FOSSEE, IIT Bombay

OpenFOAM Case Study Project

May 14, 2020

Natural Convection in an Enclosure Heated From Bottom

Shubhanshu Rai

M.Tech Scholar

Mechanical Engineering

Indian Institute of Technology, Jodhpur

email: rai.3@iitj.ac.in

ACKNOWLEDGEMENT

I want to start by expressing gratitude toward my thesis supervisor **Dr Shobhana Singh,** Assistant Professor, IIT- Jodhpur for her guidance, supervision and support that provided me with the trust and tools to create and evolve throughout this project. Her valuable inputs and suggestions helped me a lot and kept me motivated throughout the duration.

I am also very much blissful to all the other faculty, staff members of the department, Research Scholars who are working in the Mechanical Engineering department, IIT-Jodhpur for their advice and support.

Table of Contents

List of Figures

List of Abbreviations

CFD	Computational Fluid Dynamics
FEM	Finite Element Method
FVM	Finite Volume Method
OpenFOAM	Open-source Field Operation & Manipulation
Gr	Grashof Number
Pr	Prandtl Number
Ra	Rayleigh Number
H	Vertical Distance (y direction)
L	Longitudinal Distance (x- direction)
K	Thermal Conductivity (W/m-k)
k	Turbulent Kinetic Energy
C_p	Specific heat(J/Kg-K)
g	Acceleration due to gravity(m^2/s)
u	Velocity in x direction (m/s)
v	Velocity in y direction(m/s)
w	Velocity in z direction(m/s)
ΔT	Temperature Difference (K)
T	Absolute Temperature
q”	Heat flux(W/m^2)
β	Coefficient of Thermal Expansion (1/K)
ρ_0	Density of Fluid at Initial Temperature (kg/m^3)

Subscripts

t	“turbulent”
eff	“effective”

Abstract

A fluid (air) filled in an enclosure having dimesion 1m x 1m x 3mm, a vertical wall at x=0 and at x = L is subjected to ambient atmospheric temperature of 295K, bottom face is at constant temperature of 305K, while fluid initial temperature is taken as 300K. Top face is insulated/adiabatic. To simulate the 2-dimesional case front and back faces are taken as "empty" type. In Open FOAM v-7, "buoyantBoussinesqPimpleFoam" solver is merged with "buoyantPimpleFoam", although Boussinesq approximation can be applied easily by using the same solver .i.e. buoyantPimpleFoam. It is transient solver deals with natural as well as forced convection for compressible & incompressible fluid flow. k-Epsilon- turbulence model is used to solve the above-stated problem.

Geometry & Meshing: Geometry and meshing are done by using utilities offered by OpenFOAM named as blockMesh.

length = 1m

height = 1m

width = 3mm

1. Introduction

Natural convection within the enclosure is different from unbounded natural convection like natural convection from a flat plate, solid cylinder, sphere placed in an atmosphere. Enclosure natural convection is also known as internal convection, Natural convection in encloser mostly depends on geometry, shape and orientation[1].

Consider the following enclosure given in figure 1, Assume top face is insulated/adiabatic, a vertical wall at x=0 and at x = L is subjected to ambient atmospheric temperature of 295K, bottom face is subjected to constant temperature of 305K, while fulid initial temperature is taken as 300K. Length of the enclosure is L(x), and height is H(y). In our problem enclosure is squared shape hence, L = H.

