Numerical study on Buoyancy Induced Flow in Enclosures Connected by Horizontal vents

Ravi Kumar

A Dissertation Submitted to

Indian Institute of Technology Hyderabad

In Partial Fulfillment of the Requirements for

The Degree of Master of Technology

भारतीय प्रौद्योगिकी संस्थान हैदराबाद
Indian Institute of Technology Hyderabad

Department of Mechanical Engineering

July, 2012

Declaration

I declare that this written submission represents my ideas in my own words, and where others' ideas or words have been included, I have adequately cited and referenced the original sources. I also declare that I have adhered to all principles of academic honesty and integrity and have not misrepresented or fabricated or falsified any idea/data/fact/source in my submission. I understand that any violation of the above will be a cause for disciplinary action by the Institute and can also evoke penal action from the sources that have thus not been properly cited, or from whom proper permission has not been taken when needed.

(Signature)

(Ravi Kumar)

(ME10M10)

Approval Sheet

This thesis entitled Numerical Study on Buoyancy Induced Flow in Enclosures Connected by Horizontal Vents by Ravi Kumar is approved for the degree of Master of Technology from IIT Hyderabad.

(Dr. Raja Banerjee) Examiner
Dept. of Mechanical Engg.
IITH

(Dr. Vinod Janardhanan) Examiner
Dept. of Chemical Engg.
IITH

(Dr.K.Venkatasubbaiah) Adviser
Dept. of Mechanical Engg.
IITH

(Dr. Narasimha Mangadoddy) Chairman
Dept. of Chemical Engg.
IITH

Acknowledgements

I would like to express my sincere gratitude to my advisor Dr.K.Venkatasubbaiah for his continuous support, valuable suggestion and immense knowledge towards my M.Tech study. I would also like to thank him for giving me an opportunity to explore myself in the area of CFD. He helped me in always, as an advisor he enriched my technical skills, spent valuable time for discussion, provided ample time to think and implement, as a friend he was always beside me to feed with moral support, motivation, and encouragement till the fulfillment of my Thesis.

Besides my advisor I would like to thank whole faculty of Mechanical Engineering Department, for their effort in making me understand the Engineering concepts during my course work which was very useful in the accomplishment of my thesis, I owe my thanks to the institute IIT HYDERABAD for providing high class facilities properly and in time towards my M.Tech study.

I express my thanks to Ph.D. scholar R.Harish for his suggestions and comments in the preparation of my thesis report.

My sincere thanks to Project associates V.Srikanth and Madhu Pondicheri for helping me in CAE lab whenever I was into some troublesome with system errors.

My special acknowledgement to all my classmates Amit Dighe, Omprakashraj, Raja Jayasingh, Mrunalini for their help and support.

I would like to extend my thanks to all 600 series hostel mates for setting up a fun filled environment during my stay.

Last but not the least I would like to pay high regards to my beloved parents Mahadevarao Salagar and Annapurna Salagar and all my family members for their support, blessings and inspiration throughout my work.

Dedicated to my parents

Abstract

The buoyancy-induced flow generated by a heat source such as fire in square enclosures connected by horizontal or ceiling vent has been studied numerically. A two-dimensional laminar and turbulent natural convection flow is investigated with the buoyancy term represented by the Boussinesq approximation. The physical model consists of two enclosures connected by horizontal vent with a finite-size of heat source located centrally on the bottom wall of enclosure. The effects of vent aspect ratio and Rayleigh number on flow field characteristics in enclosures are studied in detail. The results show a significant change in flow behavior for varying vent aspect ratio at a fixed Rayleigh number. A bidirectional and oscillating flow across the vent occurs due to buoyancy. The intensity of thermal plume increases and flow becomes chaotic in enclosures with increase in Rayleigh number. The thermal plume quickly enters to the top enclosure with increase in vent width and decrease in vent thickness.

Nomenclature

D	width of enclosure
g	acceleration due to gravity
Gr	Grashof number $= \frac{g\beta\Delta T H^3}{\nu^2}$
H	height of enclosure
h	heat transfer coefficient
k	thermal conductivity
l	heat source length
L	height of went
L/D	aspect ratio of vent
Nu	Nusselt number
Nu_{avg}	average Nusselt number
p	pressure
Pr	$Prandtl\ number = \frac{\nu}{\alpha}$
Ra	Rayleigh number $= \frac{g\beta\Delta T H^3}{\nu\alpha}$
t	time
T	Temperature
T_∞	initial fluid temperature
T_s	heat source temperature
ΔT	initial temperature difference between heat source and fluid
u, v	components of velocity in X and Y directions
V_y	vertical velocity component
X, Y	horizontal and vertical coordinates
α	Thermal diffusivity
β	Volumetric thermal expansion co-efficient
ν	kinematic viscosity of fluid
ρ	density
c_p	specific heat

Contents

Chapter 1

Introduction

1.1 Introduction

Buoyancy induced flows occur due to density differences in the fluid resulting from temperature difference. Study of Buoyancy induced or Natural convection flows are important because most of the heat transfer applications involve natural convection such as cooling of electronic devices and IC engines etc. Researchers has shown a lot of interest in natural convection flow in enclosures because of its lot of practical engineering applications such as nuclear reactor cooling, solar collector, flows arising in rooms due to due to thermal sources, flow encountered in chimneys, fluid entrapped in large elevator shafts etc.

One of the important and practical applications as fires in building where natural convection induces strong buoyancy induced flows. Study of natural convection effects that arise in such rooms would help in predicting the spread and understanding of fire. In such configuration the flow of air and combustion products across vents governs the growth and spread of fires in compartments and buildings. The rate of inflow of oxygen from the ambient determines the combustion process and the energy release rates in fires for many practical circumstances. Similarly, the spread of fire to adjoining areas is strongly dependent on the resulting flow through vents. In the context of fire safety in buildings, the first thing is to evacuate the smoke and heat released by the fire. The hot gas flow characteristics in a building due to fire are important for the design of fire safety system such as water spray sprinklers. This problem can be modeled as buoyancy driven flow in partial enclosures. Flow through apertures connecting two enclosures has been a subject of study for more than four decades. Earlier studies were limited to vertical partition. The fundamental difference between flow through a vent in a vertical partition and flow through a vent in a horizontal partition is the stable stratification of fluid in the former case, whereas the configuration is unstable for the latter. It is important to understand the transport process which arises due to density differences across horizontal or ceiling vents. The underlying mechanisms may cause the flow to become unstable, giving rise to complicated flow patterns and significant effect on the fire or heat removal.

1.2 Literature survey

The thermal plume and hot gas flow characteristics in buildings due to fire are important for the design of fire safety system such as water spray sprinklers. This problem can be modeled as buoyancy induced flows in partial enclosures.

Natural convection heat transfer is being studied since few decades and in this era a lot of work can be found in the literature [1], John Patterson and Jorg Imberger [2] studied the unsteady natural convection in rectangular enclosure with differential wall heating and for aspect ratio less than 1, they considered non dimensional parameters Rayleigh and Prandtl number in determining the different flow regimes of the flow.

Orhan Aydin and Wen-Jei Yang [3] were the first to study natural convection of air in a rectangular enclosure heated from below with different sized heat source and symmetrical cooling of side walls and they showed that average Nusselt number increases with an increase in Rayleigh number and also with an increase in heat source size.

Kandaswamy et al. [4] studied the Buoyancy effect inside a square cavity by changing the orientation of heated plate, their observation was with increase in Grashof number heat transfer increases in both vertical and horizontal orientation of plate, also heat transfer is found to increase with increase in aspect ratio of heat source, one important aspect of orientation of heated plate observed by them was an increase in heat transfer rate for vertical positioning of heat source than the horizontal.

Pravez Alam et al.[5] performed numerical investigation of natural convection in a rectangular enclosure due to partial heating and cooling at vertical walls, by varying the parameters like Rayleigh number and aspect ratio. They concluded that the heat transfer rate and the thermal buoyancy force increases with Rayleigh number, and with increase in aspect ratio from 0.5 to 1, average heat transfer rate increase and is found maximum for an aspect ratio of 1, beyond this it decreases.

Anil et al.[6] studied turbulent natural convection flow in enclosure with localized heating from below. They have considered the varying heat source length with two different boundary conditions imposed on heat source such as isothermal and isoflux or constant heat flux. They found that Nusselt number increases with increase in width of heat source for isothermal condition while it decreases for the isoflux condition.

Brown [7] was the first to study natural convection with square openings in horizontal partitions both analytically and experimentally, he considered heavier fluid on top of lighter fluid and air as the fluid medium with constant Prandtl number for all the cases, he recognized that heat and mass transfer increases with increase in partition thickness.

Ranganathan Kumar et al. [8] studied computationally the unsteady buoyancy flow in a large vertical enclosures connected by horizontal partition, in their work they fixed the vent aspect ratio to 1 and studied the fluid behavior for different Rayleigh number, based on the flow patterns in the vent, three flow regimes were identified conduction, counter current and the oscillatory flow regime.

Ahmed Sleiti [9] studied the effect of unsteady buoyancy induced flow with same configuration of large vertical enclosure with horizontal partition but this time with varying vent aspect ratio and constant Rayleigh number of 10000 unlike his first case, in this work he showed that increasing partition thickness decreases the flow exchange between the two enclosures and flow regime becomes conduction with low frequency, he also traced the velocity and temperature fields with respect to time to study the fluid behavior. Sleiti in his both work considered that enclosure is filled with lighter density fluid at the bottom and higher density fluid at the top and both the fluid in the enclosure is at rest at time t=0.

Effect of Partition thickness on buoyant exchange flow through horizontal opening in enclosures for different vent aspect ratio is studied by Harrison and Spall [10], they considered water as the fluid medium and concluded that the flow exchange increases with increasing partition thickness over the range $0.0376 \leq L/D < 0.3$, and decreases in the range of $0.3 < L/D \leq 1.0$. similar work is been done by Mercer and Thompson [11], but using brine solution and fresh water as their fluid medium and they found that the flow rates through inclined tubes decreased over the range $3.5 \leq L/D \leq 18$.

Singhal and Kumar [12] presented numerical results for the two-dimensional, unsteady, buoyant exchange flow through an opening in a horizontal partition and identified three different flow regimes, depending on the magnitude of the Rayleigh number, their opening was slightly offset from the geometric centerline consequently asymmetric flow patterns developed.

Markatos and Malin [13], and Abib and Jaluria [14] performed numerical simulations of counter current flow through a vertical partition. In their work they considered air as the working fluid to model fires in the room of a large building, investigations revealed that circulatory flow patterns develop, with the warmer air passing in one direction through the upper part of the opening and the cooler air passing in the opposite direction through the lower part of the opening, their models showed good agreement with previously attained experimental data.

Smoke produced in tunnels due to a fire catastrophe resembles fluid flow in enclosure, this flow regime is extensively studied by Kashef et al. [15].

Yapieng He and Paula Beever [16] experimentally studied buoyancy driven flow in building enclosures, subjected to fires and they tried to explain the concentration of the species and their velocity magnitudes entrapped in the enclosures.

Prahl and Emmons [17] conducted an experimental and theoretical study of the fire induced flow through an opening in vertical partition. McCaffrey and Quintiere [18] studied buoyancy driven counter current flows, generated by a fire source of constant heat release rate, in a full scale corridor burn room facility, their velocity profiles and mass balance analysis revealed a strong mixing and entrainment of flows in the corridor.

Other experimental studies on the transport process across horizontal vents in room fires have been carried out by Jaluria et al [19-21]. Tan and Jaluria [21] reported that, in the absence of a pressure difference but with heavier fluid overlying lighter fluid, a bidirectional flow arises across the vent due to buoyancy effects. The critical pressure has been reported at which transition from bidirectional to unidirectional flow arises across the vents. Any perturbations disturbing the equilibrium would lead to air movement and might cause an onset flow oscillations at the vent. The oscillations can be sustained or amplified when heat input is continued.

Venkatsubbiah and Jaluria [22] performed a numerical investigation of enclosure fire with single and multiple horizontal vents in their work they reported oscillatory bidirectional flow at the vent and also mentioned the critical Grashof number above which flow becomes chaotic nature in the enclosure.

Kerrison et al.[23] conducted a model test on enclosure fire with single ceiling vent and tried comparison with experimental observations, they concluded that their exist a linear relationship between frequency of vent temperature oscillation and heat release rate and this relationship exist only for heat release rate below a critical value.

Chow and Gao [24,25] studied the air flow motion induced across the vent due to thermal sources in an enclosure. They considered the ratio of buoyancy to inertia forces as the key parameter in determining air flow motion. In their similar work on oscillating behavior of fire induced air flow through the vent [25] they also developed a semi empirical formula using the parameters for the fire safety design purpose.

1.3 Motivation

Most of the earlier work on natural convection is limited to single enclosure, enclosures with differential wall heating, and partial enclosures with vertical vents. In the literature not much work has been reported on natural convection in two enclosures connected with either single or multiple ceiling vent, this was the main motivation for the present work, as this configuration also resembles a fire in two enclosures connected by horizontal vent, the flow field characteristics of this work may also be helpful for the design of water spray sprinklers.

1.4 Objectives of present study

- ❖ The effects of different parameters listed below on the flow field characteristics due to fire in enclosure are to be studied numerically.
    - Rayleigh Number
    - Effect of vent width
    - Effect of vent thickness
    - Multiple vents
    - Effect of turbulence

1.5 Overview of thesis

- Chapter 2 gives the description about the problem formulation, governing equations and boundary and initial conditions.
- Chapter 3 gives a brief description of mesh and some theory of numerical schemes and their solution methodology.
- Chapter 4 depicts about grid independence and validation.
- Chapter 5 Results and discussion single and multiple vent cases for both laminar and turbulent cases.
- Chapter 6 & 7 conclusion and scope for future work.

Chapter 2

Problem formulation

2.1 Physical model

The hot gas flow characteristics due to fire in a building or any enclosure that are connected by vents are important for the design of fire safety system such as water spray sprinklers. Such a problem can be modeled as buoyancy-driven flow in partial enclosures. The physical model consist of two square enclosures and these two enclosures are connected with a common horizontal rectangular vent, hence all communication between the two enclosures is carried out by this common vent, also a heat source of 0.2H is located centrally on the bottom wall of lower enclosure. Figure 2.1 represents the schematic diagram of the model.

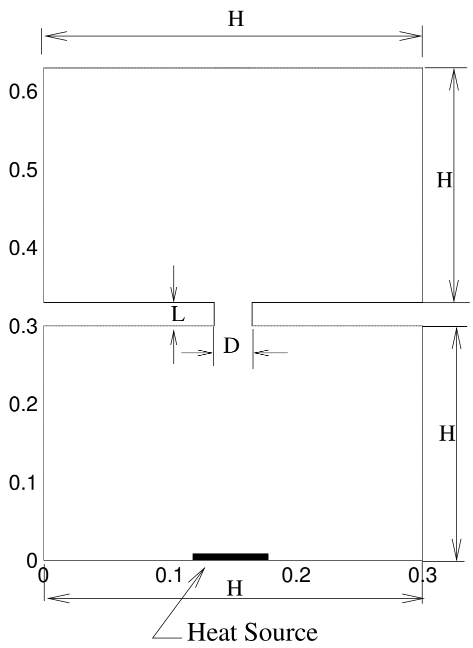


Figure 2.1: Schematic diagram of the physical model

Dimension: Side of square enclosure 0.3m.

The enclosure is filled with air at atmospheric condition at time t=0, the dimensions of the vent are adjusted so as to obtain different vent aspect ratios.

2.2 Governing equations

2.2.1 Boussinesq approximation

The buoyancy forces that arise as a result of temperature differences cause fluid flow in free convection. For low speed flows, the density changes associated to temperature changes are at most 1 percent. So we may treat the density as constant in all terms of the governing equation except in the buoyancy force term. This is known as the Boussinesq approximation.

As the density variation is only due to temperature variation we can use a variable β, known as volumetric thermal expansion co-efficient.

$$\beta = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p \tag{2.1}$$

the above equation can also be written as

$$(\rho_\infty - \rho) \sim \rho\beta(T - T_\infty) \tag{2.2}$$

From the above equation the density term in the momentum equation is replaced by the temperature terms which will contribute to buoyancy forces.

2.2.2 Continuity equation

This equation is obtained by applying the principle of conservation of mass on the fluid flow which states that mass can neither be created nor destroyed. For a two dimensional incompressible flow, the continuity equation is expressed as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2.3}$$

2.2.3 Momentum equation

Momentum equations are obtained by the application of Newton's second law, which states that rate of change of momentum of a body, is equal to the applied force. The application of this law on a fluid element gives rise to momentum equations which are also termed as Navier Stokes equations. For a two dimensional incompressible flow the momentum equations can be written as in equation (2.4, 2.5).

x- Momentum

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2.4}$$

y – Momentum

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) + \beta g(T - T_\infty) \tag{2.5}$$

2.2.4 Energy equation

The energy equation is derived from the first law of thermodynamics which states that the rate of change of energy is equal to rate of heat addition plus the rate of wok done. For a two dimensional, incompressible flow, with constant fluid properties the energy equation can be written as

$$\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} = \left(\frac{k}{\rho c_p}\right)\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \tag{2.6}$$

2.3 Boundary and Initial condition

- Boundary condition

  No slip boundary condition is applied on all the walls for the velocity field and a constant surface temperature T_s is applied on the heat source, and also all the walls except heat source are considered as adiabatic.

  - On all the walls

    $$u = v = 0$$

  - Adiabatic walls

    $$\frac{\partial T}{\partial n} = 0$$

  - Heat source

    $$T_s = \text{constant}$$

- Initial condition

  - At time t = 0.

    All the velocity components of fluid is equal to zero.

    $u = v = 0$

  - At time t = 0.

    Temperature of fluid is T_∞.

