UN VER ITI
TE JOL GI
PE RO AS

FREE/ MIXED CONVECTION IN VERTICAL POROUS CHANNEL

BY

ALI IBRAHIM KABBASHI BALOW

Final Report

Submitted to the Chemical Engineering Program

In Partial Fulfillment of the Requirements

For the Degree

Bachelor of Engineering (Hons)

(Chemical Engineering Program)

May 2011

Universiti Teknologi Petronas

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Free/Mixed Convection in Vertical Porous Channel

By

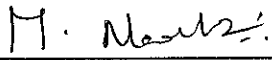Ali Ibrahim Kabbashi Balow

A project Final report submitted to the

Chemical Engineering Program

In partial fulfillment of requirements for the Bachelor of Engineering

(Hons)

(Chemical Engineering)

Approved by:

M. Narahari

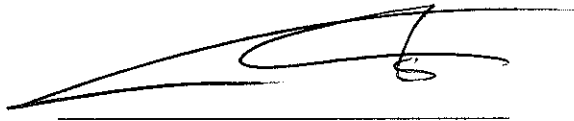**Dr, MARANENI NARAHARI**

Project Supervisor

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

May 2011

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 that the original work contained herein have not been undertaken or done by unspecified sources or person.

ALI IBRAHIM KABBASHI BALOW

ABSTRACT

Convective heat transfer or simply convection is the study of transport processes Effected by the flow of fluids. Convective heat transfer has grown to the status of a contemporary science because of people desire to understand and predict how a fluid flow will act as a carrier or conveyer belt for energy and matter. How ever

Convective heat transfer is clearly, afield at interface between two older fields: heat transfer and fluid mechanics. For this reason, the study of any convective heat transfer problem must reset on a solid understanding of basic heat transfer and fluid mechanics principles.

Since this research will cover free convection and porous medium we discuss briefly about it. Fluid flow and convection heat transfer in porous media have been widely investigated experimentally and numerically for many years due to the many important applications, such as catalytic and chemical particle beds, petroleum processing, geothermal energy extraction, transpiration cooling, packed-bed regenerators, solid matrix heat exchangers, micro-thrusters and many others. Porous media can intensify fluid mixing and increase the surface area in contact with the coolant, so porous structures are an effective heat transfer augmentation technique. There have been numerous investigations with theoretical analyses and numerical simulations of convection heat. However in this project the simulation have been done and the results have been obtained and the maximum velocity of the hot air is about 0.0003m/s moreover the velocity in figure11showed the maximum near the heated wall and in natural convection the maximum velocity is on the heated wall. therefore the velocity is very identical to the assumption of Darcy flow is valid when the fluid is not too fast ref [7] For the temperature is about 1270K and the temperature we can manipulate it by supply heat flux at high or low see the appendices 1&2 check temperature contour for different values for heat flux therefore we can conclude that the velocity obtained by simulation is satisfied as well as the temperature.

ACHNOWLEDGEMENT

In the name of Allah, the most Gracious and the most Merciful

First and foremost, Praise to Allah the almighty for bestowing me with the strength, health as well as intensity to accomplish this project.

However, there are many people who helped me while working on this project with their help and   continuous support I could be able to come to the end of my final year project endeavor. Hereby I would like to relish this opportunity to express my heartily gratitude towards all of them. First of all, I would like to thank Dr. MARANENI NARAHRI, the project supervisor for his support as well as for his guidance throughout the time of doing this project. and I would like to thank Dr RAJASHEKAR PENDYAlA for his support and guidance as well.

Also I would like to acknowledge all my friends, who have given me invaluable support and advice as well as feedbacks on my project. Lastly and most of all I would like to express my gratitude to my parent for their care and their endless motivation which given me strength in all my undertakings as well as in finishing this project.

May Allah, the most generous, bless all of you, Thank you very much?

TABLE OF CONTENTS

CHAPTER 4

CHAPTER 5

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

Cp	heat capacity, J/kg k		Greek symbols
g	gravitational acceleration, m/s2	α	Thermal diffusivity ,m2/s
H	length, m	β	Thermal coefficient of expansion ,1/k
Kp	thermal conductivity of porous medium w/mk		
K	permeability ,m2	δ_T	Thermal boundary layer thickness ,m
L	height ,m	Φ	Porosity
m`	mass flow rate ,kg/s	v	Viscosity ,m2/s
Nu	Nusselt Number	ρ	Density ,kg/m3
P	Pressure ,Pa	ψ	Stream function m2/s
q``	Heat flux ,w/m2		
Q`	Heat transfer rate, W		
Ra	Rayleigh number		
T	Temperature, k		
u, v	Velocity components ,m/s		
x, y	Coordinates ,m		

Table 1: Nomenclature ref [7]

CHAPTER 1

1.1. INTRODUCTION

A vertical channel is attached to the exterior wall of a building .solar radiation warms up the surface And upward self driven flows are triggered by the temperature difference between the collector and the ambient (natural convection) this air stream can be used for ventilation and heating of building .its common this system works in conjunction with a fan to enhance the flow intensity when required (mix convection).Yanik.Boutin (2009) however The geometry of vertical open ended channels can be optimized to maximize the heat transfer rate and facilitate fluid flow such channels can be used as solar wall and solar chimney. These systems are usually fan assisted and expected to achieve high energy efficiency.

Therefore in this research the channel shape and the operating condition (Reynolds number, Nusselt Number) will be studied carefully to know the effects on the heat transfer and energy recovery.

Moreover we model vertical channel such as solar wall filled with porous medium(metallic foams) to enhance heat transfer under mixed convection, and porous structures in thermal system such as solar wall and solar chimney can play an important role for improving energy efficiency.

The effects of the channel breadth .Rayleigh number and heat loss coefficient and the optimization of the pressure drop imposed by the ventilation system will be studied.

1.2 Porous medium

By a porous medium we mean a material consisting of a solid matrix with an interconnected void .we suppose that the solid matrix is either rigid (the usual situation) or undergoes small deformation .the interconnectedness of the void (the pores) allows the flow of one or more fluids through the material .in the simplest situation (single phase flow) the void is saturated by a single fluid in two phase flow, a liquid and a gas share the void space.

In natural porous medium the distribution of pores with respect to shape and size is irregular. Examples of natural porous media are beach sand, sandstone, and limestone, and rye bread, wood and human lung.

