Journal of Applied Mathematics & Bioinformatics, vol.4, no.2, 2014, 85-97
ISSN: 1792-6602 (print), 1792-6939 (online)
Scienpress Ltd, 2014

Natural Convection in a Rectangular Enclosure with Colliding Boundary Layers

J.W. Mutuguta[1], F.K. Gatheri[2] and M.M. Kiragu[3]

Abstract

The movement of the fluid in natural convection results from the buoyancy forces imposed on the fluid when its density in the proximity of the heat transfer surface is decreased as a result of the heating process. The objective of this paper is the computational study of the flow initiated by natural convection. The problem being investigated is the colliding boundary layer in a rectangular enclosure. The dimensional governing equations are first transformed to non-dimensional equations. The purpose of this transformation is to reduce the effort required to make a study over a range of variables. A non-dimensional scheme was chosen which led to improved iterative convergence for faster transients. The three dimensional analogue of the stream function-vorticity formulation was used where the scalar vorticity was replaced by a vector and the scalar stream function by a

[1] School of Pure and Applied Sciences, Murang'a University College, P.O. Box 75 – 10200 Murang'a. E-mail: jmutuguta@gmail.com
[2] School of Mathematics and Statistics, Technical University of Kenya, P.O. Box 43844-00200 Nairobi. E-mail: kgatheri@yahoo.com
[3] P.O. Box 474 Karatina. E-mail: mkiragu@ymail.com

Article Info: *Received* : February 15, 2014. *Revised* : March 29, 2014.
Published online : June 15, 2014.

vector potential. The equations were then solved using the method of variable false transients. The results indicated that the flow region is stratified into three regions; a cold upper region, a hot region in the area of the confluence of hot and cold streams and a warm lower region.

Mathematics Subject Classification: Applied Mathematics and Modeling
Keywords: Colliding boundary layer; Turbulent Natural Convection; Heat transfer

1 Introduction

When a viscous fluid flows past a stationary solid boundary, a layer of fluid which comes into contact with the boundary surface adheres to it and the condition of no-slip implies that the velocity of the fluid at a solid boundary must be the same as that of the boundary. Thus the layer of fluid that cannot slip away from the boundary undergoes retardation. This retarded layer further causes retardation on the adjacent layers thus developing a thin layer in the vicinity of the boundary surface at which the velocity of the fluid increase rapidly from zero at the boundary surface to the main stream velocity. Such a layer is known as the boundary layer.

Laminar boundary layers come in various forms and can be loosely classified according to their structure and the circumstances under which they are created. The thin shear layer that develops on an oscillating body is an example of the stokes boundary layer, while Blasious boundary layer (Liao and Campo [2]) refers to the similarity solution of the steady boundary layer attached to a flat plate held in an oncoming unidirectional flow. When fluid rotates, viscous forces may be balanced by coriolis[4] effect rather than convective inertia leading to the formation

[4]Apparent deflection of moving objects when viewed from rotating frame of reference.

of an Ekma layer[5]. Thermal boundary layers also exist in heat transfer. Multiple types of boundary layers can coexist near the surface. The thickness of the velocity boundary layer is normally defined as the distance from the solid boundary at which the flow velocity is 99% of the free stream velocity i.e. the velocity that is calculated at the surface boundary in an inviscid flow solution. The boundary layer represents a deficit in mass flow compared to an inviscid case with slip at the wall. It is the distance by which the wall would have to be displaced at the inviscid case to give the total mass flow as the viscous case.

The no-slip condition requires that the velocity at the solid boundary be zero and the fluid temperature be equal to that of the boundary. The flow velocity will then increase rapidly within the boundary layer as governed by the boundary layer equations. Thermal boundary layer thickness is the distance from the solid boundary at which the temperature is 99% of the temperature found from an inviscid solution. The ratio of the velocity and thermal boundary layers is governed by the Prandtl number. If the Prandtl number is 1, the two boundary layers are of the same thickness. If the Prandtl number is greater than 1, the thermal boundary layer is thinner than the velocity boundary layer. If Prandtl number is less than 1, the thermal boundary layer is thicker than the velocity boundary layer. For air of Prandtl Number Pr = 0.71, the thermal boundary layer is 141% of the velocity boundary layer.

The present study is designed to investigate the room air distribution. The importance of the present study lies not only in the room air distribution but also in room ventilation. Ventilating is the process of replacing air in the room to control temperature or to remove moisture, odor, dust, smoke airborne bacteria, carbon dioxide and also to replenish oxygen. It includes both the exchange of air to the outside as well as circulation of air within the building. In the absence of the

[5] A layer in which there exist a balance of forces between pressure gradient, coriolis forces and turbulent drag.

heater, cold air flows into the room and down to the floor. Warm air in the room rises and flows out through the window. The room is therefore filled with cold air.