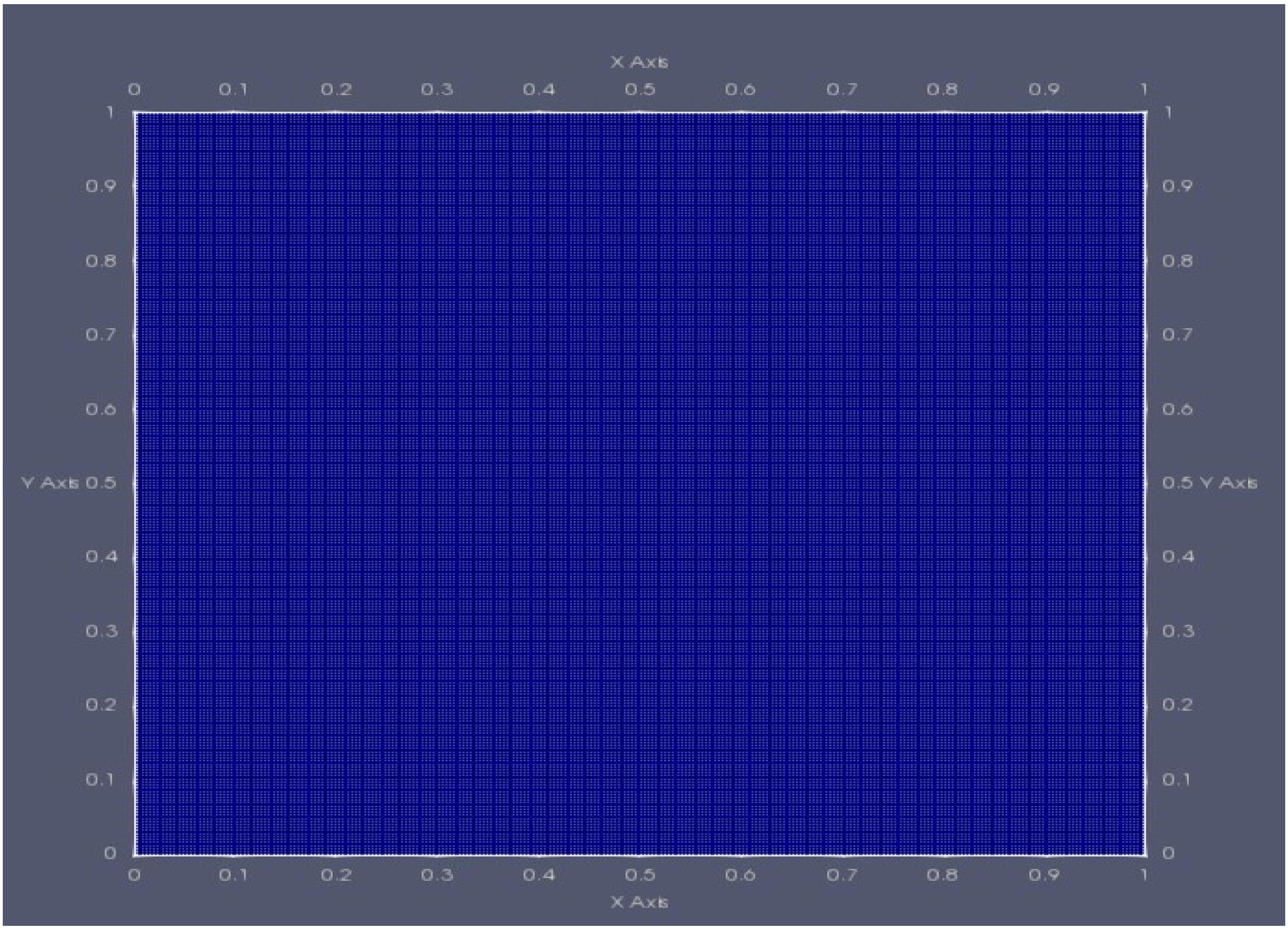

Fig1.1: enclosure with mesh elements

1.1 Enclosure having heat supply from the bottom

As mentioned above in this case just merely the existence of temperature difference is not sufficient condition for the formation convection current, which is a result of buoyancy effect .i.e. difference in density of the fluid. In this analysis, Grashof number(Gr) is $12.898\text{x}10^8$, Prandtl number(Pr) is 0.705 and Rayleigh number which is defined as product of Gr and Pr is $9.093\text{x}10^8$.

When temperature difference exceeds the critical value, then only convection current is formed. Experimental results show that when Ra ≥ 1708, where Ra is Rayleigh number, Ra is defined as the product of Grashof number(Gr) and Prandtl number(Pr). 1708 value for Ra is also known as critical Rayleigh Number[2].

