International Journal of Science and Research (IJSR)
ISSN: 2319-7064
ResearchGate Impact Factor (2018): 0.28 | SJIF (2018): 7.426

A Study of Enhancement of Natural Convection Heat Transfer in a Square Enclosure with Localized Heating from Above

Edwin Nyagwonda Jephiter[1], Johana Kibet Sigey[2], Jeconia Okelo Abonyo[3], P. Roy Kiogora[4]

[1, 2, 3, 4]Department of Pure and Applied Mathematics, Jomo Kenyatta University of Agriculture and Technology, Nairobi, Kenya

Abstract: ***A numerical study of natural convection of heat transfer in a three dimensional square cavity is considered. The two opposite vertical walls and the bottom wall have been considered cold constant temperature with a source of heat fixed at the corner top surface and the non heated part of the top surface considered insulated. The study is aimed at examining the velocity flow and temperature distribution. The central finite difference method is used in solving the energy and momentum equations. The differential equations are solved by the central difference method and the forward difference method .The solutions are presented at various Reynolds number, Froude number, Eckert number with constant Prandtl number 0.71. When varying Re number, the velocity is seen to decrease, while when the Fr number is varied, the velocity decreases with increase in room depth. The behavior of the flow fields are analyzed by 2D graphs. When the Fr number is varied, the velocity decreases with increase in room depth. When the Re number is varied for both horizontal and vertical velocity, the velocity increases with increase in room depth. It also seen that an increase in Eckert number results into an increase in temperature distribution.***

Keywords: Central difference method, Reynolds number, Froude number, Eckert number and Prandtl number

1. Introduction

The study of natural convection in enclosure cavities has been extensive both experimentally and numerically. Its importance is felt in many engineering applications such as solar energy systems, air conditioning, heat transfer in buildings, heat removal in micro-electronics, and cooling of nuclear reactors and many others. The cooling of electronic components is essential for their reliable operation. Therefore, the enhancement of heat transfer is an important subject in an engineering field. Heat transfer involves flow of heat and temperature. Heat flow is the movement of thermal energy from one point to another due to temperature difference. Temperature is the amount of thermal energy available. A fluid is any substance that flows. Heat transfer from one point to another by the movement of fluids is known as convection. Convection is usually the dominant form of heat transfer in fluids. This can be categorized as free convection, natural convection or forced convection. Natural convection is when a fluid motion is caused by buoyancy .Buoyancy forces are induced from density changes caused by temperature gradient in the fluid region close to heat transfer surface. The molecules of the fluid separate and scatter causing the fluid to be less dense. The hotter fluid rises and the cooler fluid gets denser and it sinks. Forced convection is when a fluid is forced to flow over the surface by an external source such as fans and pumps creating an artificially induced convection current. The difference in temperature brings about the changes in cooling and heating effects of a fluid. The boiling of water for instance is due to convection currents. The higher the velocity of the flowing fluid, the higher the rate of heat transfer rate. In case of combined convection, one could often like to know how much of the convection is due to external constraints, such as the fluid velocity and how much is due to free convection occurring in the system. In the present work, natural convection of fluid that takes place in a square cavity with localized heating from horizontal top surface is considered. The source of heat is placed at one corner of the top surface. The bottom and the two vertical opposite walls are considered cold, while, the other vertical walls and part of the top surface are considered insulated.

1.1 Geometry of the problem

The fig.1 below shows the source of heat and the two opposite vertical cold walls and the bottom cold surface .Convection is investigated in a room, 1m long, 1m width and 1m height. The heater is placed at the corner of the top surface and opposite vertical walls and bottom wall kept at cold constant temperature.

