection phenomena are strongly enhanced with the Rayleigh number increasing.

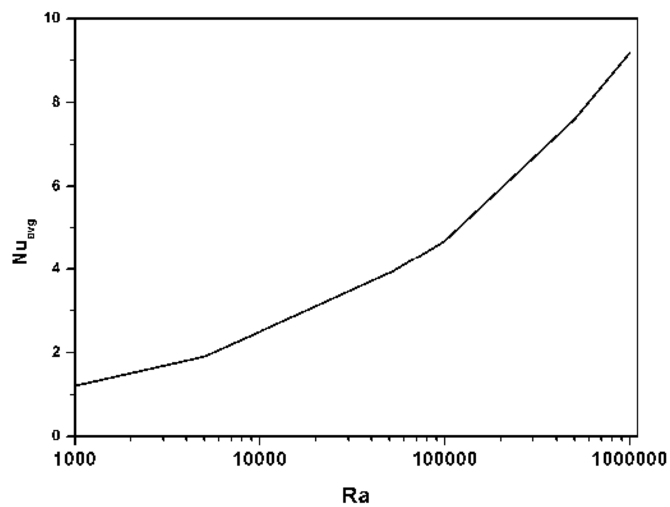


Fig. 8. The average Nusselt number as a function of the Rayleigh number.

7. Conclusions

The present numerical work investigates the natural convection inside rectangular cavity which has a top wall with a triangular roof shape and when a SAS inserted inside it. Here, the working fluid considered is air. The aim of this study is to investigate the fluid flow behavior and heat transfer phenomenon due to the presence of the SAS inside a rectangular cavity with one sidewall subjected to heat flux boundary condition. The sidewall maintained at constant cold temperature.

- It is concluded that due to the presence of the solid adiabatic strip, the lower portion of the cavity remains very cold compared to the upper portion.
- The flow vortices move nearer to the SAS as the Rayleigh number increases.
- The particles have higher kinetic energy near the hot surface compared to the case of the particles near the cold surface. From the other hand, the narrow area between the walls and the strip shows a high velocity.

- Also, when the Rayleigh number increases the velocity magnitude increases inside the cavity.
- The results indicated the usual fact that when the Rayleigh number increases the average Nusselt number also increases.
- In addition, the model is found to be in a good agreement with the previous experimental work.

References

1. Ostrach, S. (1972). Natural convection in enclosures. *In Advances in Heat Transfer, P.H. James and F.I. Thomas, Editors, Elsevier*, 161-227.
2. Ostrach, S. (1988). Natural convection in enclosures. *Journal of Heat Transfer*, 110(4b), 1175-1190.
3. Ostrach, S. (1982) Low-gravity fluid flows. *Annual Review of Fluid Mechanics*. 14(1), 313-345.
4. Hoogendoom, C.J. (1986). Natural convection in enclosures. *Proceedings of the Eighth International Heat Transfer Conference, (San Francisco, Hemisphere publishing Corp, Washington, DC)*, 111-120.
5. Ganzarolli, M.M.; and Milanez, L.F. (1995). Natural convection in rectangular enclosures heated from below and symmetrically cooled from the sides. *International Journal of Heat and Mass Transfer*, 38(6), 1063-1073.
6. Aydin, O.; Ünal A.; and Ayhan, T. (1999). Natural convection in rectangular enclosures heated from one side and cooled from the ceiling. *International Journal of Heat and Mass Transfer*, 42(13), 2345-2355.
7. Sezai, I.; and Mohamad, A.A. (2000) Natural convection from a discrete heat source on the bottom of a horizontal enclosure. *International Journal of Heat and Mass Transfer*, 43(13), 2257-2266.
8. Corcione, M. (2003). Effects of the thermal boundary conditions at the sidewalls upon natural convection in rectangular enclosures heated from below and cooled from above. *International Journal of Thermal Sciences*, 42(2), 199-208.
9. Aydin, O.; and Yang, W.J. (2003). Natural convection in enclosures with localized heating from below and symmetrical cooling from sides. *International Journal of Numerical Methods for Heat & Fluid Flow*, 10(5), 518-529.
10. Sharif, M.A.R.; and Mohammad, T.R. (2005). Natural convection in cavities with constant flux heating at the bottom wall and isothermal cooling from the sidewalls. *International Journal of Thermal Sciences*, 44(9), 865-878.
11. Türkoglu, H.; and Yücel, N. (1995). Effect of heater and cooler locations on natural convection in square cavities. *Numerical Heat Transfer, Part A: Applications*, 27(3), 351-358.
12. Lakhal, E. K.; Hasnaoui, M.; Vasseur, P.; and Bilgen, E. (1995). Natural convection in a square enclosure heated periodically from part of the bottom wall. *Numerical Heat Transfer, Part A: Applications*, 27(3), 319-333.

