Study of Natural Convection in Rectangular Enclosure Varying Wall-Temperature

by

Wan Nashiha Binti Wan Adenan
13424

Dissertation submitted in partial fulfilment of
the requirements for the
Bachelor of Engineering (Hons)
(Chemical Engineering)

JANUARY 2014

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Study of Natural Convection in Rectangular Enclosure Varying Wall-Temperature

by

Wan Nashiha Binti Wan Adenan

A project dissertation submitted to the

Chemical engineering program

Universiti Teknologi PETRONAS

In partial fulfillment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

CHEMICAL ENGINEERING

Approved by,

____

(Dr. Rajashekhar Pendyala)

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons

WAN NASHIHA BINTI WAN ADENAN

ABSTRACT

This research is aimed to analyze the behavior of natural convection in the rectangular enclosure by varying wall temperature and using different types of fluid such as liquid metals, gasses, water and oil. In order to predict the behavior, Computational Fluid Dynamics (CFD) simulation tool ANSYS Fluent 14.0 is used to perform 2D and 3D simulation for various types of fluid (air, water, mercury and gasoline) with varying the temperature of one side of the wall in transient order. The boundary condition and operating condition for different type of fluid is defined too see the foreseeable effect of its changes to the simulation result. The result has shown various behavior of temperature contour and variation of velocity inside the enclosure. It also shows that the transient dispersion of temperature within rectangular enclosure is affected by the buoyancy forces acting upon the fluid, thermal expansion coefficient, density as well as the viscosity of the fluids. . Navier Stokes equation is further used to describe the temperature behavior of different fluids and different temperature changes schemes.

ACKNOWLEDGEMENT

The author would like to express the utmost acknowledgment for the opportunity and support given by Chemical Engineering Department of Universiti Teknologi PETRONAS in providing the Blok 5 Simulation Lab as the venue for my project simulation for 2 semesters. My sincerest gratitude is extended to my supervisor namely Dr Rajashekhar Pendyala for his assistance in my final year project entitled "Study of Natural Convection in Rectangular Enclosures Varying Wall Temperature". Dr Raja had also guided me the usage of the ANSYS Fluent 14.0 software to simulate simple fluid dynamics problems. His consistent attention from week to week never fails to motivate me to perform in this project. With the guidance, advice and support of everyone which includes FYP Coordinators, FYP Colleagues and all FYP Examiners, this thesis has turned out to be a successful and meaningful paper for fundamental research understanding of natural convection behavior in rectangular enclosures varying wall temperature.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLE

CHAPTER 1

INTRODUCTION

1.1 Background Study

Natural convection in a rectangular enclosure or cavities appears in many practical and industrial devices. This spontaneous mode of heat transfer is applied in engineering for decades. For example, in electronics industry, this Buoyancy-driven flows is applied in many thermal engineering application since passive cooling of components by natural convection is cheaper, quietest and also the most reliable method of heat rejection alternatives (Morini and Spinga, 2001). The applications involving rectangular enclosures are electronic components in computer, multi pane window, buildings and energy storage systems. Agricultural sectors also use natural convection as a mean for drying process applications and storage.

As a fundamental topic of thermal science, transient natural convection have been analyzed and many studies and papers were published by many authors to study the theoretical and analytical solutions for different geometries, more or less complex parameters and operating conditions. Morini and Spinga(2001) cited some studies that focus on the transient laminar natural convection along rectangular ducts. Hasnaoui et al. (1998) published a study on natural convection in cavity heated at its base. Altec et al. (2007) study the effect of tilted rectangular enclosure with a vertically situated hot plate inside and its application. Ganguli and Pandit (2009) produced a paper on simulation result of natural convection performed for various tall slender vertical geometries with varying gap temperature and compare the result with the available literature results.

It can be seen that present study analyzed natural convection numerically in a rectangular cavity heated on one side while the other side is cooled. Also, the temperatures of the heated wall and cold wall are assumed to be constant over time or in steady state. Some questions raised on this issue because by using experimental result, it is difficult to determine the parameters related to convection and to the behavior of the fluid flow itself. Hence, this study is important and relevant to compare the behavior of natural convection with different wall temperature with respect to time.

1.2 Problem Statement

The present analysis of natural convection in rectangular enclosure with different wall temperature has not been the attention of studies both experimentally or numerically. There is no publication and literature of studies conducted for rectangular enclosure for various fluids with varying wall temperature.

Thus, to provide more comprehensive study, the Computational Fluid Dynamics software is used to study the behavior of this transient free flow convection and the governing parameters related to convection.

1.3 Objectives

The main objective of this study is to use Computational Fluid Dynamics (CFD) simulation tool ANSYS Fluent 14.0 to formulate a 2-Dimensional and 3-Dimensional numerical model that can be used to stimulate the behavior of natural convection in the rectangular enclosure.

This model can also be used to stimulate the effects of temperature variations on one side of the wall with respect to time. To vary the wall temperature of the selected side of the rectangular enclosure, a User Defined Function (UDF) program would be created to perform the specific task to vary the wall temperature manually.

The fluid flow behavior and characteristic of different fluid such as liquid metals, gases, water and also oil will also be studied within the rectangular enclosure.

1.4 Scope of Study

This study would be a significant for the fundamental of numerical study of natural convection in rectangular enclosure. This study will focus on the natural convection in vertical rectangular enclosure with temperature variation on one-sided of the wall with respect to time.

The heat flow distribution in the rectangular enclosure is analyzed under multiple temperature differences of the wall and also by using the different parameters such as the effect of aspect ratio, Rayleigh, Nusselt and also Prandlt number to the heat flow regimes and distribution. The heat transfer and fluid flow characteristic of different type of fluid will also be studied.

CHAPTER 2

LITERATURE REVIEW/ THEORY

2.1 Natural Convection

Incopera and Dewit (2002), states that convection is the term utilized to describe the transfer of energy between surfaces and also the fluid movement over respective surface. Conditions in which the fluid is not forced to flow over a surface but still remain to have the convection current within the fluid is regarded as natural convection. Natural convection originated as a force acts on a fluid with gradients of specific mass. The net effect is a buoyant force including the free convection currents. Studies show that the occurrence of specific mass gradient in natural convection is due to the temperature gradient and the force is due to the gravitational field.

Heat transfer into a static fluid leads to a local volumetric expansion. As the gravity induced pressure gradient effect takes place, the expanded fluid parcel would become buoyant and displaced, resulting in heat transfer by fluid motion in addition to conduction. In natural convection, motion of the fluid arises solely due to the temperature differences existing within the fluid for example; hot air rising off the surface of a radiator.

Studies on natural convection inside enclosure began around seven decades ago. Elder (1965) conduct a study to investigate the natural convection of silicon oil and paraffin oil in a vertical slot to stimulate the relationship between buoyancy and shearing forces. Yin et al. (1978) conduct studies to investigate Nusselt number predictions on tall cavities to examine the effect of Aspect Ratio (AR) to the temperature field and heat transfer rate in rectangular enclosure. The result shows that Nusselt number was directly proportional to Grashof number but inversely proportional to AR because the dependent of AR was weak. Elsherbiny et al. (1982)

conducted a study on the effect of temperature gradient and AR to the heat transfer inside vertical enclosure. The findings from their studies shows that heat flow inside enclosures were weakly dependent on AR but strongly dependent on temperature gradient.

Recent studies in 21st centuries also studied vertical and rectangular enclosure in different parameters and objectives. Morini and Spinga (2001) study the effect of varying Nu, Pr and AR to the velocity distribution inside the enclosure. Ganguli et al. (2009) conducted studies to study the effect of varying gap width and temperature differences the conclusions to their findings is tabulated in the table below. Noguiera et al. (2011) conducted a study to investigate the effect of varying Rayleigh number and aspect ratio to the thermal boundary layer of a vertical enclosure.

Table 2.1 Present Studies on Natural Convection

Author	**Fluid**	**Geometry**	**Parameters**	**Variables**	**Conclusion**
Morini and Spiga (2001)	Air	Rectangular ducts(open both end)	Aspect ratio Nusselt Number Prandtl number	(Uniform Wall Temperature) Varying dimensionless number	• Transient T distribution depends on AR • Nusselt number average are time dependent • Velocity distribution depends on Prandtl and increase linearly with Grashof number
Ganguli and Pandit (2009)	Air	Vertical enclosure	Aspect ratio Nusselt Number Prandtl number	Varying gap width Varying temperature difference	• At low Ra number, depending on the value of AR and Pr=0.73, the cells form and remain steady for infinite time. • At high Ra number depending on the value of AR and Pr=0.73, the cells move and reappear due to self-generated pressure gradient.
Nogueira et al (2011)	Air	Rectangular cavities	Rayleigh number Aspect ratio	Varying aspect ratio Varying Rayleigh number	• Rayleigh number influenced the flow profile and heta transfer within cavity • As well as thermal boundary layer thickness • Nusselt number depends on aspect ratio and linerly proportional with respect to aspect ratio
Yin et al (1978)	Air	Vertical enclosure cavities	Aspect ratio Nusselt Number	Different Nu number	• Value of Nu is directly proportional to Grashof number but inversely proportional l to AR • The dependence of AR in fluid flow is weak

2.2 Natural Convection in Enclosed Space

Natural convection flow phenomena inside an enclosed space are an example of very complex fluid system that may leads to analytical, empirical and numerical solution. Figure 2.1; consider a system where a fluid is contained in between two vertical plates separated by distance ***x.*** When a temperature difference of ΔT_w= T_1 –T_2 is applied on the fluid, a heat transfer q will be experienced with the approximate flow regions. This concept is applied in 2D case study. The natural convection inside the enclosure convey the buoyancy forces and temperature change in heat transfer fluid between the walls is accomplished by displacement of the fluid.

