CFD ANALYSIS OF STEADY AND TRANSIENT NATURAL CONVECTION IN AN ENCLOSED CAVITY

Authors

Tanveer Hassan Mehedi	121412
Rahat Bin Tahzeeb	121406

Supervisor

Dr. A.K.M Sadrul Islam

Professor & Head of The Department

Department of Mechanical & Chemical Engineering.

Islamic University of Technology

A thesis submitted to the Department of Mechanical & Chemical Engineering in partial fulfillment of the requirements for the degree of Bachelor of Science in Mechanical Engineering

November, 2016

CERTIFICATE OF RESEARCH

This is to certify that the work presented in this thesis paper is the outcome of the research carried out by the candidates under the supervision of Dr. A.K.M Sadrul Islam, Professor & Head of the Department of Mechanical & Chemical Engineering. It is also declared that, neither this thesis nor any part thereof has been submitted anywhere else for the award of any degree or any judgement.

Authors

______________________	________________________
Tanveer Hassan Mehedi	Rahat Bin Tahzeeb

Signature of Supervisor

____

Dr. A.K.M Sadrul Islam

Professor & Head of The Department
Department of Mechanical & Chemical Engineering.
Islamic University of Technology

ABSTRACT

The research is conducted to find out the critical width of insulation in air insulated walls seen in residential buildings and industrial furnaces.

Natural convection between two differentially heated walls have been simulated using ANSYS FLUENT in both steady and transient conditions.

To simulate different heat transfer and fluid flow conditions, Rayleigh number ranging from 10^3 to 10^5 has been maintained (i.e. Laminar flow.)

In case of steady state analysis, the cfd predictions are in very good agreement with the literature reviewed data.

Transient simulation process has been performed by, using USER DEFINED FUNCTIONS, where the temperature of the hot wall varies with time in a sinusoidal manner.

To obtain and compare the heat transfer properties, Nusselt Number has been calculated at the hot wall at different conditions.

The buoyancy driven flow characteristics have been investigated by observing the flow pattern in a graphical manner.

The characteristics of the system at different temperature differences between the wall has been observed and documented.

Dedicated To

To our respected mentor

Professor Dr. A.K.M SADRUL ISLAM

ACKNOWLEDGEMENTS

We would like to express our sincere gratitude to **Dr. A.K.M SADRUL ISLAM** for all the guidance and support he has provided us throughout the year. We would like to thank our Faculty **Dr. Arafat Ahmed Bhuiyan** for sharing his knowledge. We would like to thank our faculty **Mr. Sayedus Salehin** for giving us constant inspiration for a publication. We also cordially grateful to our faculty **Mr. Ifat Rabbil Qudrat Ovi** who introduced us with CFD.

Contents

1 Introduction

Natural convection is a type of heat transfer through fluid medium which does not occur due to any external source (i.e. fan, pump, suction device), rather the sole driver is the density differences within the fluid occurring due to the temperature gradient. The highly ordered fluid motion characterized by smooth layers of fluid is called laminar flow. Any fluid motion occurs by natural means such as buoyancy. The fluid motion in natural convection is often not noticeable because of the low velocities involved.

1.1 Aims and Objectives

The objective of this study is to find out the heat transfer characteristics of air inside an enclosed cavity in both steady and transient state with respect to different temperature gradients and different widths of air gaps.

At first CFD simulations for the steady state analysis was done to compare with the published literature and verify the results.

Then the simulation was carried out for the transient state and the Nusselt numbers for different conditions were compared to observe the change in the characteristics of heat transfer.

2 Literature Review

Despite the deluge of publications on natural convection heat transfer, the literature shows that there have been relatively few studies of the fundamentals of transient natural convection in enclosures.

This state of affairs seems surprising if we consider the fact that most of the confined flows driven by buoyancy in architectural, environmental, and solar energy engineering applications are time dependent·

The determination of buoyancy-driven flow in an enclosed cavity provides a suitable comparison problem for evaluating the performance of numerical methods dealing with viscous flow calculations·

2.1 Problem Setup

N C Markatos et al. has given a very definitive schematic for the proposed problem where the boundary conditions and the surface body conditions are clearly stated. The overall characteristics of an enclosure can be briefly described as a system which consists of 4 walls which can either be insulated or have different thermodynamic conditions. Generally fluid inside enclosures do not remain stationary. Due to the temperature difference between walls, convection currents are observed. Fluid adjacent to the hot surface rises up and adjacent to the relatively colder surface falls down.

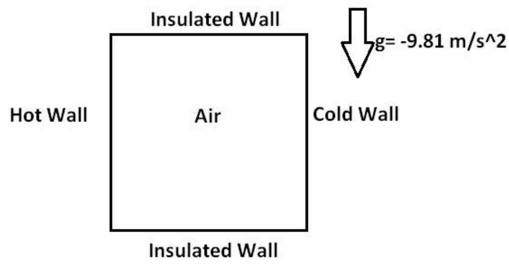


Figure 1 Enclosure considered for the study

2.2 Governing Equations

The Boussinesq approximation is applied to problems where the fluid varies in temperature from one place to another, driving a flow of fluid and heat transfer. The fluid satisfies conservation of mass, conservation of momentum and conservation of energy.

In the Boussinesq approximation, variations in fluid properties other than density are ignored.

If *u* is the local velocity of a parcel of fluid, the continuity equation for conservation of mass is

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0. \qquad (1)$$

If density variations are ignored, this reduces to

(2)

$$\nabla \cdot \mathbf{u} = 0.$$

The general expression for conservation of momentum (the Navier–Stokes equations) is

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + \frac{1}{\rho} \mathbf{F} \qquad (3)$$

where $\boldsymbol{v}$ is the kinematic viscosity and **F** is the sum of any body forces such as gravity. In this equation, density variations are assumed to have a fixed part and another part that has a linear dependence on temperature:

$$\rho = \rho_0 - \alpha \rho_0 \Delta T \qquad (4)$$

where α is the coefficient of thermal expansion. The resulting conservation equation is

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} - \mathbf{g} \alpha \Delta T \qquad (5)$$

In the equation for heat flow in a temperature gradient, the heat capacity per unit volume, ρC_p, is assumed constant. The resulting equation is

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = \kappa \nabla^2 T + \frac{J}{\rho C_p} \qquad (6)$$

where ***J*** is the rate per unit volume of internal heat production and ***k*** is the thermal diffusivity.

