Simulation of Turbulent Natural Convection in a Rectangular Enclosure by Use of k-ω-sst Model

M. N. Maithima [1], K. O. Awuor [2], J. E. Nafula [3]
[1]Researcher, Department of Mathematics, Kenyatta University,
P.O Box 43844 – 00100 Nairobi, Kenya
[2]Researcher and Senior Lecturer, Department of Mathematics, Kenyatta University,
P.O Box 43844 – 00100 Nairobi, Kenya
[3]Researcher, Department of Mathematics, Kenyatta University,
P.O Box 43844 – 00100 Nairobi, Kenya

Abstract. Turbulent flow is a type of flow that is highly unsteady, undergoes irregular fluctuations and three dimensional in motion. Naturally, turbulent flow is diffusive and dissipate energy. The aim of this study is to numerically investigate natural turbulent convection in a three dimensional rectangular enclosure using k-ω-sst model. The research is to investigate temperature and velocity distribution in a rectangular enclosure with constant Rayleigh number ($Ra = 7.80 \times 10^{11}$) and varying aspect ratios ($1.5 \geq A.R. \geq 0.75$). The equations governing the flow are Momentum, Continuity and Energy equations. The equations are first time averaged. The averaging process introduced non-linear terms; Reynolds stress and heat flux which are modelled using k-ω-sst model. The emerging equations after modelling are non-dimensionalized and then discretized by finite difference method and the results solved using ANSYS Fluent. The results showed that, increasing the height of the enclosure decreases the aspect ratio, which in turn strengthens the stream function consequently increasing the number of vortices.

Keywords: turbulence, natural convention, rectangular enclosure, k-ω-sst model

Introduction

Most fluids flows occurring in nature and created in engineering applications are turbulent. Turbulent flow is random, diffusive, dissipative, chaotic and irregular. These features have made turbulent flow to be highly and widely used in energy systems in industry. Due to its vast application in day to day activities, buoyancy driven natural convection in an enclosure is receiving more and more research attention. In most turbulent natural convection flows, investigations of velocity and temperatures profiles, heat transfer and turbulent intensities are mostly obtained by means of either experimental or modelling. Zeidan (2017) did a numerical study of turbulent cavitating flows in thermal regime and he modeled how heat is conducted in an enclosure. Nafula and Awuor (2020) studied numerical simulation of turbulent natural convective heat transfer in a rectangular enclosure using the $k - \omega$ model. Roy *et al.* (2020) investigated flow and heat transfer in an enclosed domain bounded by two concentric square cylinders. The governing equations were solved using the finite difference method. Prof. Ying Zhang *et al.* (2021) studied natural convection in a permeable square cavity with a couple of isothermally hot and cold blocks in the cavity. This work is aimed at investigating the effects of heating and cooling on the velocities and stream function in a rectangular enclosure using the $k - \omega - sst$ model.

Mathematical Formulation

We consider a 3 D enclosure shown in Figure 1. The vertical walls are isothermal while the horizontal walls are adiabatic. The hot wall is kept at 308K while the cold wall is kept at 288K creating a temperature difference of 18K between them.

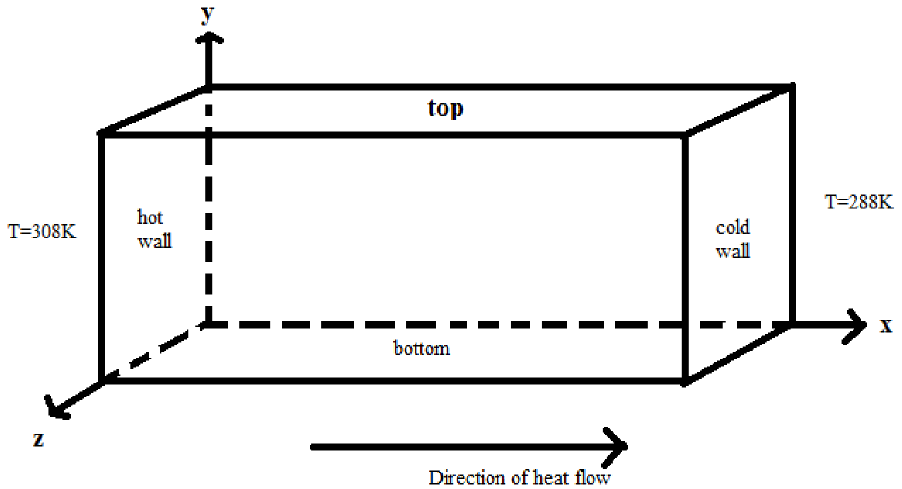


Figure 1

Governing Equations

The set of governing equations in two-dimensional rectangular coordinates which are continuity, momentum and energy equations are derived by Anderson *et al.* (1984).