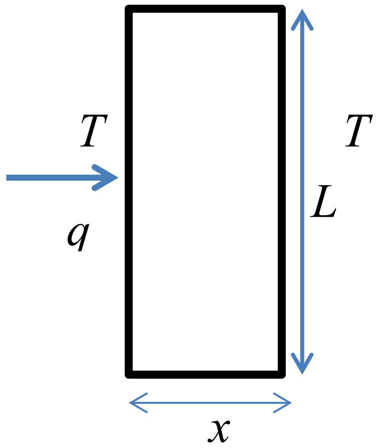


Figure 2.1 Nomenclature of Natural Convection in Enclosed Vertical Space

In study where the constant fluid properties are assumed, the Boussinesq approximation can be applies with negligible viscous dissipation and internal heat sources of the fluid. Newtonian and incompressible fluid is also assumed.Incopera and DeWitt (2002) also state that, the larger the temperature differences between a fluids to two different surface with significant different in temperature, the larger the buoyancy force and the stronger the natural convection currents.

2.3 Mathematical Formulation

To study the behavior of heat transfer inside enclosure by using FLUENT simulation, several assumptions is taken into considerations such as incompressible and Newtonian fluid is used due to the small variation in pressure and hence the flow is accounted for by buoyancy variation (Ganguli et al., 2007). In addition, the temperature of the fluids are assumed to operate in within Boussinesq approximation and the density in the buoyancy term is assumed to vary with temperature hence, this resulted in higher heat transfer rate.

In determining the heat transfer, aspect ratio (AR) is the most important parameter affecting the heat and fluid flow (Varol et al., 2006). Varolet. al also state that the higher rate of heat transfer is obtained at lower aspect ratio (AR) and vice versa

The heat transfer rate inside enclosure is defined as follow:

The rate of heat transfer inside enclosure:

$$Q = hA_s\left(T_{walls} - T_f\right) = mC_p\Delta T \qquad (1)$$

Where the convective heat transfer coefficient, h:

$$h = \frac{Nuk}{L_c} \qquad (2)$$

h is convective heat transfer coefficient (W/m^2°C); Nu is the Nusselt number; k is the air thermal conductivity (W/m°C) and Lc is the characteristic length of the channel (m).

Also, the derived incompressible two-dimensional Navies-Stokes equation can be used together with other supplementary equation such as conservation of mass; Continuity Equation, conservation of energy; Energy Equation and Newton’s second law; Momentum equation can be used to express the behavior of this fluid movement.

Conservation of Mass (Continuity Equation):

$$\rho\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)=0 \tag{3}$$

Newton's Second Law (Momentum Equation):

In *x* direction

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=-\frac{\partial P}{\partial x}+\left\{\frac{\partial}{\partial x}\left[\mu\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[\mu\frac{\partial u}{\partial y}\right]\right\} \tag{4}$$

In *y* direction

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right)=-\frac{\partial P}{\partial y}+\left\{\frac{\partial}{\partial x}\left[\mu\frac{\partial v}{\partial x}\right]+\frac{\partial}{\partial y}\left[\mu\frac{\partial v}{\partial y}\right]\right\}+g\beta\Delta T \tag{5}$$

Conservation of Energy (Energy Equation):

$$\frac{\partial T}{\partial t}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\alpha\left[\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right] \tag{6}$$

Navier-Stokes general equation

$$\rho\left(\frac{\partial v}{\partial t}+v\bullet\nabla v\right)=-\nabla p+\mu\nabla^2+f \tag{7}$$

Navier-Stokes equation assume the fluid studied is indefinitely divisible and do not composed of any particles such as atoms or molecules and that it is static at relativistic velocities (McGraw-Hill, 2008). From the equation mentioned above, a dimensionless model can be developed to determine the parameters characterizing natural convection. Several assumptions can be made before the analysis which is the system is considered to be at steady state.

2.3.1 Prandtl Number (Pr)

Thickness of thermal boundary layer is proven to increase in the flow direction of liquid. Development of velocity boundary layer in relative to the thermal boundary layer will affect the convection heat transfer. Prandtl number is a dimensionless number used to describe the relative thickness of velocity and thermal boundary layers in natural convection. Prandtl number is defined in equation:

$$\Pr = \frac{\upsilon}{\alpha} = \frac{\mu C_p}{k} \tag{8}$$

Some typical ranges of Prandtl number for different types of fluid are tabulated below;

Table 2.2 Prandtl Number Range

Fluid	**Prandtl Number (Pr)**
Liquid metals	0.001-0.030
Gases	0.7-1.0
Oil	50-2000
Water	1-10

2.3.2 Grashof Number, Gr

To measure the relative magnitudes of the buoyancy forces and the friction force opposing on the fluid, Grashof Number (Gr) is used. Gr is dimensionless and it is the ration between the buoyancy force to the viscous force inside an enclosure. The equation is expressed as:

$$Gr = \frac{g\beta\left(T_{walls} - T_f\right)D_h^3}{v^2} \tag{9}$$

All the fluid properties are usually evaluated at the film temperature, T_f as shown below:

$$T_f = \frac{T_{out} - T_{in}}{2}(K) \tag{10}$$

Where

g = Gravitational acceleration, 9.8 m/s²

β = Coefficient of volume expansion, $\beta = 1/T_f$

T_{walls}= Temperature of hot surface (absorber), °C

T_f = Film temperature, °C

D_h = Hydraulic Diameter, m

v = Kinematic viscosity, m²/s

Grashof Number also very important in providing the information in determining whether the fluid flow is laminar or turbulent

2.3.3 Rayleigh Number (Ra)

For an enclosure, Rayleigh Number (Ra) is used to determine the heat transfer correlation. Rayleigh Number is the product of GrashofNumber(Gr) and Prandtl Number (Pr). The equation is expressed as follows:

$$Ra = Gr \cdot \Pr = \frac{g\beta\left(T_{walls} - T_f\right)D_h^3 \Pr}{v^2} \qquad (11)$$

2.3.4 Reynolds Number (Re)

Relation between the inertial force and viscous force in fluid flow is given by the Reynolds Number (Re). This information is used to characterize different flow regimes such as laminar or turbulent flow. At lower Reynold Number, laminar flow occurs while turbulent flow occurs at higher Reynolds Number.

The governing equation of Reynolds Number

$$\mathrm{Re} = \frac{\rho V L_c}{\mu} = \frac{V L_c}{v} = \frac{V D_h}{v} \qquad (12)$$

Where

V = free stream velocity (m/s)

μ = dynamic viscosity of air ($kg/m \cdot s$)

ρ = density of air (kg/m^3)

D_h= hydraulic diameter

2.3.5 Nusselt Number (Nu)

Nusselt Number is a very important parameter in determination of rate of heat transfer as well as the heat transfer coefficient. The correlation of Nusselt number is different for various heat transfer applications. The correlation of Nusselt number based on different case is as follows:

i. Horizontal rectangular enclosures

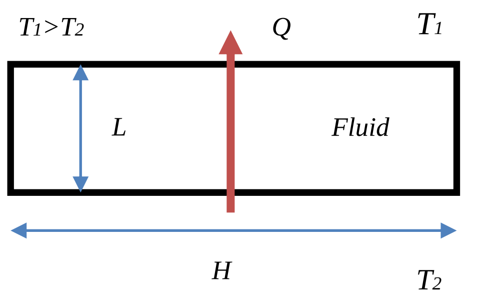


Figure 2.2 Horizontal rectangular enclosures

Figure above shows a horizontal rectangular enclosure at which $= 0°$, the Nusselt number is calculated using the following equation.

$$Nu = 1 + 1.44\left(1 - \frac{1708}{Ra}\right) + \left[\left(\frac{Ra^{1/3}}{5830}\right)\right] - 1 \quad (13)$$

This correlation of Nusselt Number is only valid for Ra $<10^8$

ii. Inclined rectangular enclosures

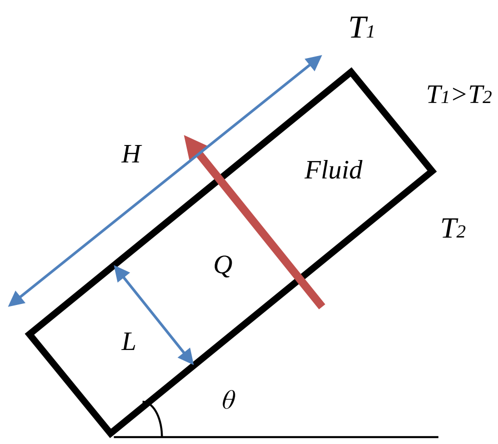


Figure 2.3 Inclined rectangular enclosure

Critical angle for inclined rectangular cavities is tabulated as follows

Table 2.3 Critical Angle for Inclined Rectangular Cavities

(H/L)	**1**	**3**	**6**	**12**	**>12**
$\tau *$	**25°**	**53°**	**60°**	**67°**	**70°**

Transition between two types of motion occurs at critical tilt angle $\tau *$ with corresponding change in Nu. For Pr ≈ 0.7; Ra $<10^5$; aspect ratio: H/L >> 12 and inclination angle at **0° < θ≤ 70°** for system as in figure, the Nusselt number can be determined from equation below:

$$Nu = 1 + 1.44\left[1 - \frac{1708}{Ra\ \cos\theta}\right]\cdot\left[1 - \frac{1708(\sin 1.8\,\theta^{1.6})}{Ra\ \cos\theta}\right] + \left[\left(\frac{Ra\ \cos\theta}{5380}\right)^{1/3} - 1\right] \quad (14)$$

For all aspect ratio beyond the critical tilt angle, the following relation can be used for all aspect ratio (H/L);

$$Nu = Nu\left(\tau = 90^\circ\right)(\sin\tau *)^{1.4} \qquad \left(\tau * \le \theta < 90^\circ\right) \quad (15)$$

iii. Vertical rectangular enclosures

In case of vertical rectangular enclosure where $\theta = 90°$ as shown in figure, the Nusselt number is dependent on the aspect ratio H/L of the enclosure.