13. Sarris, I.E.; Lekakis, I.; and Vlachos, N.S. (2004). Natural convection in rectangular tanks heated locally from below. *International Journal of Heat and Mass Transfer*, 47(14–16), 3549-3563.
14. Lage, J.L.; and Bejan, A. (1993). The resonance of natural convection in an enclosure heated periodically from the side. *International Journal of Heat and Mass Transfer*, 36(8), 2027-2038.
15. Sarris, I.E.; Lekakis, I. and Vlachos, N.S. (2002). Natural convection in a 2d enclosure with sinusoidal upper wall temperature. *Numerical Heat Transfer, Part A: Applications*, 42 (5), 513-530.
16. Roy, S.; and Basak, T. (2005). Finite element analysis of natural convection flows in a square cavity with non-uniformly heated wall(s). *International Journal of Engineering Science*, 43(8–9), 668-680.
17. Bilgen, E.; and Yedder, R.B. (2007). Natural convection in enclosure with heating and cooling by sinusoidal temperature profiles on one side. *International Journal of Heat and Mass Transfer*, 50(1–2), 139-150.
18. Cheikh, N.B.; Beya, B.B.; and Lili, T. (2007). Influence of thermal boundary conditions on natural convection in a square enclosure partially heated from below. *International Communications in Heat and Mass Transfer*, 34(3), 369-379.
19. Basak, T.; Roy, S.; and Balakrishnan, A.R. (2006). Effects of thermal boundary conditions on natural convection flows within a square cavity. *International Journal of Heat and Mass Transfer*, 49(23–24), 4525-4535.
20. Sathiyamoorthy, M.; Basak, T.; Roy, S.; and Pop, I. (2007). Steady natural convection flows in a square cavity with linearly heated side wall(s). *International Journal of Heat and Mass Transfer*, 50(3–4), 766-775.
21. Schmidt, F.W; Giel, P.W.; Phillips, R.E.; Wang, D.F. (1986). A comparison of experimental and predicted results for laminar natural convection in an enclosure. *International Journal of Heat and Fluid Flow*, 7(3), 183-190.
22. Zhong, Z.Y.; Yang, K.T.; and Lloyd, J.R. (1985). Variable Property effects in laminar natural convection in a square enclosure. *Journal of Heat Transfer*, 107(1), 133-138.
23. Nithyadevi, N.; Kandaswamy, P.; and Lee, J. (2007). Natural convection in a rectangular cavity with partially active side walls. *International Journal of Heat and Mass Transfer*, 50(23–24), 4688-4697.
24. Mahmud, S.; Das, P.; Hyder, N.; and Islam A. S. (2002). Free convection in an enclosure with vertical wavy walls. *International Journal of Thermal Sciences*, 41(5), 440-446.
25. Sharma, A.K.; Velusamy, K.; Balaji, C.; and Venkateshan, S. (2007). Conjugate turbulent natural convection with surface radiation in air filled rectangular enclosures. *International Journal of Heat and Mass Transfer*, 50(3–4), 625-639.
26. Rout, S.K.; Choudhury, B.K.; Sahoo, R.K.; and Sarangi, S.K.; (2014) Multi-objective parametric optimization of Inertance type pulse tube refrigerator using response surface methodology and non-dominated sorting genetic algorithm. *Cryogenics*, 62, 71-83.

27. Henkes, R.A.W.M.; Vlugt, F.F.V.D.; and Hoogendoorn, C.J. (1991). Natural-convection flow in a square cavity calculated with low-Reynolds-number turbulence models. *International Journal of Heat and Mass Transfer*, 34(2), 377-388.
28. Hsieh, K.J.; and Lien, F.S. (2004). Numerical modeling of buoyancy-driven turbulent flows in enclosures. *International Journal of Heat and Fluid Flow*, 25(4), 659-670.
29. Markatos, N.C.; and Pericleous, K.A. (1984). Laminar and turbulent natural convection in an enclosed cavity. *International Journal of Heat and Mass Transfer*, 27(5), 755-772.
30. Omri, M.; and Galanis, N. (2007). Numerical analysis of turbulent buoyant flows in enclosures: Influence of grid and boundary conditions. *International Journal of Thermal Sciences*, 46(8), 727-738.
31. Calcagni, B.; Marsili, F.; and Paroncini, M. (2005). Natural convective heat transfer in square enclosures heated from below. *Applied Thermal Engineering*, 25(16), 2522-2531.
32. Rout S. K.; Mishra, D. P.; Thatoi, D.N.; and Mishra, A. K. (2012). Numerical analysis of mixed convection through an internally finned tube, *Advances in Mechanical Engineering*.
33. Emery, A.F.; and Lee, J.W. (1999). The effects of property variations on natural convection in a square enclosure. *Journal of Heat Transfer*, 121(1), 57-62.
34. Newell, M.E.; and Schmidt, F.W. (1970). Heat transfer by laminar natural convection within rectangular enclosures. *Journal of Heat Transfer*, 92(1), 159-167.
35. Patankar, S.V. (1980). *Numerical Heat Transfer and Fluid Flow*. McGraw-Hill, New York.
36. Wu, W.; Ewing, D.; and Ching, C.Y. (2006). The effect of the top and bottom wall temperatures on the laminar natural convection in an air-filled square cavity. *International Journal of Heat and Mass Transfer*, 49(11–12), 1999-2008.
37. Arnold, J. N.; Catton, I.; and Edwards, D. K. (1975). Experimental Investigation of natural convection in inclined rectangular region of differing aspects ratios. *ASME*, 75-62.