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

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

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

$$\rho C_p\left(\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) + \Phi \tag{4}$$

Where

$$\Phi = \mu\left\{2\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2\right] + \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right)^2\right\}$$

The above equations were non-dimensionalised to reduce the number of parameters. The resulting equations in general form become;

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \tag{5}$$

$$U\frac{\partial U}{\partial X} + V\frac{\partial U}{\partial Y} + \frac{\partial U}{\partial \tau} = -\frac{\partial P}{\partial X} + Pr\left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}\right) \tag{6}$$

$$U\frac{\partial V}{\partial X} + V\frac{\partial V}{\partial Y} + \frac{\partial V}{\partial \tau} = -\frac{\partial P}{\partial Y} + Pr\left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2}\right) + Ra.Pr.\theta_f \tag{7}$$

$$\frac{\partial \theta_f}{\partial \tau} + u\frac{\partial \theta_f}{\partial X} + v\frac{\partial \theta_f}{\partial Y} = k\left(\frac{\partial^2 \theta_f}{\partial X^2} + \frac{\partial^2 \theta_f}{\partial Y^2}\right) + \Phi$$

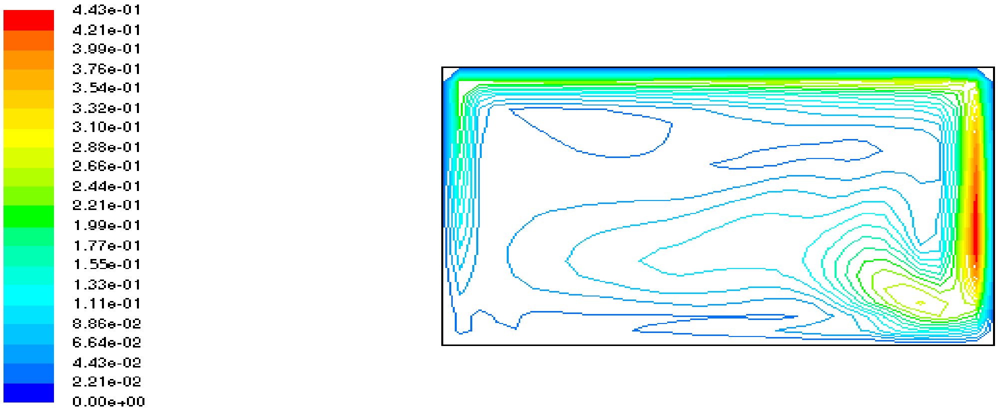
Results and Discussion

The results presented here were obtained by use of finite difference method to solve the governing equations numerically and together with the boundary conditions give the numerical solutions for variables in $SST\ k - \omega$ model. The Rayleigh numbers is kept constant by keeping the length constant.

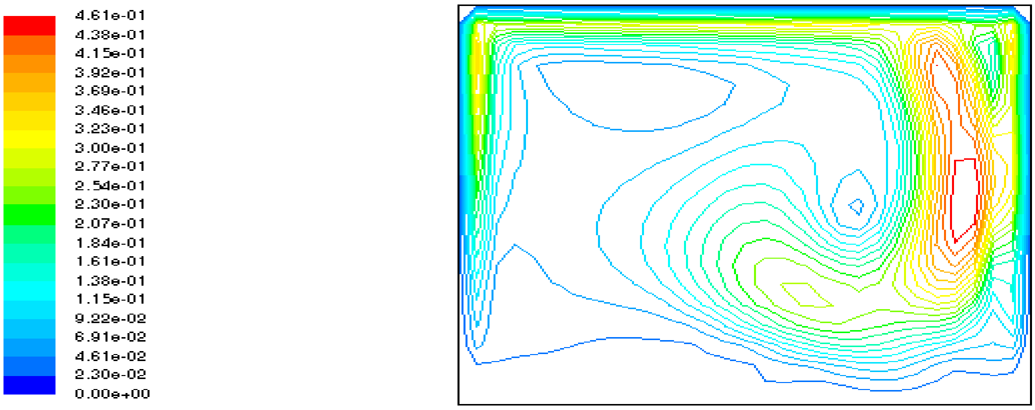
a) Contours of velocity magnitude (m/s) at $Ra = 7.80 \times 10^{11}$

The velocity contours of velocity magnitude shows a concentration of the vortices around the top wall, left hot wall and the right cold wall. At constant temperature, as the aspect ratio is decreased the vortices become bigger and become less as they disappear along the walls.