The three numbered equations are the basic convection equations in the Boussinesq approximation.

The approximation's advantage arises because when considering a flow of, say, warm and cold water of density ρ_1 and ρ_2 one needs only to consider a single density ρ, the difference $\Delta\rho=\rho_2-\rho_1$, is negligible.

3 Methodology

This section provides the basic equations, assumptions, properties and specific heat model used in the current study.

3.1 Background

The basic equations to solve the fluid motion and the heat transfer are mass conservation (Continuity), momentum conservation (Navier-Stokes) and energy equation. These basic equations are listed below:

Continuity:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \quad (7)$$

X- Momentum:

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{xz}}{\partial z}\right] \quad (8)$$

Y- Momentum:

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{yz}}{\partial z}\right] \quad (9)$$

Z- Momentum:

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}=-\frac{\partial p}{\partial z}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}\right] \quad (10)$$

Total Energy:

$$\frac{\partial(E_T)}{\partial t}+\frac{\partial(uE_T)}{\partial x}+\frac{\partial(vE_T)}{\partial y}+\frac{\partial(wE_T)}{\partial z}=-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}$$

$$+\frac{1}{Re_r}\left[\frac{\partial}{\partial x}(u\tau_{xx}+v\tau_{xy}+w\tau_{xz})+\frac{\partial}{\partial y}(u\tau_{xy}+v\tau_{yy}+w\tau_{yz})+\frac{\partial}{\partial z}(u\tau_{xz}+v\tau_{yz}+w\tau_{zz})\right]$$

$$-\frac{1}{Re_r Pr_r}\left[\frac{\partial q_x}{\partial x}+\frac{\partial q_y}{\partial y}+\frac{\partial q_z}{\partial z}\right] \quad (11)$$

These equations do not have a direct analytical solution except for very simple cases, and hence have to be solved in the discretized form in a domain. In this study, they are solved using finite volume technique using commercial computational fluid dynamics (CFD) software FLUENT.

3.2 Assumptions

- Fluid is assumed to be Newtonian.
- Incompressible flow.
- Surface is considered to be smooth and no-slip condition is used on all wall surfaces.
- Boussinesq model has been considered to generate the effects of buoyancy.
- Heat transfer through radiation has been neglected.

3.3 Boundary Conditions

The two vertical walls in the geometry were kept at constant temperature, one being in a higher temperature than the other in the steady state analysis.

Whereas in the transient analysis, the temperature of the hot wall was kept varying with time linearly through the use of USER DEFINED FUNCTION.

As for the two horizontal walls, they were kept as insulated with zero heat flux input.

The geometry was considered to be planar for this study and the effect of gravity was taken into consideration in the y axis.

3.4 Fluid Properties

The fluid properties were taken at the mean temperature of the two vertical walls which was 288K.

All the values were considered to be constant except the density of air taken into consideration under the boussinesq model to account for the effect of buoyancy.

Property	Value
Density	1.225 kg/m^3
C_p , Specific Heat	1007j/kg-K
Thermal Conductivity	0.02476 W/m-K
Viscosity	1.802e-05 kg/m-s
Thermal expansion co-efficient	0.003472 1/K

3.5 Flow and Heat Transfer Parameters

Flow and heat transfer characteristics are generally specified using a set of dimensionless numbers. These dimensionless numbers were calculated in different ways, depending upon the geometry and the flow conditions. The flow parameters used and the ways they were calculated are explained in this section.

Nusselt Number

Nusselt number is one of the very important parameters for analyzing heat transfer performance. Higher the Nusselt number, more effective is the heat transfer process. This dimensionless number essentially indicates the effectiveness of the convective heat transfer process taking place between the walls and the fluid. It is based on the convective heat transfer coefficient and is given by

$$\boldsymbol{Nu = \frac{hL_c}{k}} \qquad (12)$$

A Nusselt number of 3 for an enclosure indicates that heat transfer through the enclosure by natural convection is three times that by pure conduction.

Grashof Number

The governing equations of natural convection and the boundary conditions can be nondimensionalized by dividing all dependent and independent variables by suitable constant quantities.

The dimensionless Grashof number represents the ratio of buoyant forces to viscous forces.

$$\boldsymbol{Gr_L = Grashof\ Number = \frac{Buoyant\ Forces}{Viscous\ Forces}} \qquad (13)$$

Prandtl Number

The relative thickness of the velocity and the thermal boundary layers is best described by the dimensionless parameter Prandtl number, defined as

$$Pr = Prandtl\ Number = \frac{Molecular\ Diffusivity\ of\ Momentum}{Molecular\ Diffusivity\ of\ Heat} \quad (14)$$

The Prandtl numbers of gases are about 1, which indicates that both momentum and heat dissipate through the fluid at about the same rate. Heat diffuses very quickly in liquid metals (Pr << 1) and very slowly in oils (Pr >> 1) relative to momentum. Consequently, the thermal boundary layer is much thicker for liquid metals and much thinner for oils relative to the velocity boundary layer.