Mohammed O. and Galvanis [3] studied natural turbulent convection in a differentially heated 2D cavity which they simulated using the Shear Stress Transitional (SST) $k-\omega$ turbulence model. Comparisons with experimental benchmark values show that, in this case, conduction in the horizontal walls has a significant effect on the calculated results. They also show that agreement between calculated and measured values at mid-height of the cavity does not suffice to establish the validity of the model and numerical procedure.

Visser et al [4] investigated buoyancy driven viscous flow through saturated packed pebble beds using a set of homogeneous volume averaged conservation equations. Comparison of the simulated results and experimental data of Niessen and Stocker [7] indicated an acceptable correlation.

Kandaswamy et al [1] studied natural convection heat transfer in a square cavity induced by heated plate. The top and bottom of the cavity were adiabatic. The study was performed for different values of Grashof number ranging from 10^3 to 10^5 for different aspect ratios. They found that the rate of heat transfer increases with increasing Grashof number.

Rundle and Lightstone [5] investigated turbulent natural convection in a square cavity heated and cooled on opposite walls and obtained the solution using three turbulent models i.e. $k-\varepsilon$, $wilcox\ k-\omega$ and Shear Stress Transitional models (SST). They found that the $wilcox\ k-\omega$ model was superior to $k-\varepsilon$ and SST models due to its high accuracy.

Nomenclature

τ	Viscous stress tensor
μ	First coefficient of viscosity
μ_s	Second coefficient of viscosity
g	Force due to gravity
∂	Differential operator

ρ	Density
Φ	The dissipation function or scalar transport unknown
λ	Thermal conductivity
β	Volumetric coefficient of expansion
C_P	Specific heat at constant pressure
η	Coordinate normal to the boundary
k	Kinetic energy of turbulence
T	Thermodynamic temperature
x, y, z	Coordinate direction in the $\hat{i}$, $\hat{j}$, $\hat{k}$ direction
Θ	Non-dimensional mean temperature
P	Thermodynamic pressure.
Ra	Rayleigh Number $Ra = \frac{\rho_R^2 C_{PR}\, g\, \beta \Delta T L_r^3}{\mu_R \kappa_R}$
Pr	Prandtl number $\Pr = \frac{\mu_R C_{PR}}{\kappa_R}$
$\bar{\varepsilon}$	Dissipation rate of turbulent kinetic energy

2 Mathematical formulation

The problem being considered is shown schematically in Figure 1 below. The dimensions of the model are 3.0m long, 3.0m wide and 3.0m high. The heater is mounted below the window and the remaining walls are adiabatic.

Some of the assumptions used in this analysis are that fluid is Newtonian and that there are no heat sources within the enclosure. The change in the fluid density is incorporated through the Boussinesq approximation.

Turbulent flow of air is described mathematically by the Reynolds Averaged Navier Stokes equations (RANS) including the time averaged energy equation for the mean temperature field that drives the flow by buoyancy force. These equations are;

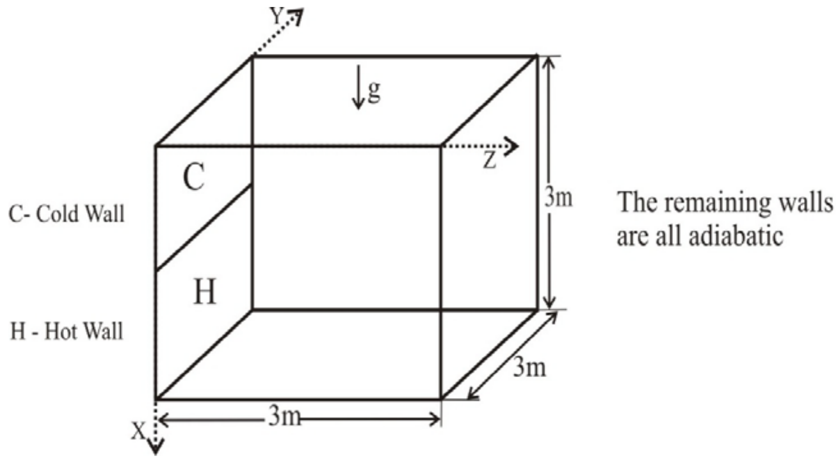


Figure 1: The schematic diagram for the case of colliding boundary layers with buoyancy effects

i) Equation of Continuity.