On the pores scale (microscopic scale) the flow quantities (velocity ,pressure etc) will be clearly irregular .But in typical experiments the quantities of interest are measured over area that cross many pores and such space –averaged (macroscopic) quantities change in regular manner with respect to space and time ,and hence are amenable to theoretical treatment.

How we treat a flow through a porous media structure is largely a question of distance – the distance between problem solver and actual flow structure (Bejan 2004) when the distance is short the observer sees only one or two channels, or one or two open or closed cavities .in this case it is possible to use conventional fluid mechanics and convective heat transfer to described what happens at every point of the fluid and solid filled spaces.

When the distance is large so that there are many channels and cavities in the problems solver's field of vision, the complication of the flow paths rule out the conventional approach in this case volume averaging and global measurements (eg permeability ,conductivity) are useful in describing the flow and in simplifying the description.

As engineers focus more and more on designed porous media at decreasing pore scales, the problems tend to fall between the extremes noted above .in intermediate range, the challenge is not only to describe coarse porous structures but also to optimize flow elements and to assemble them the resulting flow structures are

designed porous media (Bejan ,2004b) figure below illustrate more on porous medium
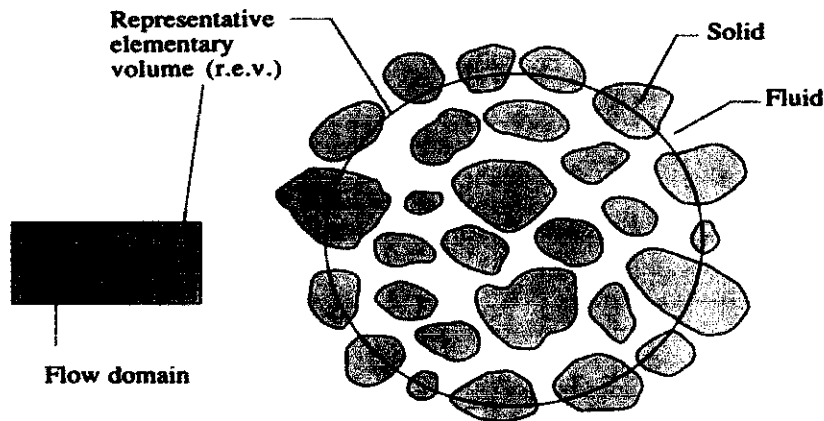

Figure 1: porous medium

Many research activities in physics have been recently focused on convective heat transfer in porous media because of its relevance to variety of situation occurring in engineering application and nature among these works, natural and forced convection studies occupy the majority of the investigation.

The inter-facial area of mixed convection ,which connects natural and forced convection ,however ,has not been given due attention in porous media ,most of available studies are related to stable laminar flow and comprehensive review of it .understanding the dynamics behavior of mixed convective fluid flow through porous media ,especially flow transition using a proper model is still a challenge in fluid dynamics research.

1.3 Fluid flow in porous channel

The flow problem in porous tubes/channels received much attention in recent years because of its various applications in biomedical engineering, for example in the dialysis of blood in artificial kidney, in the flow of blood in the capillaries, in the flow in blood oxygenators, as well as in many other engineering areas such as the design of filters, the general porous pipe design, in transpiration cooling boundary layer control and gaseous diffusion. In boundary layer control, the decelerated fluid particles in the boundary layer are removed through slits in the wall into the interior

of the body. Separation can be prevented through sufficiently strong suction. Alternatively, separation can be prevented by supplying additional energy to the fluid particles which are being retarded in the boundary layer. Also in separating $U235$ from $U238$ by gaseous diffusion, the uranium is first converted to the gas $UF6$. By pressure gradient, the gas is then forced through a porous wall. The difference in the molecular weights causes differences in the rates of diffusion through the porous material. Furthermore, the blowing is used to add reactants, prevent corrosion and reduce drag. However suction is helpful to remove reactants in chemical processes. Extensive work is available for the flow of a Newtonian

Fluid in a porous channel. However, made an interesting, study for the hydrodynamic Maxwell fluid in aporous channel.

The magnetohydrodynamic (MHD) flow between parallel plates is a classical problem that occurs in MHD power generators,MHDpumps, accelerators, aerodynamic heating, electrostatic precipitation, polymer technology, petroleum industry, purification of crude oil and fluid droplets and sprays. Especially the flow of non-Newtonian fluids in channelsare encountered in various engineering applications. For example, injection molding of plastic parts involves the flow of polymers inside channels. During the last few years the industrial importance of non-Newtonian fluids in widely known. Such fluids in the presence of a magnetic field have applications in the electromagnetic propulsion, the flow of nuclear fuel slurries and the flows of liquid state metals and alloys. (Sarparkaya) presented the first study for MHD Bingham plastic and power law fluidsT. Hayat et al (2008).

Due to non-linear dependence, the analysis of the behavior of non-Newtonian fluids presents exciting challenges to engineers, mathematicians and physicists alike. The equations which govern the flow of non-Newtonian fluids are of higher order , much more complicated and subtle in comparison with that of Newtonian fluids. But there is not a single constitutive equation which can describe the flow behavior of all the non-Newtonian fluids. Because of the complex microstructure of fluids, various models have been proposed to predict the non-Newtonian behavior. Several investigators are now engaged in getting the solutions under different physical aspects. One class of non-Newtonian fluids which has gained considerable attention in recent years is the differential type. Among these fluids one special subclass for which one can reasonably hope to obtain an analytic solution is the second grade

fluid. Although the second grade fluid model for steady flow is able to show the normal stress effects but it does not take into account the shear thinning or shear thickening phenomena that many fluids show. The third grade fluid model represents a further, although inclusive; attempt toward a more comprehensive description of the behavior of non-Newtonian fluids.Keeping this importance of third grade fluid. T. Hayat et al (2008)

1.4. BACKGROUND

Convection flows in porous media have been extensively investigated during the last several decades and they have included several different physical effects. This interest is due the many practical applications which can be modeled or approximated as transport phenomena in porous media.

In flows appears in wide variety of industrial applications, as well as in many natural circumstances such solar wall ,solar chimney (in buildings) geothermal extraction, storage of nuclear waste material ,ground water flows ,industrial and agricultural water distribution ,oil recovery process ,thermal insulation engineering ,pollutant dispersion in aquifers ,cooling of electronic components ,packed bed reactors ,food processing ,casting and welding of manufacturing process ,the dispersion of chemical contaminants in various processes in chemical industry and in environment, soil pollution ,fibrous insulation ,liquid metal flow through endric structures in alloy casting and even for obtaining approximate solution for flow through turbo machinery. This topic is of vital importance in all these application , therefore comprehensive literature concerns these process can be found and most recent references Bejan(2004) Ingham and pop (1998).