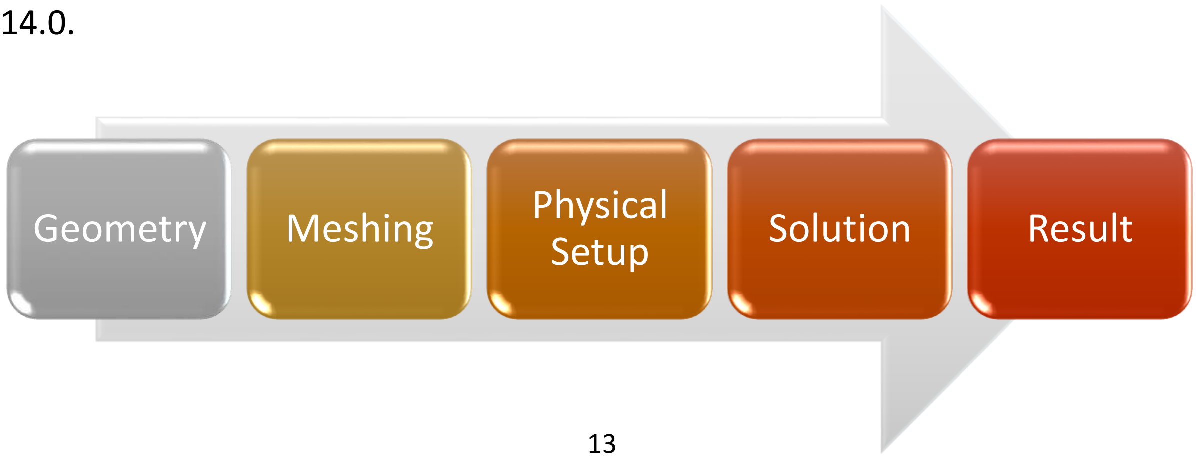
Rayleigh Number

Rayleigh number, which is the product of the Grashof number, which describes the relationship between buoyancy and viscosity within the fluid, and the Prandtl number, which describes the relationship between momentum diffusivity and thermal diffusivity. Hence the Rayleigh number itself may also be viewed as the ratio of buoyancy forces and (the products of) thermal and momentum diffusivities.

$$Ra_L = Gr_L Pr = \frac{g\beta(T_H - T_C)L^3}{v^2} Pr \quad (15)$$

4.0 Modeling Procedure

The entire simulation has been performed by using ANSYS 14.0. The heat transfer and laminar flow modeling for both the steady and transient state analysis has been done using computational fluid dynamics software FLUENT. The meshing has been done in the preset design modeler provided by ANSYS 14.0.



4.1 Geometry

The geometry has been defined as a square where the left wall defines the hot wall, the right wall defines the cold wall and the upper and lower wall are defined as the insulated walls.

The surface body for the geometry is fluid which accounts for the air present in between the two walls.

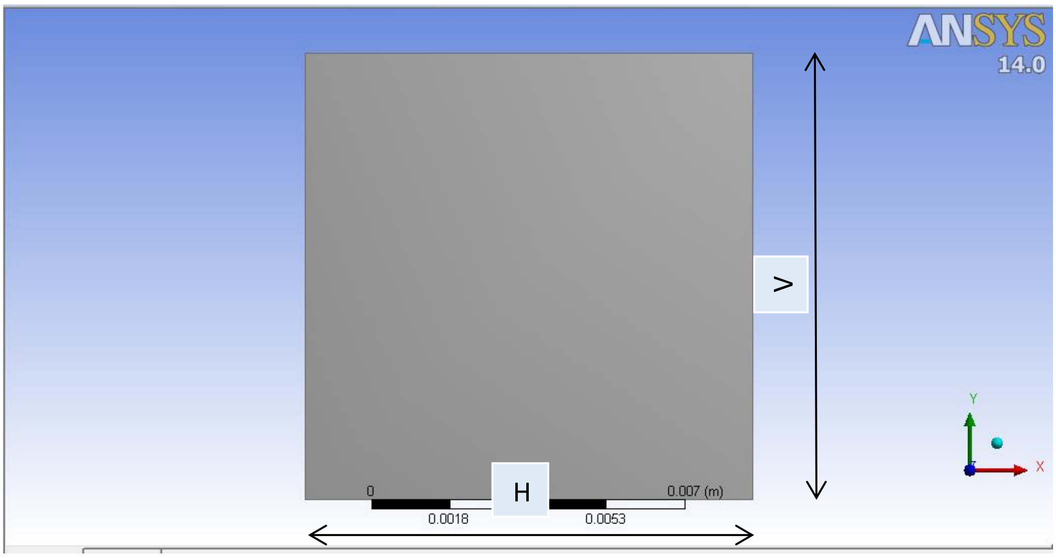


Figure 2 ***Geometry file for the study***

V/H ratios used for steady state analysis:

Condition	V/H
Ra 10^3	0.01/0.01
Ra 10^4	0.02/0.02
Ra 10^5	0.03/0.03

V/H ratios used for transient state analysis:

Condition	V/H
ΔT= 4^{o}C	0.01/0.01
ΔT= 6^{o}C	0.02/0.02
ΔT= 8^{o}C	0.03/0.03

4.2 Meshing

One of the most important tasks for any simulation related to CFD analysis is the meshing part. The results obtained must be mesh independent which validates the findings of a study.

For the purpose of this study, intensive mesh analysis has been done to ensure the mesh independence. The results that were extracted from the software were validated against the date obtained from the reviewed literature [3].

The initial solutions were done to confirm the legitimacy of the defined physical setup and then the number of divisions of the mesh were gradually increased until the results reached up to a constant value.