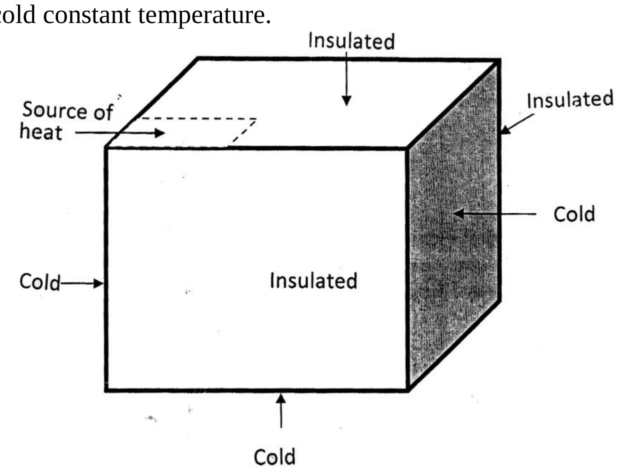

Figure 1: Geometry of the problem

2. Literature Review

Over the years, many numerical and experimental studies have been performed and quite interesting results have been submitted. A natural convection heat transfer experiment in a tall vertical rectangular enclosure (aspect ratio 16.5) with an array of eleven discrete flush heaters has been done by Keyhani *et al.* (1988) It was found that the discrete heating in the enclosure results in a significantly augmented local heat transfer rate over that for an enclosure with the uniformly heated vertical wall. Aydin and Yang (2000) numerically investigated the natural convection of air in a vertical square cavity with localized isothermal heating from below and symmetrical cooling from sidewalls. The top wall as well as non-heated parts of the bottom walls was varied. Two counter rotating vertices were formed in the flow domain due to natural convection. The average Nussselt number at the heated part of the bottom wall was shown to increase with increasing Rayleigh number as well as with increasing length of the heat source. The work done by Sigey *et al*.(2011) who both investigated in detail turbulent flow in a three dimensional enclosure in the form of a room with a convectional heater built into one of the walls and having a window in the same wall. The size of the window was varied to pave way for two cases of study but centre fixed with respect to the heater. The result indicated that the rate of heat transfer is higher for a larger window than for a small window as the Rayleigh number increases. Deng *et al*. (2002) studied numerically a two dimensional lamina natural convection in a rectangular enclosure with discrete heat sources on walls. A new combined temperature scale was suggested to non-dimensionalize the governing equations of natural convection induces by multiple temperature difference, the Rayleigh numbers used were Ra = 10^3 to 10^6. Ampofo *et al.* (2003) conducted an experiment study of low-level turbulence natural convection in an air filled vertical square cavity. The cavity was 0.7m high × 1.5 deep giving rise to a 2D flow. The hot and cold walls of the cavity were isothermal at 50^0 and 10 ^{0}c respectively, that is, a Rayleigh number equals to 1.58 × 10^9. The experiments that were carried out on Ampofo work and Karayiannis (2003) were conducted with very high accuracy and as such the results formed experimental benchmark data and were useful for validation of computational fluid dynamics codes. Calcagni *et al*. (2005) studied the natural convection heat transfer numerically and experimentally in square enclosure heated from below. The study was focused on the calculation of heat and average Nusselt number on the heat source. Martorell *et al.,* (2003) work dealt with the natural convection flow and heat transfer from horizontal plate cooled from above experiments were carried out for rectangular plated leaving aspect ratios between$\emptyset$ =0.36 and 0.43 and Rayleigh numbers in the range of 290 ≤ Raw ≤ 3.3× 10^5. The values of Raw and $\emptyset$ were selected, to the design of printed circuit boards. The results showed that such a low raw effect could be accounted for in a physically consistent manner by adding a constant term to the heat transfer correlation. Massimo *et al.* (2008) studied the natural convective heat transfer generated by a source with a height located in two different positions inside a square enclosure. A natural convection in a square enclosure with a hot source experiment with three different heights was carried out by Corvaro *et al.* (2009). The height of the strip influenced the distribution of the velocity fields, and consequently the heat transfer efficiency. The study shows how the natural convective heat transfer worsens with the increase in the source height. Ahmed *et al.,* (2001), performed experiment on natural convection in a trapezoidal enclosure with partial heating from below and symmetrical cooling from the sides. The heating was simulated by a centrally located heat source on the bottom wall and four different values of the dimensionless source length 0.2, 0.4, 0.6, 0.8 were considered. The results show that the average Nusselt number increases with the increases on the source length. Patrick *et al*. (2011) performed experiment on a numerical study of the effect of a below-window convective heater on the heat transfer rate from a cold recessed window. Convective heaters are often mounted below a cold window in buildings in cold claimants. The presence of the heater alters the flow and temperature distributions in the room near the window and the rate of convective heat transfer to the window. The result shows that when the heater output is low the cold downward flow from the window reaches the floor leading to a cold air layer forming near the floor. At a higher heater output the hot upward flow from the heater is strong to divert the cold flow from the window away from the floor. The work by Sigey *et al.* (2013) investigated the enhancement of convective heat transfer in a square enclosure with localized heating from below has been done. In their work the numerical study of natural convection of heat transfer in a three dimensional square cavity was considered. The two opposite vertical walls and the top wall were considered cold constant temperature with source of heat fixed at the middle bottom surface while the other two vertical walls and the non-heated part of the bottom surface were considered insulated. In their work the temperature decreased as the fluid moved from the source of heat towards the cold walls. Numerical study undertaken by Robins *et al*, (2014) of Mixed Convection Flow Inside Ventilated Enclosure with bottom wall uniformly heated, two vertical walls maintained at constant cold temperature and top wall insulated indicated that the strength of circulation increases with the increase in value of Richardson number irrespective of the Reynolds number and Prandtl number and as the value of Richardson number increases, there occurs a transition from conduction to convection dominated flow at Richardson number 1. Momanyi *et al.* (2015) recently studied the Effect of Forced Convection on Temperature Distribution and Velocity Profile in a Room and the results showed that temperature decreases with increase in room height also the velocity decreases as fluid particles flow up the room. Okewa *et al*. (2017) recently studied Forced convection in a three dimensional rectangular enclosure was considered with heaters placed on the opposite wall on the y-z planes, two windows on the adjacent opposite walls on x-z planes and one fan centrally fixed at the top (ceiling) x-y plane. The fan was set to rotate, the speed was varied. Results showed that temperature increases with increase in room depth. As the fan speed increases, temperature increases with increase in room depth at a lower rate. However the rate of increase in temperature is higher with increase in the room's depth. At any particular room depth, temperature is higher at lower Reynolds number and lower at high Reynolds number. Temperatures within the room are generally lower when the fan speed is increased. With respect to velocity profile, velocity of air within the room decreases with decrease in rooms' depth. The rate at which velocity decreases is higher at lower Reynolds number. As the fan speed decreases the rate at which velocity decreases lowers. Results also indicate that velocity is lower directly beneath the fan. The lowest velocity is registered when Reynolds number is high and highest at low Reynolds number. Due to little focus by researchers on localized heating from above, it then attracts our present study. In this work investigation of flow field and temperature particularly with the source of heat placed at the top wall and on one of the corners is done. The heat creates convection current and a study is thus done.

