Modelling Turbulent Natural Convection Using the Staggered Grid and k-ω SST

[1]Kimunguyi, K. J., [2]Awuor, K. O., and [3]Gatheri, F. K.
[1]Researcher and Lecturer, School of Mathematics and Actuarial Science, Technical University of Kenya, Nairobi, Kenya
[2]Researcher and Senior Lecturer, Department of Mathematics, Kenyatta University, Nairobi, Kenya
[3]Executive Dean and Professor of Mathematics, Faculty of Applied Sciences and Technology, Technical University of Kenya, Nairobi, Kenya

Abstract. In this paper, a computational study of turbulent, natural convective flow of an incompressible fluid in a rectangular cavity is done. At the hot wall, the temperature distribution is a function of temperature gradients. The objective of this study is to investigate numerically, turbulent natural convection in a 3-D cavity using the staggered grid and the k-ω SST model. The statistical-averaging process of the mass, momentum and energy governing equations introduces unknown turbulent correlations into the mean flow equations which represent the turbulent transport of momentum, heat and mass, namely Reynolds stress $(\overline{u_i u_j})$ and heat flux $(\overline{u_i \theta})$, which are modeled using k-ω SST model. The Reynolds-Averaged Navier-stokes (RANS), energy and k-ω SST turbulent equations are first non-dimensionalized and the resulting equations are discretized using a staggered grid. From the results, for both the experimental data and simulation using the staggered grid and k-ω SST model, the discretization error has diminished to zero and the grid independence has been reached. Furthermore, we have reduced computational cost by finding an optimum grid size of the control volume, without compromising with the accuracy of the solution. The investigated Rayleigh number of this study lies at Ra = 1.58×10^9.

Keywords: turbulence, natural convection, staggered grid, k-ω SST model

Introduction

In fluid dynamics, turbulence is a flow regime characterized by chaotic and stochastic changes. These include low momentum diffusion, high momentum convection and rapid variation of pressure and velocity in space and time. As a result, turbulent flows have high diffusivity, are non-linear and characterized by a strong, three-dimensional vortex stretching mechanism which makes them rotational, 3-D and transfers energy and vorticity to increasingly smaller scales until the flow gradients become so big that they are smeared out by molecular viscosity. Because of these properties, turbulent flows are very important to many engineering applications and industrial and agricultural systems. Specifically, convection flows are one of the fundamental problems in fluid dynamics due to their role in meteo- rology where they appear as wind as an outcome of solar radiation in the atmosphere and in industrial applications, where they are used in cooling systems to reduce possible noise exposures and technical failures. Many analytical, experimental and numerical investigations have been performed in the past, but the findings are not exhaustive and so these flows are still of substantial interest. The focus of this paper is computational study of the lift due to turbulent natural convection and the resultant thermal mixing of the flow in an enclosure described in Figure 1 below.

Natural turbulent convection in cavities attracts considerable interest from thermal scientists given that it can be found in many industrial and/or civil engineering applications such as energy transfer in rooms and buildings, nuclear reactor cooling, solar collectors and electrical component cooling. A significant number of experimental and theoretical works have been carried out in the past in an attempt to understand turbulent natural convective flows in enclosures.

Among the work is that of Kulacki (1975) who studied natural convection in a horizontal fluid layer boundary bounded by upper isothermal surface and bottom insulated plate. For Prandtl numbers varying from 2:75 to 6:85 and Rayleigh numbers up to 2 10^{12}, the experimental data of Kulacki (1975) were correlated by the following expression: $Nu_{top} = 0.403Ra^{0.226}$. Further research on natural convection in an enclosure with localized heating and cooling has been studied by Gatheri *et al.* (1994). Gatheri *et al.* (1994) further investigated how to use false transient factors for the solution of natural convection problems and has well done a parametric study of an enclosure with localized heating and cooling (Gatheri, 1997). *Sigey et al.* (2004) not only did research on numerical free convection turbulent heat transfer in an enclosure but also carried out parametric studies on a rectangular enclosure using the standard k-ε model. Further work on natural turbulent convection has been about the use of mesh generation for the solution of natural convection problems (Gatheri, 2005), use of a variable false transient for the solution of coupled elliptic equations and use of buoyancy driven natural convection heat transfer in an enclosure and magnetohydrodynamic (MHD) free convective flows past an infinite vertical porous plate with Joule heating (Gatheri, 2005).