Further, Gr and Pr are defined as:

$$Gr = \frac{\rho^2 H^3 \beta g \Delta T}{\mu^2} \tag{1}$$

$$Pr = \frac{\mu C_p}{K} \tag{2}$$

$$Ra = Gr.Pr \tag{3}$$

Where

ρ is density of fluid

H is height of enclosure(m)

β is coefficient of thermal expansion

ΔT is difference of temperature

μ is dynamic viscosity(Pa.s)

K is conductivity of fluid(W/m.k)

C_p is heat capacity of fluid (j/kg.K)

2. Problem Statement

In this problem, an enclosure is created using blockMesh utility offered by OpenFOAM, and meshing is done using the same. The heat is supplied at the bottom of the enclosure in the form temperature of magnitude 305K. Vertical faces named as “left” & “right” is supplied at uniform temperature of 295K. Front and back faces are “empty” type to impletment 2D simulation. Top wall is adiabatic/insulated to avoid the loss of heat from the system. Initial temperature of the fluid is taken as 300K and all the thermophysical properties are taken at 300K, all the properties of air, which is used as the fluid in our case is assumed as constant but density of water is considered as variable, because it is only because of change in density of water, convection current form, so to incorporate the variable density, boussinesq approximation is used, according to this approximation change in density is assumed as negative liner function of temperature.

Mathematically

$$\Delta\rho = -(\rho_0 \beta \Delta T) \tag{4}$$

Where

ρ_0 is initial density of fluid

β is coefficient of thermal expansion

ΔT = T - T_{ref}

The above approximation mentioned in equation is valid only when $\Delta\rho \ll \rho_0$

3. Governing Equations

3.1) Mass Conservation

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho.v) = 0 \tag{5}$$

3.2) Momentum Equation

x-direction

$$\frac{\delta u}{\delta t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = -\frac{1}{\rho}\frac{\delta p}{\delta x} + \frac{\mu}{\rho}(\nabla^2 u) \tag{6}$$

y-directionn

$$\frac{\delta v}{\delta t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = -\frac{1}{\rho}\frac{\delta p}{\delta y} + \frac{\mu}{\rho}(\nabla^2 v) - g[1 - \beta(T - T_{ref})] \tag{7}$$

z-directionn

$$\frac{\delta w}{\delta t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} = -\frac{1}{\rho}\frac{\delta p}{\delta z} + \frac{\mu}{\rho}(\nabla^2 w) \tag{8}$$

3.3) Energy Conservation(Sensible)

$$\frac{\delta T}{\delta t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z} = \alpha_{eff}(\nabla^2 T) \tag{9}$$

$$\alpha_{eff} = \frac{\nu}{Pr} + \frac{\nu_t}{Pr_t} \tag{10}$$

Where,

ν is kinematic viscosity

Pr_t turbulent Prandtl number

α_{eff} effective thermal diffusivity

4) Simulation Procedure

When user goes for the CFD simulation, it merely means that fluid dynamics simulation by finite volume method. One can solve CFD simulations using Finite Element method, but FVM is more common. Almost all famous commercial as well as open-source software tools such as Fluent, OpenFOAM uses FVM as their solving procedure. FVM method is useful to solve an algebraic form of partial differential equations for simulation. FVM considers the small tiny region of volume and this region is called as mesh/node. The governing equations of flow are applied and solved numerically at each node defined in OpenFOAM polymesh. OpenFOAM case setup generally contains three folders named as 0, constant and system.

a) “0” folder contains initial and boundary condition for velocity, temperature, pressure etc. and these properties are calculated during the simulation.

b) “constant” folder has files for thermophysical properties, turbulence modelling and also contains information about the mesh.

c) “system” folder contains files like blockMeshDict, controlDict, fvSchemes, fvSolution, these files control simulation. It includes information on time step, various schemes used, tolerances relaxation factor etc.

4.1 Geometry and Mesh

First of all, model of the enclosure is created by blockMesh and meshing is done by using the same. Structured hexagonal shape mesh is created. Three hundred elements are used in x,y direction and one element is used in the z-direction.