The convective heat transfer potential of flow through porous media is relatively new as the technology of porous insulation, gas cooled electric machinery, and nuclear reactors grew out of the contemporary concern with the cost of energy and minimization of cooling scheme

1.5. Convection heat transfer theory

Heat energy transfers between a solid and a fluid when there is a temperature difference between the fluid and the solid. This is known as "convection heat transfer". Generally, convection heat transfer cannot be ignored when there is a significant fluid motion around the solid. The temperature of the solid due to an external field such as fluid buoyancy can induce a fluid motion. This is known as "natural convection" and it is a strong function of the temperature difference between the solid and the fluid. Blowing air over the solid by using external devices such as fans and pumps can also generate a fluid motion. This is known as "forced convection". Fluid mechanics plays a major role in determining convection heat transfer. For each kind of convection heat transfer, the fluid flow can be either laminar or turbulent. Laminar flow generally occurs in relatively low velocities in a smooth laminar boundary layer over smooth small objects, while turbulent flow forms when the boundary layer is shedding or breaking due to higher velocities or rough geometries.

1.6 .PROBLEM STATEMENT

A vertical open ended channel filled with a porous medium with an imposed heat flux and heat loss coefficient on one of its walls. Has been studied by using fluent software. However A fan can enhance the self driven flow then mixed convection regime is considered other wise the flow is completely natural convection. The objective is to study characteristics of the flow and heat transfer in the channel as well as the behaviour of natural convection. The potential applications include solar wall and solar chimney, used for ventilation and preheating of makeup air in building.

1.7. OBJECTIVE

THE OBJECTIVE OF THIS PROJECT IS

- ❖ To study the characteristics of the flow in vertical channel
- ❖ natural flow in vertical channel
- ❖ To know the maximum velocity and temperature for hot air in outlet of the channel.

1.8 .SCOPE OF STUDY

The nature of this project requires fundamentals of mathematics mainly ordinary differential equation (ODE) and partial differential Equation (PDE) beside the chemical engineering discipline. Heat transfer, natural convection, force convection and mixed convection. And computer literature (familiar with fluent software) Hence considerable amount of time should be spent on acquiring this knowledge from relevant sources, books, internet, journals etc. This information will become indispensable when the actual analysis is performed later.

In this research the channel shape and the operating condition (Reynolds number, Rayleigh number) will be studied carefully to know the effects on the heat transfer and energy recovery.

CHAPTER 2

2.1 LITERATURE REVIEW

2.1.1 Description of The vertical channel

The configuration of vertical open ended channel considered in this project is shown in figure 2 the collector surface is made of a flat vertical plate separated from the wall by certain distance in such a way that air can flow in this channel moreover we considered 2D channel filled with porous material of uniform and isotropic permeability K assuming that the flow is slow enough to follow Darcy law ref [7]
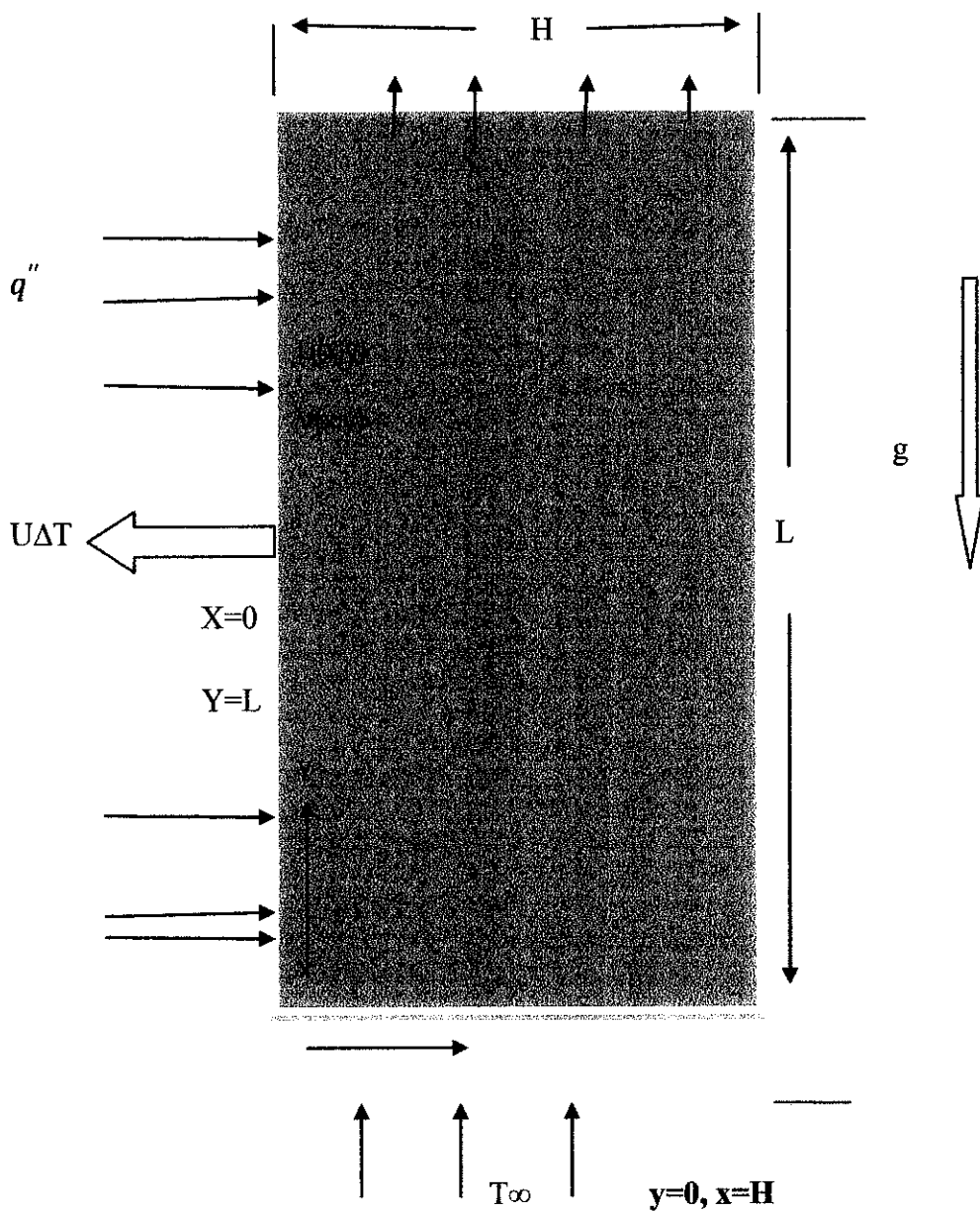