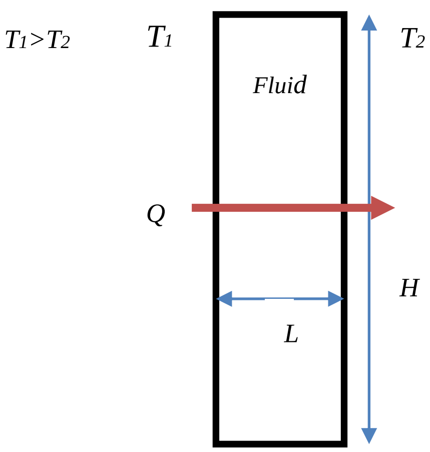


Figure 2.4 Vertical rectangular enclosures

In this research study however, the focus would be given only to the vertical rectangular enclosure varying wall temperature for different fluids. Finite volume method would also be used to solve the numerical problem where the partial differential equations will be represented and evaluated in the form of algebraic equations.

2.4 Mechanism of Flow and Heat Transfer in Vertical Enclosure in Steady-State Cases.

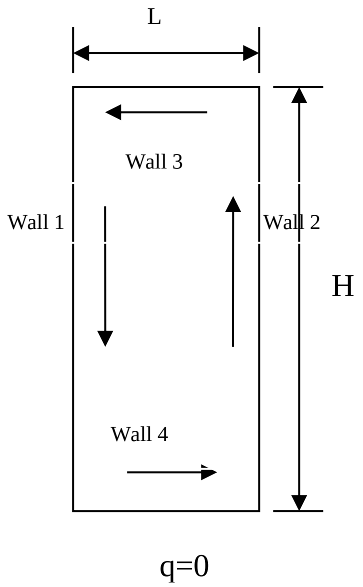


Figure 2.5 Rectangular Enclosures in Steady State Case

Figure 2.5 shows a vertical enclosure with height H and width L. constant temperature is maintained at wall 2, 3 and 4 while the temperature of wall 1 will varies by from 373 K and decrease by 10 K (323 K $<\Delta T<$363 K). Initially, the temperature inside the enclosure is kept at a uniform temperature 373 K however, due to the difference in temperature between wall 2 and wall 1, the fluid rises along the hotter wall 2, turns at the top end, sinks along the colder wall 1 and turns again at the bottom. This mechanism this mode of transfer is termed as unicellular convection.

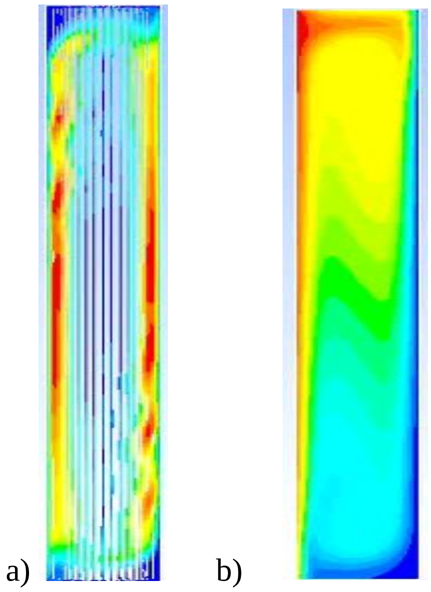


Figure 2.6 a) Streamline b) Temperature Profile

Based on this result, Rayleigh number (Ra), Prandtl number (Pr) and Aspect Ratio, AR can be taken into consideration for the analysis. In general, flow in this problem can be classified into three regimes that are conduction regimes, transitions regimes and boundary layer regimes. Conduction regimes are conduction near the walls and convection in core of the enclosure. The boundary layer regime is convection in the core and conduction limited to a very thin boundary layer near the wall.

It can also be observed that convection cell was formed for the predetermined Rayleigh values. The flow can be classified as deficient and heat transfer principally occurred via conduction through the fluid. Observation also shows that there is no movement in the center region.

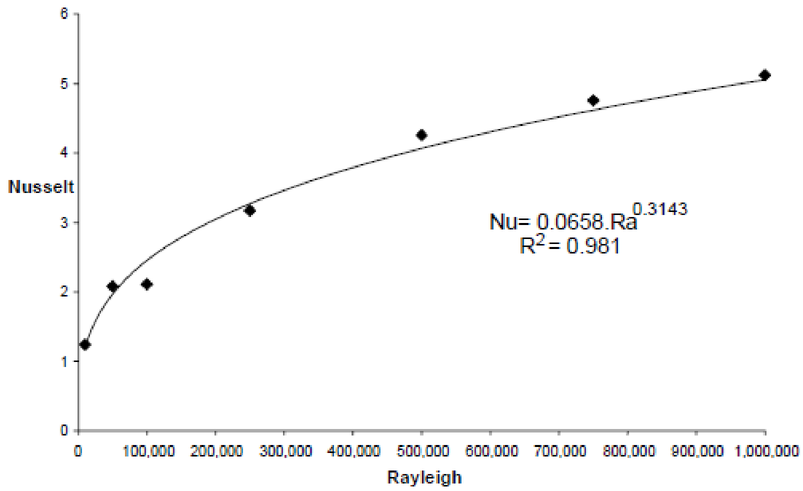


Figure 2.7 Correlation between Rayleigh number and Nusselt number

From the result, the formation of a thermal boundary layer along the hot and cold wall shows the denser behavior towards increment of Rayleigh number. Based on data of the Nusselt number as a function of Rayleigh number, a graph for correlation between the Nusselt and Rayleigh number is plotted. It can be concluded that the flow profile and heat transfer within the cavity as well as thermal boundary layer inside rectangular cavity is influenced by the Rayleigh number and Nusselt number is heavily influenced by the Aspect Ratio, AR of the geometry itself.

2.5 Properties of Different Fluids Selected For Study

There are four groups of fluids being selected as the parameter to stimulate the heat transfer behavior inside rectangular enclosure that is gases, liquid, oil and liquid metals. The purpose for varying types of fluids inside the enclosure is to study the effect of changes in fluid properties such as the density of fluid, viscosity, thermal conductivity, and the effect of different Prandtl number of different fluid to the result.

Table below shows the tabulated properties of four different types of fluids at normal temperature and pressure selected for this study.

Table 2.4 Properties of Different Type of Fluids

Fluid	**Dry Air (gas)**	**Water (liquid)**	**Gasoline (oil)**	**Mercury (liquid metals)**
Density, ρ (kg/m^3)	1.225	998.2	719.7	13529
Dynamic viscosity, μ (kg/ms)	2.485e-05	1.003e-03	3.32-03	0.001523
Prandtl Number, Pr	0.717	7.01	1132	0.029
Specific heat, Cp, (J/kg.K)	1021	4182	2130	139.3
Thermal Conductivity (W/mK)	0.0371	0.6	0.135	8.54
Thermal expansion coeifficient	0.00343	0.00043357	0.000950	0.000182

2.6 User Define Function (UDF) for Transient Heat Transfer within Rectangular Enclosure

In order to stimulate the transient heat transfer of rectangular enclosure, the temperature of one selected wall should be varies with respect to time. In order to define the change in temperature a User Define Function program should be created. User Defined Function is a function provided by the user where the function is built into the system or environment.

In basic programming language, UDF are defined by using the DEF FN syntax. Once created, the function can be used to define the UDF may be used in expression in SQL statements. This also includes SELECT statement where the data can be used together with the data stored in tables in the database for instance, temperature changes data per unit time could be defined in this case.

In this study, the temperature schemes are defined by four different equations which are linear, exponential, logarithmic and sinusoidal. The function for the respected equation is obtained based on the tabulated data and plotted graph.

The following graph is plotted and the equation is obtained from the graph before it is written in C language and interpreted into the simulation.

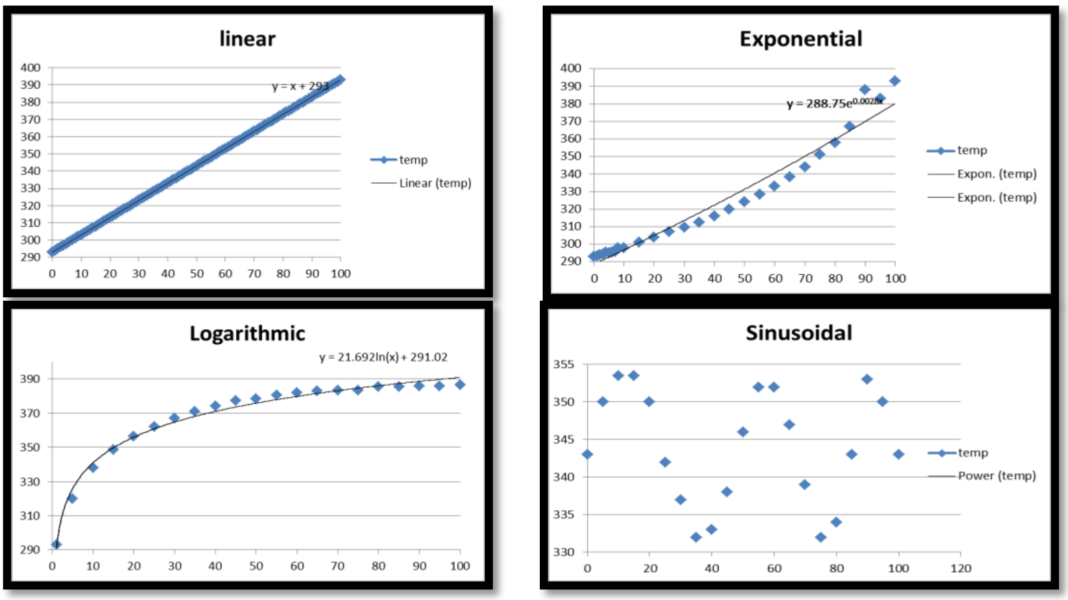


Figure 2.8 Temperature Changes Graph

CHAPTER 3

METHODOLOGY

3.1 Research Methodology

In this study, natural convection within vertical rectangular enclosures will be studied numerically by using Computational Fluid Dynamics (CFD) ANSYS 14.0 to formulate a 2-D and 3-Dimensional numerical model which can be used to analyze the behavior of fluid flow within enclosure, heat transfer and also temperature variations by varying its wall temperature and using different fluids.