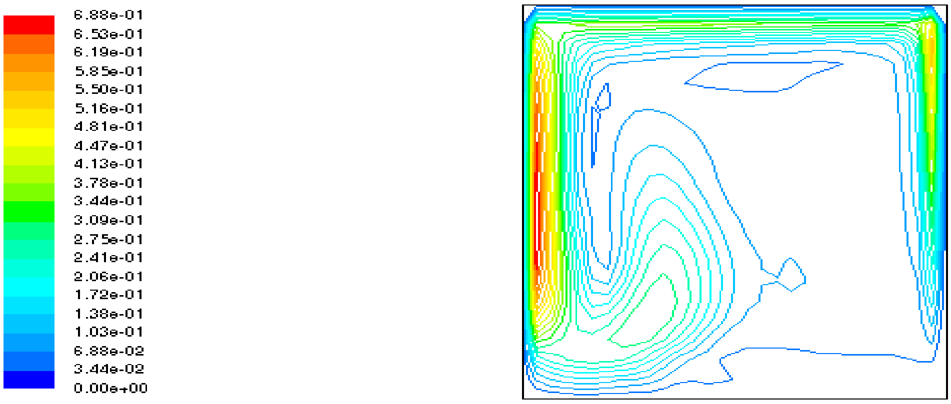
$i)$ ***Aspect Ratio*** $= \mathbf{1.5}$



$ii)$ ***Aspect Ratio*** $= \mathbf{1}$



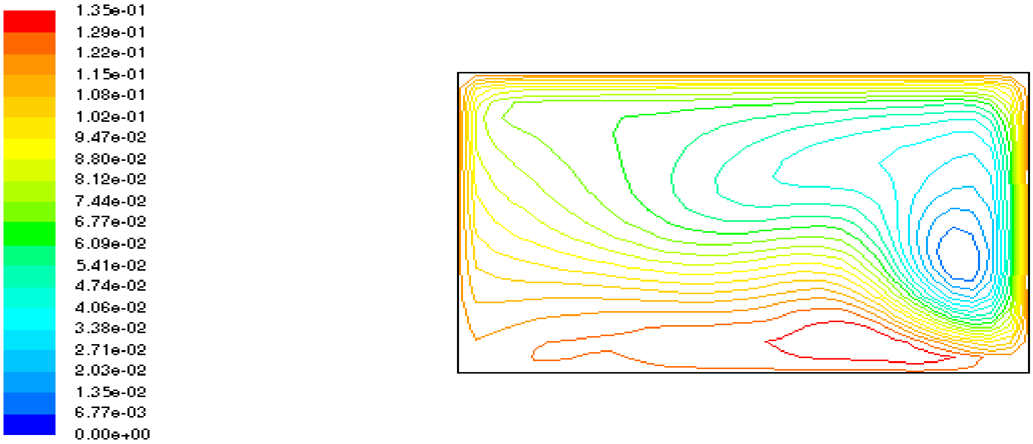
$iii)$ ***Aspect Ratio*** $= \mathbf{0.75}$



