Validation of Turbulence Convection in a 3-D Enclosed Chamber by Use of *k-ω* Model and SIMPLEC

S. G. Musyoki[1], K. O. Awuor[2], J. K. Kimunguyi[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 and Lecturer, Department of Applied Mathematics, Technical, University of Kenya, P.O Box 52428 – 00200 Nairobi, Kenya

Abstract. Turbulent flows are highly unsteady (random), chaotic and three dimensional in motion. Turbulent flows have high Reynolds number, non-zero vorticity. Turbulent flows are diffusive and dissipate energy naturally. The aim of this project is to numerically investigate natural turbulent convection in a three dimensional enclosed chamber using k-ω model and SIMPLEC method. The research is to investigate temperature and velocity distribution in an enclosure. 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 model. The emerging equations after modelling are non-dimensionalized and then discretized by finite volume method and the results solved using SIMPLEC algorithm. The numerical method is validated using benchmark experimental data provided by Markatos and Pericleous (1984). In this numerical investigation, the imbalance of the residual, by SIMPLEC became negligible after 350 iterations where done in period of 5 hrs while the simulation yielded non-dimensional temperature of 0.46, while the mean temperature profiles show a uniform distribution in the enclosure core.
Keywords: turbulence, natural convention, *k-ω* model, SIMPLEC

Introduction

Turbulent flow is an erratic motion of fluid particle with violent transverse interchange of momentum. A high Reynolds number constitutes to turbulent flow, in which, inertial force dominates over viscous forces. Turbulence is a feature of fluid flow but not of fluids. Molecular properties of the fluid do not control majority of characteristics of turbulent flow. Turbulent flow is experienced in aspects like water flowing past an obstacle in a river, the flow through pumps and turbines, flow of blood through veins, smoke and dust flowing in the atmosphere only to mention just but a few. Turbulent flows are highly unsteady (random) and three dimensional in motion. It consists of fast mixing and increased heat, momentum, and mass transfer rate, in other words, diffusive. Turbulence contain a vast range of spatial and temporal scales and also has a large deal of fluctuating vorticity (rotational). It is also dissipative, that is, it involves the transformation of energy from an initial form to a final form. Turbulent flow has wide applications especially in the field of engineering, automobile and machinery systems involving fluid flow.

Literature Review

Omri and Galanis (2006) investigated on natural turbulent convection in a two dimensional enclosure which is differentially heated and simulated by use of the SST k-ω turbulent model. The results showed that the conduction in the horizontal walls, after comparing with the Ampofo and Karayiannis Experimental benchmark results, significantly affected the calculated results. At the middle of the cavity, it was found that the measured values agreed with the calculated values. That agreement was not enough to establish the model validity and the procedure of obtaining numerical data. The comparison showed that if choosing of the distribution in the boundary layer of the smaller nodes is well done, it will yield the grid independent benchmark values.

Research was done by Awour (2012) in assessing the performance of k-ω, k-ε and k-ω SST turbulence models in predicting the transfer heat of due to natural convection in an enclosed chamber. The modeling of the heat transfers in the three dimensional enclosed rectangular air filled cavity included turbulence effect on Rayleigh number greater than or equal to 10^9. The emerging terms $\overline{u_i u_j}$ from averaged momentum and $\overline{u_i \theta}$ from averaged energy equation were modeled using the three turbulent models in order to close the governing equation. A temperature of 323k and 283k was maintained in the cavity on the hot wall and the opposite cold wall respectively. The remaining walls where adiabatic. The pressure term was eliminated by formulating the vorticity vector potential. The resulting equations were solved using the method of finite difference. The comparison of the performance of the three models with the experimental results showed that k-ω SST model performed better than the other two models in prediction of heat transfer in the cavity. He also heated and cooled the same wall in a room and used k-ω SST model due to its performance. The room was divided into three portions; the upper cold, the middle hot (area between the window and heater) and the lower warm.

Two-dimensional natural convection stratified flow in an enclosed parallelogram was studied by Salin (2015). The enclosure was insulated on the upper wall, a cold sidewalls and hot lowest wall at uniform temperature. Velocity and pressure components were regarded as the dependent variables in the equation of momentum. He used finite volume method on collated grid. A Rayleigh number ranging from $10^3 \leq Ra \leq 10^6$ and $0.5 \leq AR \leq 4$ was used. The outcome showed that Rayleigh number and Aspect ratio was directly proportional to average Nusselt number. Also, His conclusion was that Aspect ratio of the flow field defined the rate of heat transfer.