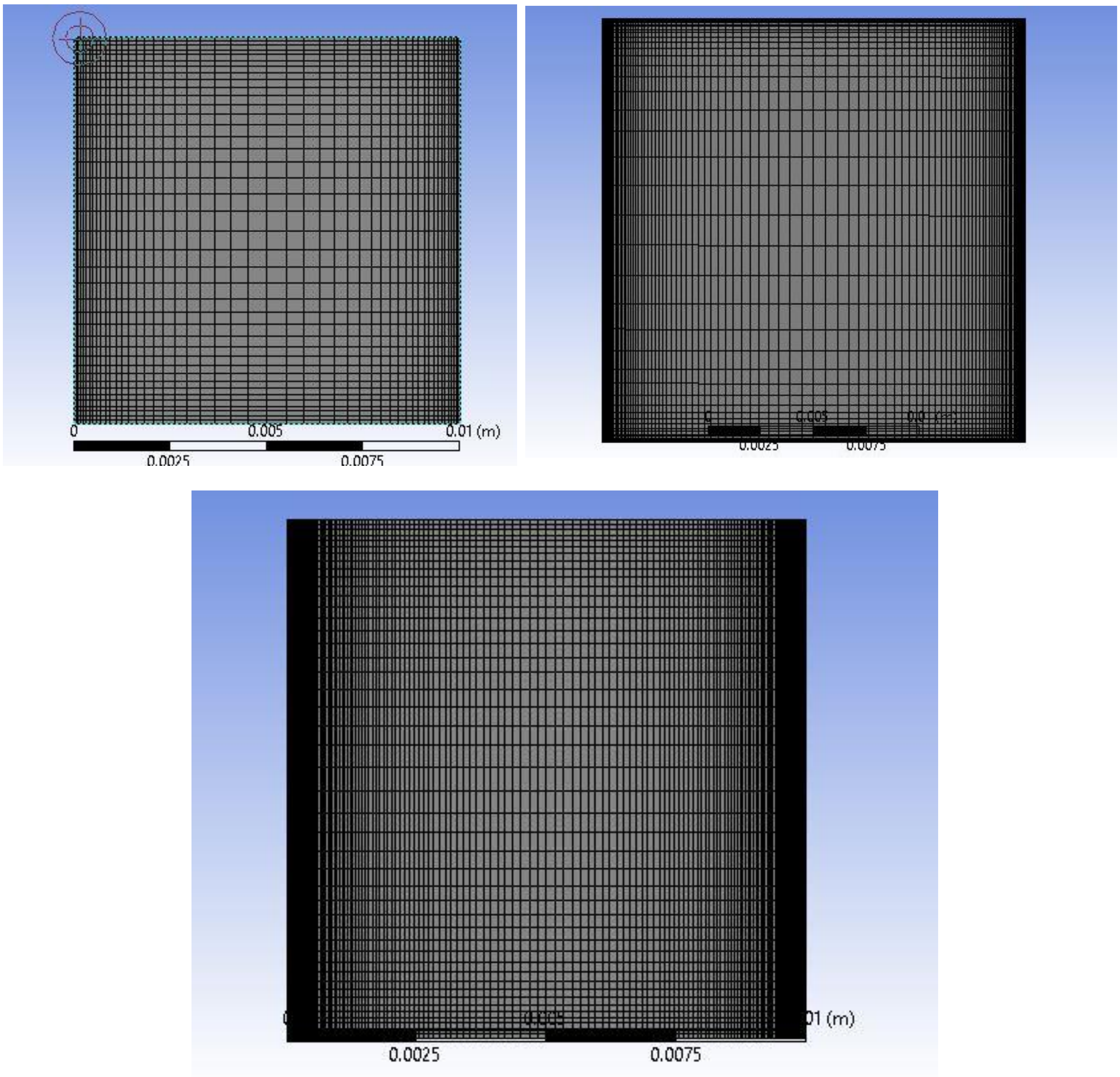


Figure 3 ***Different mesh configurations***

4.3 Physical Setup

The physical setup of the simulation is the part where the bonding between the computer simulation and the real world is made.

Once the grid was generated the physical setup was made in FLUENT 14.0 with the following properties.

Material Properties:

The material properties that were required to be defined in this study was basically the property of air at the mean temperature.

The mean temperature for the study as stated earlier was 288K.

The properties were defined in the following manner

Property	**Value**
Density	1.225 kg/m^3
C_p, Specific Heat	1007j/kg-K
Thermal Conductivity	0.02476 W/m-K
Viscosity	1.802e-05 kg/m-s
Thermal expansion co-efficient	0.003472 1/K

Boundary conditions:

Steady state Analysis:

For the steady state analysis temperatures of both the hot and wall were kept constant. For example, for Ra= 10^3, the temperature of the hot wall was taken as 282.5K and for the cold wall it was 293.5K. Subsequently, the temperature gradient and the distance between the two walls were altered to maintain Ra= 10^4 and Ra= 10^5 which ensures laminar flow of air.

Transient Analysis:

The transient analysis was done in two parts. One where the temperature of the hot wall increases gradually as a function of time and then in the 2nd part it reduced following the same trend.

User Defined functions had to be used in this case so that the software could take the instructions regarding the varying input function. The temperature of the cold wall remained constant throughout the time period. The total time of simulation was for 1 hour, where the temperature was raised for the first half an hour and then it fell for the next half an hour.

4.4 Result Extraction

The results of this study were extracted from CFD post. The main output from this module was the wall heat flux of the hot wall, which in further steps helped to calculate the Nusselt number.

In addition to that, the velocity and temperature contours and the velocity profiles were also obtained from CFD post.

The wall heat flux was calculated from the built-in function calculator which gave direct value of the required parameter.

After extracting the value of the wall heat flux, the Nusselt number was obtained by using the following equation:

$$\boldsymbol{Nu} = \frac{\boldsymbol{q_{wall}}}{\boldsymbol{k}} \times \frac{\boldsymbol{D}}{\boldsymbol{T_H} - \boldsymbol{T_C}}$$

5 Results and Discussion

Temperature Contour (Steady State)