2.4 Grashof number

It is the ratio of buoyancy force to viscous acting on the fluid element and it is mathematically expressed as

$$\text{Grashof number} \quad = \frac{g\beta\Delta T H^3}{\nu^2}$$

2.5 Rayleigh number

Rayleigh number determines whether the flow of fluid is laminar or turbulent due to heat transfer, and it is given by the product of Grashof number and Prandtl number.

$$\text{Rayleigh number} \quad = \frac{g\beta\Delta T H^3}{\nu\alpha}$$

2.6 Nusselt number

It is the ratio of convective heat transfer rate to conduction heat transfer rate, it signifies whether conduction or convection heat transfer is dominant in a particular heat transfer application, if Nusselt number is greater than 1 convection heat transfer is dominant and if it is less than 1 than conduction heat transfer will be dominant. Sometimes it is also known as dimensionless temperature gradient at the surface.

$$Nu = \frac{hl}{k}$$

Chapter 3

Numerical Method

3.1 Meshing

The physical model consists of two enclosures connected by horizontal vent. Quadrilateral cells are used in the whole domain, the size of the cells is varied depending on the condition of flow i.e. laminar or turbulent in order to capture all the desired parameters. Figure 3.1 shows a schematic of the mesh.

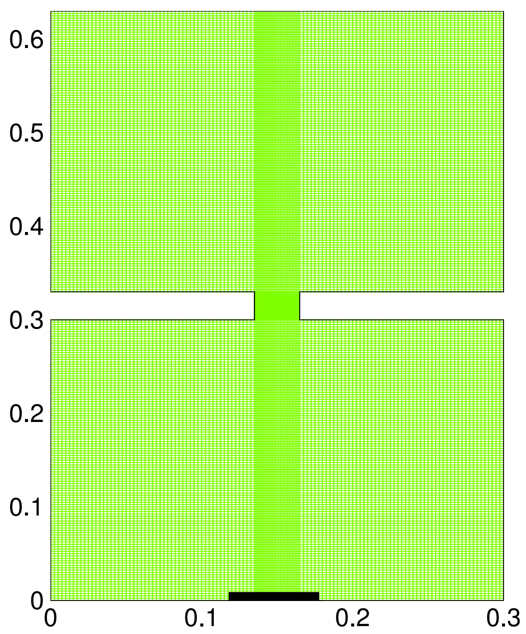


Figure 3.1: shows a schematic of the mesh.

The number of cells in the domain varies with different aspect ratio of the vent, Table 3.1. gives the information of number of cells in the domain for different aspect ratios.

Table 3.1: Represents number of cells for different vent aspect ratios

SI	Case	Vent thickness L	Vent width D	Aspect ratio L/D	No. of cells
1	Laminar	0.1H	0.1H	1	34020
2		0.1H	0.15H	0.66	36630
3		0.1H	0.2H	0.5	39240
4		0.2H	0.1H	2	34920
5		0.2H	0.15H	1.33	37980
6		0.2H	0.2H	1	41040
7		0.4H	0.1H	4	36720
8		0.4H	0.15H	2.67	40680
9		0.4H	0.2H	2	44640
10	Multiple vent	0.1H	0.1H	1	39240
11		0.1H	0.15H	0.66	44460
12		0.2H	0.1H	2	41040
13	Turbulence	0.1H	0.1H	1	180900
14		0.2H	0.1H	2	181800
15		0.1H	0.2H	0.5	181800

3.2 Numerical schemes

The governing equations are solved using commercial CFD solver FLUENT 13. Pressure based solver with second order unsteady formulation is employed. PISO algorithm is used for pressure-velocity coupling and the discretization of the momentum and energy equation is done using second order upwind scheme. The fluid properties are considered constant and Boussinesq approximation is made which considers density as constant in all the equations, except the buoyancy term. For turbulent flows Large Eddy Simulation (LES) with subgrid-scale model is employed.

3.3 Description of Numerical schemes

- Body force weighted
- PISO Algorithm

3.3.1 Body force weighted

The pressure terms in the governing equations are discretized using body force weighted technique, this technique computes the face pressure by assuming that the normal gradient of the difference between pressure and body forces is constant. This method is best suited for flows that involve buoyancy and axisymmetric swirl calculations.

3.3.2 PISO Algorithm

Piso stands for Pressure-Implicit with Splitting of Operators (PISO) and is a scheme for pressure velocity coupling. One of the limitations of the SIMPLE and SIMPLEC algorithms is that new velocities and corresponding fluxes do not satisfy the momentum balance after the pressure-correction equation is solved and hence the calculation must be repeated until the balance is satisfied [27]. So in order to improve the efficiency of this calculation, the PISO algorithm solves one predictor step and two pressure correction steps and hence it is an improved version of SIMPLE scheme, it is also strongly recommended for larger time step.

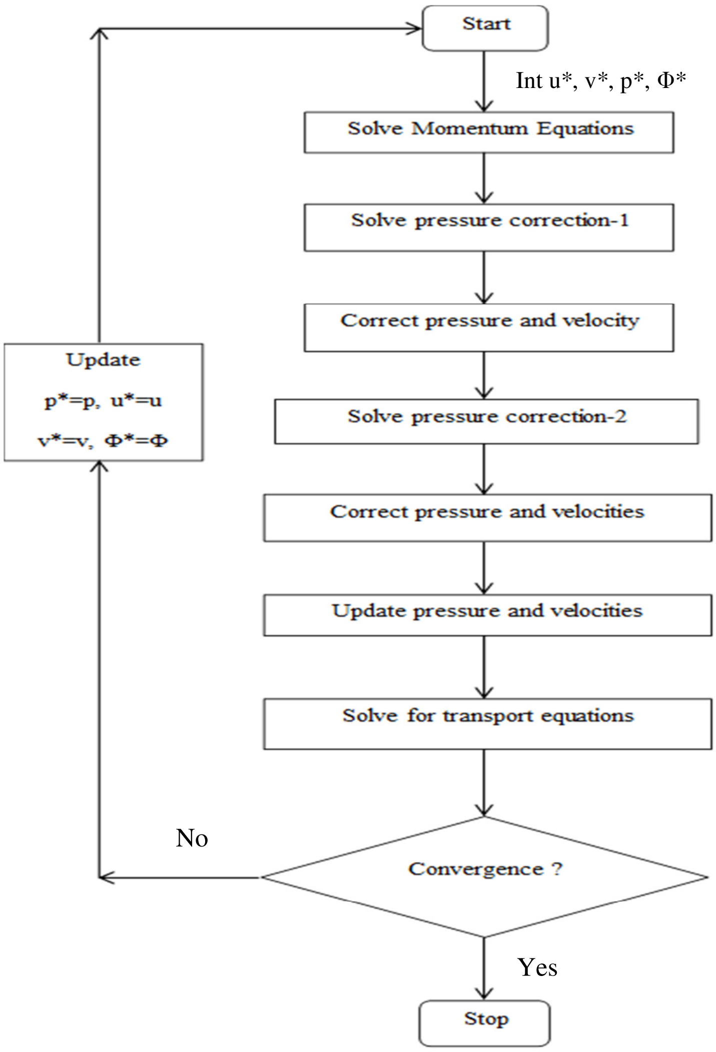


Figure 3.2: Flow chart of PISO Algorithm

Chapter 4

Grid independence and Validation

4.1 Grid independence

Grid independency study is carried out with four grid sizes and details are given in Table 4.1

Table 4.1: Different grid sizes

Sr.no	Mesh	Each Enclosure	Vent	Total cells
1	Fine	200x200	20x20	80400
2	Medium	120x120	12x12	28944
3	Coarse	100x100	10x10	20100
4	Medium (Fine vent)	138x120	30x30	34020

It is observed that, the variation in the velocity and temperature values inside the enclosure, for different grid sizes when calculated falls less than 1% and hence all the grids mentioned in Table 4.1 will approximately give the same value up to two decimal points. Therefore the grid with medium-fine vent is a good choice considering accuracy and computational time, hence all the calculations for laminar case is done using this grid. In order to analyze the grid independence quantitatively the temperature and velocity distribution plot on the horizontal surface at the center of bottom enclosure is plotted and is shown in Fig 4.1 and Fig 4.2. From these figures it can be observed that grid independence is achieved.

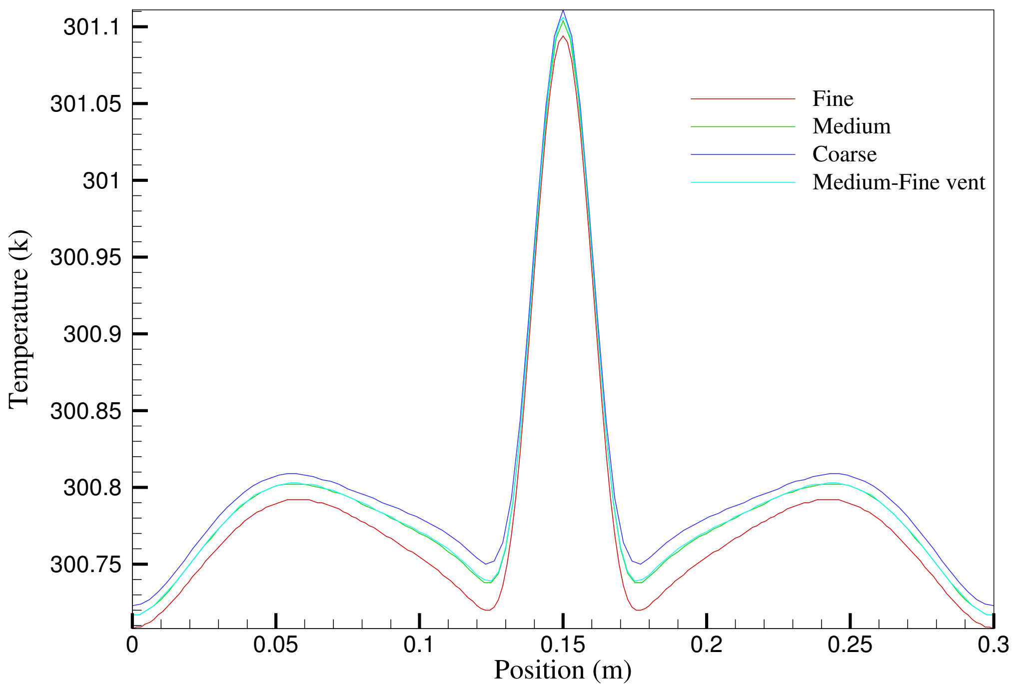


Figure 4.1: Temperature variation on horizontal surface at the center of bottom enclosure.

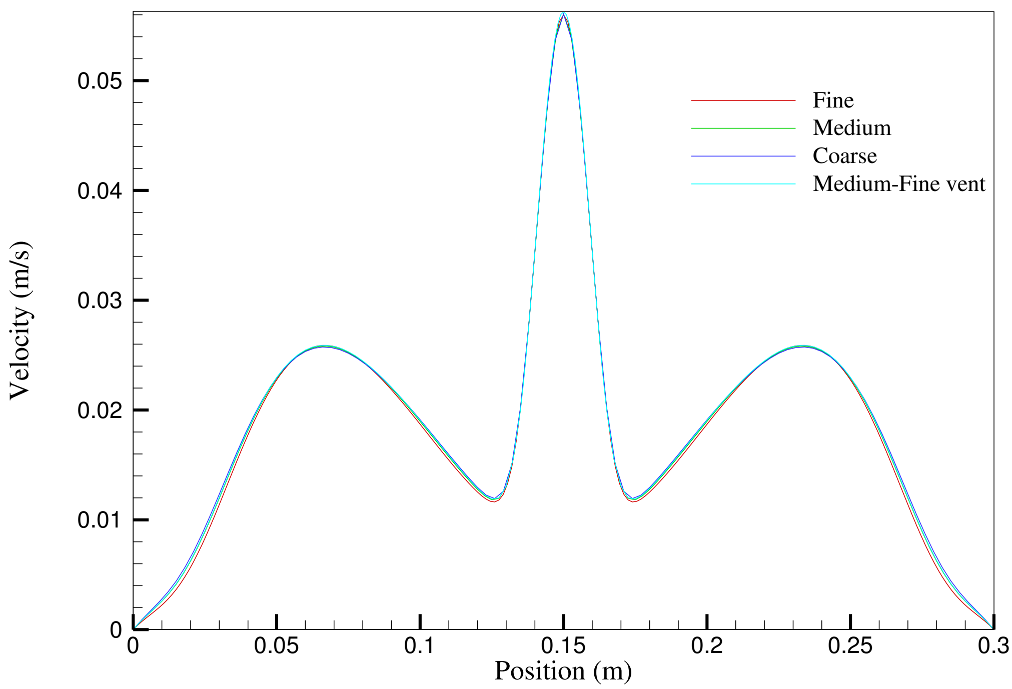
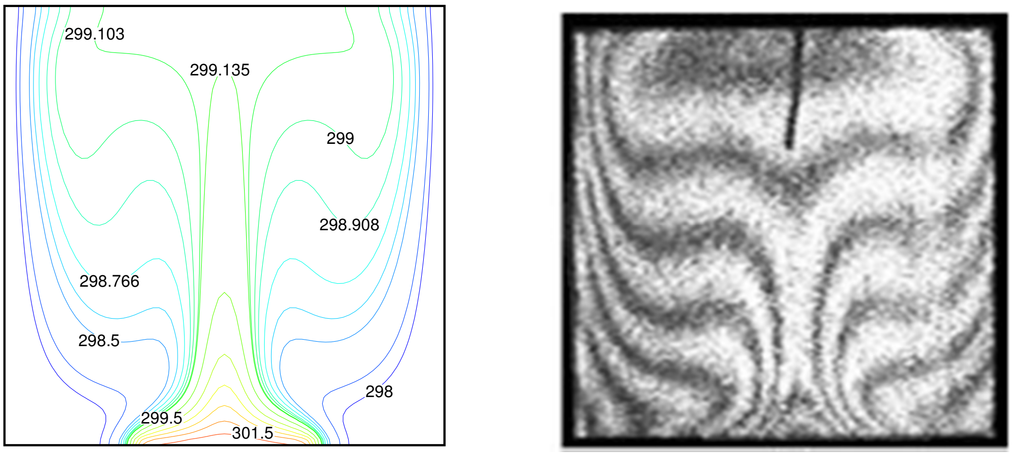


Figure 4.2: Velocity variation on horizontal surface at the center of bottom enclosure.

4.2 Validation

To validate the numerical methods, results are obtained for the experimental work of Calcagni et al. [26], experimentally studied the natural convection flow in a square enclosure heated from below. The experimental set up considered consisted of top and bottom surfaces made of plexiglass and considered as adiabatic except for the heated section, while the lateral vertical walls were made of aluminum and are cooled by circulating thermo- static fluid which maintains the walls at a constant temperature T_c. The heat source is located at the center of the bottom wall with a size of 0.4H and is maintained at a constant temperature T_s. Results for the case of Ra = 1.86 x10^5 are compared, and it is found that temperature contours are closely matching with Calcagni et al. [26] work, and average Nusselt number along the heat source obtained for the present case Nu_{avg} = 9.44 and this value is also closely agreed with experimental value Nu_{avg} = 11.6.



a) Numerical b) Experimental

Figure 4.3: Temperature contour for validation

Chapter 5

Results and Discussion

A two dimensional laminar and turbulent natural convection flow in square enclosures connected by horizontal vent is studied numerically and reported for different Rayleigh numbers. The value of Prandtl number Pr for air is taken as 0.72 for all the cases. The size of each enclosure is 0.3 m x 0.3 m and a constant size heat source is located at the center of bottom surface of size 0.2H. The effects of vent aspect ratio, Rayleigh number and turbulence on flow field characteristics in enclosures are reported here.

5.1 Effect of variable fluid properties

To see the effect of temperature on the properties of fluid, few simulations are performed assuming fluid properties as a piecewise function of temperature and the results for one particular case L/D=1, where L=0.2H, and Ra = 2.5×10^8 with corresponding temperature difference of $\Delta T = 100\ ^0C$ is tabulated in Table 5.1. The results are taken from the average values examined by creating different surfaces in the bottom enclosure as these surfaces will be near to heat surface the fluid in this region will be very much inclined to changes in temperature.

Table 5.1: Percentage change in fluid properties

Si No	Property	Constant value	Variable value	$\frac{\Delta\rho}{\rho}$
1	Density	1.1614(kg/m^3)	1.0660(kg/m^3)	0.0821

And also to see the effect of variable fluid properties on the temperature and velocities, plots are generated near the heat source and are shown in Fig 5.1 and Fig 5.2. From these results it can be visualized that the change in temperature and velocities are not varying abruptly, besides change in properties of fluid is also much less than one, hence Boussinesq approximation model [31] is assumed to give comparable results and hence is adopted in all the cases.

