Mathematical Theory and Modeling
ISSN 2224-5804 (Paper) ISSN 2225-0522 (Online)
Vol.8, No.6, 2018

Turbulent Natural Convection in an Enclosure with Colliding Boundary Layers

Mutuguta John Wanjau[1*]

1. School of Pure and Applied Sciences, Murang'a University of Technology, P.O box 75 - 10200, Murang'a, Kenya

* E-mail: jmutuguta@mut.ac.ke

Abstract

When a heated plate is exposed to ambient room air without an external source of motion, a movement of air is experienced as a result of density gradients near the plate. This process is known as natural convection. The main objective of this paper is the computational study of the flow initiated by natural convection. We investigate the colliding boundary layer in a rectangular enclosure. The solution procedure in this problem involves the transformation of the dimensional governing equations to non-dimensional equations whose purpose is to reduce the effort required to make a study over a range of variables. In this study a non-dimensional scheme is chosen which leads to improved iterative convergence for faster transients. The three dimensional analogue of the stream function- vorticity formulation was used in which the scalar vorticity was replaced by a vector and the scalar stream function by a vector potential. The equations were then solved using the method of variable false transients in which the elliptic equations for the components of the vector potential were replaced by parabolic equations by insertion of false transient terms. The insertion of these multiplying factors before each time derivative effectively enabled different time steps to be used for each equation. The results of the numerical simulation showed that the flow region was stratified into three regions; a warm lower region, a cold upper region and a hot region in the area of the confluence of hot and cold streams.

Keywords: Colliding boundary layer, Natural convection

1. Introduction

A boundary layer is the layer of fluid in the immediate vicinity of a boundary surface. The boundary surface effect occurs at the field in which all changes occur in the flow pattern. The boundary layer distorts surrounding non-viscous flow. 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 2002) 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 Effect rather than convective inertia leading to the formation of an Ekma layer. 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 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.

Kandaswamy et al 2007 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 103 to 105 for different aspect ratios. They found that the rate of heat transfer increases with increasing Grashof number. The study conducted to investigate the buoyancy driven field and heat transfer in four wall solar chimney using different models by Evangellos et al 2007 showed that k - ε model is more reliable and likely to provide a superior performance for flow boundary layers under strong adverse pressure gradients. This was confirmed by comparing the results of the numerical simulation with the experimental results. Rundle and Lightstone 2007 investigated turbulent natural convection in a square cavity heated and cooled on opposite walls and obtained the solution using three turbulent models i.e. *wilcox* $k-\omega$, $k-\varepsilon$ 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. Sigey et al. 2004 investigated free convection turbulent heat transfer in an enclosure while Mohamed and Galvanis 2007 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 showed that in this case, conduction in horizontal walls have significant effect on the calculated results. They also showed that agreement between calculated and measured values at mid height does not suffice to establish validity of the model and numerical procedure. The study conducted by Mutuguta et al. 2014 to investigate the buoyancy driven flows showed that placing a heater beneath the window minimized condensation and created a convective air current in the room that resulted in a warmer room.

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.

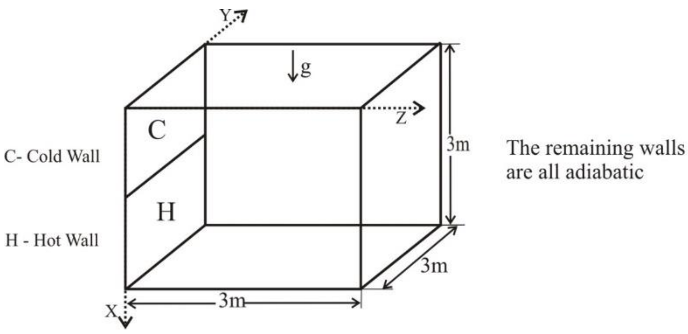


Figure 1: The schematic diagram for the case of colliding boundary layers with buoyancy effects.

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;

i) Equation of Continuity.

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

ii) Equation of Momentum

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

iii) Equation of Energy

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

Where $\tau = \mu\left(\nabla \cdot u + \nabla \cdot v\right) + \mu_S \delta \nabla \cdot w$ and $\Phi = \tau \nabla U + \overline{\mu\left(\nabla \cdot u + \nabla \cdot v\right)}\nabla \cdot u$

iv) Conservation Equation for the Turbulent Kinetic Energy

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

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

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

The boundary conditions describing this motion are;

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

On the hot boundary, $T = T_H = 50^o c$

On the cold boundary, $T = T_C = 10^o c$

The non dimensional temperature is defined by $\Theta = \frac{T - T_*}{T_H - T_C}$. 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 $\frac{\partial \Theta}{\partial n} = 0$ 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, 1963). Using this method, each time step was divided into three levels. At each level, one dimensional implicit equations were solved for each of the three directions in turn. 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