The validation of $\kappa - \varepsilon, \kappa - \omega,\ SST$ and a coarse DNS models for turbulent natural convection in a differentially heated cavity containing a fluid with P r = 0:71 and Rayleigh numbers ranging from 1.58×10^{9} to 10^{12} was performed by Aounallah *et al*. (2007). The conclusion of Aounallah *et al*. (2007) was that the k-ω-SST provided superior outcomes than the other models analyzed, though it was not able to reproduce accurately the mean flow.

Awuor (2012) did a study to assess the performance of three numerical turbulence models; $\kappa - \varepsilon, \kappa - \omega,\ and\ \kappa - \omega\ SST$ to find the model with a better approximation to the experimental data in predicting heat transfer profiles due to natural convection inside an air filled cavity. The results showed that $\kappa - \omega\ SST$ model is a more accurate layer simulation under high temperature gradient as compared with the $\kappa - \varepsilon\ and\ \kappa - \omega$ models. A numerical data was then obtained for a test problem using the best model, $\kappa - \omega\ SST$ which gave the outcome that the resultant numerical data stratified into three regions: a cold upper region, a hot region in the area between the heater and a warm lower region. The research work by Awuor (2012) is the motivation behind this paper.

Methods

In this paper, modelling turbulent heat transfer in a natural convection flow using the staggered grid within a cavity is conducted. The geometry is illustrated in Figure 1. It consists of a hot surface, located on the left side of the rectangular cavity wall, and a cold surface on the right side. The enclosure is heated on the hot wall (Red color) and cooled on the cold wall (blue color). The measurements of Ampofo and Karyiannis (2003) were used. The walls measure 0.75m by 0.75m wide by 1.5m. The hot and cold walls of the cavity were isothermal at 323$\pm$0.15K and 283$\pm$0.15K respectively, giving a Rayleigh number of 1.58×10^{9}. Each of the remaining walls are adiabatic. Fluid flow will depend only on the temperature difference given as $\Delta T = T_h - T_w$. Aspect ratio is 0.5. Furthermore, the Boussinesq Approximation (1903) is assumed and is presented below. In this research, we will study the variables as used by Ampofo and Karyiannis (2003).

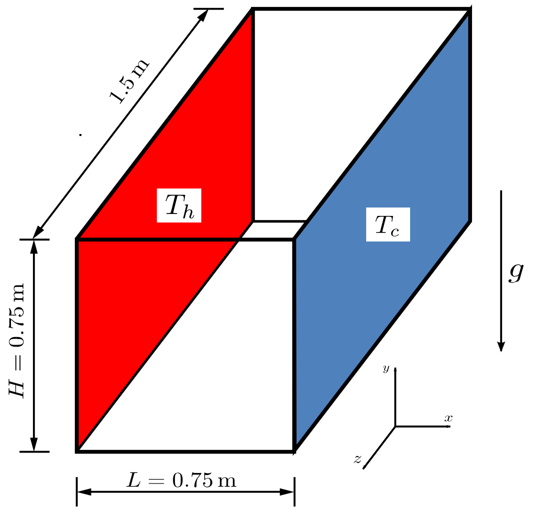

Figure 1. Geometry of the 3-D numerical model

The Boundary Conditions

The following boundary conditions are admissible in this problem:

i. The choice of the non-dimensional Θ temperature was such that $0 \leq \Theta \leq 1$.

ii. The Dirichlet boundary conditions apply on the heater and the window, while the Neumann boundary condition applies on the adiabatic walls.

iii. No slip boundary condition is used at the solid wall boundary of the 3-D enclosure.

iv. Free slip boundary holds for the component of velocity normal to the impermeable wall surfaces.