3. Methodology

3.1 The governing equations

The general governing equations governing the model, that is, continuity, momentum and energy equations are described below.

3.2 Continuity equation

A continuity equation is a differential equation that describes the transport of some kind of conserved quantity, in particular-mass. According to Curie (1984), (assuming the fluid density is constant), the continuity equation can be written as;

$$\frac{\partial u_x}{\partial x}+\frac{\partial v_y}{\partial y}=0 \qquad (1)$$

Where x and y are the distances measured along the horizontal and vertical directions respectively, u and v are the velocity components in x and y direction respectively.

3.3 Momentum equation

Momentum equations describe the motion of fluid substances. These equations are derived from the Newton's second law of motion which states that the sum of the body and surface forces acting on a system is equal to the rate of change of linear momentum of the system. The momentum equation in rectangular coordinates (x, and y) is given by:

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

The y-component momentum equation in a dimensionalised form is:

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

3.4 Energy equation

The energy equation is derived from the first law of thermodynamics which states that the rate of energy increase in a system is equal to the heat added to the system and the work done on the system. The energy equation is derived from the first law of thermodynamics which states that the rate of energy increase in a system is equal to the heat added to the system and the work done on the system. From Currie (1974) assuming no external heat source, the energy equation is often written as;

$$u\frac{\partial\theta}{\partial x}+v\frac{\partial\theta}{\partial y}=\frac{1}{\mathrm{Pr}}\frac{\partial^2\theta}{\partial x^2}+E_c\left(\frac{\partial u}{\partial y}\right)^2 \qquad (4)$$

3.5 Method of solution

The finite difference method is used to solve the continuity, energy and momentum equations and proper boundary conditions will be applied numerically to achieve the results. We develop the central difference numerical scheme for both energy and momentum equations. In this study the main parameters of interest are the Reynolds, Froude number and Eckert number. The Prandtl number is taken as 0.71 for air. Vector plots are used to study the effects of the parameters on fluid flow and heat transfer.