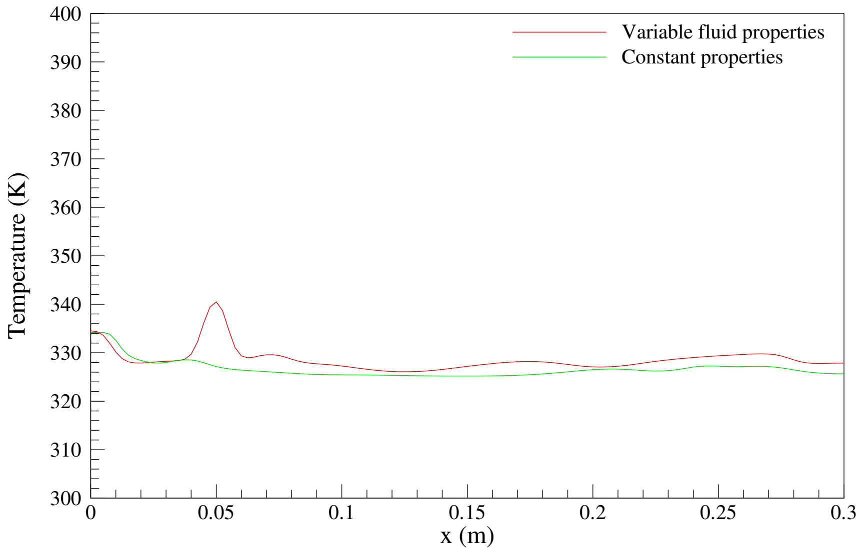


Figure 5.1: Comparison of temperature between constant and variable fluid properties

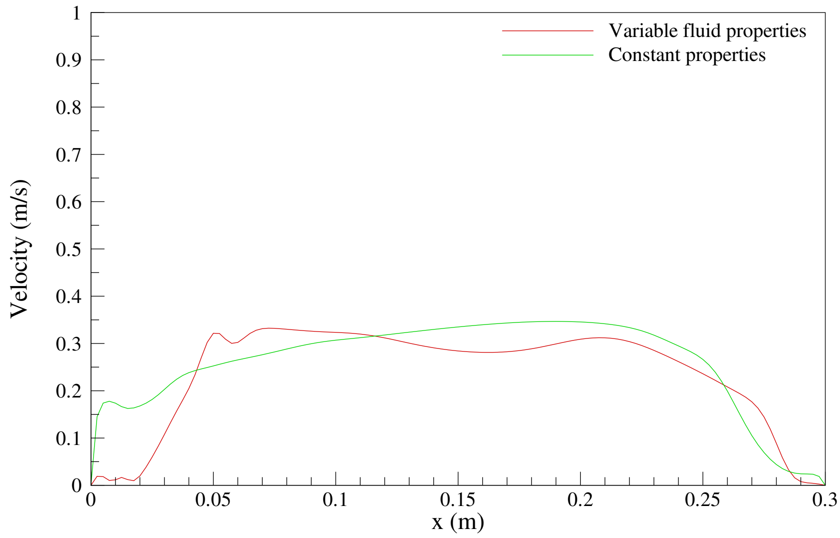


Figure 5.2: Comparison of velocity between constant and variable fluid properties

5.2 Time evolution

To examine how a thermal plume will initiate, grow and spreads in the enclosure with time, temperature and velocity contours at different times are shown in Fig. 5.3 and Fig.5.4 for vent aspect ratio L/D = 0.5, Rayleigh number Ra = 1.25×10^8, where the width of the vent D = 0.2H and temperature difference between heat source and fluid ΔT = 50 K.

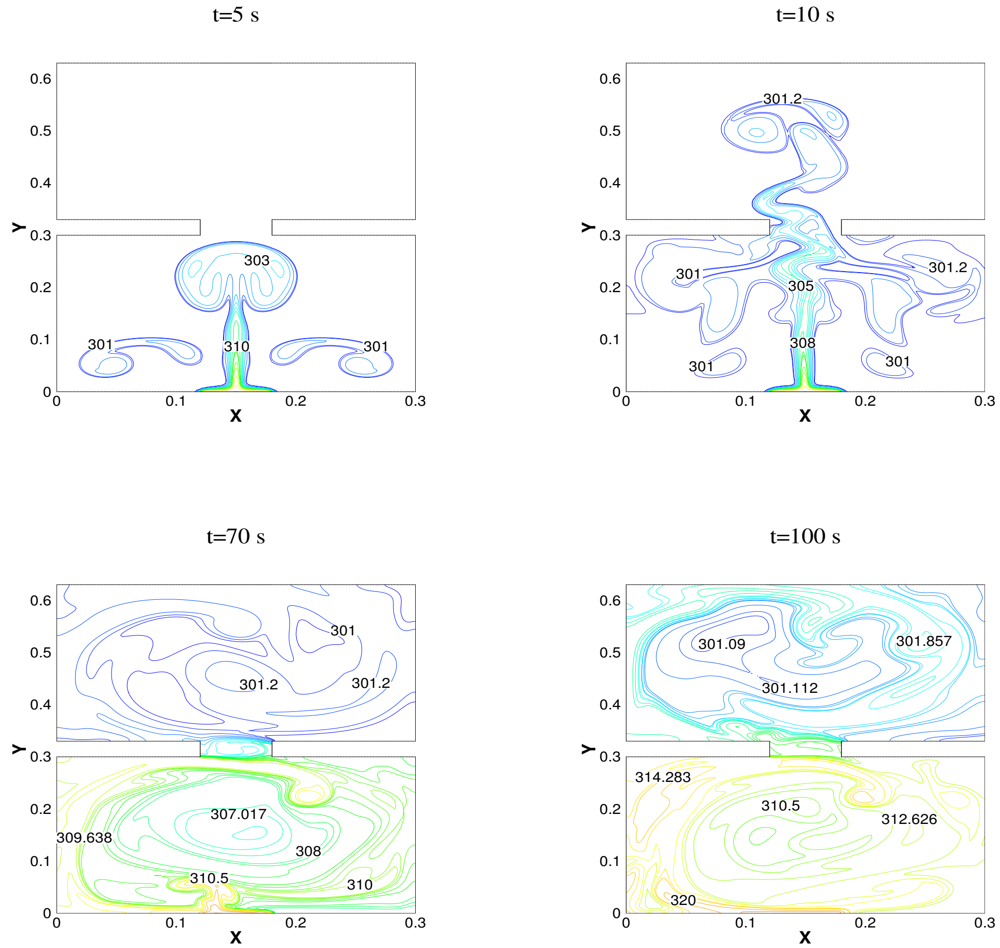


Figure 5.3: Time evolution temperature contours for L=0.1H D=0.2H and Ra=1.25×10^8.

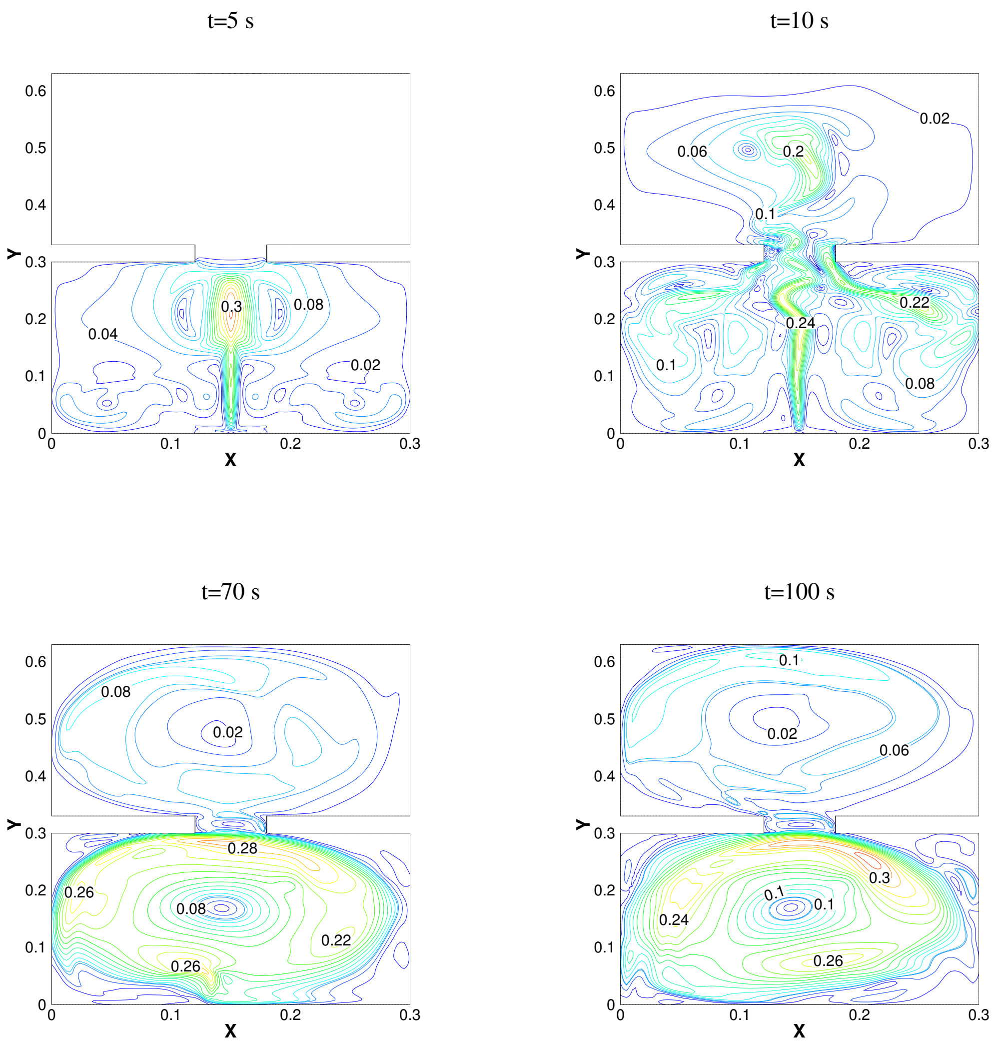


Figure 5.4: Time evolution velocity contours for L=0.1H D=0.2H and Ra=1.25×10^8.

From Fig. 5.3, it can be interpreted that the thermal plume grows and moves upwards from heat source with increase in time. The flow enters the top enclosure at time $5 < t < 10$ s and a quasi-steady state condition is reached at time $t \geq 100$ s. A density difference arises between hot fluid in bottom enclosure and cold fluid in top enclosure at the vent hence flow is bidirectional at the vent due to buoyancy effect. The strength of the plume increases with increase in time in the top enclosure. From Fig.5.4, the convection currents are formed due to temperature difference between heat source and fluid. The fluid gains the momentum and moves upwards from the heat source with increase in time. In enclosures, the change in velocity is small compared to change in temperature due to the differences between momentum and thermal diffusivities which represents $Pr < 1.0$.

5.3 Effect of Rayleigh number

A study on the effect of Rayleigh number on the fluid flow inside the enclosure is made and the effects on the temperature and velocities are interpreted. In order to show the effect, results for one such case with an AR=0.5, where width of the vent is 0.2H, and temperature difference varying from $\Delta T = 10$ K to $\Delta T = 100$ K is presented in Fig [5.5 - 5.7] which represent the temperature and velocity contour at different time levels. As the Rayleigh number increases from Ra = 2.5×10^7 to 2.5×10^8, the intensity of thermal plume increases and flow becomes chaotic as time progresses in the enclosures. The flow in the vent is found to oscillate for most of the time for Rayleigh number Ra $\leq 5 \times 10^7$ and chaotic at higher Rayleigh numbers due to higher intensity of buoyancy, also the velocity of the fluid is found to increase with increase in Rayleigh number. The sudden growth of vortices occurs at high Rayleigh number compared to low Ra flows. From Fig 5.5 and Fig 5.7 it is observed that with increase in Rayleigh number the fluid gains higher temperature and momentum and hence starts moving quickly towards the top enclosure, while moving towards the top enclosure the hot fluid displaces the cold fluid and this displaced fluid due to higher density moves downwards in the bottom enclosure. It can also be emphasized that higher the Rayleigh lower the time needed by the fluid to enter into the top enclosure, this can be clearly viewed from the temperature and velocity contours.

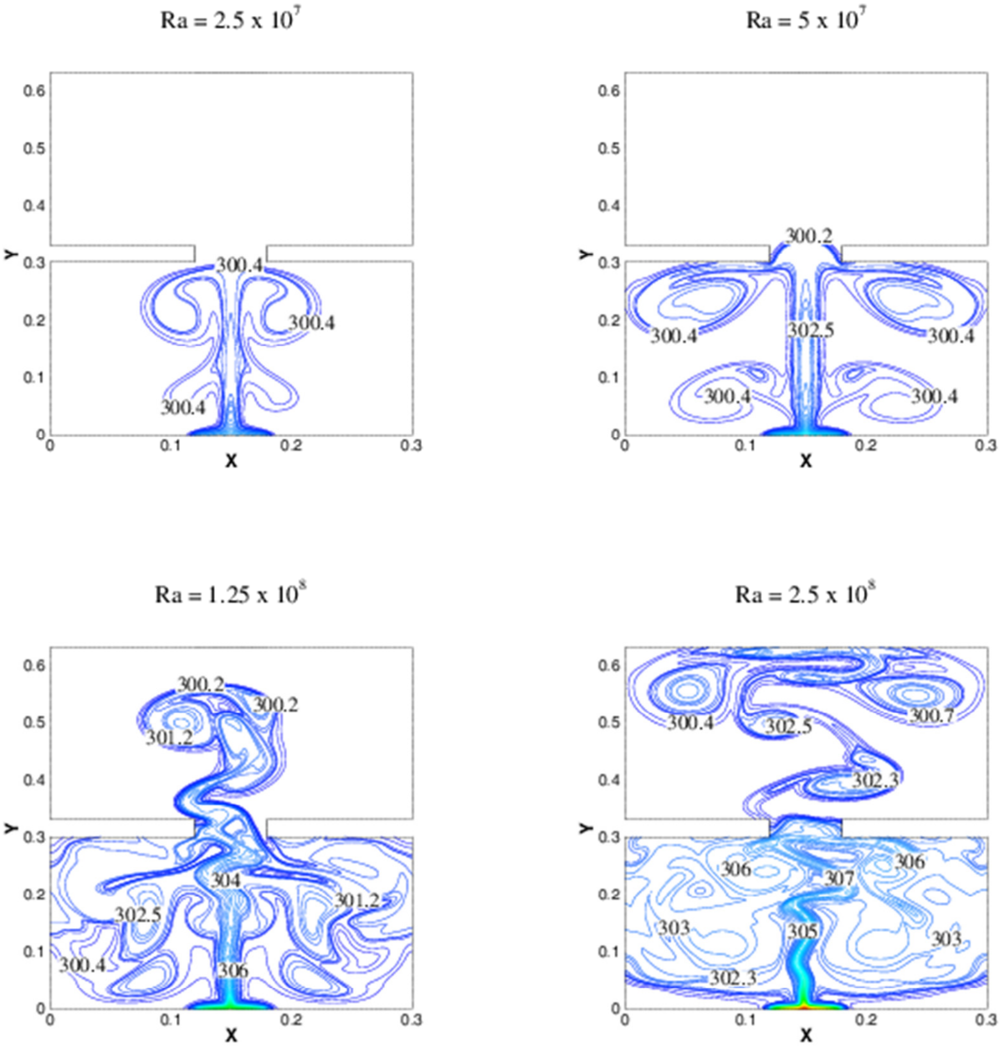


Figure 5.5: Temperature contour for different Rayleigh for the case L=0.1H D=0.2H and time t=10 sec.

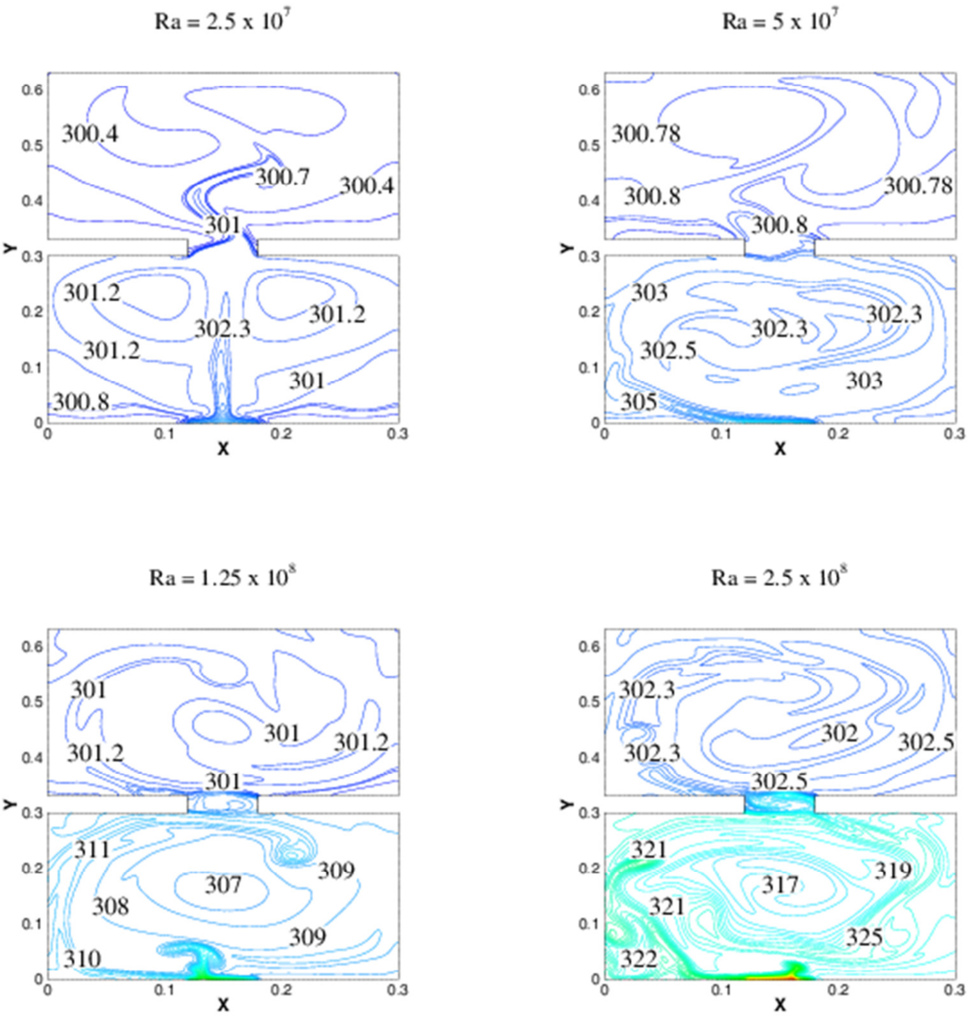


Figure 5.6: Temperature contour for different Rayleigh for the case L=0.1H D=0.2H and at time t=70 sec.