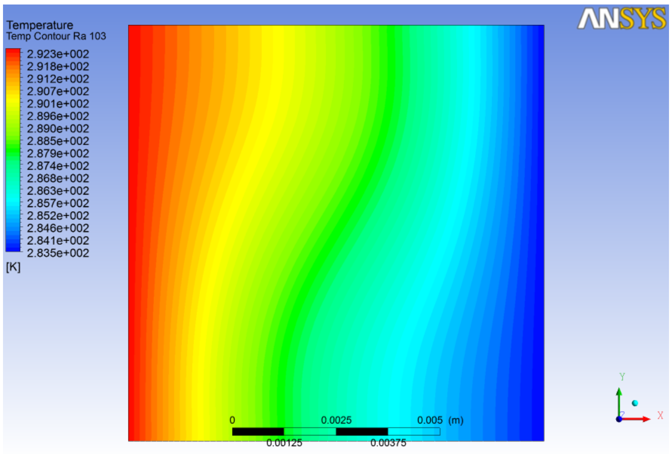


Figure 4 ***Temperature Contour for Ra 10^3 in steady state.***

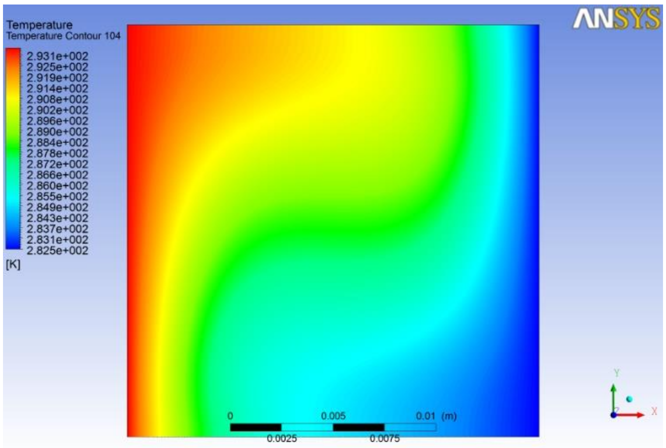


Figure 5 ***Temperature Contour for Ra 10^4 in steady state.***

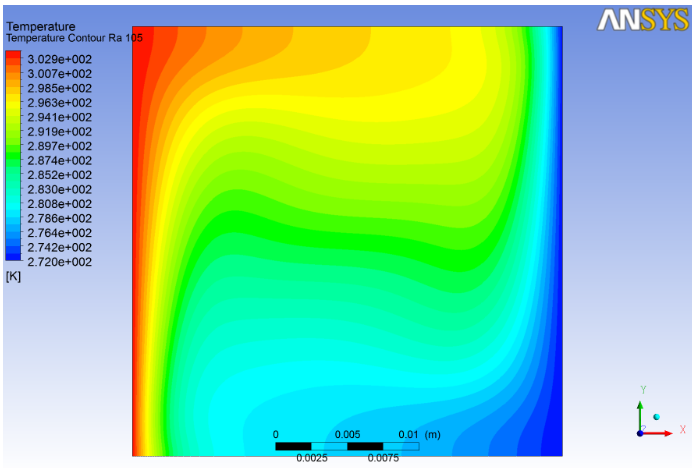


Figure 6 ***Temperature Contour for Ra 10^5 in steady state.***

Velocity Profile (Steady State)

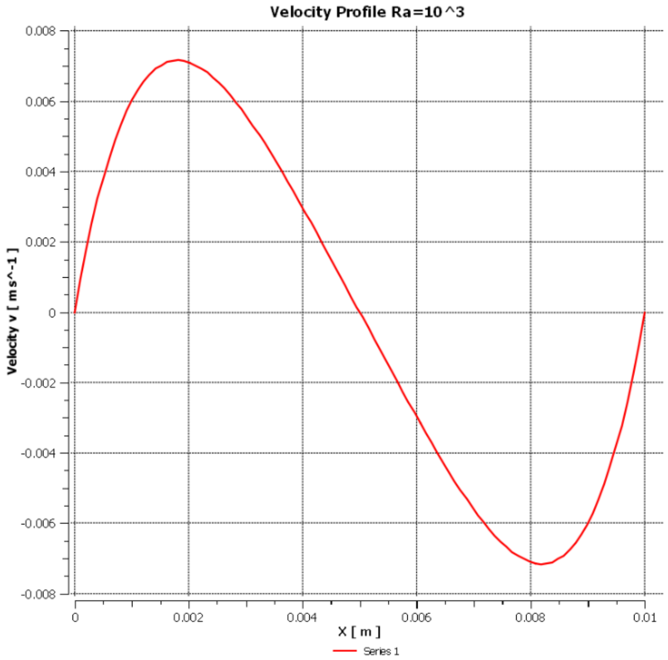


Figure 8 ***Velocity Profile for Ra 10^3 in steady state.***

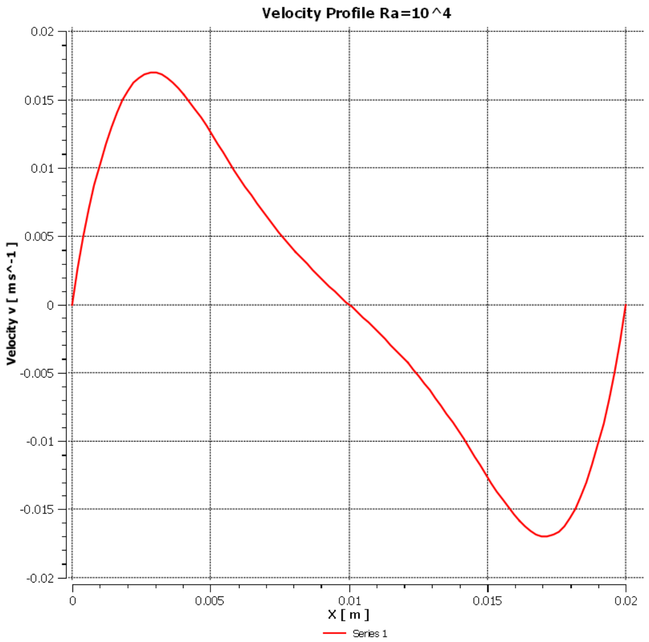


Figure 7 ***Velocity Profile for Ra 104 in steady state***.