The numerical results we have found with these boundary conditions are numerical solutions for variables in $k - \omega\ SST$ model, which will be validated against the experimental data provided by Ampofo and Karayiannis (2003). This benchmark is at a Rayleigh number of 1.58×10^9.

Governing Equations

The final set of equations for turbulent natural convection flow, which represent the turbulent transport of momentum, heat and mass, are

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j + \underline{\rho u_i}\right) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}\left(\rho U_i + \underline{\rho u_i}\right) + \frac{\partial}{\partial x_j}\left(\rho U_i U_j + U_i \underline{\rho u_i}\right) = -\frac{\partial P}{\partial x_i} + p g_i + \frac{\partial}{\partial x_j}\left(\tau_{ij} - U_i \underline{\rho u_i} - \rho \underline{u_i u_j} - \underline{\rho u_i u_j}\right) \quad (2)$$

$$\frac{\partial}{\partial t}\left(C_P \rho T + C_P \underline{\rho T}\right) + \frac{\partial}{\partial x_j}\left(C_P \underline{\rho U_j T}\right)$$
$$= \frac{\partial p}{\partial t} + U_j \frac{\partial p}{\partial x_j} + \underline{u_j \frac{\partial p}{\partial x_j}} + \frac{\partial}{\partial x_j}\left(\lambda \frac{\partial T}{\partial x_j} - C_P \underline{u_t T} - C_P \underline{u_i T}\right) + \Phi \quad (3)$$

where:

$$\tau_{ij} = \mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_i}{\partial x_i}\right) + \mu_s \delta_{ij} \frac{\partial u_k}{\partial x_k} \quad (4)$$

$$\Phi = \tau_{ij} + \frac{\partial U_i}{\partial x_j} + \underline{\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}} \quad (5)$$

$$\frac{\partial}{\partial t}\rho k+\frac{\partial}{\partial x_j}(\rho U_i k)=\underline{u_j\frac{\partial}{\partial x_j}\mu\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i})}-\frac{1}{2}\frac{\partial}{\partial x_j}\underline{\rho u_i u_i u_j}-\rho\underline{u_i u_j}\frac{\partial u_i}{\partial x_j}+\underline{\rho u_i}g_i-\underline{u_j\frac{\partial p}{\partial x_i}} \quad (6)$$

$$\frac{\partial \omega}{\partial t}+U_i\frac{\partial \omega}{\partial x_j}=-\frac{\partial}{\partial x_i}\left[\left((\mu+\frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right]+\left[\beta-\frac{k^2}{\sigma_\omega\sqrt{c_u}}\right]\frac{\omega}{k}P_k-\beta\omega^2 \quad (7)$$

Notice that the unknown turbulent correlations in the mean flow equations namely Reynolds stress $(\underline{u_i u_j})$ and heat flux $(\underline{u_i}\theta)$, as outcomes of the statistical averaging process of mass, momentum and energy governing equations, have been modeled using k-ω SST model.

The Staggered Grid

For the case of choice of arrangement on the grid, we used a staggered grid arrangement instead of the collocated grid, in order to evaluate the velocity components at the control volume faces while the rest of the variables governing the flow field, such as the pressure, temperature, and turbulent quantities, are stored at the central node of the control volumes., as depicted in Figure 2 below.