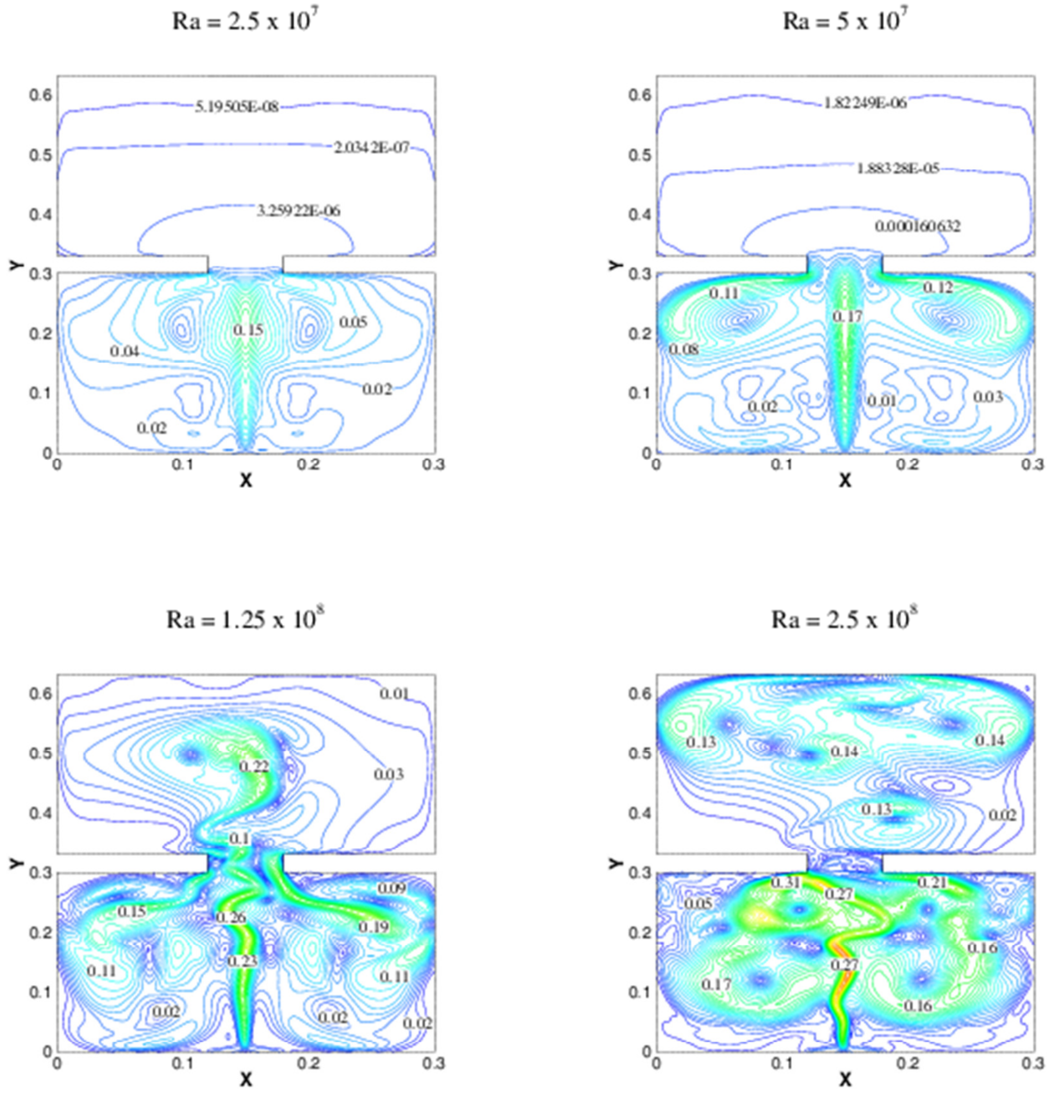


Figure 5.7: Velocity contour for different Rayleigh for the case L=0.1H D=0.2H and at time t =10 sec.

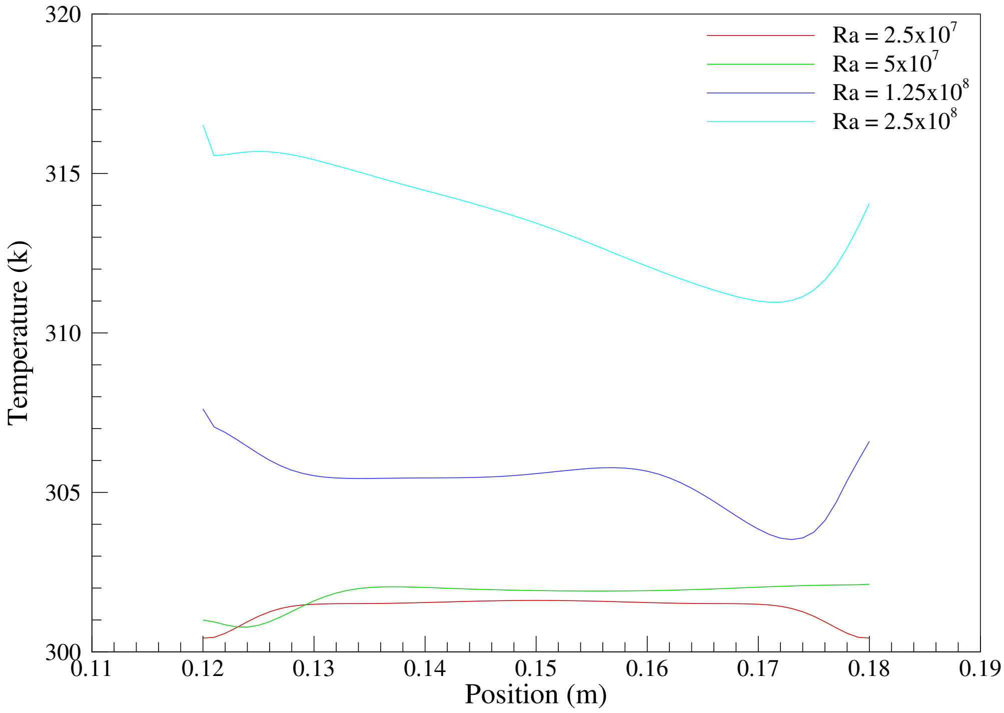


Figure 5.8: Variation of temperature at vent bottom with increase in Rayleigh for the case L=0.1H D=0.2H and at time t=50 sec.

A surface is created at the bottom of the vent and the variation of temperature on this surface is observed and is presented in the Fig 5.8. It can be interpreted from this that the effect of Rayleigh number on the fluid inside the enclosure is to increase its temperature intensity.

5.3.1 Different flow regimes

The variation of instantaneous vertical velocity at a point located at the center of the vent with time for different Rayleigh numbers is shown in Fig. 5.9. It depicts, that different regime of flow occurs in the vent with an increase of Rayleigh number such as no bulk fluid motion, counter current and oscillatory flows which depends on the magnitude of buoyancy force. For low Rayleigh it can be observed that the velocity is very small hence there will be very mild convection currents and flow will mostly be unidirectional, for Ra=$5x10^7$ it is observed that flow will be oscillatory for most of the time and only small amount of bidirectional flow occurs, for higher Ra=$2.5x10^8$ flows both oscillatory and bidirectional flow with bulk fluid motion takes place.

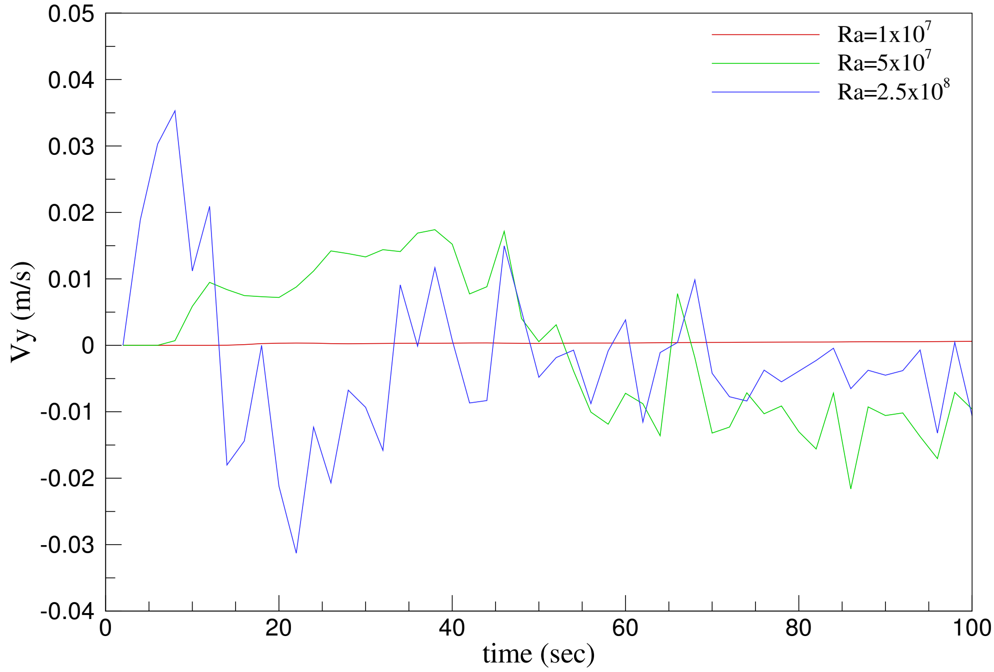


Figure 5.9: Variation of instantaneous normal velocity with time at the center of the vent for different Rayleigh number.

5.4 Effect of vent width

A study on the effect of vent width on the fluid flow inside the enclosure is made and the effects on the temperature and velocities are interpreted. Quasi steady state temperature and velocity contours for different vent widths is shown in Fig 5.10 for Ra = 2.5 x 10^8 and corresponding ΔT=100 K, where the thickness of the vent L = 0.1H. The flow becomes more oscillatory in the enclosures with increase of width from D = 0.1 H to 0.2 H due to increase of interactions between hot and cold fluids. From Fig. 5.10 (a &b), it can be noticed that the thermal plume enters the top enclosure quickly and also increases the intensity of the plume in the top enclosure with increase in vent width. From Fig. 5.10 (c & d) it is observed that, the velocity of fluid increases in the vent with increase in vent width and hence increases the amount of fluid entering the top enclosure.

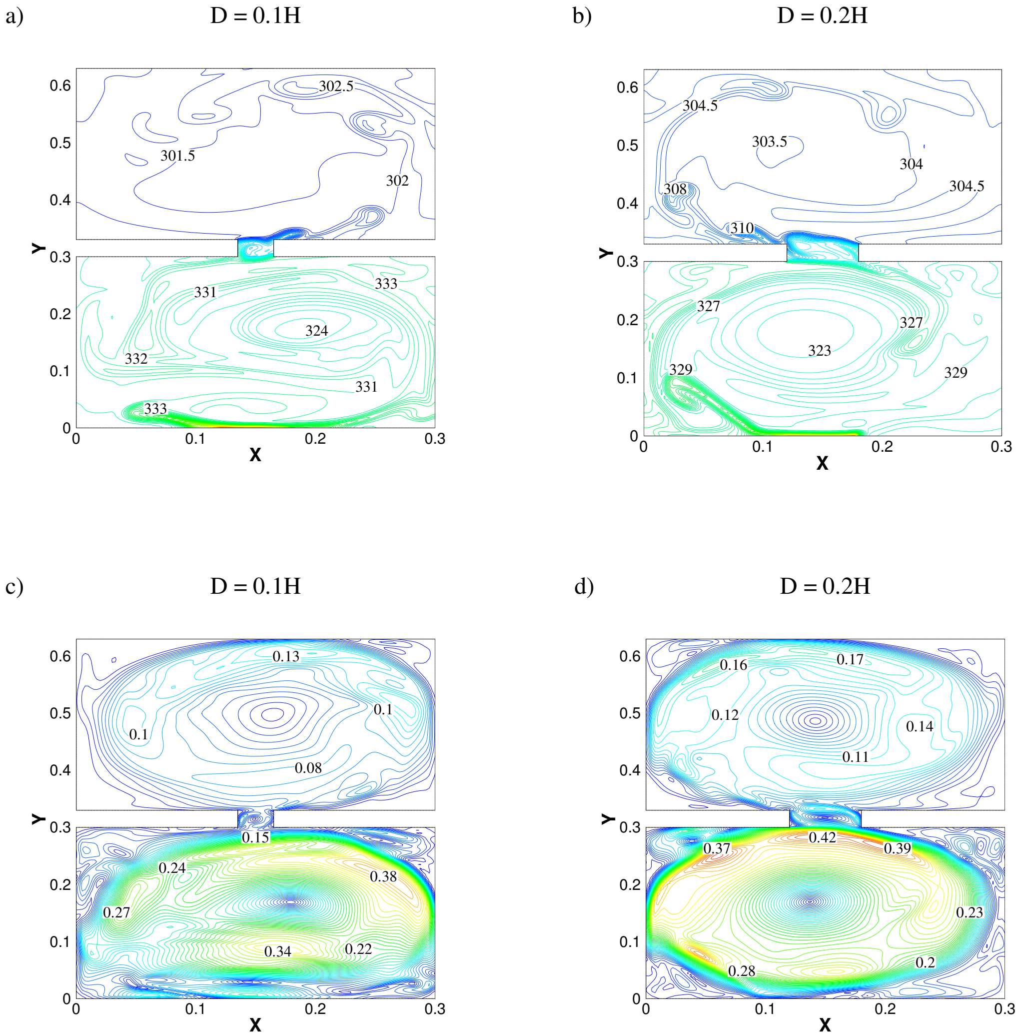


Figure 5.10: Temperature and velocity contour for different vent width and for $Ra = 2.5 \times 10^{8}$.

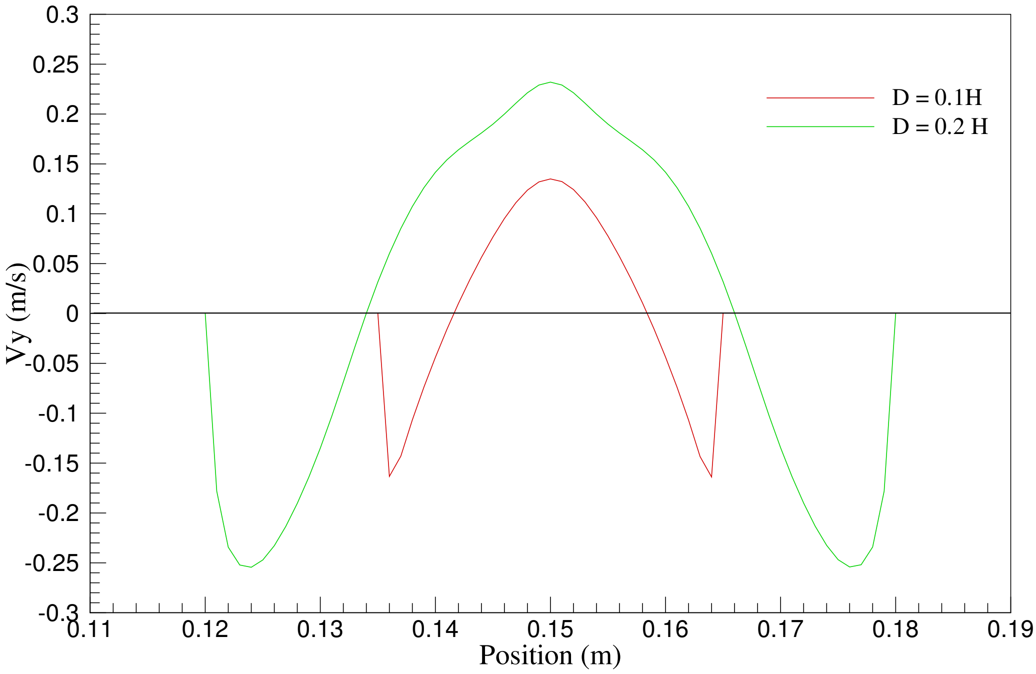


Figure 5.11: Variation of normal velocity at the vent bottom for different vent width and for Ra = 2.5 x 10^8

The normal velocity profile along the vent bottom at different vent widths is also shown in Fig 5.11 this plot gives an idea of the variation of velocities due to width. The normal velocity is found negative near the vent walls which indicate that cold fluid enters to the bottom enclosure through the walls adjacent to the fluid and hot fluid escapes to top enclosure through the middle portion of the vent, this plot also gives an idea of volume of fluid passing through the vent bottom of the enclosure, from this plot it can be extracted that volume of fluid passing through the vent bottom of the enclosure increases with increase in width.

5.5 Effect of vent Thickness

A study on the effect of vent thickness on the fluid flow inside the enclosure is made and the effects on the temperature and velocities are interpreted.