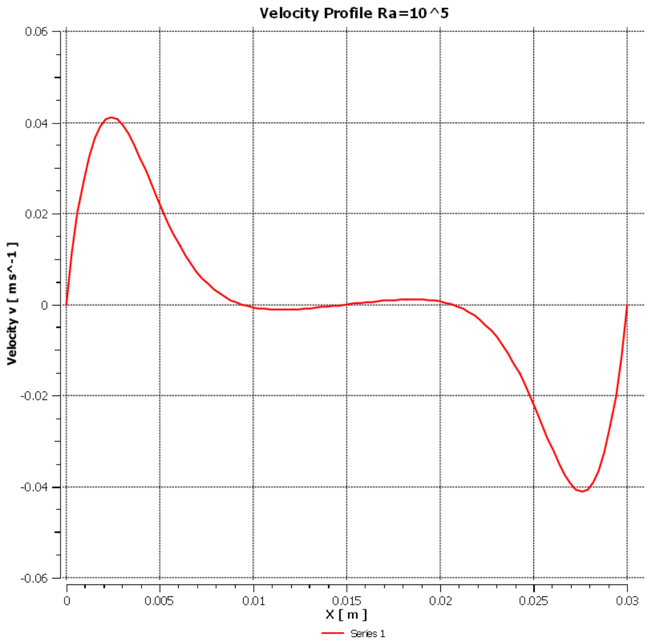
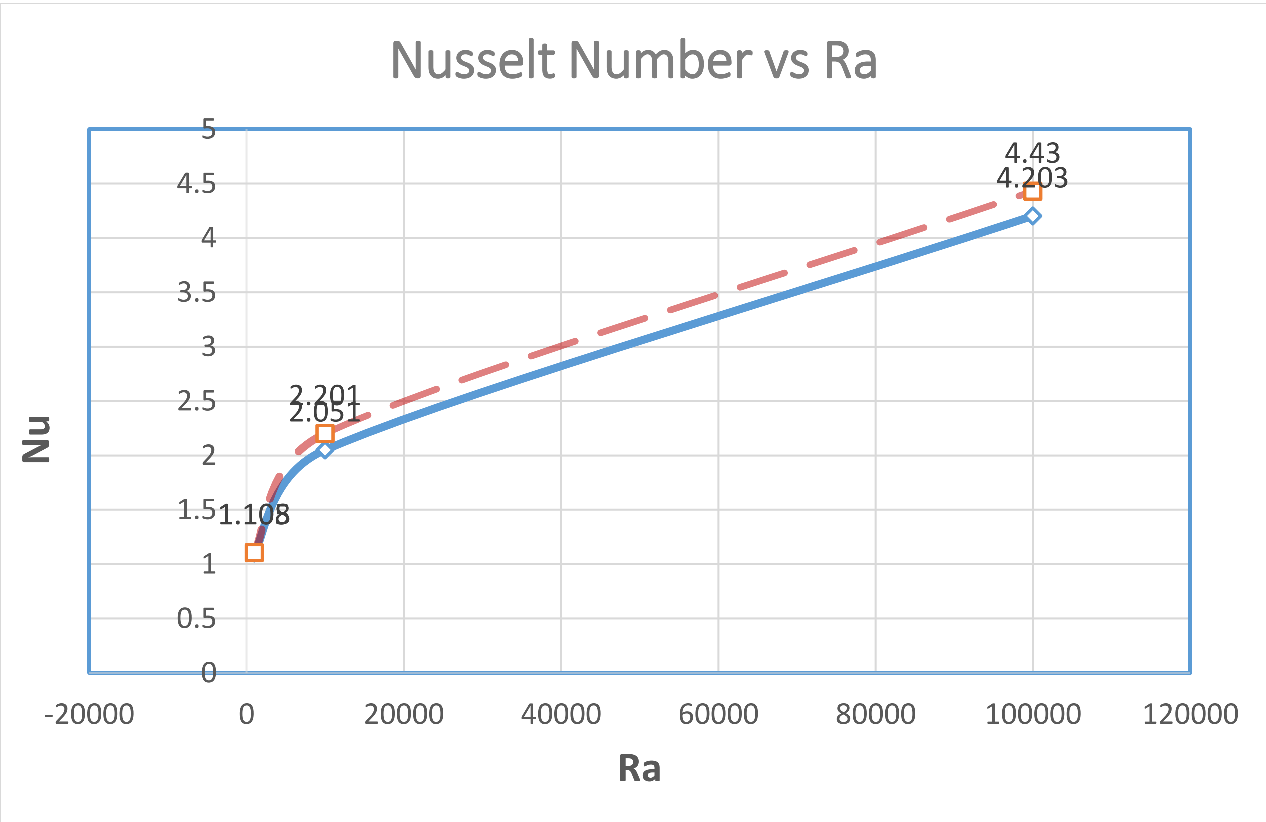


Figure 9 Velocity Profile for Ra 10^5 in steady state

Nusselt Number Validation for steady state

Parameter	Boussinesq Model Nu	Result Obtained in Markatos 1984 Nu
Ra=10^3	1.105	1.108
Ra=10^4	2.051	2.201
Ra=10^5	4.203	4.430



Transient State Analysis: (Velocity Profiles)