3.6 Effects of Reynolds number on vertical velocity

The momentum governing equations are discretized in y component. Using explicit numerical scheme, u_y is replaced by forward difference while u_{xx} and u_{yy} are replaced by explicit difference in (3).Then equation (3) becomes

$$\left[\frac{V_{i,j+}-V_{i,j}}{(\Delta y)}\right]=\frac{1}{(Fr)^2}\frac{1}{\mathrm{Re}}\left[\frac{V_{i+1,j}-2V_{i,j}+V_{i-1,j}}{(\Delta x)^2}+\frac{V_{i,j+1}-2V_{i,j}+V_{i,j-1}}{(\Delta y)^2}\right] \qquad (5)$$

Taking, $\Delta x=\Delta y=0.01$ into (5) we have the scheme

$$\left(Fr^2\,\mathrm{Re}-100\right)V_{i,j+1}+\left(400-Fr^2\,\mathrm{Re}\right)V_{i,j}-100V_{i,j-1}=100V_{i+1,j}+100V_{i-1,j} \qquad (6)$$

Taking and i = 1,2…….6 and j = 1 in (6) we form the following systems of linear algebraic equations

$$\left.\begin{array}{l}
\left(Fr^2\,\mathrm{Re}-100\right)V_{1.2}+\left(400-Fr^2\,\mathrm{Re}\right)V_{1,1}-100U_{1,0}=100V_{2,1}+100V_{0,1}\\
\left(Fr^2\,\mathrm{Re}-100\right)V_{1.3}+\left(400-Fr^2\,\mathrm{Re}\right)V_{1,2}-100U_{1,1}=100V_{3,1}+100V_{1,1}\\
\left(Fr^2\,\mathrm{Re}-100\right)V_{1.4}+\left(400-Fr^2\,\mathrm{Re}\right)V_{1,3}-100U_{1,2}=100V_{4,1}+100V_{2,1}\\
\left(Fr^2\,\mathrm{Re}-100\right)V_{1.5}+\left(400-Fr^2\,\mathrm{Re}\right)V_{1,4}-100U_{1,3}=100V_{5,1}+100V_{3,1}\\
\left(Fr^2\,\mathrm{Re}-100\right)V_{1.6}+\left(400-Fr^2\,\mathrm{Re}\right)V_{1,5}-100U_{1,4}=100V_{6,1}+100V_{4,1}\\
\left(Fr^2\,\mathrm{Re}-100\right)V_{1.7}+\left(400-Fr^2\,\mathrm{Re}\right)V_{1,6}-100U_{1,5}=100V_{7,1}+100V_{5,1}
\end{array}\right\} \qquad (7)$$

With initial and boundary conditions V(x,0) = 1 , V(0,y) = 0 and V(x,y) = 0 respectively substituted in (7), we obtain the matrix-vector equation;

$$\begin{bmatrix}
\left(400-\mathrm{Re}\,Fr^2\right) & \left(Fr^2\,\mathrm{Re}-100\right) & 0 & 0 & 0 & 0\\
100 & \ddots & \ddots & 0 & 0 & 0\\
0 & 100 & \ddots & \left(Fr^2\,\mathrm{Re}-100\right) & 0 & 0\\
0 & 0 & 100 & \left(400-\mathrm{Re}\,Fr^2\right) & \ddots & 0\\
0 & 0 & 0 & 100 & \ddots & \left(Fr^2\,\mathrm{Re}-100\right)\\
0 & 0 & 0 & 0 & 100 & \left(400-\mathrm{Re}\,Fr^2\right)
\end{bmatrix}
\begin{bmatrix} V_{11}\\ V_{1,2}\\ V_{1,3}\\ V_{1,4}\\ V_{1,5}\\ V_{1,6}\end{bmatrix}
=\begin{bmatrix}100\\100\\100\\100\\100\\100\end{bmatrix} \qquad (8)$$