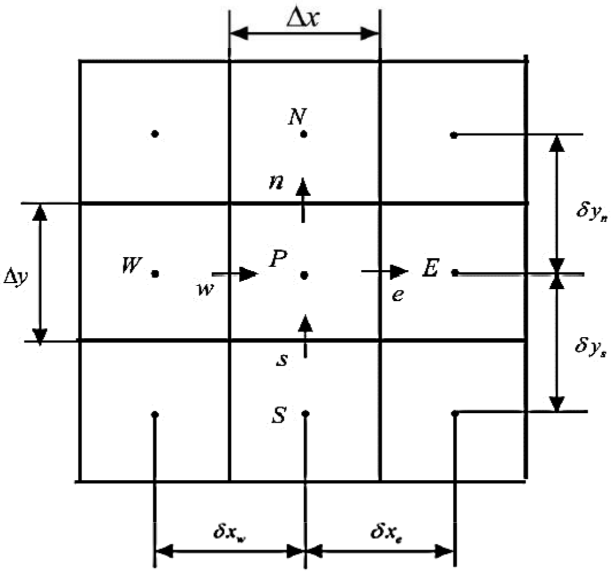

Figure 2. Variable arrangement on the staggered grid

Results and Discussion

The results presented here were obtained by solving equations (1), (2), (3), (6) and (7) after discretization using the staggered grid as shown in Figure 2 and together with the boundary conditions stated above, gave the following numerical solutions. The numerical results we have found were validated against the experimental data provided by Ampofo and Karayiannis (2003). This benchmark is at a Rayleigh number of 1.58×10^9.

Graph 1: The Computational Grid

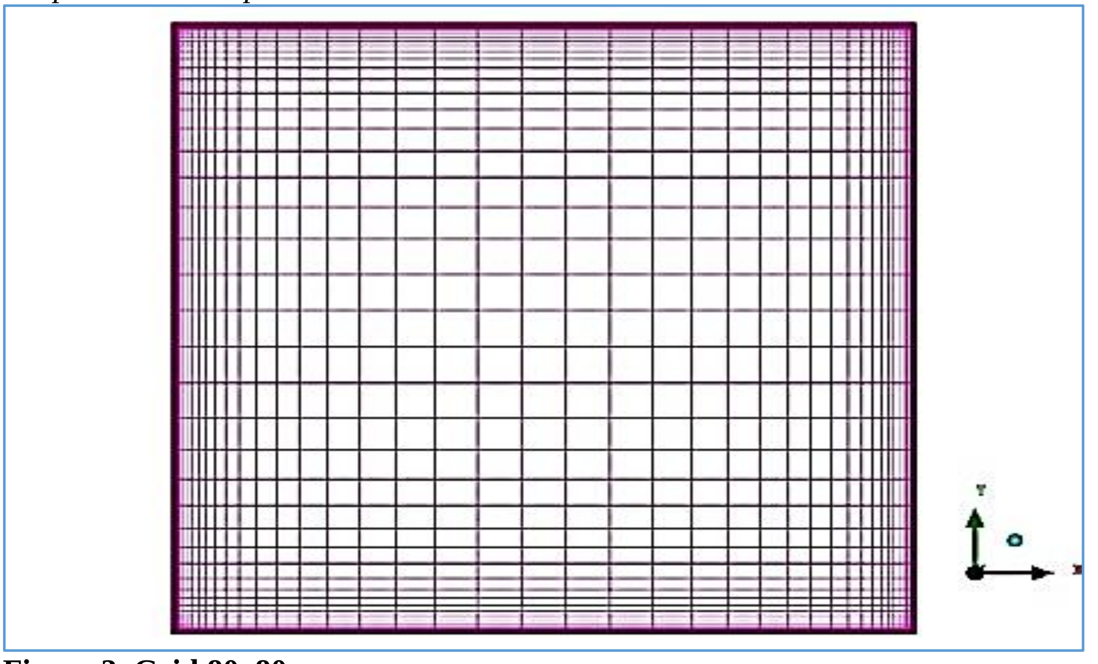

Figure 3. Grid 80x80

The computational grids shown in Figure 3 are staggered so that the scalar variables like pressure, temperature, density and turbulent quantities are stored in the cell centres of the control volume whereas vector variables like velocity and momentum are located in the cell faces. This would provide a strong coupling between velocities and pressure. Grids are clustered towards the wall capture the flow physics given that because the flow gradients are very large in the boundary layer; All variables are calculated right up to the walls without using any wall function since the $k - \omega\, SST$ model would use its blending function to switch the model to the $k - \omega$ model which is more accurate and more numerically stable in the near wall regions. On the wall surface, the boundary values for the velocity components and the turbulent kinetic energy are set to zero in conformity with no slip boundary condition.