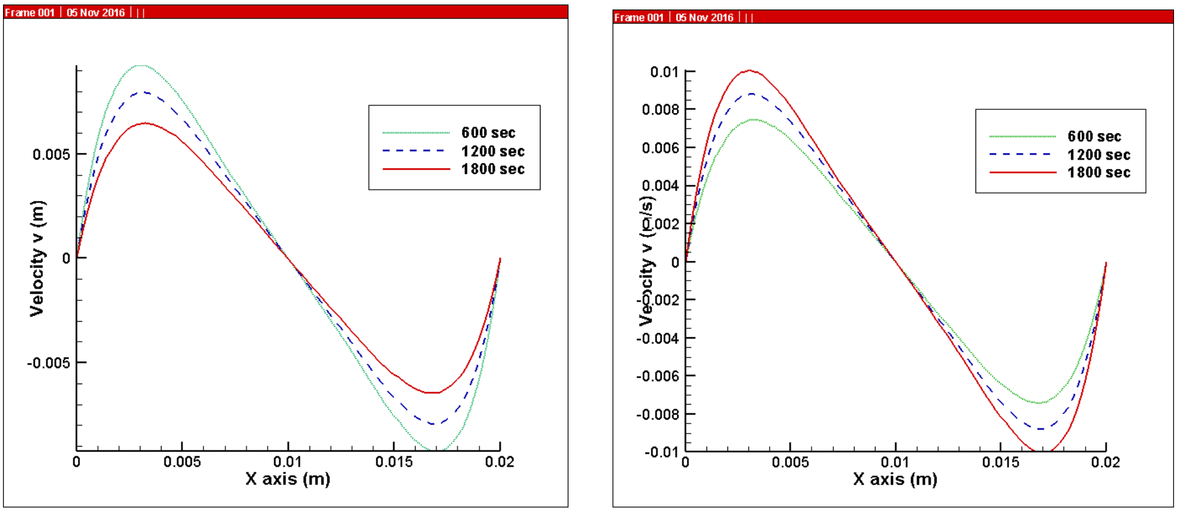


Figure 10 ***Velocity profile for ΔT= 4^oC***

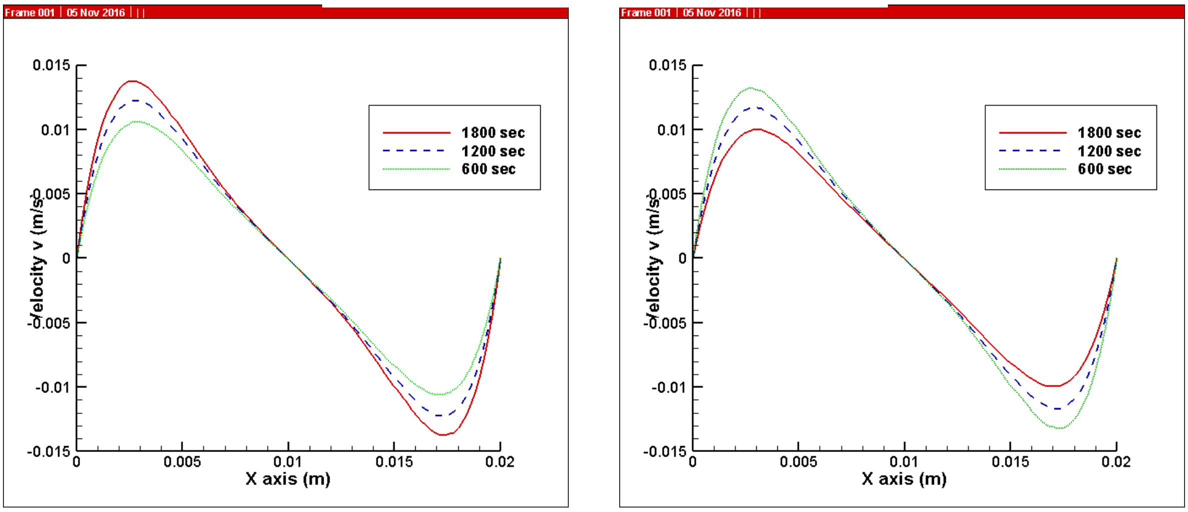


Figure 11 ***Velocity Profile for ΔT= 6^oC***

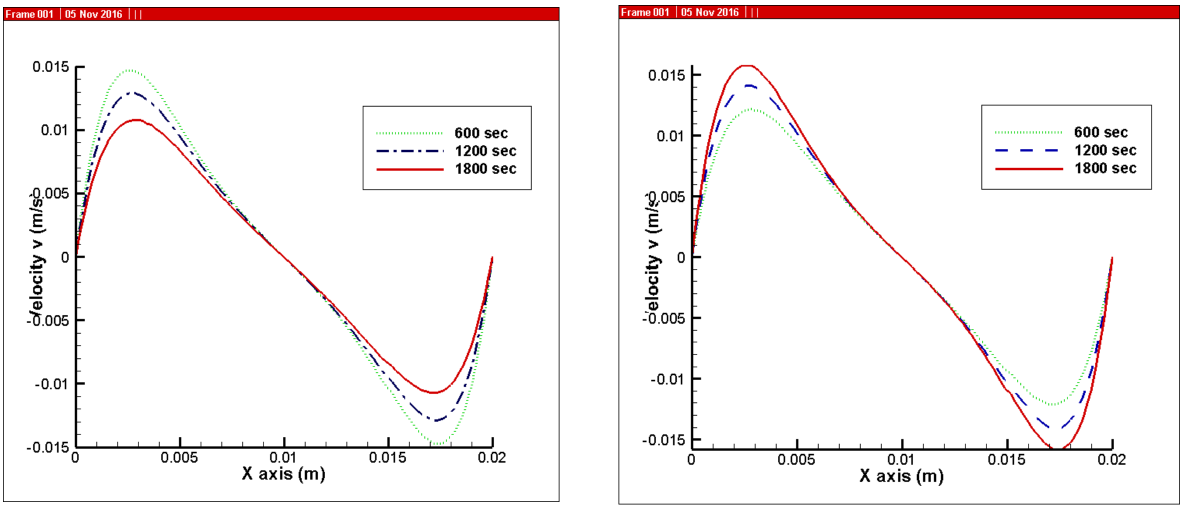


Figure 12 ***Velocity Profile for ΔT= 8^oC***

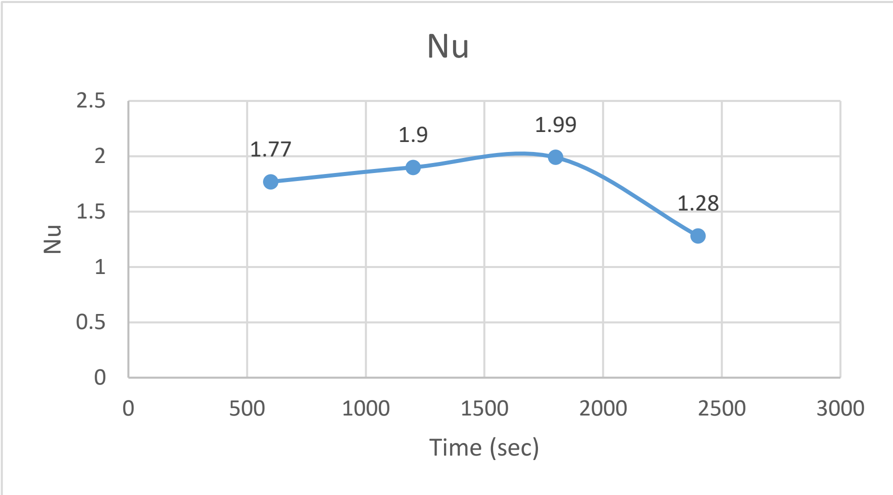


*Figure 13 Variation of Nusselt No for **ΔT= 8°C***

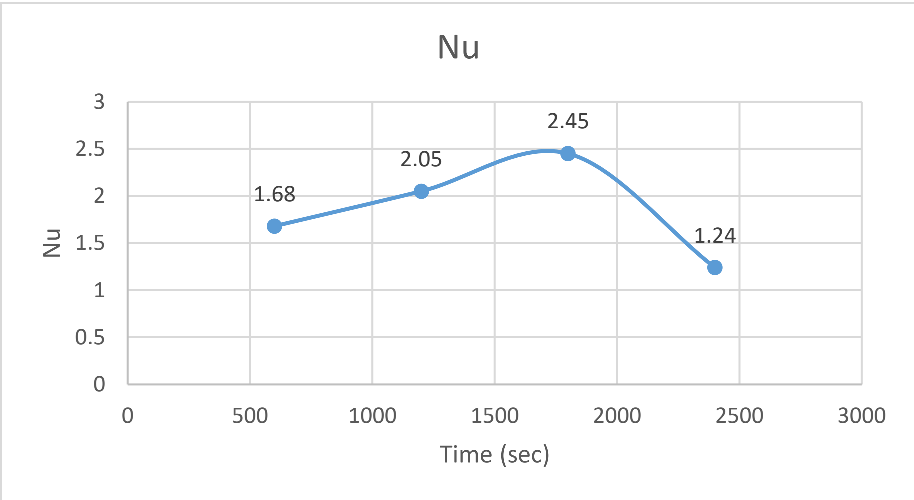


*Figure 15 Variation of Nusselt No **ΔT= 6°C***

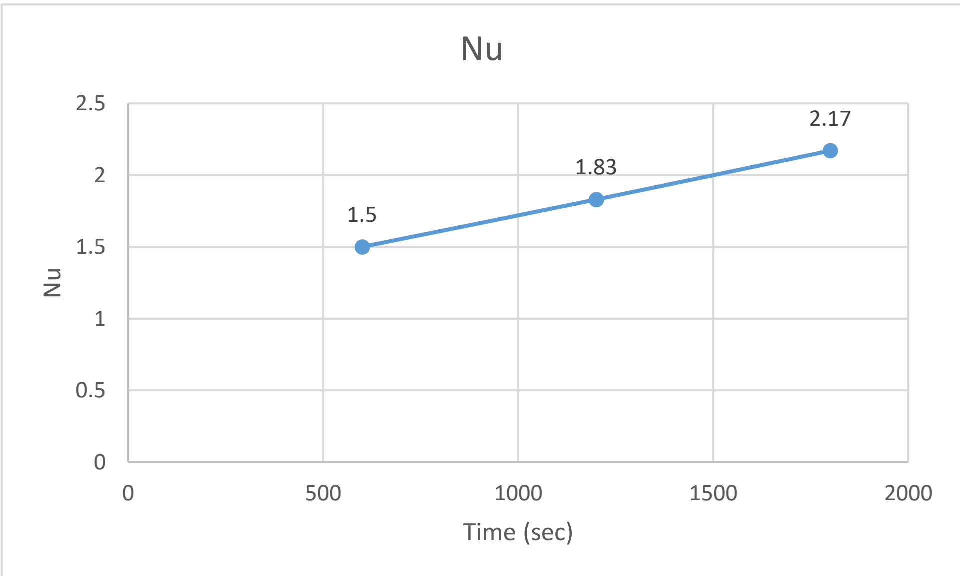


*Figure 14 Variation of Nusselt No **ΔT= 4°C***

For both the steady state and transient state analysis, the velocity profiles show that, with the increase of temperature difference (i.e. increase in Raleigh No.), the velocity in the axial direction also increases. The heat transfer rate is mainly dependent on the movement of the fluid whose convection currents drive the transfer of heat from one wall to the other.

The results obtained from the transient state also show a similar trend. The variation in Nusselt Number is in conjunction with the results obtained from the steady state analysis.

The data observed in this study can be useful in applications such as insulation of residential buildings' wall insulation and also in cases of industrial furnaces.