3.8 Effects of Eckert number on fluid temperature

The energy Equation (4) is discretized to study the effects of Eckert number for temperature profiles. Using a central difference numerical scheme, we get

$$U\frac{\theta_{i+1,j}-\theta_{i-1,j}}{2\Delta x}+V\frac{\theta_{i,j+1}-\theta_{i,j-1}}{2\Delta y}=\frac{1}{\Pr}\left[\frac{\theta_{i+1,j}-2\theta_{i,j}+\theta_{i-1,j}}{(\Delta x)^2}\right]+E_C\left(\frac{U_{i,j+1}-U_{i,j-1}}{2\Delta y}\right)^2 \tag{9}$$

We investigate the effect of E_c, on the fluid temperature. We take Pr = 0.71 since the fluid is air. Taking $\Delta x=\Delta y=0.01$ and V=1, U=1, we get the scheme

$$-284\theta_{i+1,j}+571\theta_{i,j}-286\theta_{i-1,j}=\theta_{i,j}-\theta_{i,j+1}+50E_c\left(u_{i-1,j}^2\right) \tag{10}$$

Taking i=1,2,3…….8 and j=1 we form the following systems of linear algebraic equations

$$\begin{aligned}
-284\theta_{2,1}+571\theta_{1,1}-286\theta_{0,1}&=\theta_{1,1}-\theta_{1,2}+50E_c\left(u_{0,1}^2\right)\\
-284\theta_{3,1}+571\theta_{2,1}-286\theta_{1,1}&=\theta_{2,1}-\theta_{2,2}+50E_c\left(u_{1,1}^2\right)\\
-284\theta_{4,1}+571\theta_{3,1}-286\theta_{2,1}&=\theta_{3,1}-\theta_{3,2}+50E_c\left(u_{2,1}^2\right)\\
-284\theta_{5,1}+571\theta_{4,1}-286\theta_{3,1}&=\theta_{4,1}-\theta_{4,2}+50E_c\left(u_{3,1}^2\right)\\
-284\theta_{6,1}+571\theta_{5,1}-286\theta_{4,1}&=\theta_{5,1}-\theta_{5,2}+50E_c\left(u_{4,1}^2\right)\\
-284\theta_{7,1}+571\theta_{6,1}-286\theta_{5,1}&=\theta_{6,1}-\theta_{6,2}+50E_c\left(u_{5,1}^2\right)
\end{aligned} \tag{11}$$

Taking the initial and boundary conditions $\theta(0,y)=1$,$=\theta(x,0)=1$, the above system of algebraic equation becomes

$$\begin{bmatrix} 571 & -284 & 0 & 0 & 0 & 0\\ -286 & 571 & -284 & 0 & 0 & 0\\ 0 & -286 & 571 & -284 & 0 & 0\\ 0 & 0 & -286 & 571 & -284 & 0\\ 0 & 0 & 0 & -286 & 571 & -284\\ 0 & 0 & 0 & 0 & -286 & 571 \end{bmatrix}\begin{bmatrix}\theta_{1,1}\\ \theta_{2,1}\\ \theta_{3,1}\\ \theta_{4,1}\\ \theta_{5,1}\\ \theta_{6,1}\end{bmatrix}=\begin{bmatrix}287+50E_c\\ 287+50E_c\\ 287+50E_c\\ 287+50E_c\\ 287+50E_c\\ 287+50E_c\end{bmatrix} \tag{12}$$

Solving the above matrix equation, we get the solutions for varying values of E_c

4. Results and Discussion

4.1 Introduction

The simulated results shows relationships between fluid velocity and fluid temperature with various parameters as obtained by numerical computation are given in Fig s 2 and 3. The simulation results given focus on the effects of the Re and Ec numbers on velocity and temperature distribution.

4.2 Effects of Reynolds number on vertical velocity

Solving equation (8) using MATLAB, the results of the effects of Re number on velocity of fluid are as shown in fig 2