Kimunguyi (2016) investigated numerically the natural turbulent convection in a three dimensional rectangular enclosed room where k-ω-SST model was used together with PISO algorithm. Heating produced buoyancy due to internal body forces and that generated the motion of the fluid. The time averaged equation of momentum was solved using primitive variable to study the turbulent natural convection flow instead of formulating the vorticity-vector potential. An unknown turbulent correlation is introduced into the averaged equations namely Reynold stress ($\overline{u_i u_j}$) and heat flux ($\overline{u_i \theta}$**)** in the averaged momentum and averaged energy equations respectively, which was modeled using k-ω-SST model. The governing equations were non-dimensionslized and discretized by FVM and then PISO and SIMPLEC algorithms where employed to solve these equations. From the study, the results showed that PISO method assisted in improving time and speed convergence. PISO method also reduced the effort of computing and hence the absolute error was reduced very fast in the solution of flow. The acquired temperature and velocity profiles were applicable in systems like ventilations where the flow of air in a room can be modelled.

The research done by Kimunguyi (2016) builds a foundation for this paper.

Mathematical Formulation

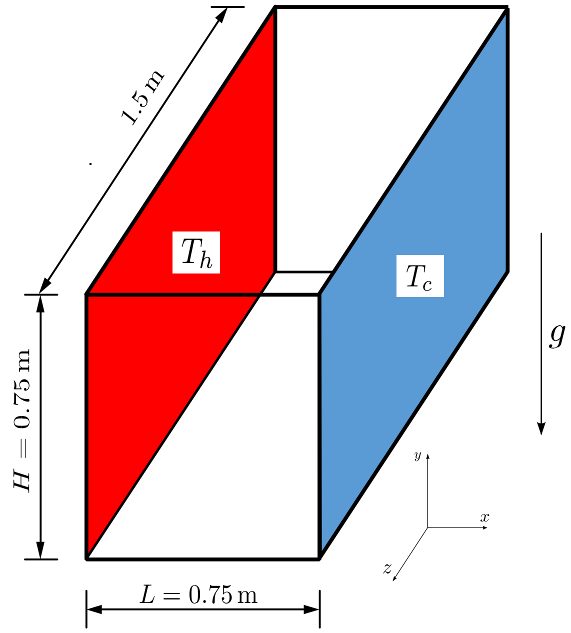

Figure 1. The geometry of the Model

The problem to be investigated is a 3-D enclosure illustrated in Figure 1. The measurements of the walls are 0.75m by 0.75m and 1.5m giving a $Ra = 1.58 \times 10^9$. The heating in the model is applied on the wall painted red and cooling on the opposite wall painted blue. All the other walls are adiabatic.

Governing Equations

The fundamental equations of continuity, momentum and energy, that govern the flow of incompressible fluids are emanated from the following general laws, namely; conservations of mass, momentum and energy as given below;

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_j)}{\partial x_j} = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho u_j) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \rho g_i + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + \mu_s \delta_{ij}\frac{\partial u_k}{\partial x_k}\right] \quad (2)$$

$$\frac{\partial}{\partial t}(C_p \rho T) + \frac{\partial}{\partial x_j}(C_p \rho u_j T) = \frac{\partial}{\partial x_j}\left(\lambda \frac{\partial T}{\partial x_j}\right) + \beta T\left(\frac{\partial P}{\partial t} + \frac{\partial u_j P}{\partial x_j}\right) + \phi \quad (3)$$

Where ϕ stands for dissipation function.

Results and Discussion

The outcomes presented here were determined by solving numerically the equations governing the flow by Finite Volume Method and together with the boundary conditions gave the numerical solutions for variables in $\kappa - \omega$ model. The benchmark experimental data provided by Markatos and Pericleous (1984) were used in order to verify.

Convergence of Solution by SIMPLEC Method

Convergence was monitored with residuals, whereby a decrease in residuals by three orders of magnitude was to indicate at least qualitative convergence whereby case residual plots would show when the residual values have reached the specified tolerance (Kimunguyi, 2016). For SIMPLEC, the criteria of residual convergence for each variable was obtained and the imbalance of the residual became negligible after 350 iterations where done in period of 5 hrs.