Graph 2: Mass Imbalance on Coarse Grid

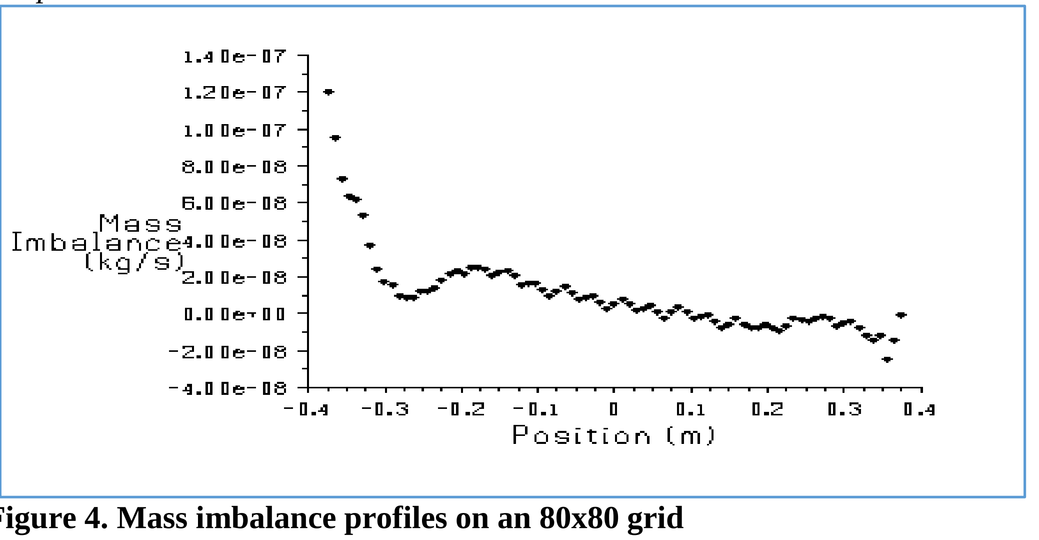

Figure 4. Mass imbalance profiles on an 80x80 grid

The dimensionless temperature of the cold and hot walls are 0 and 1 respectively.

Firstly, it was important to carry out a mesh convergence test using a grid checker. This we did by carrying out a grid independence test. This was done by computing the numerical solution on successively finer grids. The difference in numerical solution between the coarse (80x80) and finer (160x160) grid, was to be taken as the accuracy measure of the coarse grid. In this case, the 80x80 grid was refined by increasing the number of grid points to 160x160 for confirmation of grid independence. Figures 4 and 5 show a comparison of the residual mass imbalance profiles for the flow generated on each of the grids. The numerical implication is that as the mesh spacing or control volume size approached zero, the discretized equation solution matches the exact solution.

Graph 3: Mass Imbalance on Fine Grid

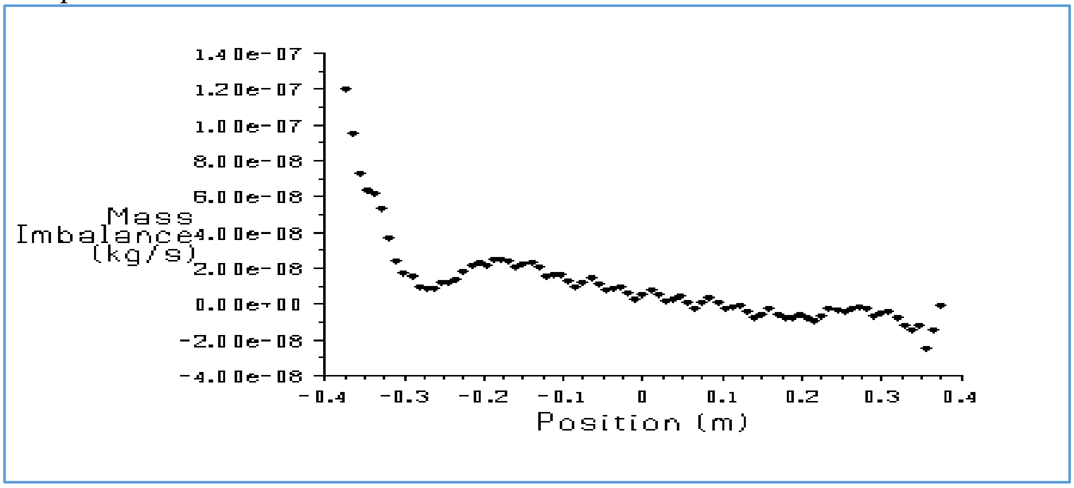

Figure 5. Mass imbalance grid on a 160x160 grid

Evidently, the results obtained on the 80x80 grid do not differ from those obtained on the 160x160 grid layout.