3.2 Work Process Flow CFD ANSYS simulation

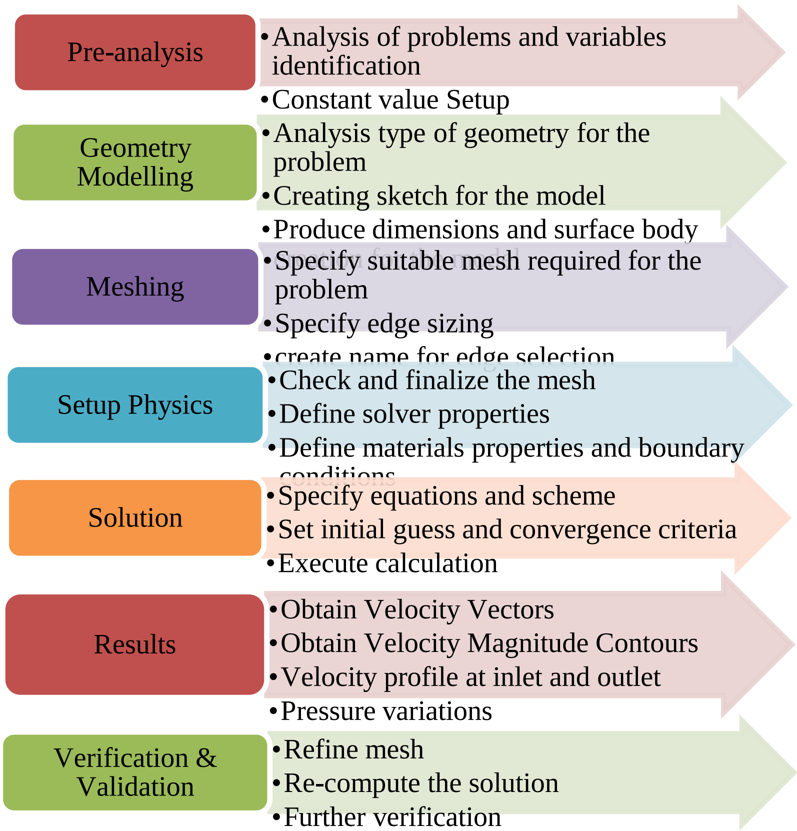
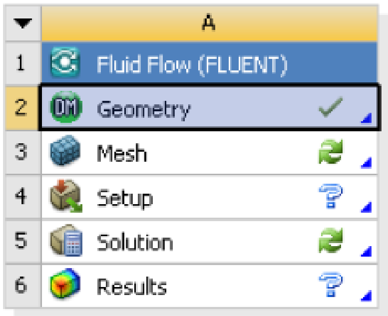


Figure 3.1 Work Process Flow

3.2.1 Pre-analysis & Geometry Modeling

In ANSYS Design Modeler, the rectangular vertical enclosure geometry is set to be done in both 2-Dimensional and 3-Dimensional. In this study, the geometry is constructed to depict the case study model that is rectangular enclosure. The geometry and dimension is set to be constant throughout the simulation for all types of fluid being studied. Surface from sketches is created by using the *surface from sketches* icon. Once the geometry is completed, the object will be imported to the mesher.



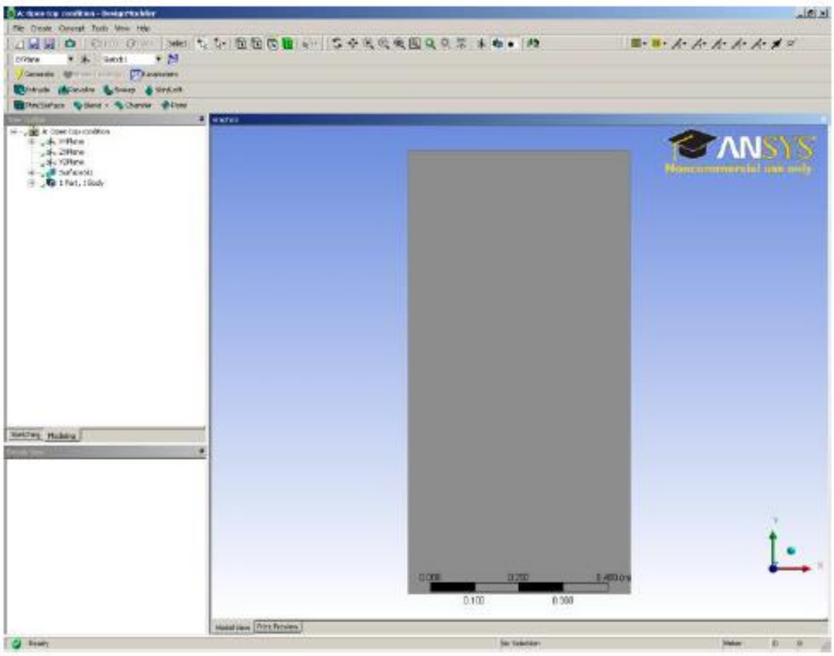
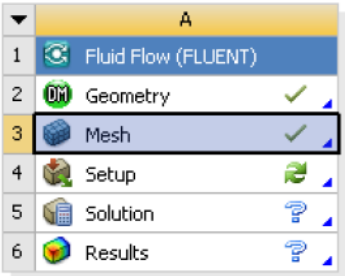


Figure 3.2 Geometry Modeling Using Design Modeler

3.2.2 Meshing

Mesh is defined as the open spaces in a net or network. After the geometry is completed, it is required to define the right meshing condition to yield more accurate display of result because meshing tells the software the mean to perform calculation.



In this study, the grid sensitivity of the mesh is increased along the hot wall and cold wall because this is the point of interest to study the convection currents and contours of temperature. Hence, by increasing the grid sensitivity close to the wall, more calculation would be performed near the wall for better accuracy. A trial and error is conducted to yield the best mesh size for each selected fluid.

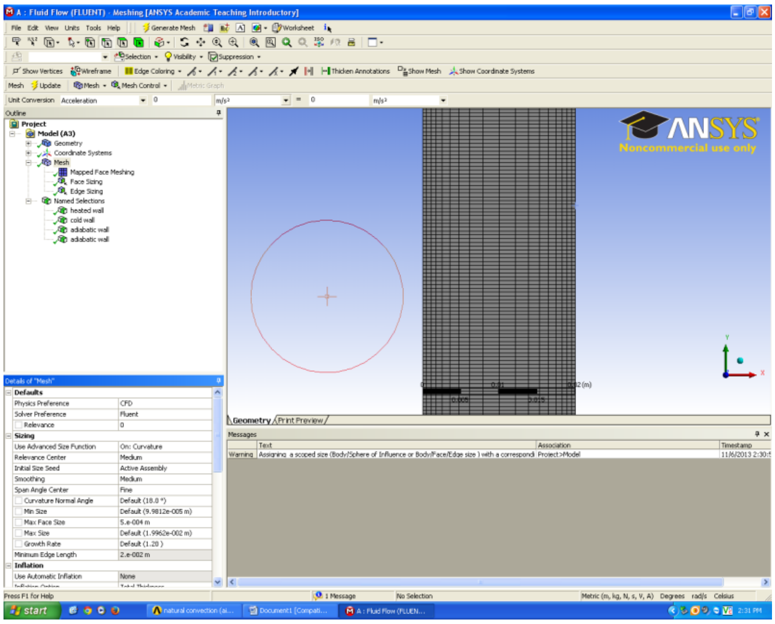


Figure 3.3 Geometry Meshing

3.2.3 Setup Physics

Upon completing the meshing, the geometry model must be checked and run in ANSYS Fluent 14.0. The first step is to specify the general setup as shown below.

3.2.3.1 General

In general setup, the item for the solver must be in pressure based, absolute velocity formulation, transient, 2D planar and with gravitational force acting downward y=-9.81 m/s²

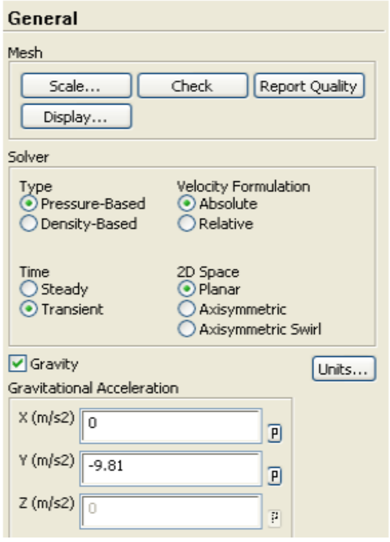


Figure 3.4 General Setup

3.2.3.2 Models

In this study, the model setup is mainly Navier Stokes related equation which is laminar flow model and energy equation.