$$\frac{\partial \rho}{\partial t} + \nabla\left(\rho \boldsymbol{U} + \overline{\rho \boldsymbol{u}}\right) = 0 \tag{1}$$

ii) Equation of Momentum

$$\frac{\partial}{\partial t}\left(\rho \boldsymbol{U} + \overline{\rho \boldsymbol{u}}\right) + \nabla \cdot \left(\rho \boldsymbol{UV} + \boldsymbol{U}\,\overline{\rho \boldsymbol{v}}\right) = \nabla \cdot \boldsymbol{p} + \rho \boldsymbol{g} + \nabla \cdot \left(\tau - \boldsymbol{U}\,\overline{\rho \boldsymbol{u}} - \rho \boldsymbol{uv} - \overline{\rho \boldsymbol{uv}}\right) \tag{2}$$

iii) Equation of Energy

$$\frac{\partial}{\partial \mathrm{t}}\left(c_p \rho T + c_p \overline{\rho T}\right) + \nabla\left(c_p \rho U T\right) = \frac{\partial p}{\partial t} + U \nabla \cdot P + \overline{u \nabla \cdot p} + \nabla\left(\lambda \nabla T - c_p U \overline{\rho\, \mathrm{t}} - c_p \rho \overline{u\, \mathrm{t}}\right) + \Phi \tag{3}$$

where

$\tau = \mu(\nabla \cdot \boldsymbol{u} + \nabla \cdot \boldsymbol{v}) + \mu_s \delta \nabla \cdot \boldsymbol{w}$, and

$\Phi = \tau\, \nabla \boldsymbol{U} + \overline{\mu\,(\nabla \cdot \boldsymbol{u} + \nabla \cdot \boldsymbol{v})} \nabla \cdot \boldsymbol{u}$.

iv) Conservation Equation for the Turbulent Kinetic Energy

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho U_j k) = \overline{u_j \frac{\partial}{\partial x_j} \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)} - \frac{1}{2}\frac{\partial}{\partial x_j}\left(\overline{\rho u_i u_i u_j}\right) - \rho \overline{u_i u_j}\frac{\partial U_i}{\partial x_j} + \overline{\rho u_i}\, g_i - \overline{u_j \frac{\partial p}{\partial x_i}} \quad (4)$$

v) Equation of rate of Dissipation of Turbulent Kinetic Energy.

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial}{\partial x_j}\rho U_j \varepsilon = -\frac{\partial}{\partial x_k}\left(\overline{\mu \mu_k \frac{\partial u_i}{\partial x_j}}\,\overline{\frac{\partial u_i}{\partial x_j}} + 2\,\mathrm{v}\,\overline{\frac{\partial u_k}{\partial x_i}\frac{\partial p}{\partial x_i}} - \mu \frac{\partial \varepsilon}{\partial x_k}\right) - 2\mu \overline{\frac{\partial u_i}{\partial x_k}\frac{\partial u_i}{\partial x_j}\frac{\partial u_k}{\partial x_j}}$$
$$-2\rho\left(\overline{\frac{\partial^2 u_i}{\partial x_k \partial x_j}}\right)^2 + 2\,\mathrm{v}\,\overline{\frac{\partial u_i}{\partial x_j}\frac{\partial \rho}{\partial x_j}} g_i - 2\mu \frac{\partial u_i}{\partial x_k}\left(\overline{\frac{\partial u_i}{\partial x_j}\frac{\partial u_k}{\partial x_j}} + \overline{\frac{\partial u_j}{\partial x_i}\frac{\partial u_j}{\partial x_k}}\right)$$
$$-2\mu\mu_k \overline{\frac{\partial^2 u_i}{\partial x_j \partial x_k}}\,\overline{\frac{\partial u_i}{\partial x_j}} \quad (5)$$

The boundary conditions describing this motion are;

$u = v = w = 0$ on each boundary

On the hot boundary, $T = T_H = 70^0\, c$

On the cold boundary, $T = T_c = 20^0\, c$

The non dimensional temperature is defined by

$\Theta = \dfrac{T - T_*}{T_H - T_C}$.

where T_* – Quiescent temperature. The choice of Θ ensures that it is bounded and its value lies between 0 and 1. Thus the non-dimensional temperature boundary conditions are;

$\Theta = 1$ on the hot boundary, $\Theta = 0$ on the cold boundary and $\frac{\partial \Theta}{\partial \eta} = 0$ on the other boundaries.