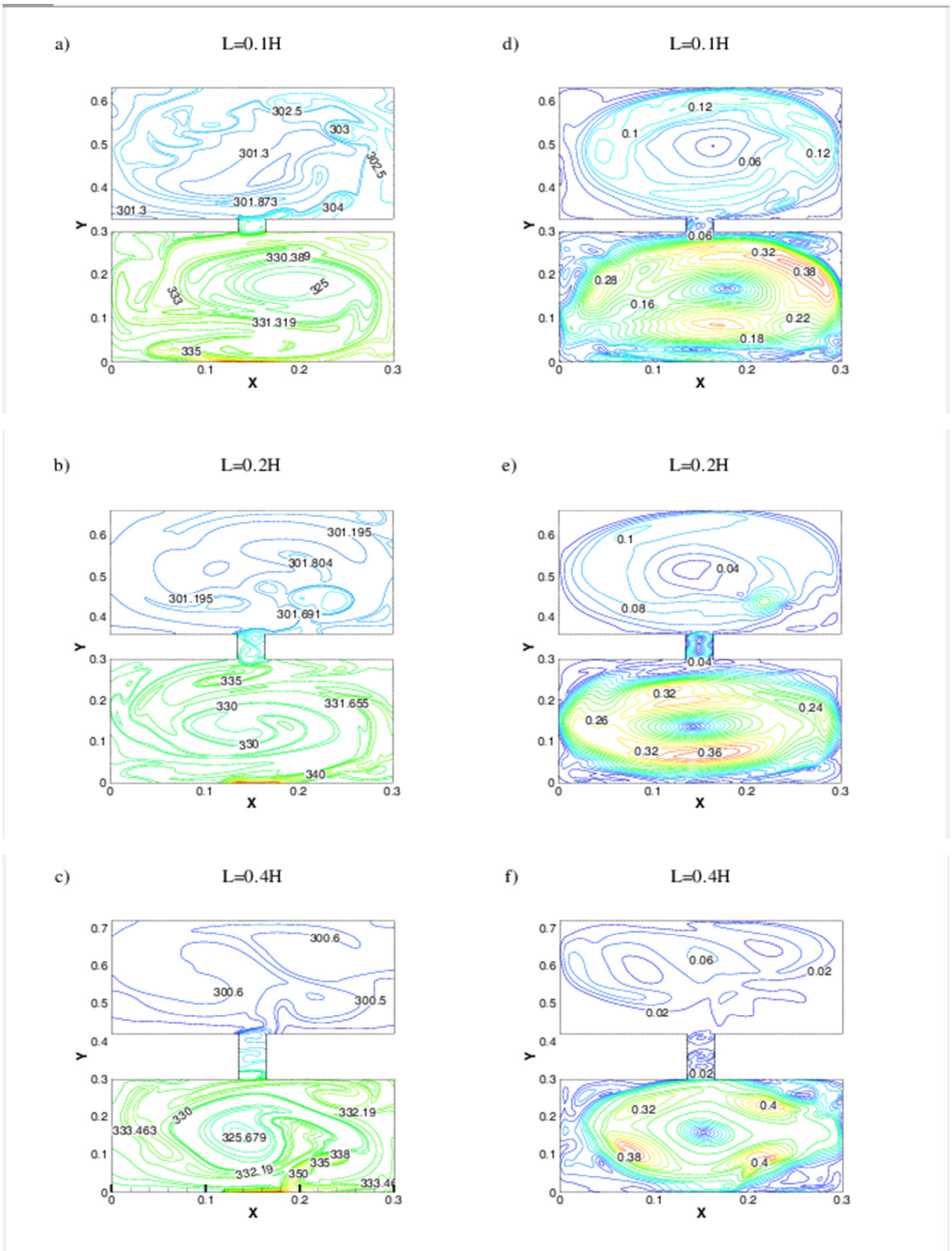


Figure 5.12: Temperature and velocity contour for different vent thickness and for $Ra = 2.5 \times 10^8$.

The temperature and velocity contours for different vent thicknesses but for the same Ra = $2.5x10^8$ is shown in Fig 5.12, the corresponding temperature difference between the heat source and fluid is ΔT = 100 K, where the width of the vent D = 0.1H. In the Fig 5.12 the left column represents temperature contours and right column represents velocity contours. The hot and cold fluid interactions occurs quickly and flow becomes chaotic in the enclosures with decrease in vent thickness. It can be visualized from Fig 5.12 (a, b, c), that the fluid temperature in top enclosure decreases while in bottom enclosure increases with increase in vent thickness due to less interactions between hot and cold fluids and from Fig 5.12 (d, e and f), fluid velocity decreases across the vent with increase in vent thickness. Small vortices are found to develop with time in the vent and with thickness the number of vortices is also found to increase this can be observed from the velocity contours and these vortices also tends to prevent the interaction of fluid between the top and bottom enclosure as a result fluid in both the enclosure is entrapped and find no way hence they start circulating in their respective enclosure.

To see about the effect on velocity, normal velocity profile along the vent bottom for different vent thicknesses at particular time is plotted and is shown in Fig 5.13.

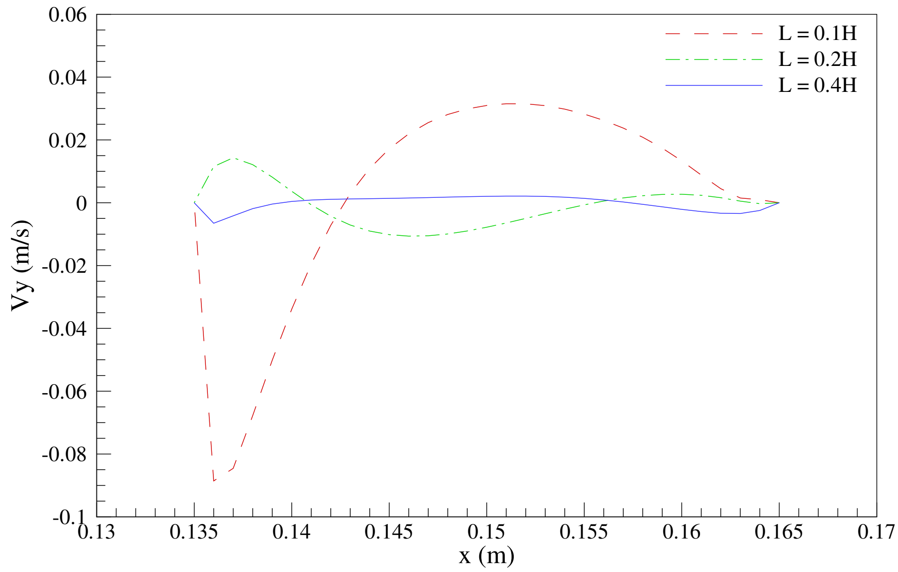


Figure 5.13: Variation of normal velocity at vent bottom for different vent thickness and for Ra = $2.5x10^8$

The normal velocity is found to decrease with increase in vent thickness, this plot gives an idea of the amount of fluid passing through the vent bottom of the enclosure, from this plot it can be extracted that the amount of fluid passing through the vent bottom of the enclosure decreases with increase in thickness.

5.6 Multiple vents

In this case, the fluid behavior inside the enclosure containing multiple ceiling vents is studied and also the effect of vent aspect ratio on the fluid flow is examined. In this study two vents are used, which are located at 25% and 75% of the top wall of bottom enclosure. It is observed that with multiple vents much fluid interaction is possible and also variation in temperature and velocities are much when compared to single vent case, also for the same Rayleigh number intensity of velocity components across the vent are higher for multiple vent case when compared to single vent.

5.6.1 Effect of vent width

The steady state temperature and velocity contours for different vent widths but for the same $Ra = 2.5x10^8$ and corresponding $\Delta T=100$ K is shown in Fig. 5.14, where the thickness of the vent L = 0.1H. The effect of change in width of the vent on the fluid flow inside the enclosure is made and the effects on the temperature and velocities are examined. In the Fig 5.14 (a & b) are temperature contours, and Fig 5.14 (c & d) corresponding velocity contours for the same $Ra = 2.5 \times 10^8$. Interaction between hot and cold fluid is higher for larger width as was found similar in case of single vent, the hot fluid particles are found to escape easily into the top enclosure due to less obstruction from top wall of bottom enclosure, as a result fluid particles in top enclosure will have higher intensities of temperature and velocities within them when compared to small width case, but in case of lower enclosure an opposite effect will take place, this is because as the hot fluid particles finds an obstruction from the top wall of bottom enclosure which prevents some amount of fluid escaping to top enclosure, this amount of fluid hence changes its direction and continues to flow adjacent to the walls of the bottom enclosure, as a result it circulates within the bottom enclosure.

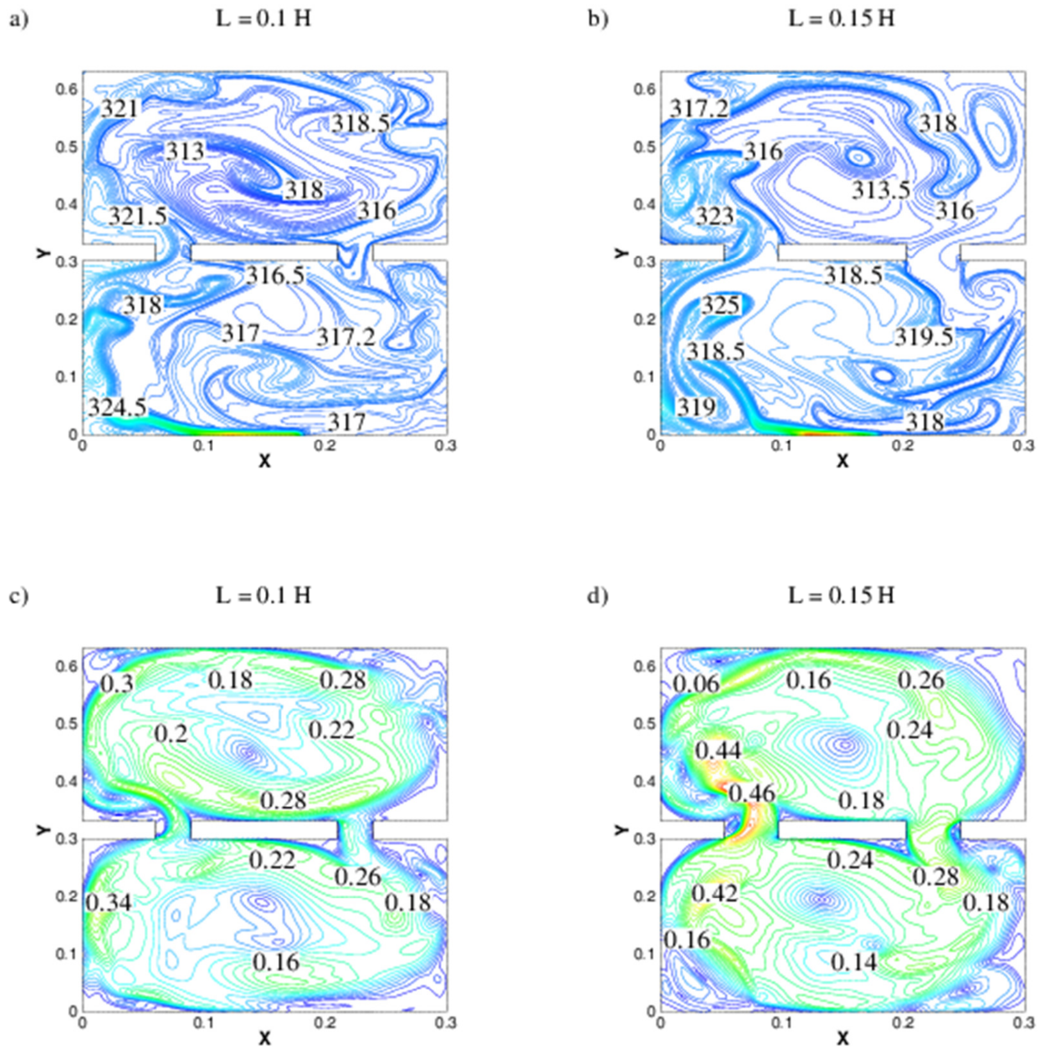


Figure 5.14: Temperature and velocity contour for different vent width for multiple vent case with Ra = 2.5 x 10^8.

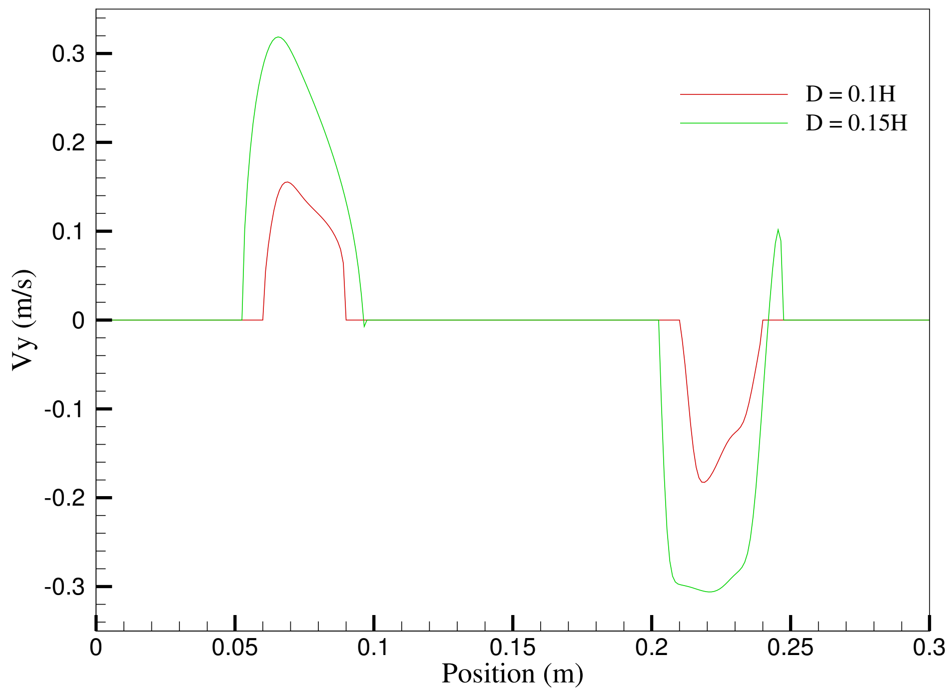


Figure 5.15: Variation of normal velocity along the vent bottom with different vent width for multiple vent case, Ra= $2.5\text{x}10^8$

The variation of normal component of velocity along the vent bottom is shown in Fig 5.15 one can notice from this figure that there is a significant change in the intensity of velocity with the change in width of the vent, which in turn has a direct effect on the amount of fluid passing across the vent. From the Fig 5.15 normal velocity is found to increase for larger width, henceforth it is concluded that amount of fluid entering the top enclosure increases with increase in width.

5.6.2 Effect of vent thickness

A study on the effect of change in thickness of the vent on the fluid flow inside the enclosure is made and the effects on the temperature and velocities are examined. In the Fig 5.16 (a & b) are temperature contours, and (c & d) are corresponding velocity contours for the same Ra = 2.5 x 10^8 and corresponding ΔT=100 K. Interaction between hot and cold fluid is higher for smaller vent thickness as was found similar in case of single vent, the hot fluid particles are found to escape easily into the top enclosure due to smaller pathway, as a result fluid particles in the top enclosure will have higher intensities of temperature and velocities within them when compared to large vent thickness case, and in case of lower enclosure an opposite effect will

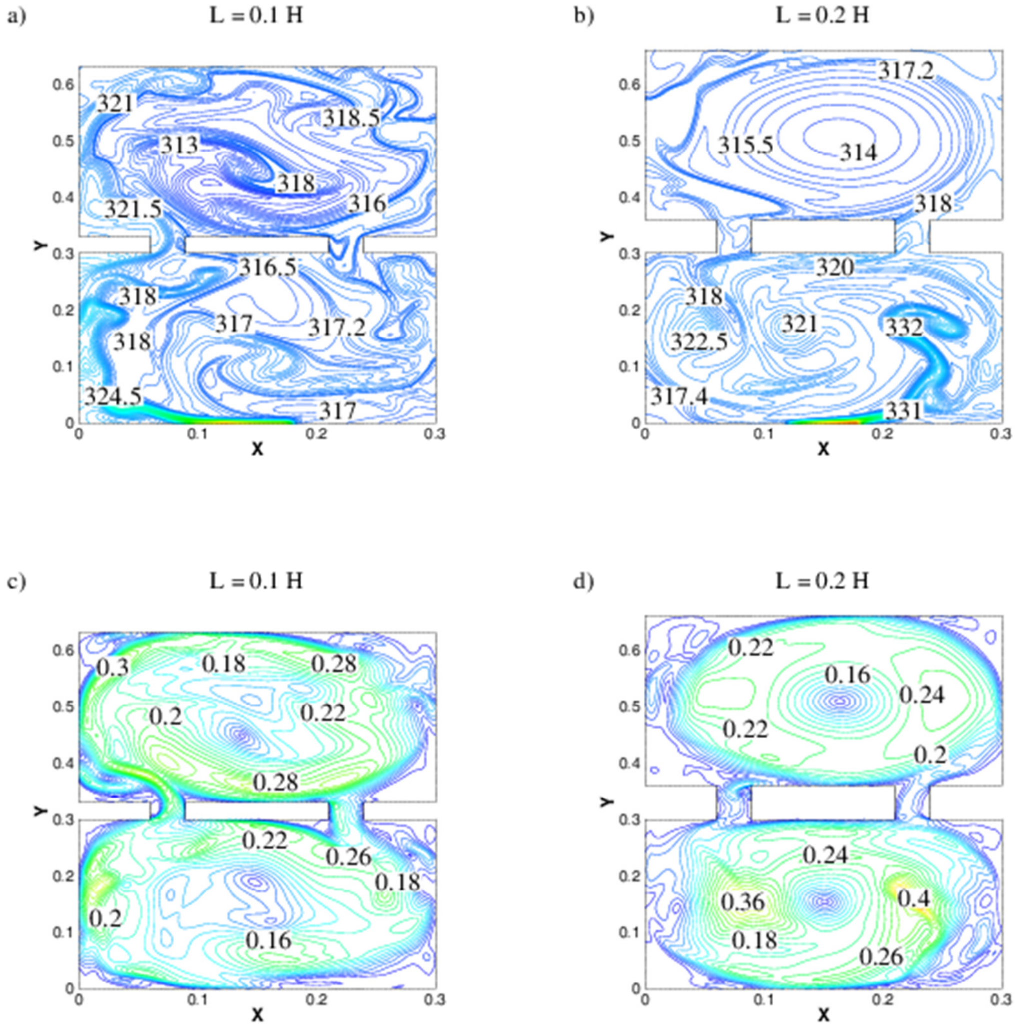


Figure 5.16: Temperature and velocity contour for different vent thicknesses for multiple vent case with Ra = 2.5 x 10^8.

take place, this is because as the hot fluid particles finds an obstruction from the larger vent thickness which prevents some amount of fluid escaping to top enclosure, this amount of fluid hence changes its direction and continues to flow adjacent to the walls of the bottom enclosure, as a result it circulates within the bottom enclosure.