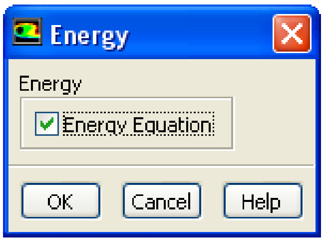


Figure 3.5 Energy Equation Model

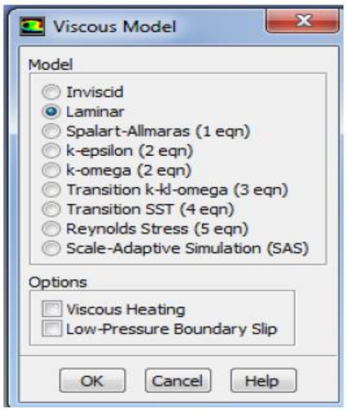


Figure 3.6 Laminar Model Setup

3.2.3.3 Materials

Air

Define the properties of air as the fluid inside the enclosure

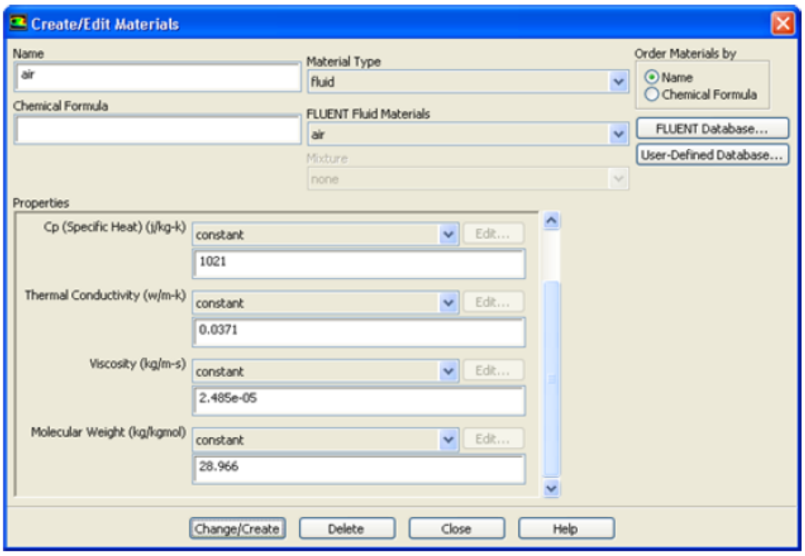


Figure 3.7 Material Setup

a) Incompressible-ideal gas is selected from the density drop down menu. This would set the density to be constant throughout the simulation

b) Set Specific heat (Cp) equal to 1021 J/kg-K

c) Thermal conductivity is set to 0.0371 W/m-K

d) Viscosity is set to 2.485e-05 kg/m-s

e) Default value of Molecular Mass 28.966 is retained

Insulation Material

To ensure the convection is occurring, an insulation material is introduced.

Solid

Insulation material is introduced with the properties below.

a) Set the density as 50 kg/m^3

b) Set the Specific Heat as 800 J/kg-K

c) Set thermal conductivity as 0.09 W/m-K

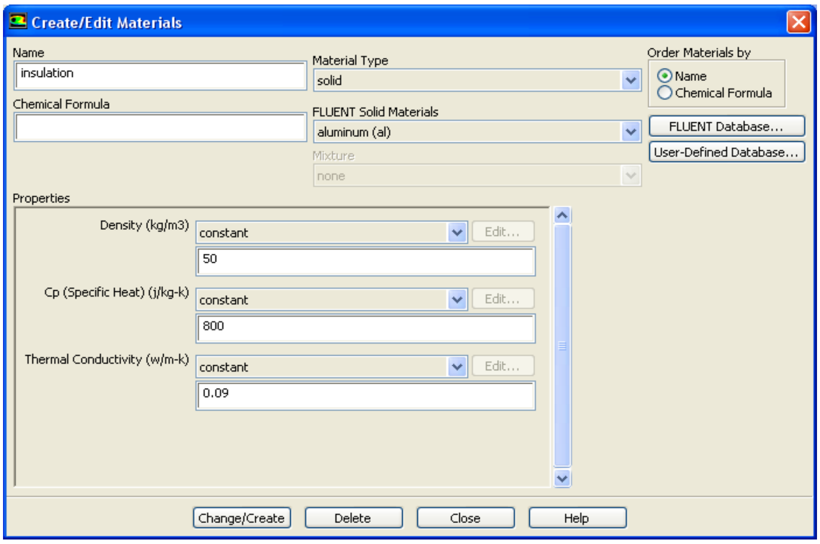


Figure 3.8 Insulation Material Setup

3.2.3.4 Boundary Condition

After completing general setup and model setup, boundary condition needs to be specified for each of the wall. The hot wall boundary condition is set up first. In thermal tab, the thermal condition is set to be temperature where a constant temperature of 308K is inserted.

Next boundary condition to be specified is insulated wall 1 and wall 2. The thermal condition in thermal tab is set to Mixed and insulation is selected as the material in the drop down menu. The following parameter is set for insulated wall 1 and wall 2 as follows;

a) Heat transfer coefficient is 3 W/m-K
b) Free stream temperature is set to be 293.15 K
c) External emissivity is 0.75
d) External radiation temperature is 293.15 K

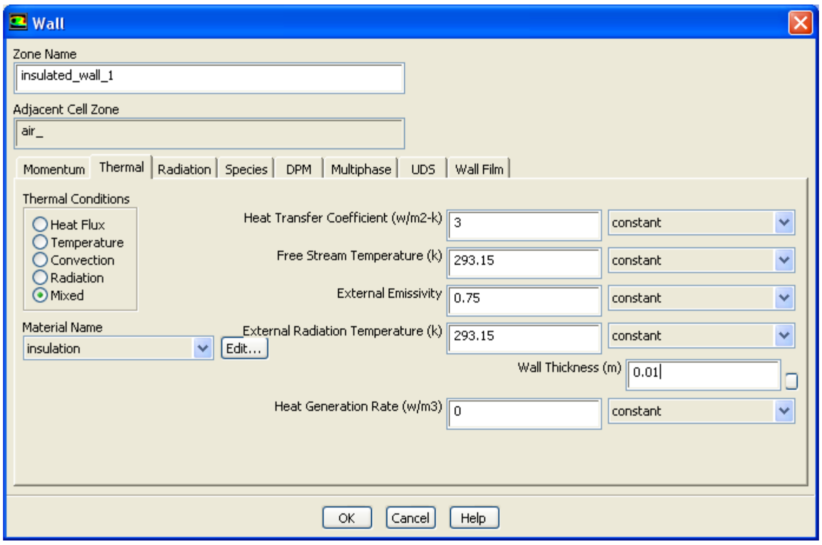


Figure 3.9 Boundary Condition Setup

The last boundary condition to be set up is the cold wall where the constant temperature of 293 K is inserted and kept constant.

3.2.4 Solution

Method of calculation and references value is set in this step. The number of iteration and calculation is important in this step.

3.2.4.1 Solution Method

For natural convection, third order MUSCL scheme is used in order to obtain better accuracy. Time step used is set as 0.01s where the simulation is auto saved every one second. Number of iterations would be varies for each cases and type of fluid depending on the number of calculation required. In transient natural convection of air, the number of iteration is set to be 100 for the solution to converge.

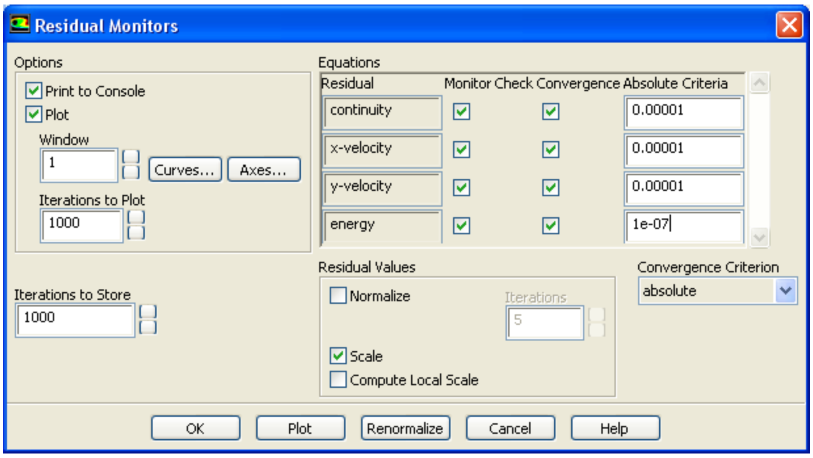


Figure 3.10 Residual Convergence

In order to obtain more accurate results, the number of absolute convergence of each residual equation is set at 10e-05 while for energy equation is set to be 10e-06.

3.2.4.2 Solution Initialization

Solution initialization is retain to be Hybrid Initialization

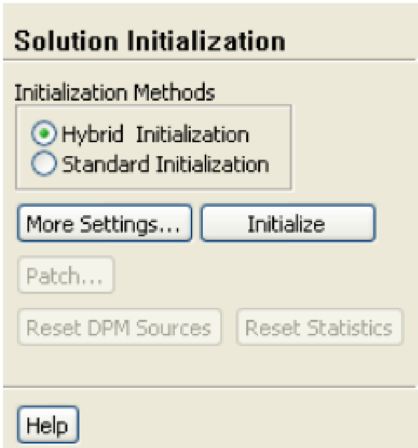


Figure 3.11 Solution Initialization

3.2.5 Results

In result, CFD post processor is launched to verify the results calculated in form of graphical result such as temperature profile, velocity profile, heat flux, graphs for Nusselt number and Rayleigh number, relation of Prandtl number to the behavior of different fluid flow and so on. The results and data calculation can also be analyzed and imported to Excel for further analysis and verification

3.2.5.1 Contours of Temperature

After calculation is converged, CFD post is used to display the temperature contour of natural convection with different temperature differences.

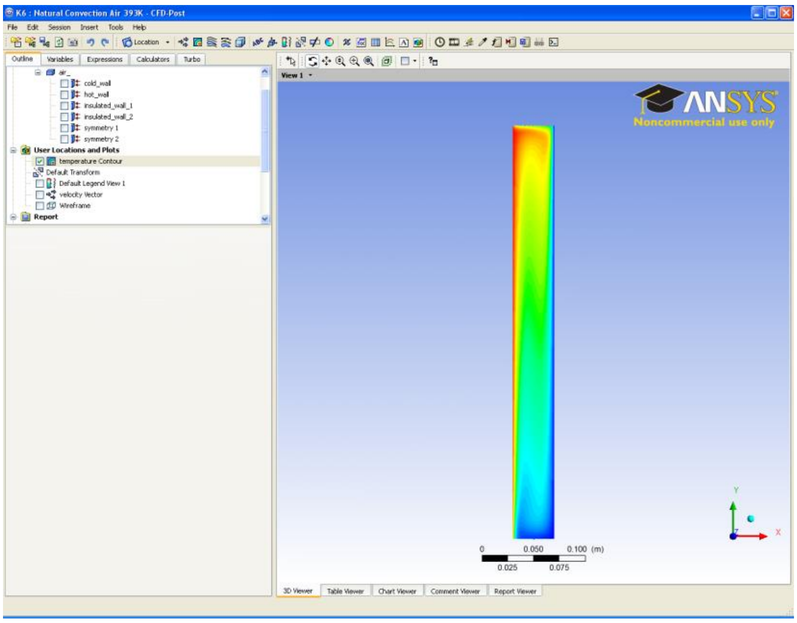


Figure 3.12 Temperature Contour

3.2.5.2 Velocity Vector

Velocity vector of the behavior of natural convection is obtained from setting the CFD post vector to symmetry 1.