Figure 2: Schematic vertical channel under study

2.1.2 THE MATHEMATICAL MODELS :

these equations have been adapted from convection heat transfer for Bejan ,2004,3rd edition pages 587,576. And Journal ,Renewable Energy 34, 2009 ref[7]

steady state governing equations Bejan(2004.)

$$\frac{du}{dx} + \frac{dv}{dy} = 0 \tag{1}$$

$$u = -\frac{k}{\mu}\frac{dp}{dx} \quad , \quad v = -\frac{k}{\mu}\frac{dp}{dy} \tag{2}$$

$$u\frac{dT}{dx} + v\frac{dT}{dy} = \alpha\frac{d^2T}{dy^2} \tag{3}$$

The Darcy flow satisfies

$$u = -\frac{k}{\mu}\frac{dp}{dx} \quad , \quad v = -\frac{k}{\mu}\left(\frac{dp}{dy} + pg\right) \tag{4}$$

By differentiating equation 4 to eliminate P

$$\frac{du}{dy} = -\frac{k}{\mu}\frac{dp}{dx} \quad , \quad \frac{dv}{dx} = -\frac{k}{\mu}\left(\frac{dp}{dy} + g\frac{dp}{dx}\right)$$

$$\frac{du}{dy} - \frac{dv}{dx} = \frac{k}{\mu}g\frac{dP}{dx} \tag{5}$$

9

Fine stream function (**B.Sunden,**)

$$u = \frac{d\psi}{dy} \tag{6}$$

$$v = -\frac{d\psi}{dx} \tag{7}$$

Mass continuity equation satisfied identical Darcy law becomes

$$\frac{d^2\psi}{dx^2} + \frac{d^2\psi}{dy^2} = \frac{k}{\mu} g \frac{dP}{dx} \tag{8}$$

If the boundary layer of the figure 2 is thin then equation 7 becomes

$$\frac{d^2\psi}{dx^2} + \frac{d^2\psi}{dy^2} = -\frac{kg\beta}{v} \frac{dT}{dx} \qquad (streamfuntion) \tag{9}$$

To get the energy equation we try to modify first of thermodynamics simplified form

$$\sigma \frac{dT}{dt} + u \frac{dT}{dx} + v \frac{dT}{dy} + w \frac{dT}{dz} = \alpha \left(\frac{d^2T}{dx^2} + \frac{d^2T}{dy^2} + \frac{d^2T}{dz^2} \right) \tag{10}$$

Where $\alpha = \frac{k}{PfCpf}$

If considered two dimension 2D and steady state flow

$$\sigma \frac{dT}{dt} = 0 \quad \text{(Steady flow)}, w \frac{dT}{dz} = 0 \quad , \quad \frac{d^2T}{dz^2} = 0$$

Then equation 9 reduce to

$$u \frac{dT}{dx} + v \frac{dT}{dy} + \quad = \alpha \left(\frac{d^2T}{dx^2} + \frac{d^2T}{dy^2} \right) \tag{11}$$

10

Boundary Condition (BC)

First BC on the left wall x = 0 and the streamline must follow the x = 0 line

$$k\frac{dT}{dx} = q`` - U(T - T\infty), \qquad \psi = 0 \tag{12}$$

Right hand side wall is adiabatic and stream line must follow x=H line

$$\frac{dT}{dx} = 0 \quad , \psi = \psi H \tag{13}$$

At the other surface BC has been set to

$$T = T\infty \qquad ,\frac{d\psi}{dy} = 0 \qquad y = 0 \tag{14}$$

$$\frac{dT}{dy} = 0 \quad ,\frac{d^2\psi}{dy^2} = 0 \qquad y = L \tag{15}$$

The net heat transfer to the fluid is obtained by

$$Q = \int_0^L \{q`` - U(T - T\infty)\}dy \tag{16}$$

$$Ra = \frac{kg\beta L^2 q``}{\alpha k v} \tag{17}$$

$$Nu = \frac{q``}{T - T\infty}\frac{H}{Kp} \tag{18}$$

11

2.2 Materials used in simulation

The main materials used in this project as in put to simulate are Air as working fluid in vertial channel (hot air) and aluminium as porous medium.

2.2.1 Air

Air is the name given to atmosphere used in breathing and photosynthesis. Dry air contains roughly (by volume) 78.09% nitrogen, 20.95% oxygen, 0.93% argon, 0.039% carbon dioxide, and small amounts of other gases. Air also contains a variable amount of water vapor, on average around 1%. While air content and atmospheric pressure varies at different layers, air suitable for the survival of terrestrial plants and terrestrial animals is currently only known to be found in Earth's troposphere and artificial atmospheres. This Air as exists in nature how ever in this project we assume the air contents of nitrogen and oxygen with percentage of (79% N2 and 21% O2).

Chemical and Physical Properties of Air @ 15C°

Properties	Values
Formula	N2=79%,O2=21%
Molecular weight	28.8g/mol
Physical state	Gas
Density	1.225kg/m3
Specific heat (Cp)j/Kg-K	1006.43 J/Kg-K
Thermal conductivity w/m-K	0.0242 W/m-k
Viscosity Kg/m-s	1.7894E-05 kg/m-s

Table 2: chemcial and physical properties of Air

2.3 Aluminium

Aluminum is a silvery white member of the boron group of chemical elements. It has the symbol **Al**, and its atomic number is 13. It is not soluble in water under normal circumstances. Aluminum is the most in the Earth's crust, and the third most abundant element, after oxygen and silicon. It makes up about 8% by weight of the Earth's solid surface. Aluminum metal is too reactive chemically to occur in nature. Instead, it is found combined in over 270 different minerals. The chief ore of aluminum is bauxite.

Aluminum is remarkable for the metal's low density and for its ability to resist corrosion due to the phenomenon of passivation. Structural components made from aluminum and its alloys are vital to the aerospace industry and are important in other areas of transportation and structural materials. The most useful compounds of aluminum, at least on a weight basis, are the oxides and sulfates.