Conclusion

For both the experimental data and simulation using the staggered grid and k-*ω* SST model, the discretization error has diminished to zero and the grid independence has been reached. Computational cost has been reduced by finding an optimum grid size of the control volume, without compromising with the accuracy of the solution.

References

Ampofo, F. & Karanyiannis, T. G. (2003). Experimental Benchmark Data for Turbulent Natural Convection in an Air-Filled Square Cavity. *Intenational Journal of Heat Mass Transfer*, *46*, 3551-3572.

Aounallah, M., Addad, Y., Benhamadouche, S., Imine, O., Adjlout, L. & Laurence, D. (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*, 1683–1693.

Awuor, K. (2012). Turbulent Natural Convection in an Enclosure: Numerical Study of Different k-epsilon models. Ph.D. Thesis, Kenyatta University, Kenya.

Berghein, C., Penot, F., Mergui, S. & Allard, F. (1993). Numerical and experimental evaluation of turbulent models for natural convection simulation in a thermally driven square cavity, *Proceedings on Adaptive Selection Mode Error Concealment (ASMEC) Conference*, 1-12.

Boussinesq, J. (1903). *Théorie Analytique de la Chaleur* (Vol. 2). Gauthier-Villars.

Davidson, L. & Nielsen, P.V. (1996). Large Eddy Simulations of the Flow in a Three-Dimensional Ventilated Room. *Proceedings Roomvent '96, 2*, 161-168.

Dol, H. S. & Hinjalic, K. (2001). Computational Study of Turbulent Natural Convention in a side Heated Near-cubic Enclosure at High Re. *International Journal of Heat and Mass Transfer, 4*, 2323-2344.

Gatheri, F. K. (2005). Variable False Transient for the Solution of Coupled Elliptic Equations. *East African Journal of Physical Sciences*, *6*(2), 117.

Gatheri, F. K., Reizes, J., Leonardi, E. & del Vahl Davis, G. (1993). The use of Variable False Transient Factors for the Solution of Natural Convection Problems. *Proceeding 5th Australian Heat and Mass Transfer Conference*, *5*, 68, University of Queensland.

Gatheri, F. K., Reizes, J., Leonardi, E., & del Vahl Davis, G. (1994). Natural Convection in an Enclosure with Localized Heating and Cooling: A Numerical Study. In: G.F. Hewitt (Ed.), *Proceedings of the 10th International Heat Transfer Conference, 2,* 361-366.

Kulacki, F. (1975). *High Rayleigh number convection in enclosed fluid layers with internal heat sources.* University of Michigan Library.

Lax, P. D. & Richtmyer, R. D. (1956). Survey of the Stability of Linear Finite Difference Equations. *Communications on Pure and Applied Mathematics, 9*, 267-293.

Pantaker, S. V. (1980). *Numerical Heat Transfer and Fluid Flow, Series in Computational Methods in Mechanics and Thermal Sciences* (1st ed.). Hemisphere Publishing Corporation, 25-39.

Sigey J. K., Gatheri F. K. and Kinyanjui M. (2004), *Numerical Study of free Convection in an Enclosure*, Ph.D. Thesis, Jomo Kenyatta University of Agriculture and Technology, Kenya.

Tian, J. & Karayiannis, T.G. (2001). Low turbulence natural convection in an air filled square cavity. *J. Heat and Mass Transfer*, *43*, 849-866.

Tian, Y.S. & Karayianins, T.G. (2000). Low Turbulence Natural Convection in an Air Filled Square Cavity, Part I: The Thermal and Fluid Flow Fields. *International Journal of Heat and Mass Transfer*, *43*, 849-866.