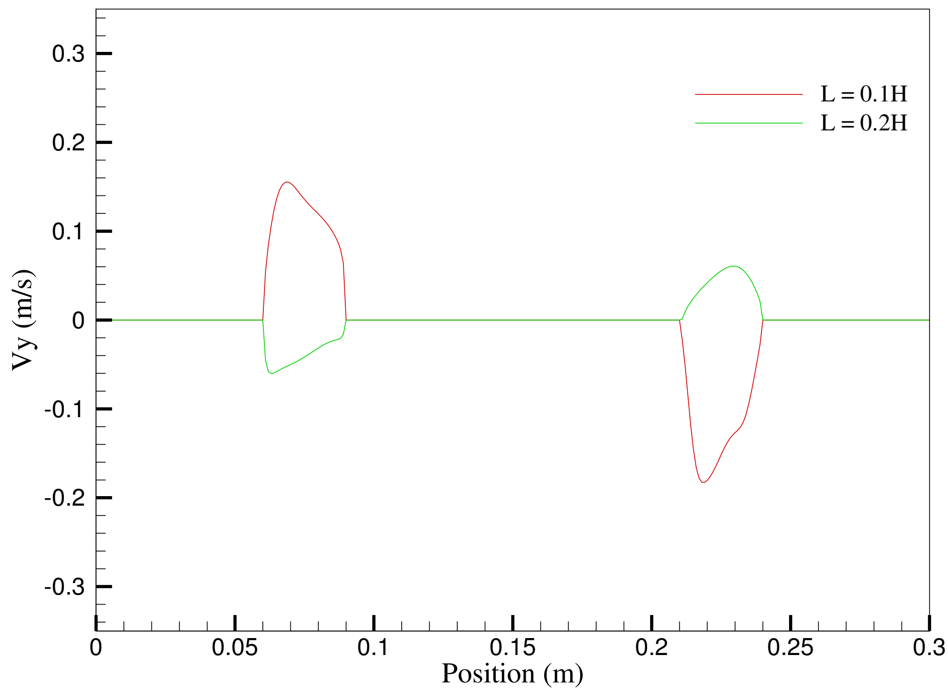


Figure 5.17: Variation of normal velocity along the vent bottom with different vent thickness for multiple vent case with and Ra= 2.5×10^8

The variation of normal component of velocity along the vent bottom is shown in Fig 5.17. One can notice from this figure that there is a significant change in the intensity of velocity with the change in height of the vent, which in turn has a direct effect on the amount of fluid passing across the vent. From the Fig 5.17 normal velocity is found to decrease with increase in thickness, henceforth it is concluded that amount of fluid entering the top enclosure decreases with increase in thickness.

5.7 Effect of Turbulence

In this case, the fluid behavior inside the enclosure due to turbulence is studied and also the effect of vent aspect ratio on the fluid flow is examined. Flows with Rayleigh number in the range $5x10^8$ – $7.26x10^8$ is considered for examination.

5.7.1 Time Evolution

Temperature contours at different times are shown in Fig. 5.18 for vent aspect ratio L/D = 0.5, Rayleigh number Ra = $5x10^8$, where the width of the vent D = 0.2H and temperature difference between heat source and fluid $\Delta T = 200^0C$.

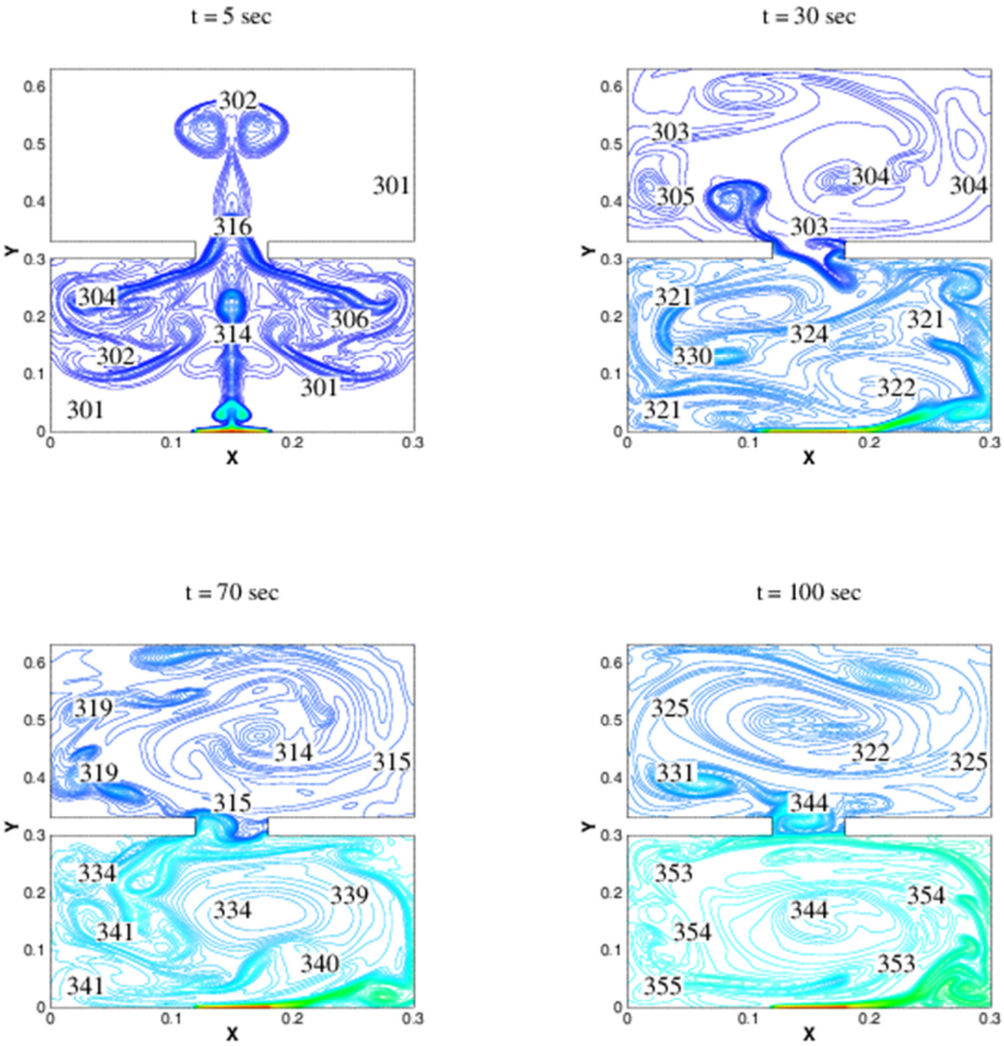


Figure 5.18: Time evolution Temperature contours for L=0.1H D=0.2H and Ra = $5x10^8$.

From Fig. 5.18, it can be interpreted that the thermal plume grows and moves upwards from heat source with increase in time. The flow enters the top enclosure very quickly at time t < 5 s and flow is found to oscillate at time t = 7 sec. At time t = 30 sec small swirls of plumes is observed in the top enclosure and these plumes grow in the later time by mixing with their neighbor and spreads the entire enclosure, temperature of the plume is found to increase in this mean time**.**

5.7.2 Effect of Rayleigh number

To see about the effect of Rayleigh number on the turbulent flow inside the enclosure simulations are performed for the case of AR = 0.5, where in D=0.2H and for different values of Rayleigh numbers. The effect of Rayleigh number on the temperature for this case but at a particular time is portrayed in Fig.[5.19 - 5.20]

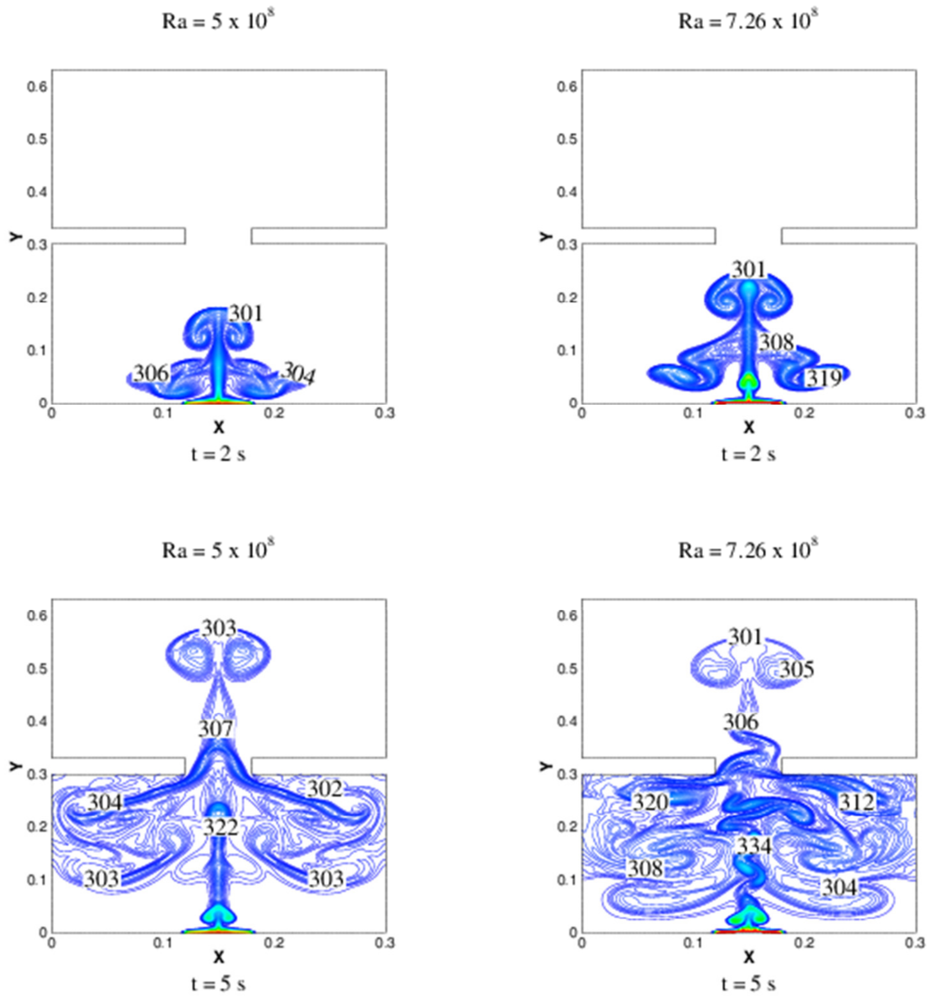


Figure.5.19: Effect of Rayleigh on turbulent flow.

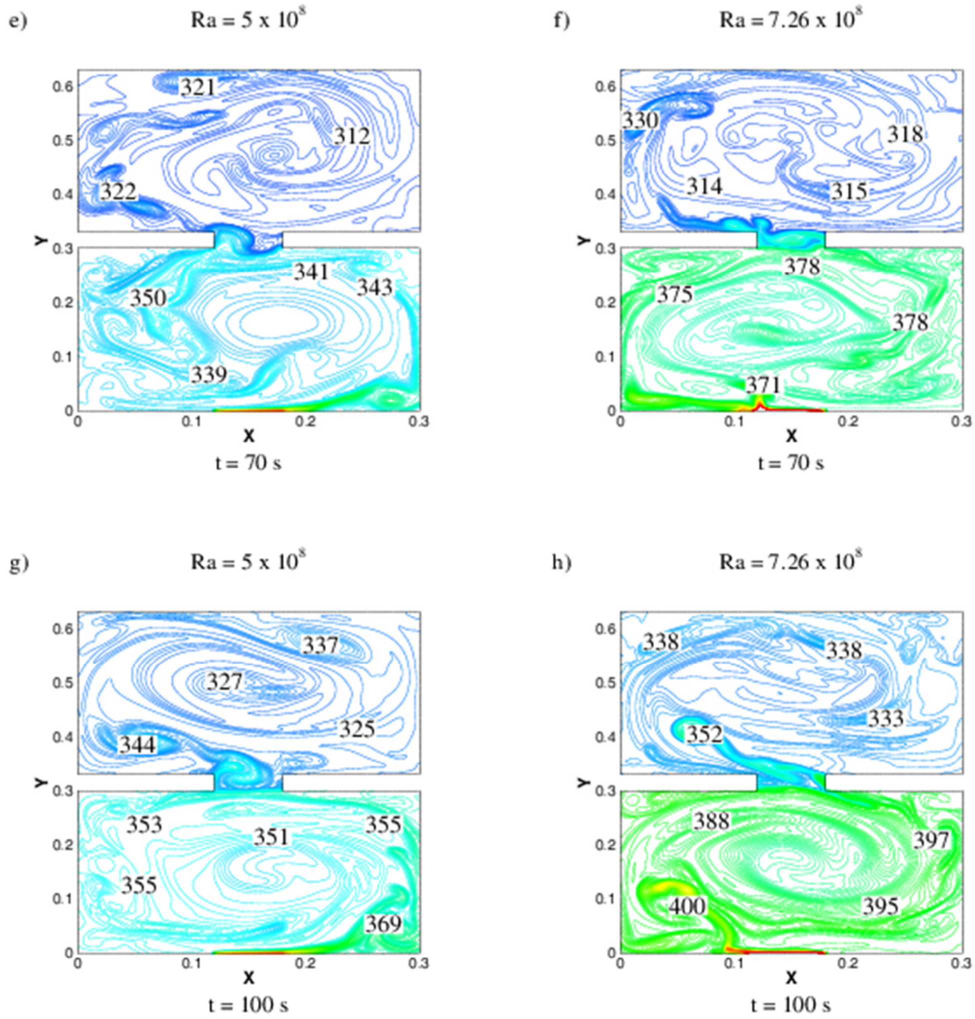


Figure.5.20: Effect of Rayleigh on turbulent flow.

From the Fig.5.19 it is observed that the temperature of the fluid increases with Rayleigh number, and hence the fluid tries to quickly enter the top enclosure with increase in Rayleigh number, the fluid starts oscillating at early time than the lower Rayleigh that can be visualized from the contour plots of Fig.5.19 (c,d) at time t= 5 sec, it depicts that the fluid flow for the lower Rayleigh number is stable during its path towards the top enclosure while for higher Rayleigh number it starts oscillating and quickly spreads the entire enclosure. Due to turbulence higher diffusion rate is observed as a result all the fluid particles gain higher temperature very quickly.

5.7.3 Effect of vent width

In order to see the effect of change in width of the vent on the turbulent flow inside the enclosure, simulations are performed on different aspect ratios. Figure.5.21 is presented to show this effect on two different aspect ratios AR =1 where in D=0.1H & AR = 0.5, where in D=0.2H, it is to be noted that the Rayleigh number was maintained constant with a value of Ra = 5×10^8 and corresponding ΔT=200 K, for both the aspect ratios. The effect of vent width on the temperature and velocity for this case at quasi steady state is portrayed in Fig.5.20. In the Fig.5.21 (a,b) represents temperature contours and (c,d) represents velocity contours. Temperature and velocities are found to attain higher values in the top enclosure for smaller aspect ratios, this is observed due to bulk motion of fluid flow across the vent for larger width. Conversely it is observed that fluid temperature and velocities are higher in the bottom enclosure for high aspect ratios, this is because of lack of bulk fluid motion. The temperature and velocities are found to attain maximum values when compared to laminar case, and also fluid particles spread quickly in the entire enclosure, and the reason for all this is the effect of turbulence which creates high mixing rate.

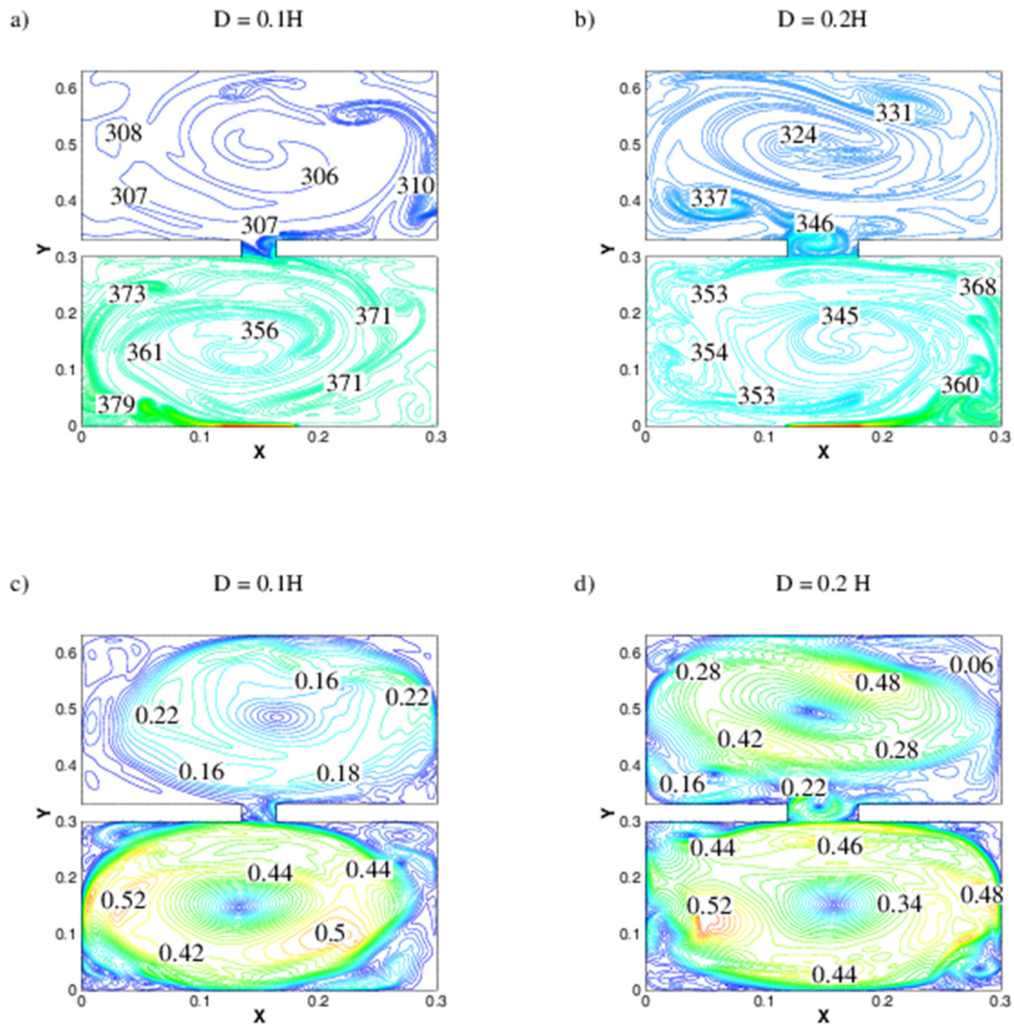


Figure.5.21: Effect of vent width on turbulent flow for Ra = 5 x 10^8.