The volume/size of each meshing element is 3.333mm x 3.333mm x 3mm

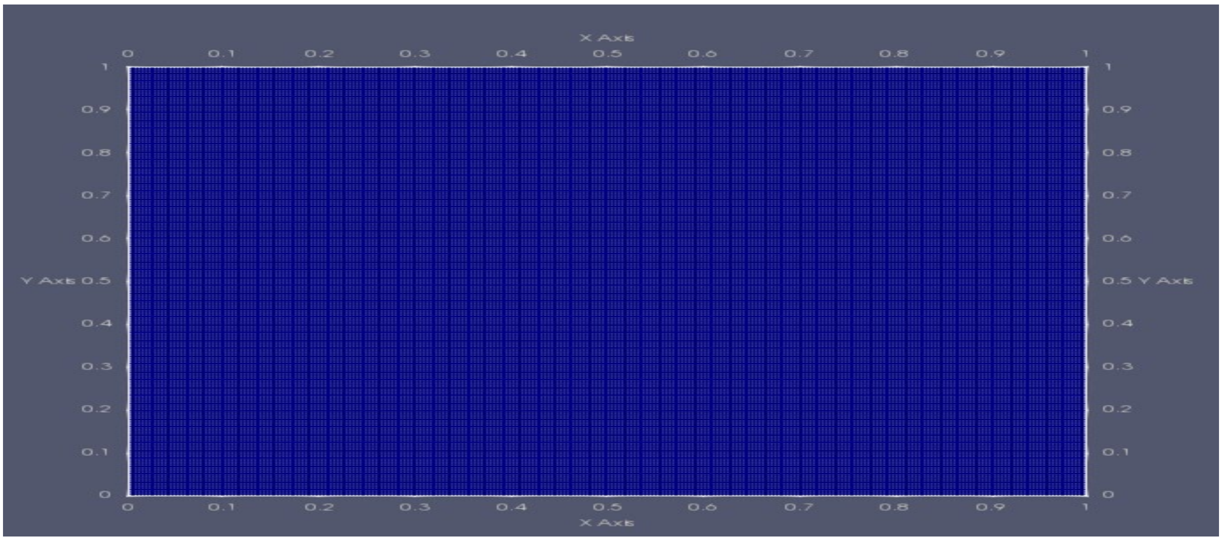

Fig4.1: Meshing of enclosure

4.2 Initial and Boundary Conditions

Initial and boundary conditions are defined in “0” folder for each properties. For temperature, “fixedValue” is used at the bottom, left & right faces of the enclosure to maintain the constant temperature condition. While top face is subjected to zero Gradient to implement adiabatic condition, for velocity no-slip boundary conditions are used for all the faces of the enclosure. For front and back faces “empty” type is defined for each variable.

4.3 Solver

The solver used is “buoyantPimpleFoam”, this is transient solver deals with natural as well as forced convection for compressible & incompressible fluid flow. I have applied Boussinesq approximation in my problem so to incorporate that we will have to change equationOfState to Boussinesq in “thermoPhysicalProperties” file located in “constant” folder. In OpenFoam v-7 “buoyantBoussinesqPimpleFOAM” solver is merged with “buoyantPimpleFoam” solver but approximation can be implemented by the method mentioned above.

Simulation done for 500second using time set of 0.01 second using OpenFoam v-7.

5. Results

Following are the images of convection current and temperature variation within the enclosure at a different timeinterval. The numerical investigation is done with study of formation and development of convection current with time and conclusion is made.

At 50second, convection current is shown by figure 5.1, the convection current here is weak. In subsequent figures the convection current is strong and can be seen clearly because cooled air moves downward and circular flow increseases resulting into Murshrrom profile is observed from 50 second onward.