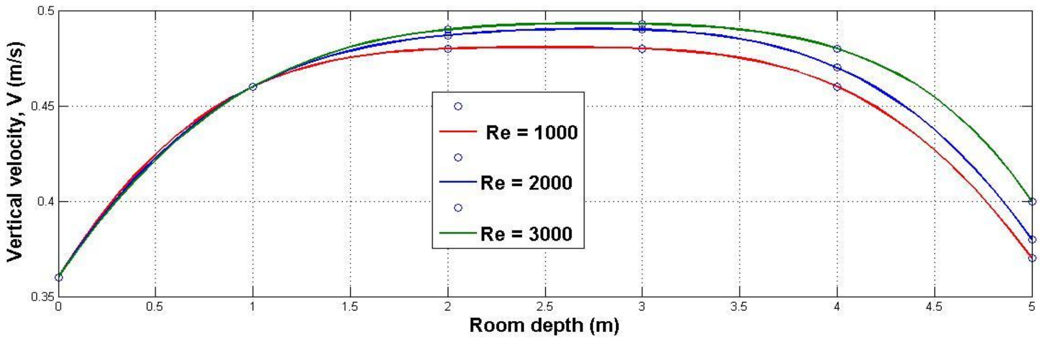

Figure 2: Graph of vertical velocity against room depth varying Re number

From fig 2 its clear that as the depth of the room increases, the vertical velocity decreases. The viscous forces became dominant**.** As the value of Re increased, the inertia forces dominated over the viscous force and the velocity increased. It can be seen from Fig 2 that, as the Re number increased the velocity along y axis also increased. As the depth increase, the velocity of the fluid also increased. The inertia forces were dominant. As the height of the enclosure increased the viscous forces became negligible, thus the velocity increased. When varying Re number, the velocity is seen to decrease with decrease in Re number as evidenced by Fig 2.

4.3 Effects of Eckert number on temperature of fluid

Equation (12) is solved using MATLAB and the results of the effects of Ec number on velocity of fluid are as shown in fig 4

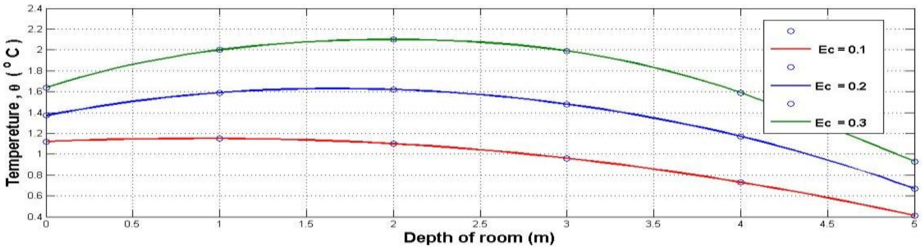

Figure 3: Graph of temperature against depth at varying Eckert number

From Fig. 3, temperature decreases with increase in room depth due to an increase in distance from the heat source. Fluid flow started at high temperature.A study on how changing Eckert number affect the distribution of temperature has been done. The results obtained have been presented in fig 3.The kinetic energy of fluid particles is higher at the top of the room. However kinetic energy reduces as depth increases in a vertical direction. This implies that there is more heat generation at the top. This outcome reveals an increase in temperature with an increase in Eckert number. However as the flow progress in the y-direction, the fluid flow is retarded hence collision of particles is reduced. This cause a gradual fall in temperature as depth from the top edge increases. When different values of distance y are plotted against corresponding temperature values, fig. 4 is obtained. It is evidence that at each Eckert number, temperature is higher near the top where heating took place and falls gradually downwards. It is therefore generally clear that an increase in Eckert number results into an increase in temperature distribution.

4.6 Validation and comparison of results

Sigey *et al.* (2013) investigated the enhancement of convective heat transfer in a square enclosure with localized heating from below. In their work the numerical study of natural convection of heat transfer in a three dimensional square cavity was considered. The two opposite vertical walls and the top wall were considered cold constant temperature with source of heat fixed at the middle bottom surface while the other two vertical walls and the non-heated part of the bottom surface were considered insulated. They established that:

- Temperature decreased as the fluid moved from the source of heat towards the cold walls.
- The position of the source of heat is one of the most important parameter on flow and temperature field
- Heat transfer is very weak at the sides of the square enclosure because the source of heat is at the centre of the enclosure.
- Flow and temperature fields are strongly affected by the cold surfaces.

In this study, similar results were found, that is;

- Temperature decreases with increase in room depth due to an increase in distance from the heat source.
- As the depth of the room increases, the vertical velocity decreases.
- Horizontal velocity increases with increase in room length.

Hence, the results are valid and can be applied in machines that work in low temperatures.