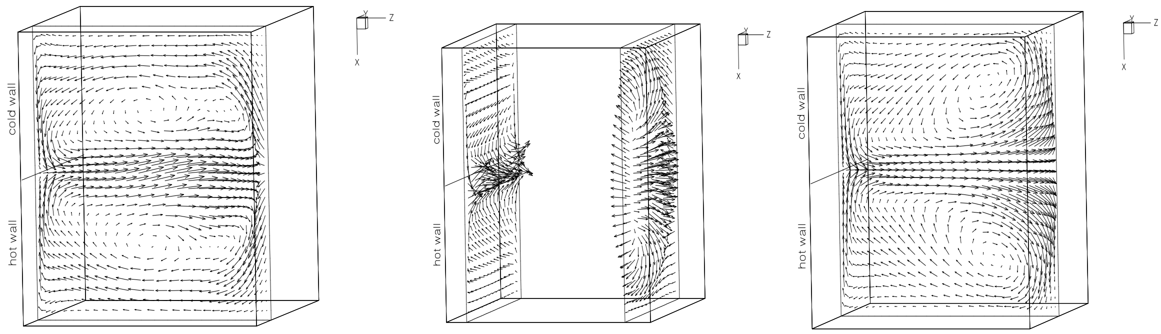
4.1. Flow Fields: $Ra=10^8$

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.

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 (c).



(a) (b) (c)

Figure 2 Velocity vectors plots at $Ra = 10^8$: (a) Side elevation at $y = 0.25$, (b) End elevation at $z = 0.1$ and $z = 0.9$ (c) Mid-plane at $y = 0.5$

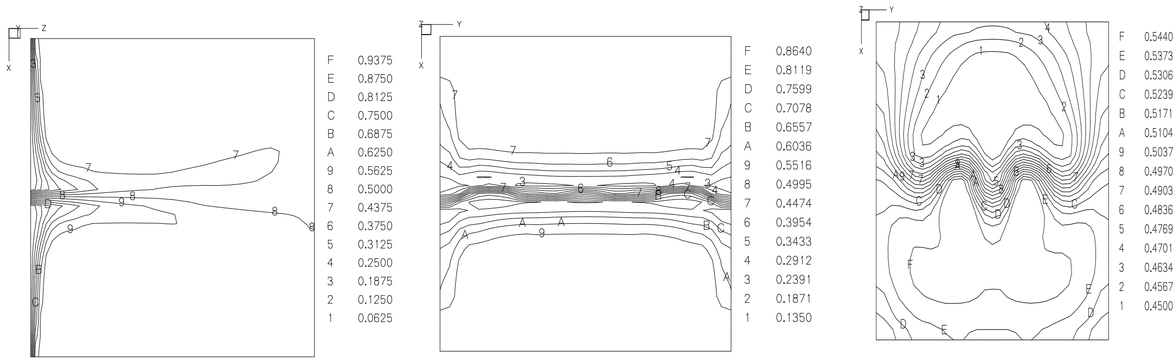
4.2. Temperature Fields.

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(a) 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 (b) and (c) indicate that the flow region may be divided into three regions.

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

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.

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



(a) (b) (c)

Figure 3. Isotherms at $Ra = 10^8$: (a) side elevation at y = 0.5 (b) End elevation at at z = 0.1 (c) end elevation at z = 0.9

5. Conclusion

In the present study, buoyancy driven turbulent flow and heat transfer inside a rectangular cavity is studied by numerical simulation. It is observed that two boundary layers are formed which collide in the middle of the region between the heater and the window after which the layers curve to form a horizontal jet that flows towards the rear of the enclosure. The jet hits the rear wall and bifurcates forming two streams. One stream flows upwards towards the ceiling while the second stream flows downwards towards the floor of the cavity. The two streams then flows back to the active wall through the ceiling and the floor of the cavity respectively. It is further observed that two large counter rotating cells are formed. One in the upper region being driven by the cold window while the lower cell is driven by the heater. The two cells collide in the region between the window and the heater resulting to the transfer of heat from the hot region to the cold region of the cavity.

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. 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 is therefore filled with cold air. On placing the heater 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.

References

Liao, S.J., Campo, A. (2002). *Analytic Solutions of the Temperature Distribution in Blasius Viscous Flow Problems*, Journal of Fluid Mechanics 453: 411–425

Kandaswamy, P., Lee, A. K. and Hakeem, A. (2007). *Natural Convection in a Square Cavity in the Presence of a Heated Plate, Non-linear Analysis*, Modeling and Control, Vol. 12 No. 2 pp. 203-212.

Rundle, C. A. and Lightstone, M. F. (2007). *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.

Samaskii, A. A. and Andreyev, V. B., (1963). *High Accuracy Difference Scheme for an Elliptic Equation with Several Space Variables*, Vol. 3, pp. 1373 – 1382.

Mutuguta, J. W., Gatheri, F.K. and Kiragu, M. M., (2014). *Natural Convection in a Rectangular Enclosure with Colliding Boundary Layers*, Journal of Applied Mathematics and Bioinformatics, Vol. 4(2) pp. 85-97

Sigey, J. K., Gatheri, F. K. and Kinyanjui, M., (2004). *Numerical Study of Free Convection Turbulent Heat Transfer in an Enclosure*, Journal of Energy Conservation and Management, Vol 45, pp. 2571-2582

Mohamed, O and Galvanis, N., (2007), *Numerical Analysis of Turbulent Natural Flows in Enclosures: Influence of Grid and Boundary Conditions*, International Journal of Thermal Sciences, Vol. 46(8), pp. 727-738.

Evangellos, B. Michalis, G. and Maria, K. (2007). *Study of Natural Convection Phenomena inside a Four Wall Solar Chimney with One Adiabatic Wall and One Wall Under Heat Flux*.