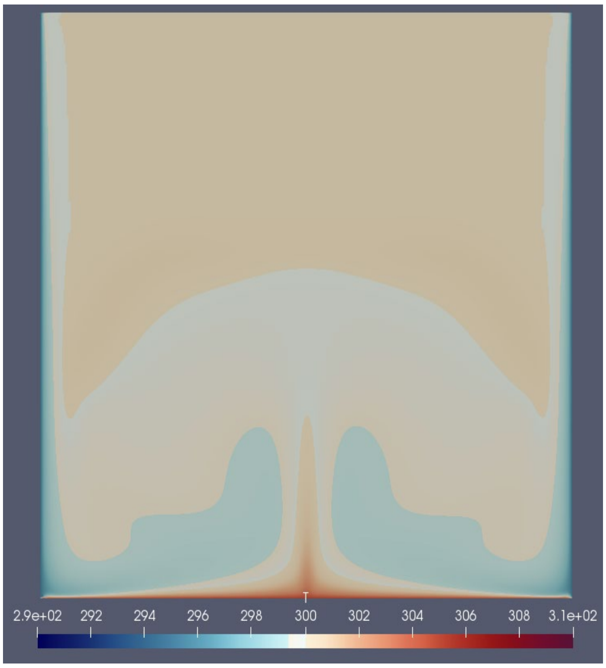

Fig:5.1: Convection current at t = 50sec.

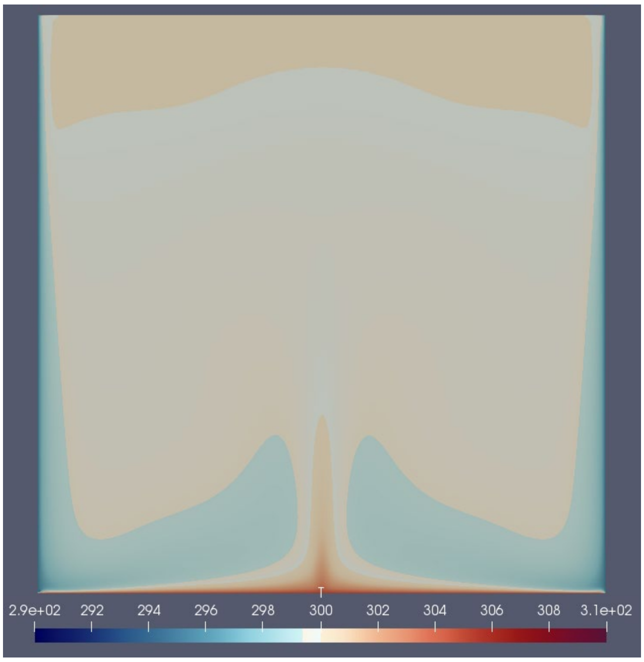

Fig:5.2: Convection current at t = 100sec.