5. Conclusion and Recommendations

5.1 Conclusion

A numerical study of natural convection in a square enclosure cavity heated from the top wall and cooled from vertical opposite walls and top is done. The source of heat has been kept constant and the location fixed. The movement of the fluid in various regions of the enclosure is brought about by temperature difference. The temperature decreases downwards .the top corner is relatively warm and as a result the hot fluid moves away from the source, since it gains energy and becomes less dense, cold fluid ascend towards the hotter zone. The main parameters of interest are Reynolds number, Froude number and Eckert number.Heat transfer was high at high Re number than low Re number. At high Re number inertial forces were predominant. Similarly at high Re number initial velocity was high than when Re number was low since at high Re number inertial forces were predominant. From Fig 2 it's clear that as the depth of the room increases, the vertical velocity decreases. The viscous forces became dominant**.** As the value of Re increased, the inertia forces dominated over the viscous force and the velocity increased. An increase in temperature with an increase in Eckert number. However as the flow progress in the y-direction, the fluid flow is retarded hence collision of particles is reduced. This cause a gradual fall in temperature as depth from the top edge increases. The position of the source of heat is one of the most important parameter on flow and temperature field.

5.2 Recommendation

a) Further studies may include experimental investigations and three dimensional turbulent problems

b) Investigations of velocity profiles and temperature distribution at various positions of the heater and in other non-rectangular enclosures.
c) Investigations on any environmental impact on forced convection.

References

[1] Ahmed W. Mustafa and Ali Ghani (2012), Natural convection in trapezoidal enclosure heated from below. *Al – Khwarizmi Engineering Journal*. **88**:76 – 86

[2] Ampofo, F. and Karayiannis T. G. (2003) *"*Experimental benchmark data for turbulent natural convection in an air filled square cavity*". International Journal of Heat and Mass Transfer* **46**: 3551 -3572

[3] Aydin, O. and Yang, J. (2000), Natural convection in enclosures with localized heating from below and symmetrically cooling from sides *Methods; Heat Fluid Flow,* **10**:518 – 529.

[4] Caronna, G., Corcione, M. and Habib, E., (2009) Natural convection heat and momentum transfer in rectangular enclosures heated at the lower portion of the sidewalls and the bottom wall and cooled at the remaining upper portion of the sidewalls and the top wall, Heat Transfer Engineering, **30 (14):** 1166–1176

[5] Calcagni, B., Marsili, F. and Paroncini, M., (2005) Natural convective heat transfer in square enclosures heated from sides by the finite difference lattice Boltzmann method, European of Scientific Research, I below, Applied Thermal Engineering, **25:**2522-2531.

[6] D.A. Olson & L.R. Glickman (1991), Transient Natural Convection in Enclosure at High Rayleigh Number, *Transition of the ASME Journal of Heat Transfer*, **113**:633– 643.

[7] Delavar, M. A. and Sedighi, K., (2011)Effect of discrete heater at the vertical wall of the cavity over the heat transfer andentropy generation using Lattice Boltzmann method, Thermal Science,**15**.**2**,423-435

[8] J.K. Sigey (1999), Buoyancy Driven Turbulent Natural Convection in an Enclosure,DAAD Scholars Workshop, Egerton University, Njoro, Kenya.

[9] Kimunguyi, K. J.,Gachigua, G. W., & Gatheri, F. K. (1994), A numerical investigation of turbulent natural convection in a 3-d enclosure using k-ω sst model and simplec method. *International Journal of Engineering Sciences & Research technology*,**7**:66-77.

[10] Kipng'eno Joel (2006), Turbulent Natural Convection with Localized Heating and Cooling on Opposite Vertical Walls of an Enclosure, MSc Thesis, Kenyatta University, Kenya.

[11] M.A. Sheremet (2011), Mathematical Simulation of Conjugate Turbulent Natural Convection in an Enclosure with Local Heat Source, Thermo physics and Aeromechanics,**18**.1:107–121.

[12] .Miguel G.,Xaman j.,Gabriela A.,(2017) Conjugate Heat Transfer Analysis in a Glazed Room Modeled as a Square Cavity, Heat Transfer Engineering, **39.2**:120-140

[13] O.A. Plumb & L.A. Kennedy (1997), Application of K-E Model to Natural Convection from a Vertical Isothermal Surface, *ASME Journal of Heat Transfer***, 99:**79–85.