3 Method of Solution

The system of non-linear partial differential equations with appropriate boundary conditions was solved using Alternating Direction Implicit method (Samaskii and Andreyev [6]). The convective terms were differenced using the hybrid scheme while the diffusion terms were differenced using the central difference scheme. The Elliptic Partial Differential Equations were converted to Parabolic Partial Differential Equations by insertion of false transient derivatives. The solution was then obtained by matching the parabolic equations through time. The procedure involves using different false transient factors in different flow regions. This results in small time steps in the boundary layer and large time steps in the rest of the enclosure.

4 Results and discussion

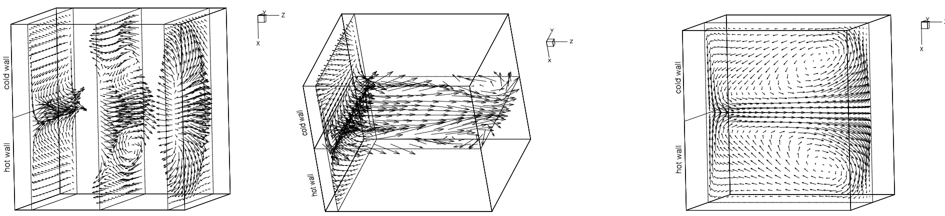
4.1. Flow Fields: Ra=10^8

The main objective of this study is to investigate the structure of the flow due to the collision of two opposed natural convection boundary layers driven by temperature difference along a vertical wall and the mechanism which causes the formulation of vertical structures in the flow that jets away from the wall. Solutions have been obtained for $Ra = 10^8$ and $Pr = 0.71$.

To reveal the fine structure of the thermal boundary layers, most of the vector plots in the $x-z$ plane have been selected on the symmetry plane and near side wall. In the case of $x-y$ the vector plots near the active wall and at the rear wall of the enclosure where most of the flow is concentrated have been selected. In the $y-z$ plane, vector plots in the borderline between the window and the heater have been selected.

Heating over the lower half and cooling over the upper half creates an unstable stratification of fluid at the colliding boundary layer. At sufficiently high Rayleigh number (high temperature difference), a strong convective motion develops and heat is transferred from the lower half of the enclosure to the upper half. Fluid in the upper half moves so that the cold fluid descends adjacent to the active wall and collides with the rising fluid from the lower half. This collision takes place halfway between the vertical walls. After collision, the boundary layers curve into the room and then join to form a horizontal jet which moves across the cavity as shown in Figure 2 (a) and (b). The jet impinges on the rear wall and bifurcates forming two streams. One of the streams moves towards the ceiling and flows back to the active wall through the ceiling while the other stream flows towards the floor and returns to the active wall through the floor of the enclosure. A clockwise cell is formed in the lower half of the enclosure between the upward moving stream on the active wall and the downward moving stream on the rear wall as shown in figure 2 (c).

In the confluence region of the colliding boundary layers, there is significant heat transfer which results from the mixing of the hot and cold streams. This is shown by the upward and downward motion of vectors in Figure 2 (b). As the horizontal jet moves towards the rear wall, there is increased mixing of fluid as seen in Figure 2 (a). As the stream moves towards the rear wall it becomes congregated towards the center forming a strong jet which impinges on the rear wall and is displaced in all directions as may be seen in Figure 2 (b).



(a) (b) (c)

Figure 2: Velocity vectors plots at $Ra = 10^8$: (a) End elevation at the plane $z = 0.1,\ 0.5$ and $z = 0.9$ respectively, (b) isometric view at $x = 0.5$ and at $\boldsymbol{z} = 0.1$ and (c) side elevation at $y = 0.5$

4.2 Temperature Fields

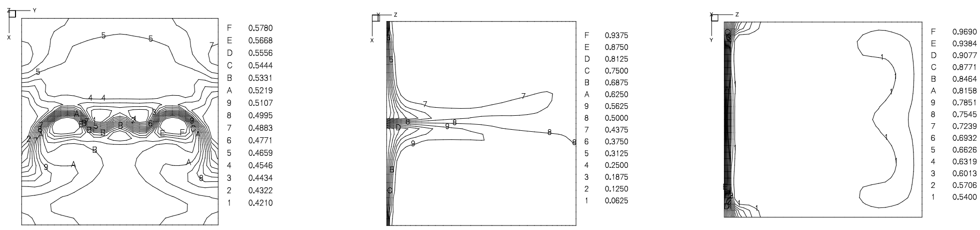
The hot and cold isotherms are congregated in a narrow strip due to fast rising and falling streams near the heater and the cold window. Figure 3(b) show temperature distribution along the heated and cooled surfaces. The isotherm 8 separates the contour plots between the upper and the lower regions. The isotherms of Figure 3 (a) indicate that the flow region may be divided into three regions.