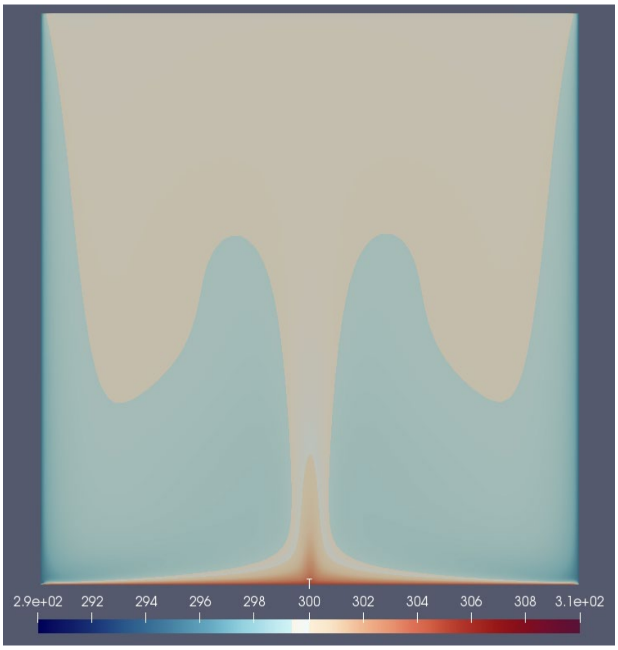

Fig:5.3: Convection current at t = 200 sec

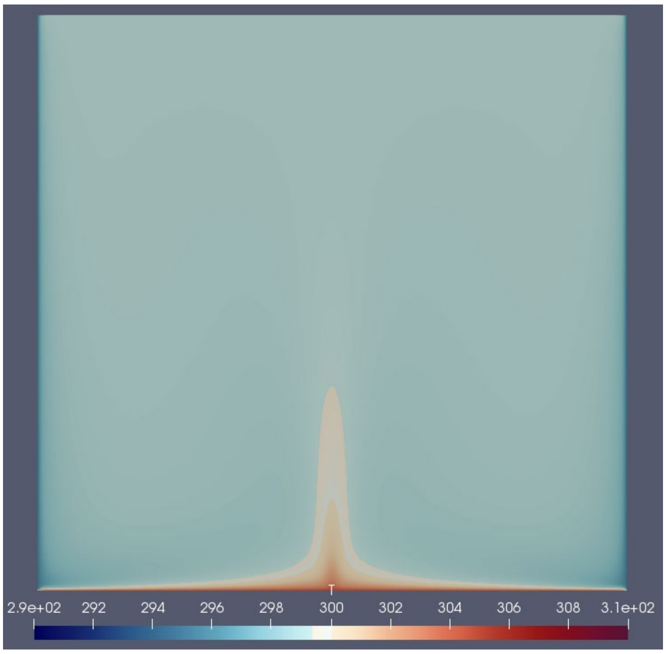

Fig 5.4: Convection current at t = 400 sec.

6. Validation of Result

Similar numerical and experimental analysis is done by M. Paroncini et al. [3], they did similar analysis with an enclosure heated at the bottom and cooled fron the vertical sides. They have used heater length at the bottom of enclosure of 20% to 80% of the total bottom surface. So our case study can be compared with heater of length of magnitude of 0.8 times of bottom length. Following are the convection current profiles are obtained from experimental analysis, which matches to a greater extent to our numerical simulation results.