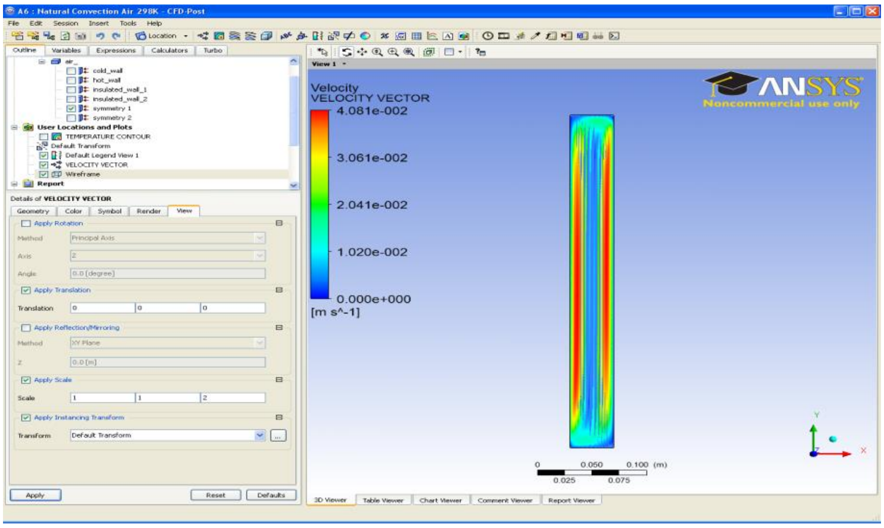


Figure 3.13 Velocity Vector

3.2.6 Verification & Validation

Verification and validation is very important to improve the accuracy and to check the solution. In this step, the project earlier is duplicated and number of mesh for the new project is refined to study the effect of increment in number of mesh towards the results.

3.3 Project Activities Flow Chart

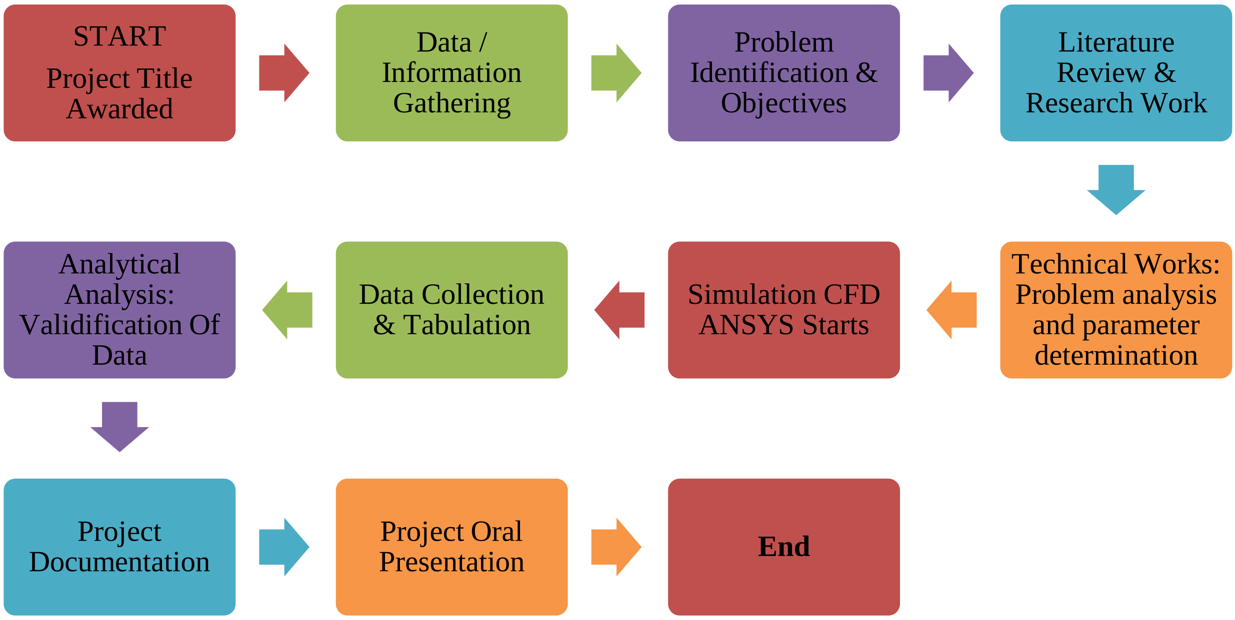


Figure 3.14 Project Activities Flow Chart

3.4 Project Gantt chart

<u>Final Year Project 1</u>

Table 3.15 Project Gantt chart FYP 1

No.	Detail/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Selection of Project Topic	■	■												
	Preliminary Research Work														
	Finding literature review		■	■	■	■									
	Basic exposure on simulation tools					■	■								
	Problem Analysis and parameter determination				■	■	■								
2	**Extended Proposal**														
	Preparation for extended proposal				■	■	■								
	Submission of Extended Proposal														
3	**Proposal Defence**														
	Preparation for proposal defence							■	■						
	Proposal Defence presentation														
4	**Project work continues:**														
	Developing ANSYS Fluent Methodology for natural convection in rectangular enclosure									■	■				
5	**Interim report**														
	Preparation for Darft Interim Report											■	■		
	Submission of Draft Interim Report													*	
	Correction of Interim Report													*	
	Final interim report submission														*

`

* Suggested Milestone

■ Process

Final Year Project 2

Table 3.16 Project Gantt chart FYP 2

No.	Detail/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Project work continues															
	carry out simulation results															
	Analysis and improvement of generated simulation result															
2	Progress report															
	Preparation and submission of progress report								*							
	Discussion of generated results with supervisor															
3	Project Work continues															
	Further analysis Specific literature review reading to verify and enhance result Finalize findings and result															
4	Pre-SEDEX											*				
5	Submission of draft report												*			
	Submission of dissertation (soft bound)												*			
6	Submission of technical paper															
7	Oral presentation														*	
8	Submission of project dissertation (Hard Bound)															*

* Suggested Milestone

Process

CHAPTER 4

RESULT AND DISCUSSION

The fundamental knowledge behind the unicellular convection inside the rectangular enclosures with different type of fluids must be studied in order to study the factor and effects of this mode of heat transfer to the energy saving and conservation. It is also important to analyze the effect of various parameters such as density, temperature changes, how different fluid behave under different temperature changes, and what are the heat transfer coefficient of the fluids under different parameter and also the effect of viscosity variation inside the enclosures.

CFD simulations were carried out for four different type of fluid which is air, water, mercury and gasoline. Air has been used in many studies numerically and experimentally by Elsherbiny et al. (1982), Batchelor (1954), Yin et al. (1978) and Wakitani (1997) and the results are compared in many literatures with respect to parameters such as aspect ratio, viscosity variation, Nusselt Number variation, Rayleigh number, Grashof number and also temperature different.

4.1 Different in Velocity Profile of Different Type of Fluids.

a) b) c) d)

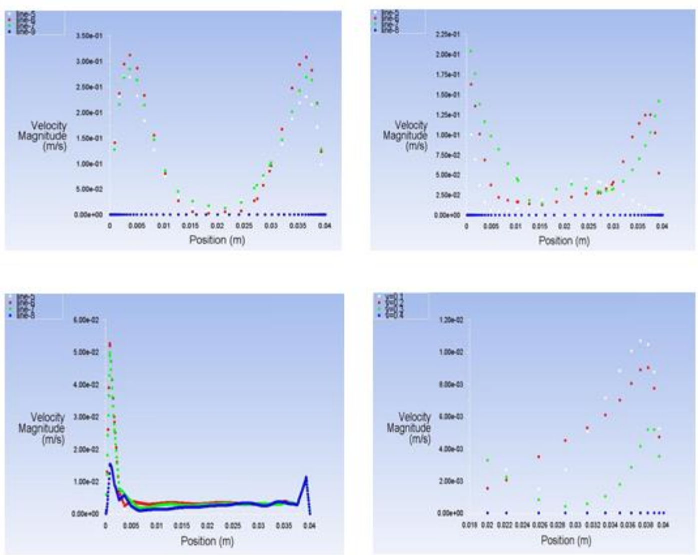


Figure 4.1 Graph of velocity vector for different fluids a) Air b) Mercury c) Oil d) Water

Figure 4.1 shows the velocity magnitude for different fluid with respect to the position at different point y=0.1, y=0.2, y=0.3, y=0.4. Four different lines are drawn on the rectangular enclosures to study the different velocity profile. In general, as temperature change, the velocity of the air inside the enclosure also increases linearly. The vector describes the movement of heat in y direction to the top and return to the bottom of the rectangular enclosures creating a unicellular convection from the graph, it can be seen that the velocity of air is the highest compared to the others and it behaves in sinusoidal manner. However, it can be concluded that, at different position of line across the enclosures, there are no significant changes from one point to another.

Figure 4.1 b) shows the velocity behavior of mercury. The velocity magnitude decreased as the heat transfer but increased as the distance to the cold wall increased. The significant different could be seen at the beginning of the convection where the velocity behaves in sine mode before it’s velocity reduced and change as the level increased.