(i) The lower region where the temperature of the fluid is everywhere above the mean value of $\Theta = 0.5$.

(ii) The unstable zone within the confluence region of the heater and the window where temperature ranges from 0.24 to 0.70. This is the region where mixing of the cold and hot streams of the fluid occurs. This region is dominated by high temperature gradients thus there is significant heat transfer between the hot and cold streams.

(iii) The upper region where temperature is below the mean room temperature $\Theta = 0.5$

Figure 3(c) shows the isotherms at the mid-plane of the hot region of the enclosure. There is steep temperature gradient near the active wall which decreases towards the rear.



(a) (b) (c)

Figure 3: Isotherms at $Ra = 10^8$: (a) end elevation at $z = 0.5$, (b) side elevation at $y = 0.5$ and (c) top view at $x = 0.75$.

5 Conclusion

In this study, buoyancy induced turbulent flow and heat transfer of air inside a three dimensional enclosure heated and cooled on the same vertical wall is investigated numerically. Different temperature conditions resulted in colliding boundary condition. It was observed that two boundary layers are formed which collide in the region between the window and the heater. After collision the two layers curve to form a horizontal jet that moves towards the rear of the enclosure. The jet then impinges on the rear wall of the enclosure and spreads to form two streams. One stream moves towards the ceiling while the other stream flows towards the floor of the enclosure. The fluid then flows back to the active wall through the ceiling and the floor of the enclosure. In the enclosure two large, equally sized, counter-rotating cells are formed. The upper cell is driven by the cold window whereas the lower cell is driven by the heater. The upper cold cell rotates in a counter-clockwise direction whereas the hot lower cell rotates in a clockwise direction. The two streams collide in the region between the window and the heater. Mixing process takes place in this region leading to the exchange of heat between the two fluid layers. This process transfers heat energy from the hot to the cold regions of the enclosure

The present study is designed to investigate the room air distribution. The results of the study indicates that placing the heater next to the window minimizes condensation and offsets the convective air current formed in the room as a result of the air next to the window. The importance of the present study lies not only in the room air distribution but also in room ventilation. Ventilating is the process of replacing air in the room to control temperature or to remove moisture, odor, dust, smoke airborne bacteria, carbon dioxide and also to replenish oxygen. It includes both the exchange of air to the outside as well as circulation of air within the building. In the absence of the heater, cold air flows into the room and down to the floor. Warm air in the room rises and flows out through the window. The room therefore is filled with cold air. When the heater is placed below the window, hot air from the heater rises and collides with the cold air from the window, mixing of the two layers takes place and as a result warm air flows into the room. The results of this analysis therefore show that the heater should be placed below the window.

References

[1] P. Kandaswamy, J. Lee and A. K. Abdul Hakeem, Natural convection in a square cavity in the presence of a heated plate, Non-linear Analysis, Modeling and Control, *ASME Journal of Heat Transfer*, **12**(2), (2007), 203-212.

[2] S.J. Liao, A. Campo, Analytic solutions of the temperature distribution in blasius viscous flow problems, *Journal of Fluid Mechanics*, **453**, (2002), 411–425.

[3] O. Mohamed and N. Galanis, Numerical analysis of turbulent buoyancy flows in enclosures: influence of grid and boundary conditions, *International Journal of Thermal Sciences*, **46**(8), (2007), 727-738.

[4] C.J. Visser, A.G. Malan, and J.P. Meyer, An artificial compressibility method for buoyancy driven flow in a heterogeneous saturated packed beds, *International Journal of Numerical Methods for Heat and Fluid Flow*, **18**(7/8), (2008), 900-918.

[5] C.A. Rundle, and M.F. Lightstone, Validation of turbulent natural convection in a square cavity for application of computational fluid dynamics modeling to heat transfer and fluid flow in atria geometries, *Second Canadian Solar Building Conference*, Calgary, June 10-14, 2007.

[6] A.A. Samaskii, and V.B. Andreyev, High accuracy difference scheme for an elliptic equation with several space variables, *USSR Computational Mathematics*, **3**, (1963), 1373 – 1382.

[7] H.F. Niessen, and B. Stocker, Data Sets of SANA Experiment: 1994-1996, JUEL-3409, (1997).