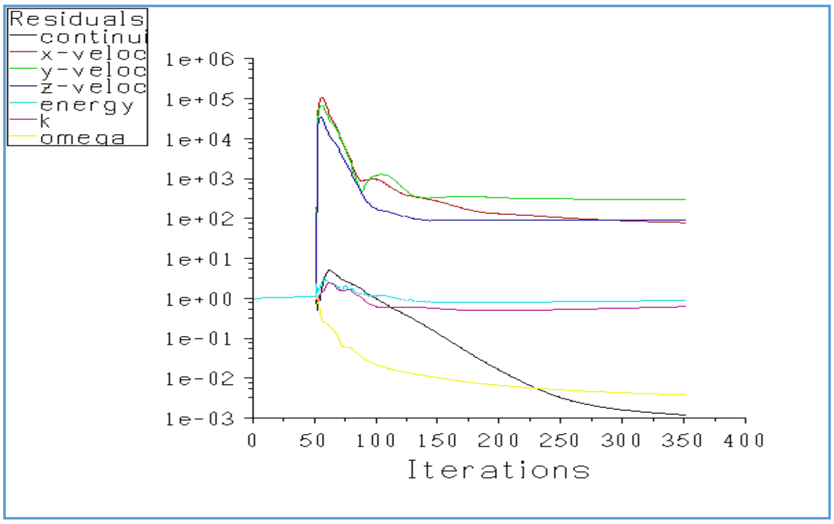

Figure 1. Scaled Residuals by SIMPLEC

Temperature Profiles

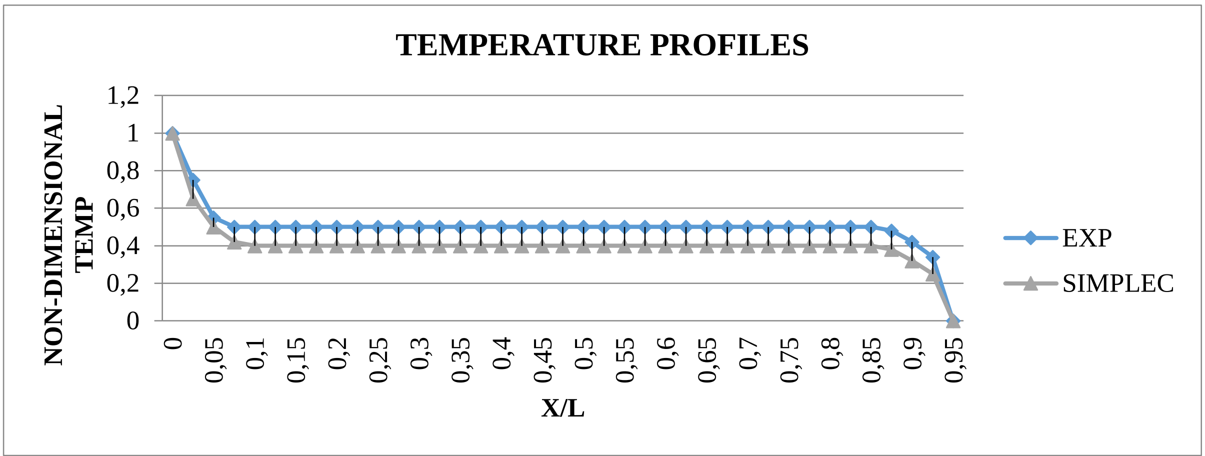

Figure 2. Mean Temperature Comparison at the core of the cavity

From Figure 2, the mean temperature profiles show a uniform distribution in the enclosure core. This confirms that there is close to zero activity in the core region of the enclosure, as the mean temperature is nearly uniform. The predicted data for temperatures by SIMPLEC shows a minimum which is less than the experimental data in the core of the cavity.The solutions produced by the numerical method were consistent with the exact solutions produced by the benchmark experimental data provided by Markatos and Pericleous (1984).

Conclusions

In this numerical investigation, the imbalance of the residual, by SIMPLEC became negligible after 350 iterations where done in period of 5 hrs while the simulation yielded non-dimensional temperature of 0.46, while the mean temperature profiles show a uniform distribution in the enclosure core

Acknowledgement

My sincere gratitude to Dr. K. O. Awuor and Dr. Kimunguyi K. Josephs for their valuable contributions and inspiration.

References

Ampofo, F., & 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*(19), 3551-3572.

Aounallah, M. Y. A. (2007). Numerical Investigation Of Turbulent Natural Convection in an inclined square cavity with a hot wavy wall. *International Journal of Heat and Mass Transfer, 50*(9-10), 1683-1693.

Awour, K. (2012), *Turbulent Natural Convection in an Enclosure. Numerical study of Different k-epsilon Model* (pp. 1-102). Kenya: Ph.D. Thesis, Kenyatta University.

Awour, K. & Gatitu, M. (2017). Numerical simulation of natural covection in a rectangular enclosure. *International Journal of innovative Research in multidisciplinary field.*

Hewitt, Duncan R., Jerume A., Lister R., & John R. (2014). High Rayleigh Number in a Three-Dimensional Porous medium. *Journal of fluid mechanics*, *748*, 879-895.

Kimunguyi, J. (2016). *Numerical Investigation of Turbulent Natural convection in a 3-D enclosure*. Kenya: PHD. Thesis, Kenyatta University.

Safaei, M., Goodarzi, M., & Mohammadi, M. (2016). Numerical modeling of turbulence mixed convection heat transfer in air filled enclosures by finite volume method. *The International Journal of Multiphysics*, *5*(4).

Sajjadi, H., & Kefayati, R. (2015). Lattice Boltzmann simulation of turbulent natural convection in tall enclosures. *Thermal Science*, *19*(1), 155-166.

Zimmerman, C. (2014). Modelling turbulent Heat Transfer In a Natural convection flow. *Journal of Applied Mathematics and Physics*, *2*, 662-670.