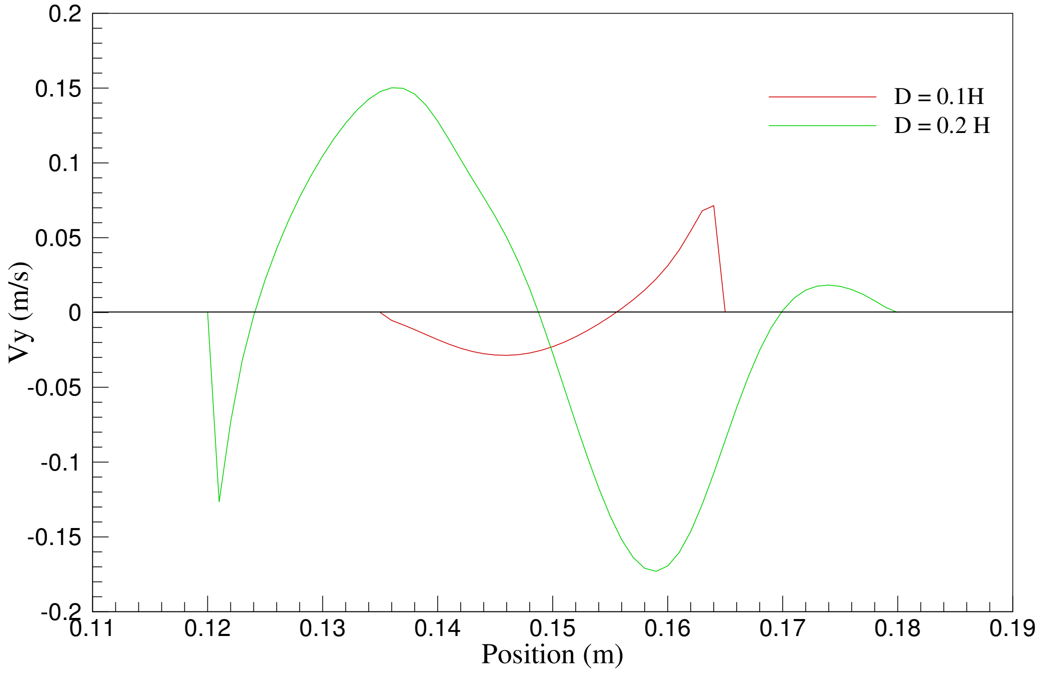


Figure 5.22: Variation of normal velocity along the vent bottom for different vent width and Ra= 5 x 10^8

The variation of normal component of velocity along the vent bottom is shown in Fig 5.22 one can notice from this figure that there is a significant change in the intensity of velocity with the change in width of the vent, which in turn has a direct effect on the amount of fluid passing across the vent. From the Fig 5.22 normal velocity is found to increase for larger width, henceforth it is concluded that amount of fluid entering the top enclosure increases with increase in width.

5.7.4 Effect of vent thickness

A study on the effect of change in thickness of the vent on the turbulent fluid flow inside the enclosure is made and the effects on the temperature and velocities are examined by performing simulations on different aspect ratios. Figure.5.23 is presented to show this effect on two different aspect ratios AR =1 where in L=0.1H & AR = 2, where in L=0.2H, it is to be noted that the Rayleigh number was maintained constant with a value of Ra = $5x10^8$ and corresponding ΔT=200 K, for both the aspect ratios. The effect of vent thickness on the temperature and velocity for this case at quasi steady state is portrayed .In the Fig.5.23 (a,b) represents temperature contours and (c,d) represents velocity contours. Temperature and velocities are found to attain higher values in the top enclosure for smaller vent thickness, this is observed because due to small vent thickness the fluid from bottom enclosure which is having higher intensities of temperature and velocities is able to quickly escape into the top enclosure and where it shares its energy with the cold fluid. An opposite phenomena happens for higher vent thickness, that is fluid temperature and velocities will be higher in bottom enclosure, this is because in the vent due to velocity gradients and the walls being closer vortices are formed which breaks the communication between hot and cold fluid.

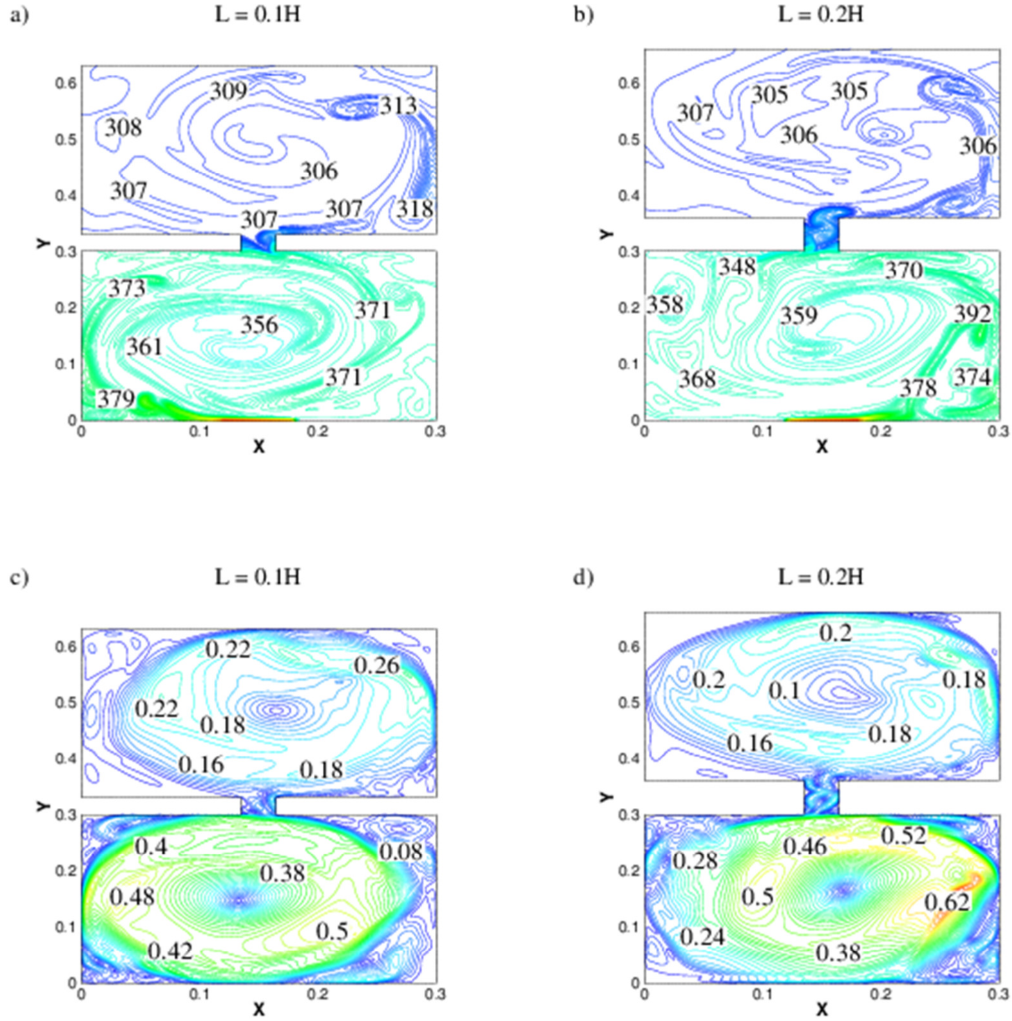


Figure.5.23: Effect of vent thickness on turbulent fluid flow and Ra = $5x10^{8}$.

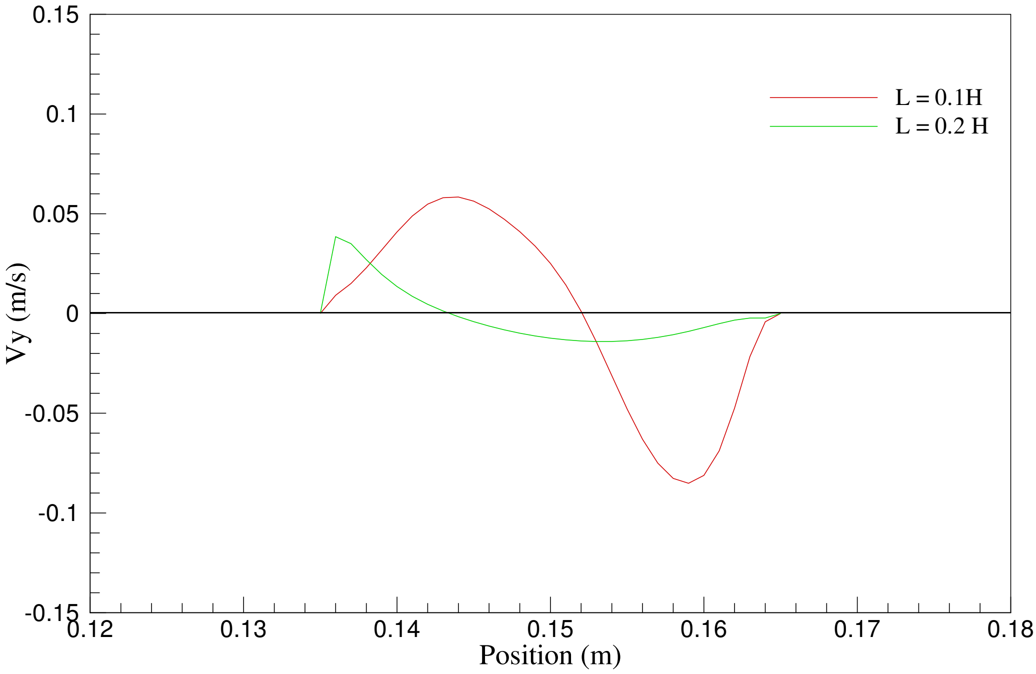


Figure 5.24: Variation of normal velocity along the vent bottom for different vent thickness and Ra= 5 x 10^8

The variation of normal component of velocity along the vent bottom is shown in Fig 5.24. One can notice from this figure that there is a significant change in the intensity of velocity with the change in height of the vent, which in turn has a direct effect on the amount of fluid passing across the vent. From this normal velocity is found to decrease with increase in thickness, henceforth it is concluded that amount of fluid entering the top enclosure decreases with increase in thickness.

5.7.5 Comparison between Laminar and Turbulence flow

In order to compare the laminar and turbulent flows in enclosure results are extracted for laminar Ra = 2.5×10^8 , ΔT=100 and Turbulent Ra = 5×10^8 , ΔT=200, for the same aspect ratio of AR = 0.5, where D=0.2H and are portrayed in Fig [5.25 – 5.30].It is visualized from both the contour and XY plots of temperature and velocities that the fluid particles will have higher intensities of both temperature and velocities at most of the times in the enclosure for the turbulent flows, as a result fluid particles quickly expose themselves into the top enclosure and at the same time they also spread and try to encompass the enclosure. From the velocity plots it can be predicted that high bulk motion of fluid exist for turbulent flows when compared to laminar flows.

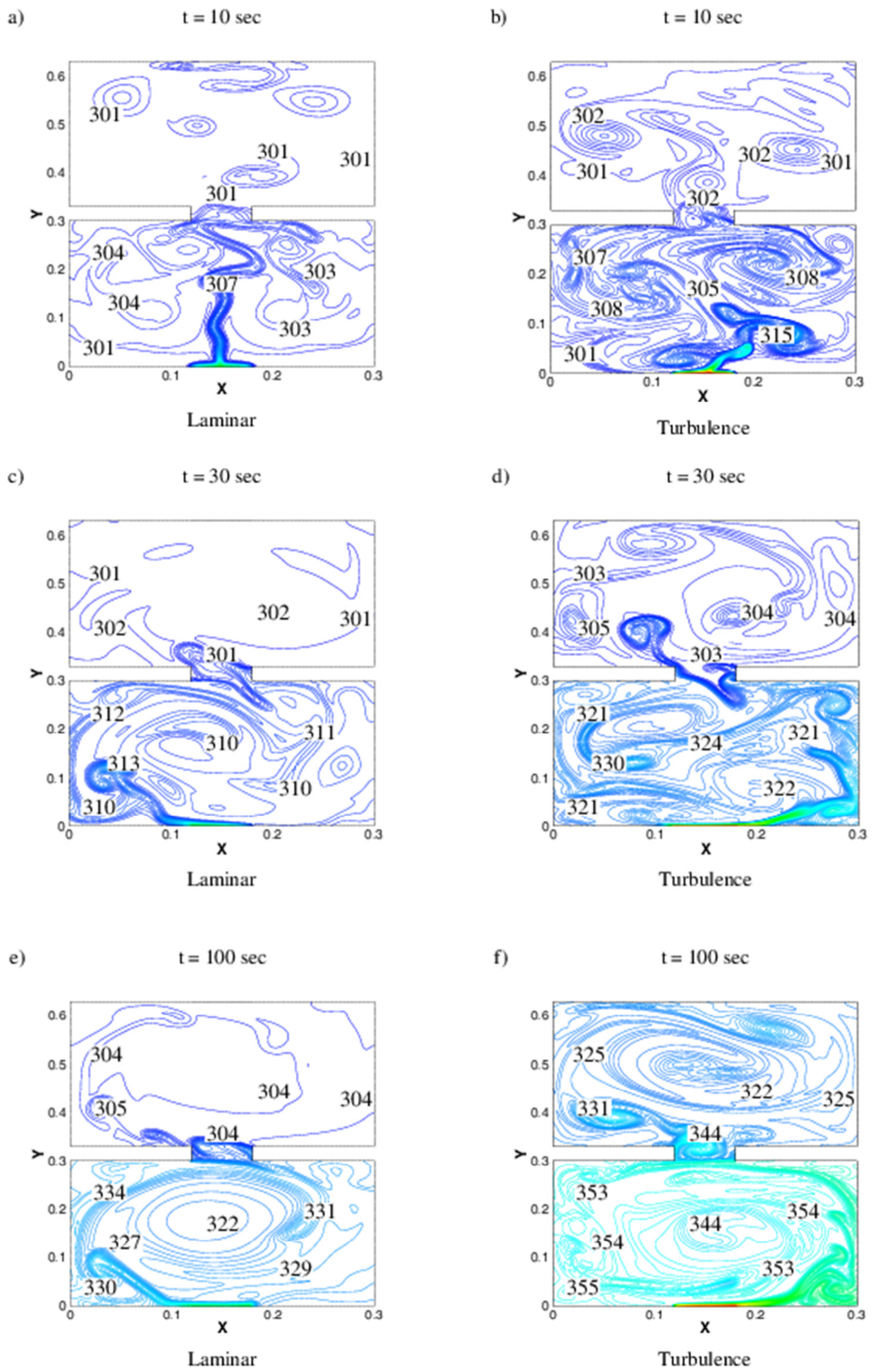


Figure 5.25: Comparison of temperature contour plots between laminar and turbulent flows of AR = 0.5 at different time levels

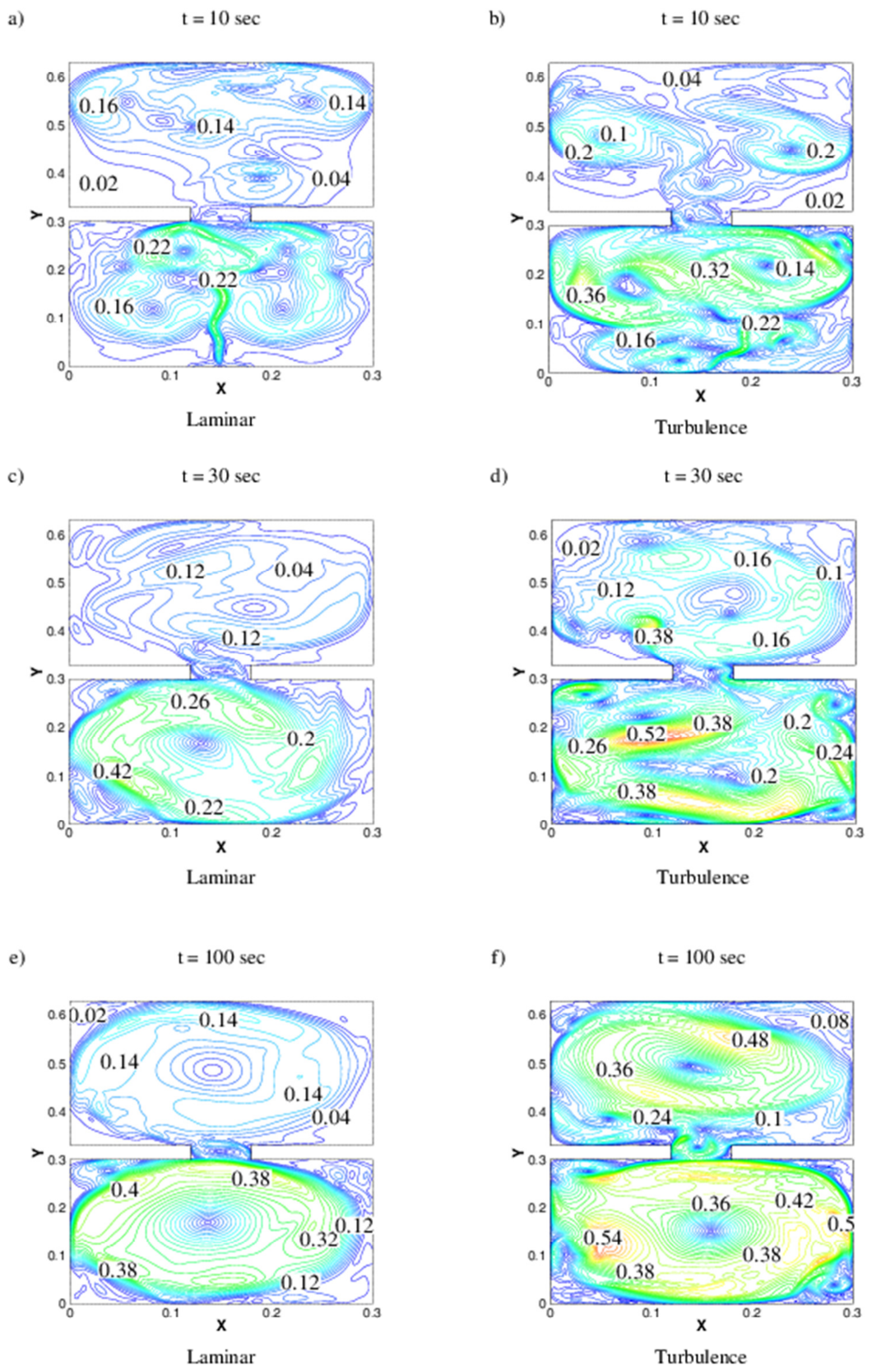


Figure 5.26: Comparison of velocity contour plots between laminar and turbulent flows of AR = 0.5 at different time levels