Despite its prevalence in the environment, aluminum salts are not known to be used by any form of life. In keeping with its pervasiveness, it is well tolerated by plants and animals. Because of its prevalence, potential biological roles, beneficial and otherwise, for aluminum compounds are of continuing interest.

Chemical and Physical properties of Aluminium

Properties	Values
Formula	Al
Molecular weight	26g/mol
Density	2719 kg/m3
Specific heat	871J/kg-k
Thermal conductivity	202.4W/m-k
Physical state	solid
@25°C/1atm Heat of vaporization Heat of fusion	294KJ/mol 10. KJ/mol
Melting point Boiling point	933.47K 2792K

Table3: chemical and physical properties of aluminum

CHAPTER 3

METHODOLOGY

Procedures identification for the project studies
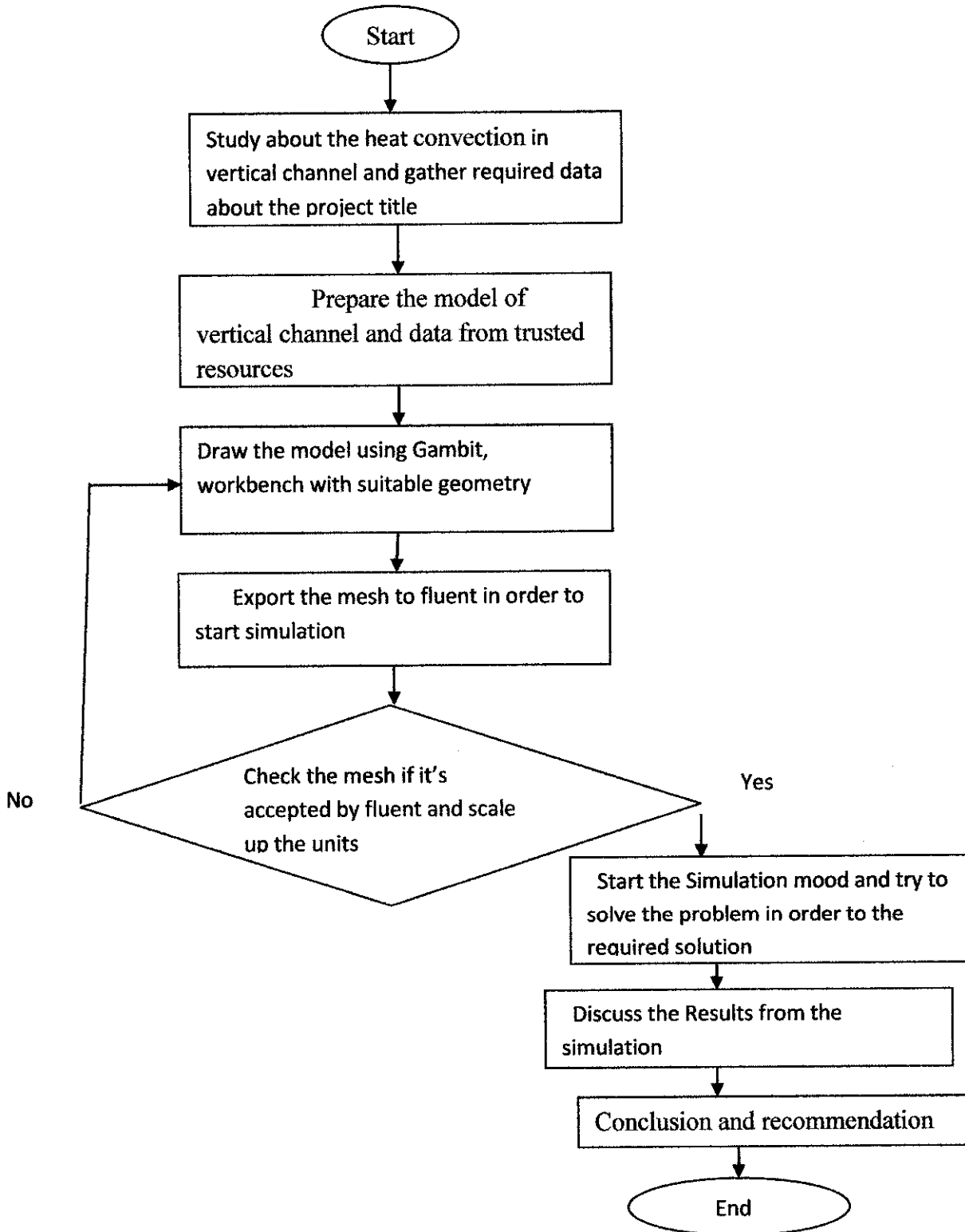

Figure3 flow chart of methodology

3.1 METHODOLOGY

This project is based on computer simulation, computational fluid dynamics (CFD) by using fluent software .therefore researcher requires to be familiar with gambit software or workbench in order to be able create the right geometry of the model and also be familiar with fluent software in order to achieve the objective of this project as mentioned in the objectives section therefore we need to collect the data or the inputs (,boundary condition, density, viscosity , temperature and velocity etc).in order to be able to simulate and get the required solution to this project

Since the project depends on the computer simulation then we need only software in this project no hardware is needed.

3.2 SOFTWARE

Name	Description
Ansys Fluent V 12	This software is the core of this project it will be used to simulate The convective heat flow in vertical porous channel
Gambit	To create the geometry of the model (vertical channel)
Work bench	Work bench is also for geometry creation and is better performance than Gambit its within fluent package
Microsoft office	For calculation , presentation ,graphs etc