Oil has the lowest velocity magnitude compared to the other type of fluids. This might be due to the high viscosity and lower thermal conductivity compared to water or mercury. Water however shows a significant increment of velocity as the convection occurs. However, as it moves closer to the cold wall, the velocity reduced abruptly with respect to distance.

4.2 Effect of Flow Patterns on Heat Transfer for Different Fluid and Different Temperature.

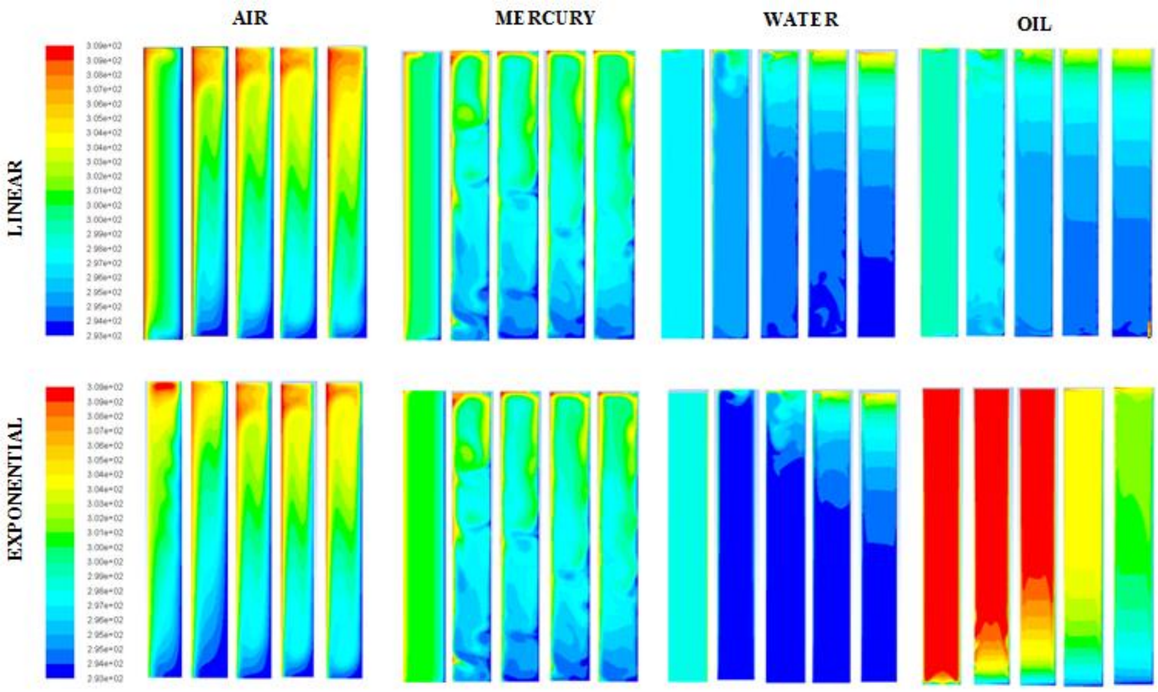


Figure 4.2 a) Temperature contour of Different fluids at time a) 0s b) 20 s c) 40 s d) 60 s e) 80 s for linear and exponential change in T

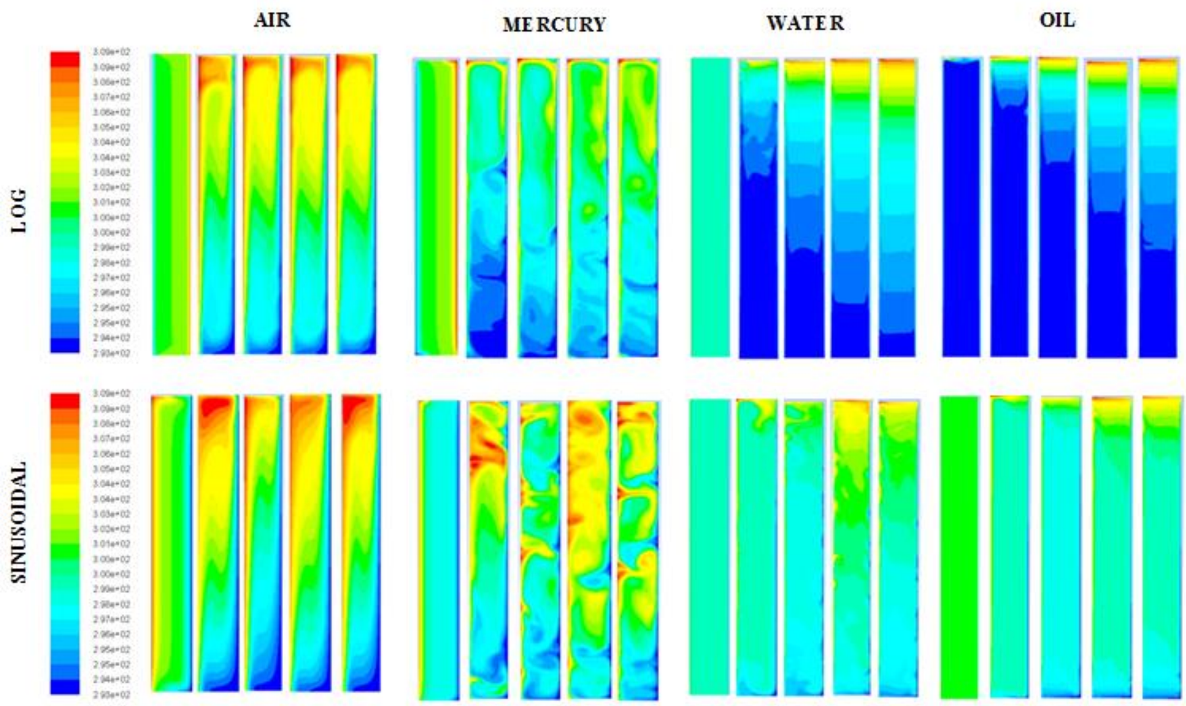


Figure 4.2 b) Temperature contours of Different fluids at time a) 0s b) 20 s c) 40 s d) 60 s e) 80 s for logarithmic and sinusoidal change in T

Understanding the fluid dynamics in the entire enclosures is possible by analyzing the temperature distribution in the enclosures. Figure 4.2 a) and Figure 4.2 b) shows the compilation of temperature distribution of four different type of fluid and four different behavior of temperature changes selected in this studies with respect to time at 0s, 20s, 40s, 60s, and 80s respectively.

In general, the theoretical unicellular mode of heat transfer can be seen in all the enclosures. Air shows the linear and continuous behavior of temperature distribution where the hot air rises along the hot wall, turns at the top and sink along the cold wall slowly. Mercury however distributes heat in vigorously along the enclosures. The irregular distribution of hot and cold fluids can be seen clearly and the rate is very high compared to the others. Water and oil behaves in almost similar manner where the hot fluid rises slowly and circulates on top part of the enclosures before uniform distribution of heat across the enclosures is achieved. However, the rate of temperature distribution in oil is about ½ times slower than that of water. This might be due to the high viscosity with lower thermal conductivity of oil compared to water itself.

In linear temperature changes of air, there are no significant different in the distribution compared to the steady state behavior that has been studied in the literature except for the rate of heat transfer. However, in exponential and logarithmic changes in temperature shows significant different in temperature distribution where the set UDF function for temperature changes in hot wall usually set the operating temperature of the hot wall to be lowest than the cold wall temperature of 293K. Hence, the distribution behavior is symmetry to the both wall and the hot wall rises from the bottom of the enclosures rather than form the hot wall before evolving into the unicellular convection. Exponential and logarithmic behavior usually is inverse of one another. Sinusoidal change in temperature signifies the rapid changes in temperature reduction and increment simultaneously.

4.3 Variation of Heat Transfer Coefficient

The data of heat transfer coefficient within the rectangular enclosure with different type of fluids from the simulation is tabulated in the table below. The graph of HTC for different fluid is plotted for different distance.

Table 4.1 Heat Transfer Coefficient

Fluids	**T Change**	**HTC**	**K**	**Lc**	**Nu**	**HTC Calc.**
Air	Linear	0.12	0.04	0.04	0.13	0.18
	Exponential	0.64	0.04	0.04	0.69	0.96
	Log	0.62	0.04	0.04	0.67	0.93
	Sine	0.02	0.04	0.04	0.02	0.03
Water	Linear	265.69	0.60	0.04	17.71	398.54
	Exponential	57.45	0.60	0.04	3.83	86.17
	Log	95.98	0.60	0.04	6.40	143.98
	Sine	0.97	0.60	0.04	0.06	1.45
Mercury	Linear	280.95	8.54	0.04	1.32	421.43
	Exponential	193.32	8.54	0.04	0.91	289.98
	Log	319.28	8.54	0.04	1.50	478.92
	Sine	9.80	8.54	0.04	0.05	14.71
Oil	Linear	27.09	0.14	0.04	8.03	40.63
	Exponential	7.85	0.14	0.04	2.33	11.77
	Log	22.91	0.14	0.04	6.79	34.36
	Sine	0.60	0.14	0.04	0.18	0.90