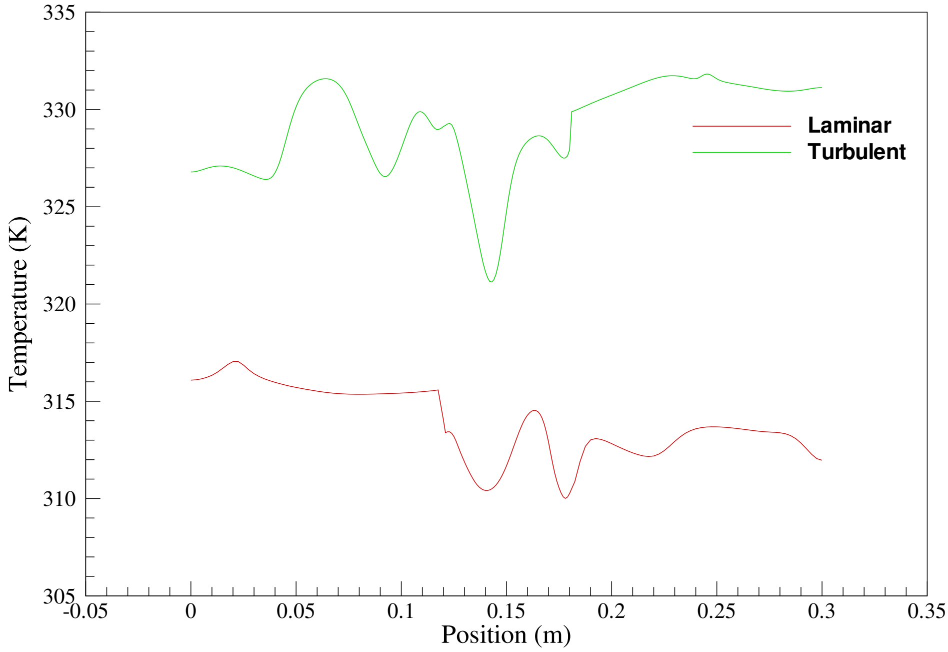


Figure 5.27: Comparison of temperature plots between laminar and turbulent flows of

AR = 0.5 at time t = 40 sec

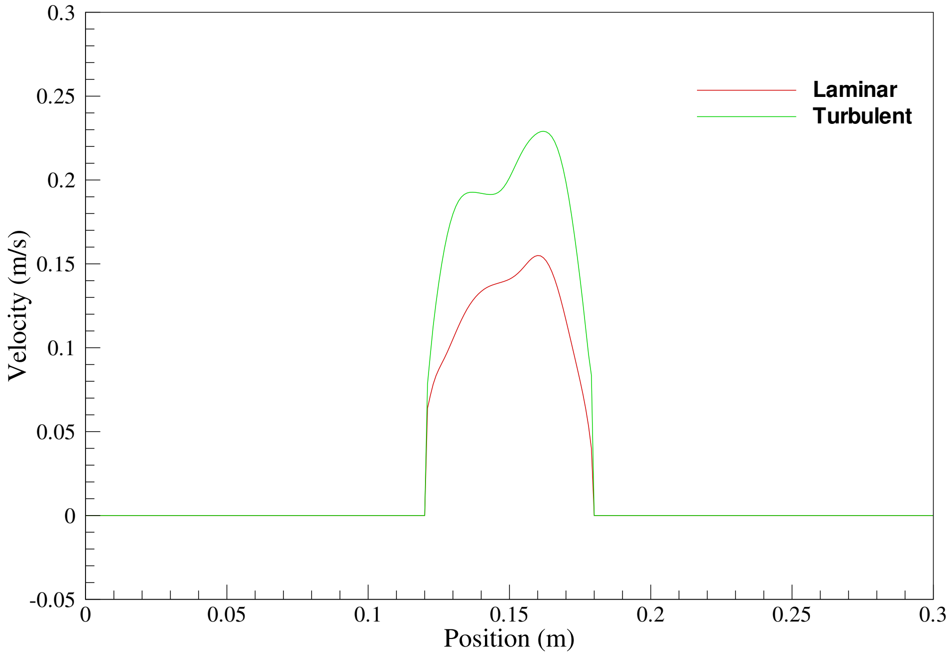


Figure 5.28: Comparison of velocity plots between laminar and turbulent flows of

AR = 0.5 at time t = 40sec

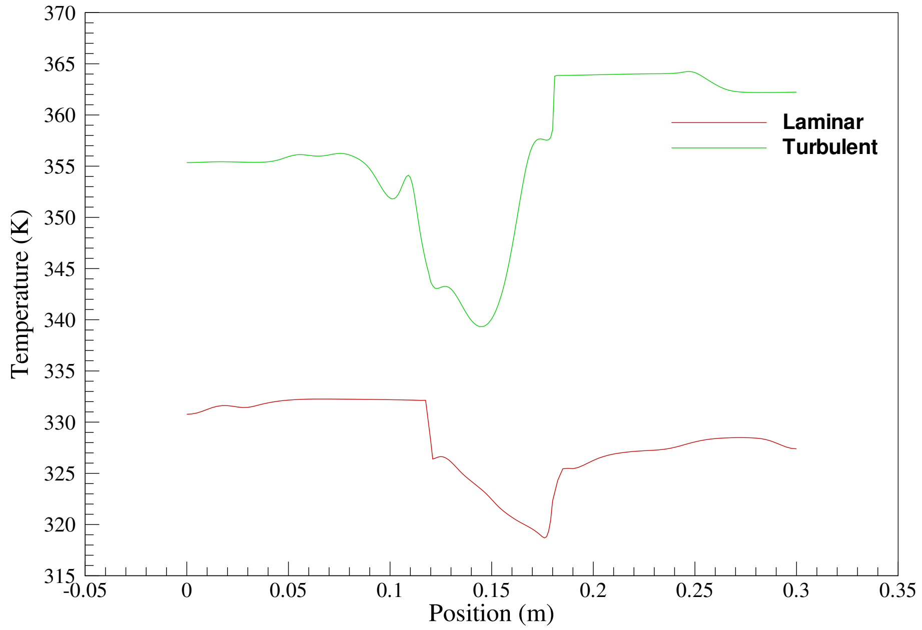


Figure 5.29: Comparison of temperature plots between laminar and turbulent flows of

AR = 0.5 at time t = 100 sec

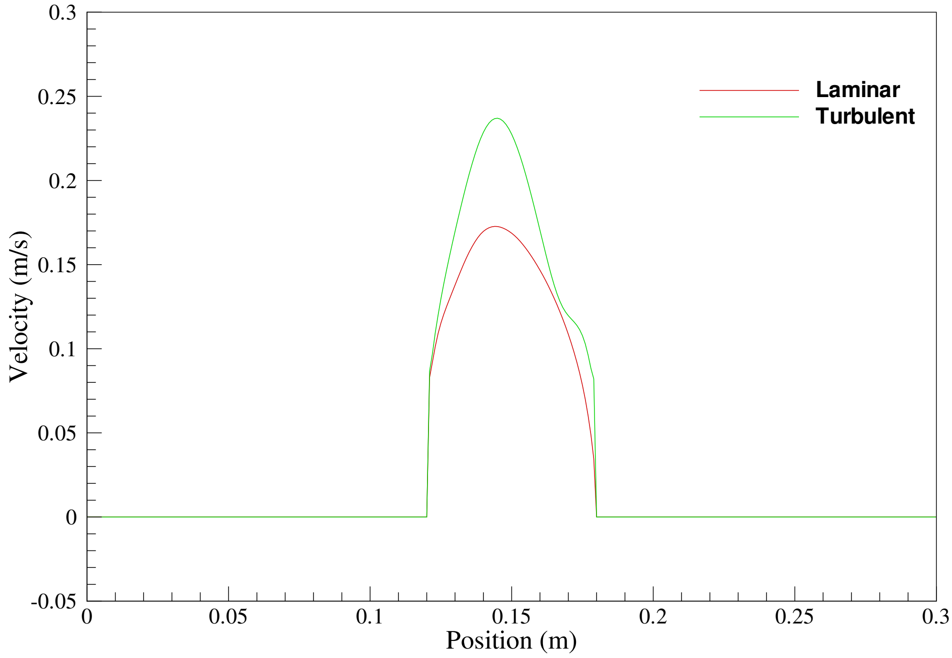


Figure 5.30: Comparison of velocity plots between laminar and turbulent flows of

AR = 0.5 at time t = 100sec

Chapter 6

Conclusion

The buoyancy-induced flow generated by a heat source such as fire in square enclosures connected by horizontal or ceiling vent has been numerically investigated and reported here. The fluid flow characteristics in the enclosures are examined for various parameters such as vent width, vent thickness, Rayleigh number, multiple vents and turbulence. Different flow regimes is been observed such as no bulk fluid motion, counter current and oscillatory flows due to change in Rayleigh number, the flow is found to be oscillating with small amount of bidirectional flow across the vent for low Rayleigh number flows, while the flow becomes unstable and chaotic in the enclosures for higher Rayleigh flows of the range Ra = $5x10^7$ to Ra = $2.5x10^8$ due to greater buoyancy.

The flow becomes unstable and more oscillatory in the enclosures with increase in vent width from D = 0.1 H to 0.2 H due to increase of interactions between cold and hot fluids, and the net effect is the increase in mass flow rate across the vent. The hot and cold fluid interactions occurs quickly and flow becomes chaotic in the enclosures with decrease in vent thickness from L = 0.4 H to 0.1 H due to increase of fluid velocity across the vent. The volume flow rate entering the top enclosure is found to decrease with increase in vent thickness.

Multiple vents is found to have much effect on the interactions of hot and cold fluid, it is observed that the rate of interaction is very high and the fluid inside the enclosure attains higher temperature and velocities, one interesting observation was the flow in these vents will be slightly oscillating and will be highly unidirectional that means from one vent hot fluid passes and from the other vent cold fluid, and it revealed that for the same vent size net mass flow rate for the multiple case will be higher than the single vent case.

Fluid quickly spreads inside the enclosure due to turbulence and is found to gain high temperature and momentum, which are the characteristics of turbulence. High amount of bidirectional flow is observed, billows of plumes swirls and fluctuates with no apparent pattern. Present results would be useful to understand the fire spread and growth in enclosures connected with ceiling vent and also useful for the design of fire safety system such as water spray sprinklers.

Chapter 7

Scope for future work

As this was the small beginning, towards the study of buoyancy-induced flow in enclosures connected by horizontal vents of different aspect ratios, where in we restricted our study in understanding only the mass flow rate, temperature and velocity fields inside the enclosure, for the case of single and multiple vents, this work can be further extended by the investigation of different cases mentioned below.

- Effect of heat source length on the fluid flow
- Effect on the fluid flow with the change in the location of heat source
- Effect on the mass flow rate with change in the location of the vent
- Radiation effects which are highly involved in fire situations
- Effects on the fluid flow with the change in vent shape
- Furthermore detailed CFD study considering species reaction

References

[1] Adrian Bejan, Convection Heat Transfer, John Wiley and Sons, New York, 1984.

[2] John Patterson and Jorg Imberger, J., “Unsteady Natural Convection in a Rectangular Cavity,” Journal of Fluid Mechanics, Vol. 100, Aug. (1980) pp. 65–86

[3] Orhan Aydin and Wen-Jei Yang “ Natural convection in Enclosure with localized heating from below and symmetrical cooling from sides”. Int. Journal of Numerical Methods for Heat and Fluid Flow, vol.10 No.5, (2000) 518-529.

[4] P. Kandaswamy, J. Lee, A.K. Abdul Hakeem “Natural Convection in a Square Cavity in the Presence of Heated Plate” . J. Nonlinear Analysis: Modeling and Control, Vol. 12, No. 2,(2007) 203–212

[5] Pravez Alam, Ashok Kumar, S. Kapoor , S.R. Ansari, Numerical investigation of natural convection in a rectangular enclosure due to partial heating and cooling at vertical walls, Commun Nonlinear Sci Numer Simulat 17 (2012) 2403–2414.

[6] Anil Kumar Sharma, K.Velusamy , C. Balaji , Turbulent natural convection in an enclosure with localized heating from below, International Journal of Thermal Sciences 46 (2007) 1232–1241

[7] W. Brown, Natural convection through rectangular openings in partitions – 2: Vertical partitions, Int. J. Heat Mass Transfer 5 (1962) 295–299.

[8] Ranganathan Kumar, Ahmad Sleiti, J.Kapat, Unsteady laminar buoyant flow through rectangular vents in large enclosures, AIAA J.Theromphysics and Heat Transfer Vol.20,No. 2, (2006).

[9] A.K. Sleiti “Effect of vent aspect ratio on unsteady laminar buoyant flow through rectangular vents in large enclosures”. International Journal of Heat and Mass Transfer Vol.51, (2008) 4850–4861.

[10] R.P. Harrison, R.E. Spall, The effect of partition thickness on buoyant exchange flow through a horizontal opening, J. Numer. Heat Transfer, part A, 44: (2003) 451-462

[11] A. Mercer and H. Thompson, An Experimental Investigation of Some Further Aspects of the Buoyancy-Driven Flow between Carbon Dioxide and Air Following a Depressurization Accident in a Magnox Reactor—Part I: The Exchange Flow in Inclined Ducts, J. Br. Nuclear Soc., vol. 14, (1975) 327–334

[12] M.Singhal and R. Kumar, Unsteady Buoyancy Exchange Flow through a Horizontal Partition, J. Heat Transfer, vol. 117, (1995) 515–520

[13] N. C.Markatos and M. R. Malin, Mathematical Modeling of Buoyancy-Induced Smoke Flow in Enclosures, Int. J. Heat Mass Transfer, vol. 25, (1982) 63–75

[14] A. H. Abib and Y. Jaluria, Turbulent Penetrative and Recirculating Flow in a Compartment Fire, J. Heat Transfer, vol. 117, (1995) 927–935

[15] A. Kashef, N. Be´nichou, G.D. Lougheed, Numerical modelling of movement and behavior of smoke produced from fires in the Ville- Marie and L.-H.-La Fontaine tunnels: literature review, Research Report, Institute for Research in Construction, National Research Council Canada, September 1, vol. 141, (2003) 66 (IRC-RR-141).

[16] Yapieng He and Paula Beever Buoyancy driven flow in building enclosures subjected to fires 13th Australasian Fluid Mechanics conference Monash University, Melbourne, Australia,13th – 18th Dec (1998)

[17] J. Prahl, H.W. Emmons, Fire induced flow through an opening, Combust. Flame 25 (1975) 369–385

[18] McCaffrey ,B,J. and Quintiere,J.,"Bouyant driven counter current Flows generated by a fire source ",in heat transfer and turbulent Buoyant convection, vol.II,D.B. Spalding and N.Afgan, ed Hemisphere pub.Co.(1977) 457-472

[19] Y. Jaluria, S.H.-K. Lee, G.P. Mercier, Q. Tan, Transport processes across a horizontal vent due to density and pressure differences, Exp. Thermal Fluid Sci.,16 (1998) 260-273.

[20] Y. Jaluria, W.K.S. Chiu, S.H.-K. Lee, Flow of smoke and hot gases across horizontal vents in room fires, Combust. Sci. Technol., 110-111 (1995) 197-208.

[21] Q. Tan, Y.Jaluria, Mass flow through a horizontal vent in an enclosure due to pressure and density differences, Int. J. Heat and Mass transfer,44 (2001) 1543-1553.

[22] K. Venkatasubbaiah, Y. Jaluria, Numerical simulation of enclosure fires with horizontal vents, Numerical Heat Transfer Part A, (In press).

[23] L.Kerrison, E.R.Galea & M.K.Patel A Two-dimensional Numerical Investigation of the Oscillatory Flow Behavior in Rectangular Fire Compartments with a Single Horizontal Ceiling Vent Fire Safety Journal 30 (1998) 357-382

[24] W.K.Chow and Y.Gao, Buoyancy and inertial force on oscillations of thermal-induced convective flow across a vent, Building and Environment 46 (2011) 315-323.

[25] W.K.Chow and Y.Gao, Oscillating behaviour of fire-induced air flow through a ceiling vent, Applied Thermal Engineering 29 (2009) 3289–3298

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

[27] FLUENT 13 manual, FLUENT incorporated.

[28] H K Versteeg and W Malalasekera, An introduction to computational fluid dynamics, Pearson education.

[29] Frank P. Incropera David P. Dewitt, Fundamentals of Heat and Mass Transfer, fifth edition, John Wiley and Sons, 2007.

[30] Patrick H. Oosthuizen, An introduction to convection heat transfer analysis, McGraw hill publication, 1999.

[31] Michel Favre - Marinet Sedat Tardu , Convective heat transfer, John Wiley and sons publication, 2009