Table 4: soft ware used in the project

3.3 Creation of geometry

After we have prepared the model in term of dimensions input and output we need to create this model by one of the software mentioned earlier to create the right geometry and mesh which will suite to our project, in this project we used gambit software to create the geometry based on the length of the channel and the width (L= 3m, H=0.5) for full steps refer to gambit user guide or the tutorials and below the models I created using gambit.
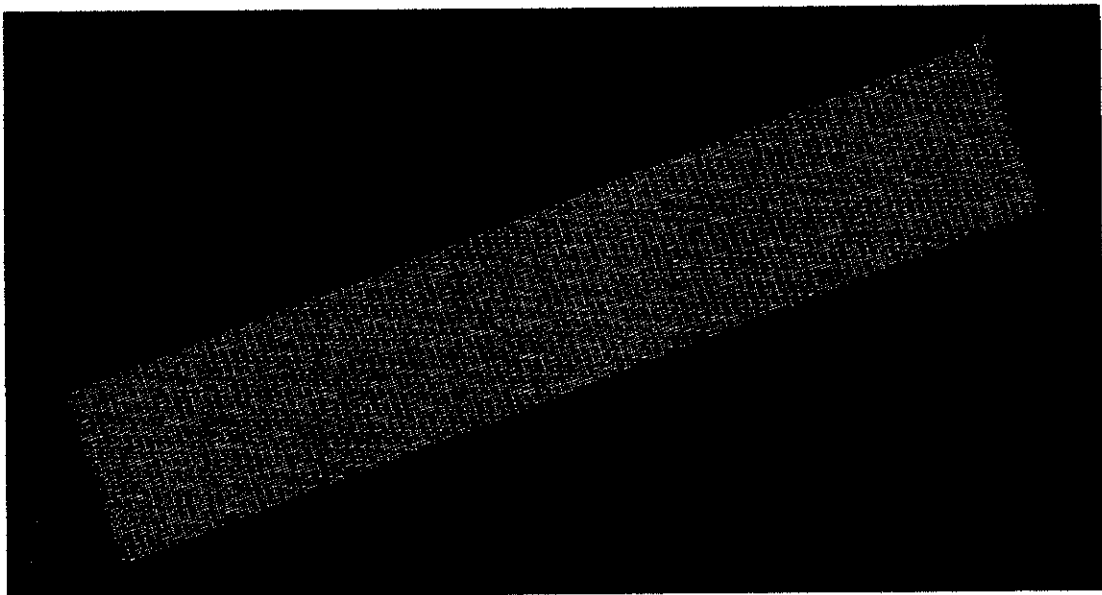

Figure 4: mesh from gambit

After the geometry has been made and meshed therefore we export the meshed model to fluent software to check the meshing is right for the simulation otherwise we recreate the model in order to suite the simulation mood.

3.4 Ansys Fluent Software

After the geometry has been made in gambit and exported to fluent therefore we import it to fluent and start to check the suitability of the mesh if its ok then continue simulation and below some introduction and capability of fluent software.

3.4.1 Introduction to Ansys fluent

Ansys flunent is a state-of-the-art computer program for modeling fluid flow, heat Transfer and chemical reactions in complex geometries (plates, cylinder, channels and ,etc).

Ansys fluent is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible. In addition, Ansys fluent uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstations and powerful computer servers. This architecture

Allows for efficient execution, interactive control, and complete flexibility between Different types of machines or operating systems.

Ansys fluent provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge/polyhedral, and mixed (hybrid) meshes.

Ansys fluent also allows you to refine or coarsen your mesh based on the flow solution.

After a mesh has been read into Ansys fluent, all remaining operations are performed within Ansys fluent. These include setting boundary conditions, defining fluid properties, executing the solution, refining the mesh, and postprocessing and viewing the results.

The Ansys fluent serial solver manages file input and output, data storage, and flow field calculations using a single solver process on a single computer. Ansys fluent also uses a utility called cortex that manages Ansys fluent's user interface and basic graphical functions. Ansys fluent's parallel solver allows you to compute a solution using multiple processes that may be executing on the same computer, or on different computers in a network. User's guide (2009).

Parallel processing in Ansys fluent involves an interaction between Ansys fluent, a host process, and a set of compute-node processes. Ansys fluent interacts with the host process and the collection of compute nodes using the cortex user interface utility.

Flowing figures illustrate the serial and parallel Ansys Fluent Architectures
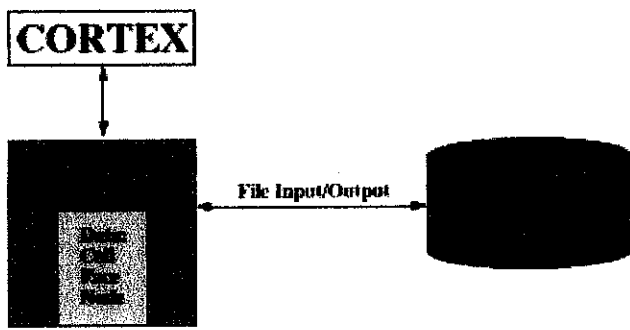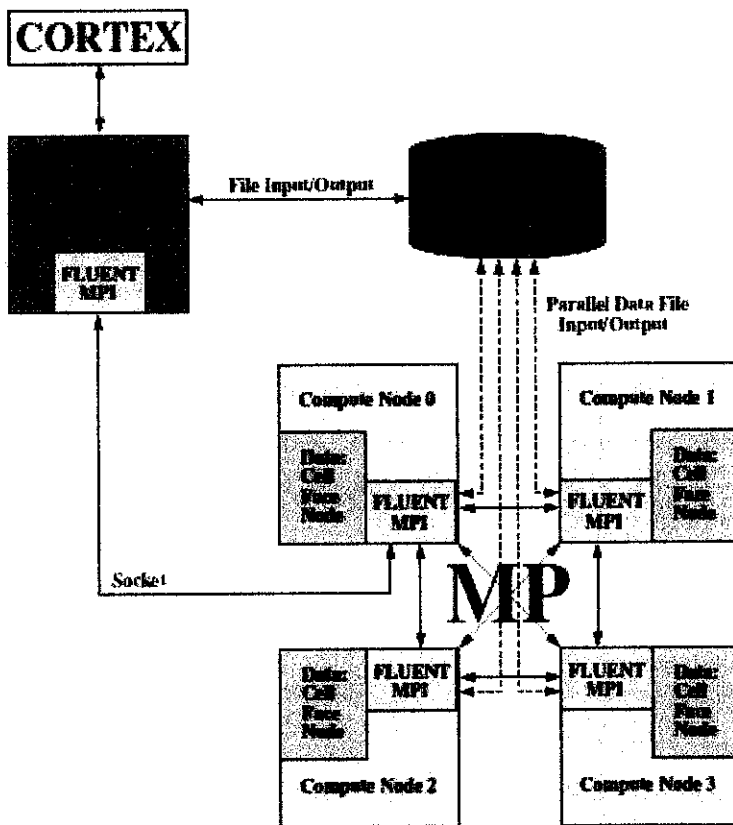

Serial Ansysfleuent Architecture

Figure 5: Parallel Ansys Fluent Architecture

19

3.4.2 Ansys Fluent Capabilities

The Ansys fluent solver has the following modeling capabilities:

- ❖ 2D planar, 2D axisymmetric, 2D axisymmetric with swirl (rotationally symmetric), and 3D flows.