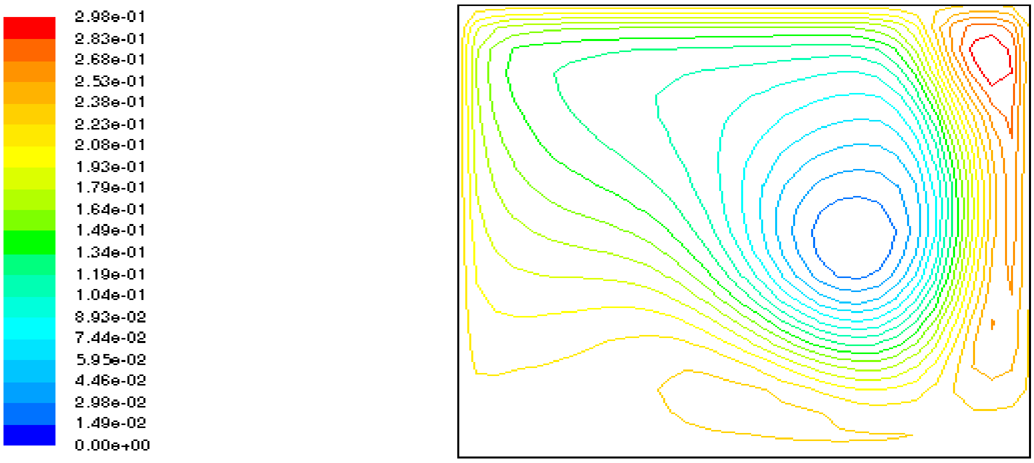
b) Contours of velocity stream function, $Ra = 7.80 \times 10^{11}$

Considering the velocity contours of stream function which shows the stream lines, the circulating vortices are concentrated around the right cold wall. As the aspect ratio decreases, the vortices move toward the left hot wall. This shows that since the aspect ratio is being varied by varying the height of the enclosure, increasing the buoyancy forces and hence leading to an increase in the strength of the stream function.

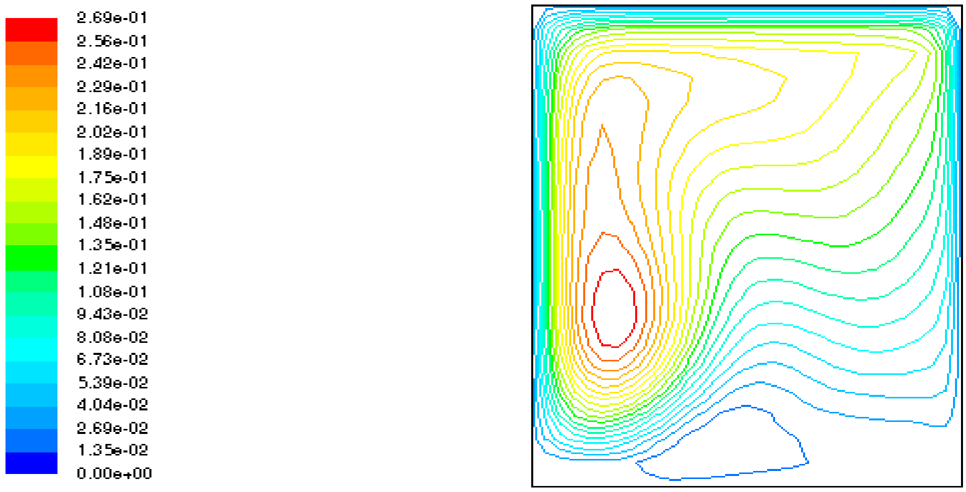
$i)\ Aspect\ Ratio = 1.5$



$ii)\ Aspect\ Ratio = 1$



$iii)\ Aspect\ Ratio = 0.75$



Conclusions

The results showed that, increasing the height of the enclosure decreases the aspect ratio, which in turn strengthens the stream function consequently increasing the number of vortices.

References

Anderson, J.D., Jr. (1990). *Modern Compressible flow: With Historical Perspective* (2^{nd} ed.). McGraw-Hill, New York.

Awuor, K.O. & Gatitu, M. (*2017)* Numerical simulation of natural convection in rectangular enclosures. *International Journal for Innovative Research in Multidisciplinary Field.*

Nafula, E. & Awuor, K. (2020). Numerical Simulation of Turbulent Natural Convective Heat Transfer in a Rectangular Enclosure using the $k - \omega$ model. *International Journal of Scientific Research and Innovative Technology*, *7*(3), 17-26.

Roy, N. C., Hossain, M., & Gorla, R. S. R. (2020). Natural convection in a cavity with trapezoidal heat sources mounted on a square cylinder. *SN Applied Sciences*, *2*(2), 143.

Zeidan, D. (2017). Numerical study of turbulent cavitation flows in thermal regime. *Journal of Numerical Methods of Heat and Fluid Flow*, *27*(7), 1487-1503.

Zhou, Yi, Wang Z., Qian Y., Yang H. & Wei Y. (2021). Numerical simulation of the flow around two square cylinders using the Lattice Boltzmann method. *Physics of Fluids, 33,* 037110.