The distance between the walls also play a significant role in the heat transfer rate where the obtained data shows that the increase in the air gap results in better insulation.

6 References

[1] N. C. MARKATOS and K. A. PERICLEOUS, "LAMINAR AND TURBULENT NATURAL CONVECTION IN AN ENCLOSED CAVITY", Concentration Heat and Momentum Limited, 40 High Street, Wimbledon, London SW19 SAU, U.K.

[2] A.A. Ganguli, A.B. Pandit∗, J.B. Joshi, "CFD simulation of heat transfer in a two-dimensional vertical enclosure", Chemical Engineering Research and Design.

[3] Osman Turan , Robert J. Poole , Nilanjan Chakraborty, "Influences of boundary conditions on laminar natural convection in rectangular enclosures with differentially heated side walls".

[4] Huimin Cui, Feng Xu, Suvash C. Saha, "A three-dimensional simulation of transient natural convection in a triangular cavity", International Journal of Heat and Mass Transfer.

[5] John D. Hall, Adrian Bejan , Jack B. Chaddock, "Transient natural convection in a rectangular enclosure with one heated side wall".

[6] Patterson, J. C. and Imberger, J. Unsteady natural convection in a rectangular cavity. J. Fluid Mechanics, 1980, 100, 65-86

[7] Ivey, G. N. Experiments on transient natural convection in a cavity. J. Fluid Mechanics, 1984, 144, 389-401