- ❖ Quadrilateral, triangular, hexahedral (brick), tetrahedral, prism (wedge), pyramid, polyhedral, and mixed element meshes

- ❖ Steady-state or transient flows

- ❖ Incompressible or compressible flows, including all speed regimes(low subsonic,transonic, supersonic, and hypersonic flows)

- ❖ In viscid, laminar, and turbulent flows

- ❖ Newtonian or non-Newtonian flows

- ❖ Ideal or real gases

- ❖ Heat transfer, including forced, natural, and mixed convection, conjugate (solid/fluid) heat transfer, and radiation

- ❖ Chemical species mixing and reaction, including homogeneous and heterogeneous combustion models and surface deposition/reaction models

- ❖ Free surface and multiphase models for gas-liquid, gas-solid, and liquid-solid flows

- ❖ Porous media with non-isotropic permeability, inertial resistance, solid heat conduction, and porous-face pressure jump conditions

- ❖ Population balance module

- ❖ Mixing-plane model for modeling rotor-stator interactions, torque converters, and similar turbo machinery applications with options for mass conservation and swirl conservation

3.5 CFD Analysis using Ansys Fluent

Steps in solving CFD problems

❖ Define the modeling goals.

❖ Create the model geometry and mesh.

❖ Set up the solver and physical models.

❖ Compute and monitor the solution.

❖ Examine and save the results.

❖ Consider revisions to the numerical or physical model parameters, if necessary. User's guide (2009)

3.6. Set up and solution in Ansys fluent

In solving any problems using Ansys fluent there are basic steps need to be done as below some figure explain these steps
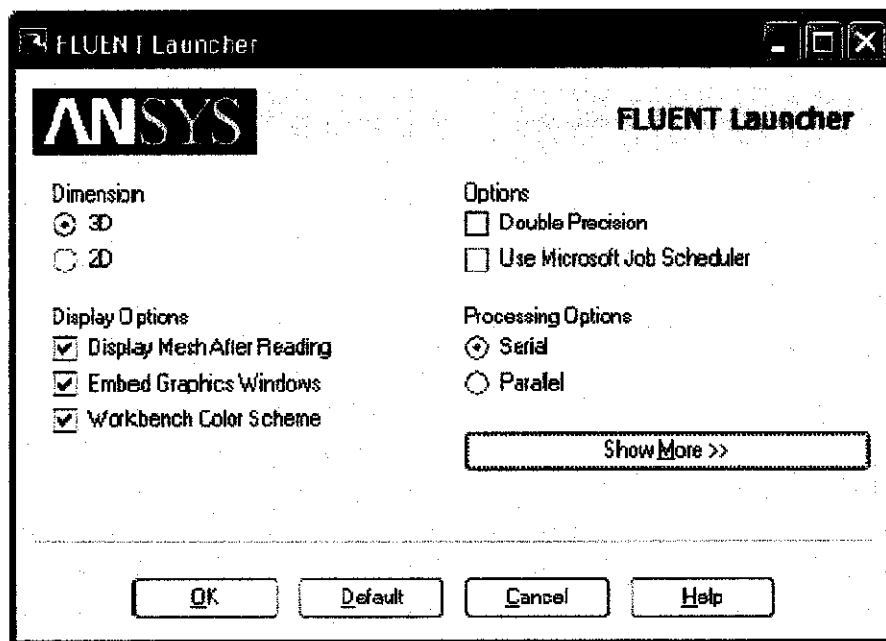

Figure 6: fluent launcher

CHAPTER 4

## 4.0 Results and discussion

4.1 The operating conditions

The data in the below table is required in the simulation process in order to the results

Boundary condition	Inlet velocity	Inlet temperature	Geometry
T_∞ =288.15K(fluid temperature) T_w = 300K(wall temperature) q' =50w/m2(heat flux)	V =0.0002m/s	T_∞ =288.15K	H= 0.5m L=3m

Table 5 operating condition

The ultimate results we are trying to obtain in this project that we need to observe the characteristics of the hot air in inside the vertical channel therefore we have to get the temperature contour, velocity contour and the velocity profile (V verse Position) and below the discussion of the above mention.

4.2 Residuals
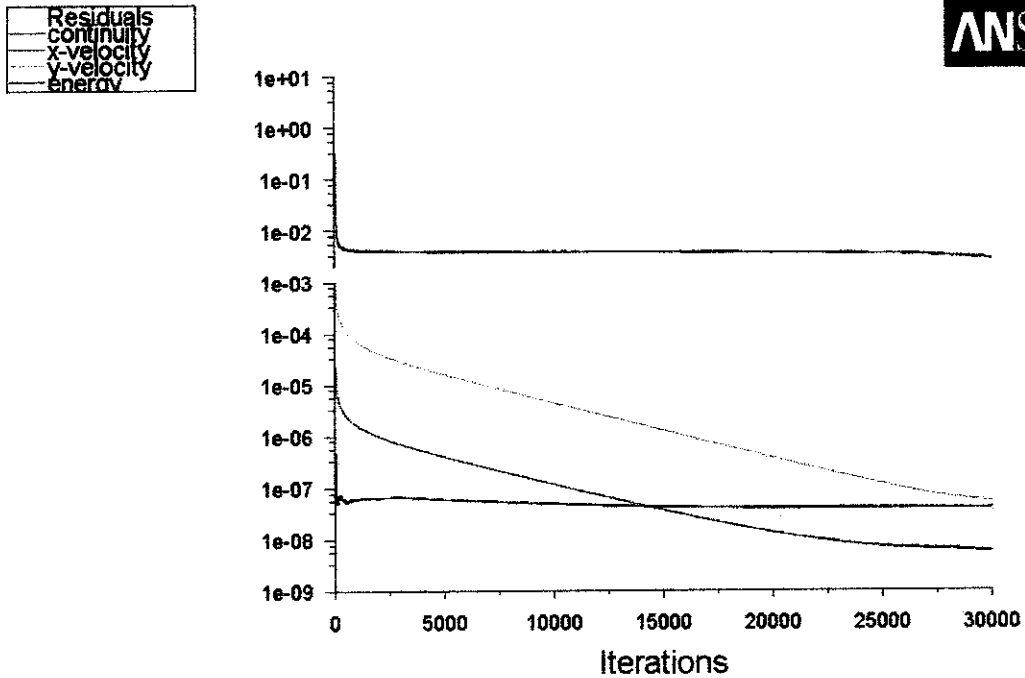
Figure 7: residuals

After set up the problem in fluent you need to get the result as accurate as much therefore we need to set the solution monitor into certain set points for energy, continuity, X-velocity etc with this the solution will converge at the set point which indicated consequently we get the temperature contour ,velocity contour and velocity profile

23

4.3 Temperature contour

After running the simulation for certain amount of time we obtained the flowing results.