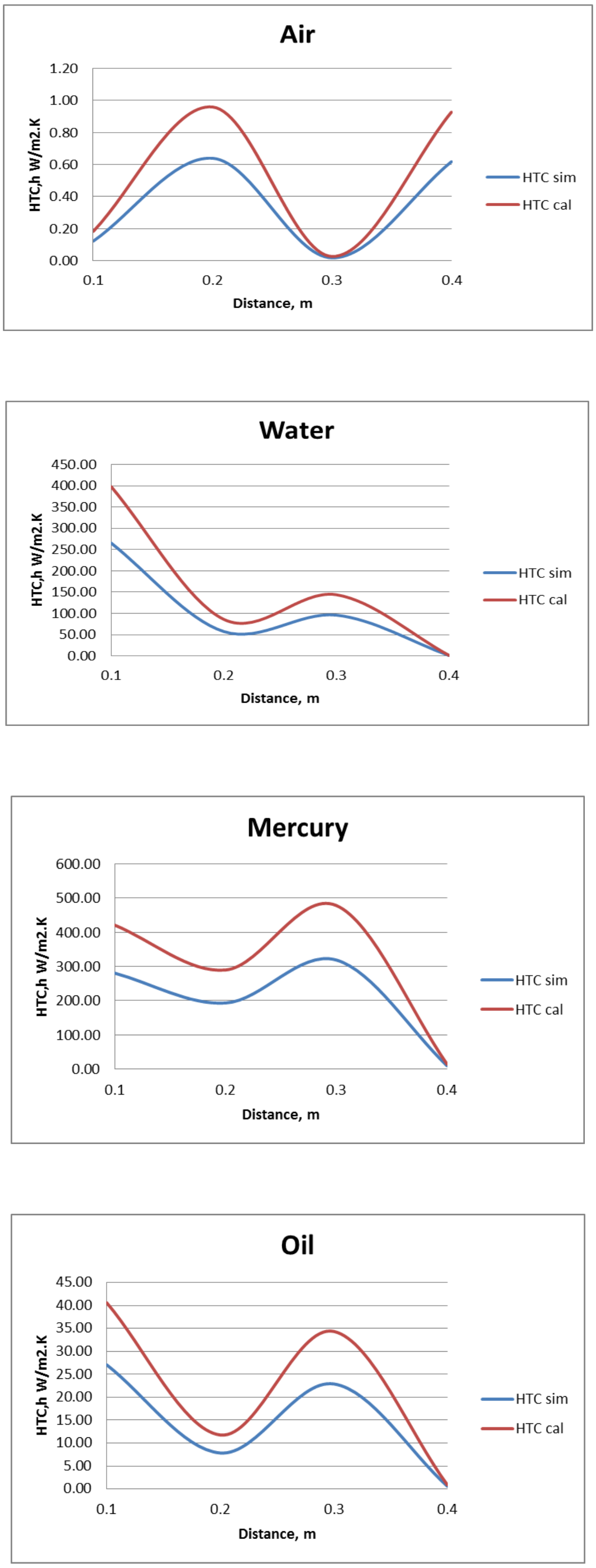


Figure 4.3 Heat Transfer Coefficients vs. Distance

From the plot above, it can be seen that the heat transfer coefficient can be estimated from the result in the simulation and compared with the HTC calculated based on the temperature correlation. In most type of fluids, the heat transfer decresed along the distance and increased after some distance. The deviated simulation result and calculated result is estimated to be ±47% and this is due to the significant changes in temperature itself.

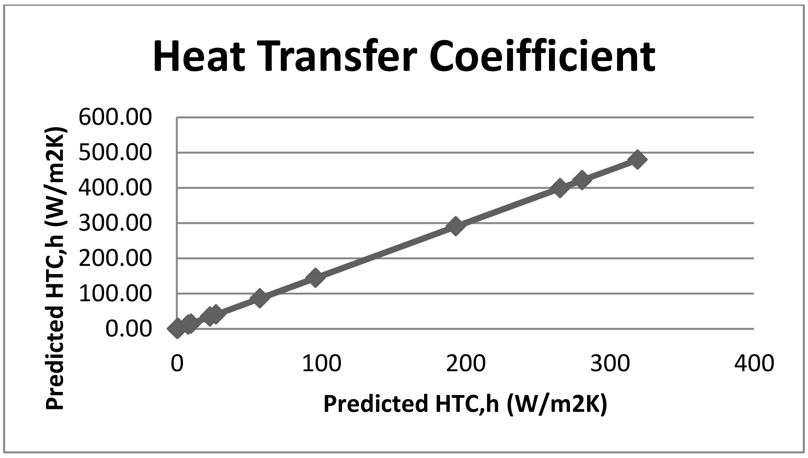


Figure 4.4 Heat Transfer Coefficients

4.4 Effect of Viscosity Variation

There are also variation in viscosity with the given temperature variation at small timescale (t < 250) compared to the assumed constant velocity. The same findings were made by Wakitani (1996) where the author investigates the effect of viscosity variation with temperature of air and silicon oil. It is found that the viscosity of air increases with temperature for air and decreased with temperature for oil. In this study, the viscosity variation is determined for four different types of fluids and the correlation between viscosity and temperature different can be drawn.

From the simulation, the air behaves in line with the findings from Wakitani (1996) studies where the viscosity of air increases with time. Oil and water however, behaves in reversed manner from air where the viscosity is decreasing as the temperature increases and that the movement of fluids inside the enclosure also slower in water and oil respectively. Mercury shows a very significant change in viscosity with respect to temperature changes. The rate of viscosity changes is highest in mercury followed by air.

CHAPTER 5

CONCLUSION

In conclusion, this work discussed on the fundamentals of transient natural convection inside rectangular enclosures with variation of temperature changes at one specified wall. The simulation is performed in 2D cylindrical geometry of 0.4 m by 0.04 m. the types of fluids selected for this study are air, water, mercury and gasoline. The CFD simulation has been performed for specific rectangular enclosure for different fluid by varying wall temperature by using the defined UDF (User Defined Function). The simulation result shows that temperature distribution and flow regimes within the rectangular enclosure are different for different temperature change and different type of fluid.

The rate of heat transfer inside the enclosures is affected by density, temperature along the hot wall, thermal coefficient different and thermal conductivity of different types of fluids. In air, the natural convection behavior obey the unicellular convection behavior when the air rises along the hot wall, turns on top and sink down alongside the cold wall creating a complete cycle of heat. Mercury transfer heat at more vigorous manner due to the high thermal expansion coefficient. Water and oil behaves almost the same in transporting the heat inside the enclosure.

Changes in heat transfer coefficient can be seen clearly with different type of fluids at different regimes of temperature change. From the analysis result of the study simulation, comprehensive analysis has successfully verified the fundamental objectives. Currently, time steps used is 0.1 s whereas the number of iteration is set to be 800 to produce 80s result. In later stages, more improved simulated result can be generated with better meshing size, better time steps and also the geometry could be analyzed in 3 Dimensional. More accurate result can be analyzed with better time steps and more iteration. Good quality video can also be produced to stimulate the behavior of heat transfer inside enclosures.

REFERENCES

Adachi, Takahiro. (2006). Stability of natural convection in an inclined square duct with perfectly conducting side walls.*International Journal of Heat and Mass Transfer, 49*(13–14), 2372-2380.

Altaç, Zekeriya, &Kurtul, Özen. (2007). Natural convection in tilted rectangular enclosures with a vertically situated hot plate inside.*Applied Thermal Engineering, 27*(11–12), 1832-1840.

Ayyaswamy, P. S., and Calton. I. (1973). "The Boundary Layer Regime for Natural Convection in a Differentially Heated, Tilted Rectangular Cavity," J. Heat Transfer, 95, 543.

Ayyaswamy, P. S., and Calton.I. (1973). "The Boundary Layer Regime for Natural Convection in a Differentially Heated, Tilted Rectangular Cavity," *J. Heat Transfer*, 95, 543.

Catton, I., "Natural Convection in Enclosure," *Proc. 6th Int. Heat Transfer Conf.*, Toronto, Canada, 1978, Vol. 6, pp. 13-31.

Catton, I., "Natural Convection in Enclosure," *Proc. 6th Int. Heat Transfer Conf.*, Toronto, Canada, 1978, Vol. 6, pp. 13-31.

Di Piazza, Ivan, &Ciofalo, Michele. (2000). Low-Prandtl number natural convection in volumetrically heated rectangular enclosures: I. Slender cavity, AR = 4. *International Journal of Heat and Mass Transfer, 43*(17), 3027-3051

Elsherbiny, S.M., Raithby, G.D. and Hollands, K.G.T., 1982, Heat transfer by natural convection across vertical and inclined air layers. Trans. ASME J. Heat Transf., 104: 96–102.

FLUENT 14.0., 2011. User’s Manual to FLUENT 6.2. Fluent, Inc. Centrera Resource Park, 10 Cavendish Court, Lebanon, USA.

Ganguli, A.A., Pandit, A.B. and Joshi, J.B., 2007, Numerical predictions of flow patterns due to natural convection in a vertical slot. Chem. Eng. Sci., 62(16): 4479–4495.

Holman, J. P. (2010) “Combined Free and Forced Convection,” *Heat Transfer*, 10th Edition, pp. 358-360, McGraw Hill Education.

Incropera, F.P. and Dewitt, D.P., (2002). Fundamentals of Heat and Mass Transfer (5th ed.). (Wiley & Sons, New York) (Appendix-A,p. 917)

Incropera, F.P., Dewitt, D. P., Bergman T. L. and Lavine, A. S. (2007) “Introduction to Heat Transfer,” John Wiley & Sons (Asia) Pte Ltd, 5th Ed., pp. 551-554.

Jakob, M., (1949). Heat Transfer (Wiley & Sons, New York) (Chapter 25)

MacGregor, R. K., and A.P. Emery (1969), “Free Convection through Vertical Plane Layers: Moderate and High Prandtl Number Fluids,” *J. Heat Transfer*, 91, 391

MacGregor, R. K., and A.P. Emery (1969), “Free Convection through Vertical Plane Layers: Moderate and High Prandtl Number Fluids,” *J. Heat Transfer*, 91, 391.

Morini, G. L., &Spiga, M. (2001).Transient laminar natural convection along rectangular ducts.*International Journal of Heat and Mass Transfer*, *44*(24), 4703-4710.

Wakitani, S., 1994, Experiments on convective instability of large Prandtl number fluids in a vertical slot. J. Fluid Mech., 116:120–126.

Wakitani, S., 1996, Formation of cells in natural convection in a vertical slot at large Prandtl number. J. Fluid Mech., 314:299–314.

Wakitani, S., 1997, Development of multicellular solutions in natural convection in an air-filled vertical cavity. Trans. ASMEJ. Heat Transfer. 119: 97–101