[14] Onyango. O. M,Sigey. J. K, Okelo. J. A, Okwoyo. J. M, (2013), Enhancement of Natural Convection Heat Transfer in a square enclosure with localized heating from Below. *International Journal of Science and Research (IJSR).ISSN (Online):* 2319-7064

[15] Patrick, H. Oosthuizen, (2011), A numerical study of the effect of a below – window convective heater on the heat transfer rate from a cold recessed window. *Frontiers in heat and mass transfer, 2,013004 Dol: 10 .5098/hmt.v2.1.3004*

[16] Paroncini, M., F. Corvaro (2009), Natural convection in a square enclosure with a hot source. *The International Journal of thermal science* **48**: 1683

[17] Rahman, M. M., Mamun, M. A. H., Billah, M.M. and Saidur, R. (2010)Natural convection flow in a square cavity with internal heat generation and a flush mounted heater on a side wall, Journal of Naval Architecture and Marine Engineering.

[18] Radhwan, M. and Zaki, G. M. (2000) Laminar natural convection in a square enclosure with discrete heating of vertical walls, *JKAU, Eng. Sci.*, **12.2**:83-99.

[19] Sigey, J. K., (2004), Three Dimensional Buoyancy driven Natural convection in Enclosure PHD thesis JKUAT Kenya.

[20] Siocha, E.N., Sigey, J.K., Okelo, J.A., and Okwoyo, J.M. (2016),A study on natural convection heat transfers between two vertical top closed walls with symmetrically heated isothermal walls. *The International Journal of Scientific Research and Engineering Studies (IJSRES),* ISBN 2349-8862.

[21] Enclosure,"*Asian Research Publishing Network,* **1(3):** 23-35

Author Profile

Edwin Nyagwonda Jephiter: Edwin holds a Bachelor of Education Science degree, and specialized in Mathematics and physics from Jaramogi Oginga Odinga University of science and technology, Kenya, He is currently pursuing Msc. in applied mathematics at Jomo Kenyatta University of Science and Technology (JKUAT), Kisii CBD Campus, Kenya. He is a High School teacher in Kenya. He has much interest in the study of fluid Mechanics and their respective applications in modeling physical phenomenon in Mathematics, sciences and engineering.

Johana K. Sigey: Prof. Sigey holds a Bachelor of Science degree in mathematics and computer science First Class honors from Jomo Kenyatta University of Agriculture and Technology, Kenya, Master of Science degree in Applied Mathematics from Kenyatta University and a PhD in applied mathematics from Jomo Kenyatta University of Agriculture and Technology, Kenya. Affiliation: Jomo Kenyatta University of Agriculture and Technology, (JKUAT), Kenya. He is currently the Director, JKuat, and Kisii CBD. He has been the substantive chairman - Department of Pure and Applied mathematics –Jkuat (January 2007 to July- 2012). He has published 9 papers on heat transfer, MHD and Traffic models in respected journals. Teaching experience: 2000 to date- postgraduate programme: (JKUAT); Supervised student in Doctor of philosophy: thesis (3 completed, 5 ongoing); Supervised student in Masters of Science in Applied Mathematics: (13 completed, 8 ongoing) .Phone number +254-0722795482.

Jeconia Okelo Abonyo: Prof Okelo holds a PhD in Applied Mathematics from Jomo Kenyatta University of Agriculture and Technology as well as a Master of Science degree in Applied Mathematics and first class honors in Bachelor of Education, Science; specialized in Mathematics with option in Physics, both from Kenyatta University. He has dependable background in Applied Mathematics in particular fluid dynamics, analyzing the interaction between velocity field, electric field and magnetic field. He has published over Forty four papers in international Journals. He is a Professor in the Department of Pure and Applied Mathematics and Assistant Supervisor at Jomo Kenyatta University of Agriculture and Technology

Phineas Roy Kiogora Dr. Phineas Roy Kiogora obtained his MSc. in Applied Mathematics from Jomo Kenyatta University of Agriculture and Technology (JKUAT), Kenya in 2007 and a PhD in Applied Mathematics from the same university in 2014. Presently he is working as a Lecturer at JKUAT. He has published four papers in international journals and guided many students in Masters Courses. His area of research is Hydrodynamic Lubrication.