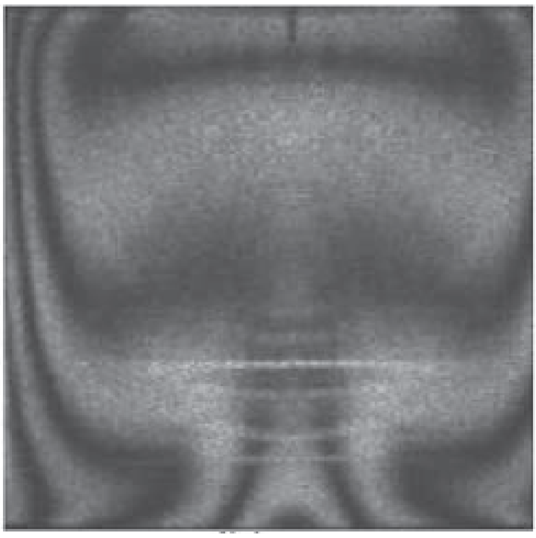

Fig 6.1: Convection current for Ra of order 10^5

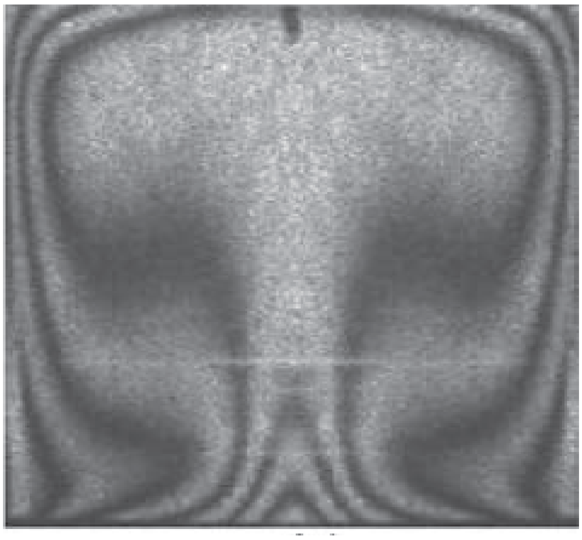

Fig 6.2: Convection current for Ra of order 10^6

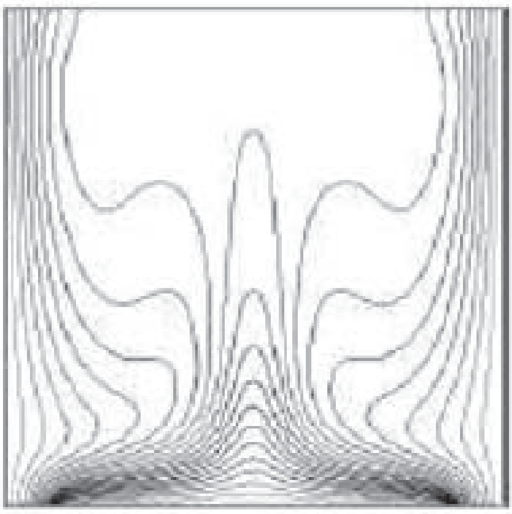

Fig 6.3: temperature contour for Ra of order 10^5

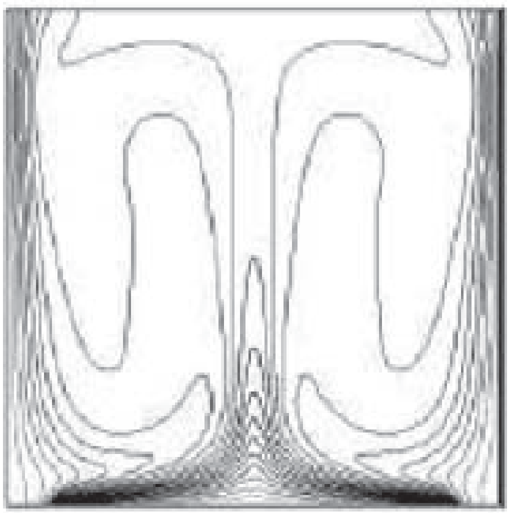

Fig 6.4: temperature contour for Ra of order 10^6

The temperature contour shown in figure 6.3 and 6.4 matches with figure figure 5.4 and 5.5 to a good extent, which validates the simulation result. Further, the figure 6.5 is temperature contour plot obtained from the simulation, it matches with result obtained by M. Paroncini et al. [3] from numerical simulation.

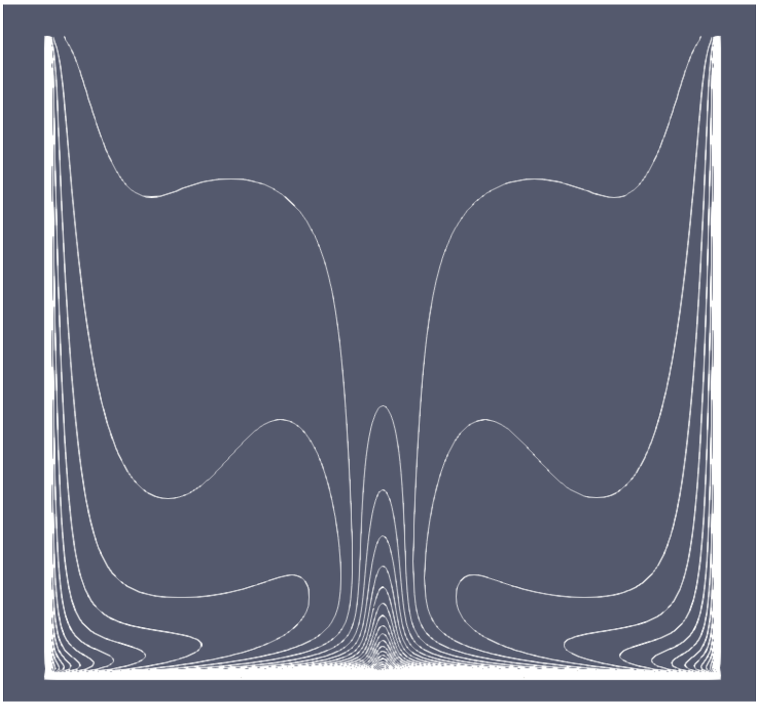

Fig 6.5: Temperature contour from simulation for Ra of order 10^8

7. References

[1] Convection Heat Transfer, Fourth Edition. *Adrian Bejan*., Published 2013 by John Wiley & Sons, Inc. pp. 248-249.

[2] A. Pellew and R. V. Southwell, On maintained convective motion in a fluid heated from below, *Proc. R. Soc.*, Vol. A176, 1940, pp. 312–343.

[3] B. Calcagni, F. Marsili and M. Paroncini, Natural convective heat transfer in square enclosures heated from below, Applied Thermal Engineering 25 (2005), pp. 2522–2531.