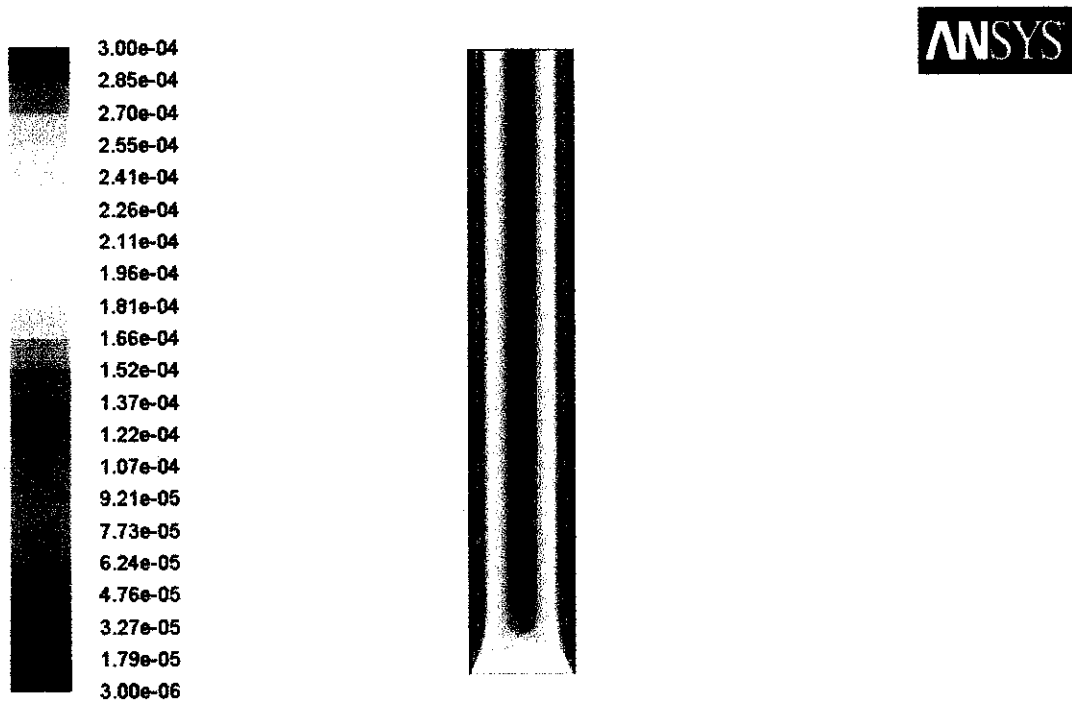| 1.27e+03 |
| 1.22e+03 |
| 1.17e+03 |
| 1.12e+03 |
| 1.07e+03 |
| 1.02e+03 |
| 9.75e+02 |
| 9.26e+02 |
| 8.77e+02 |
| 8.28e+02 |
| 7.79e+02 |
| 7.30e+02 |
| 6.81e+02 |
| 6.32e+02 |
| 5.83e+02 |
| 5.34e+02 |
| 4.85e+02 |
| 4.36e+02 |
| 3.87e+02 |
| 3.38e+02 |
| 2.89e+02 |
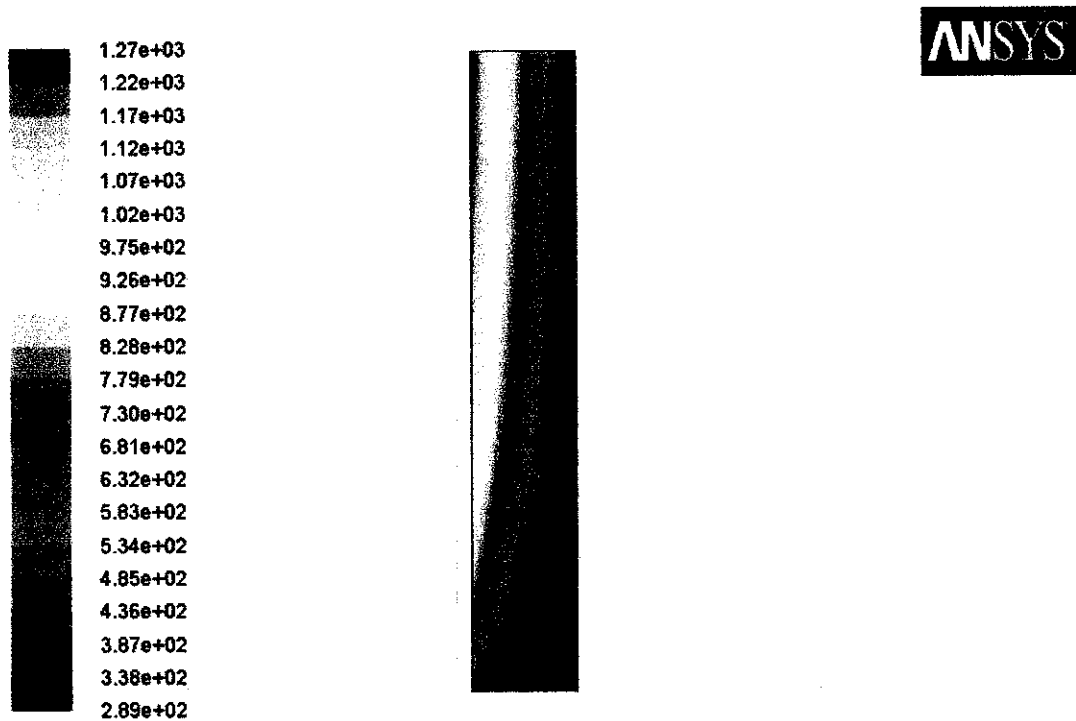
Contours of Static Temperature (k)

Jul 04, 2011
ANSYS FLUENT 12.0 (2d, pbns, lam)

Figure 8 : temperature contour

From this result the temperature contour indicate that the maximum temperature of the hot air inside the channel is about 1270 K however this considered as maximum and we can control the temperature in order to get the require heating to our requirement. More ever the channel we studying here between two walls (refer to model figure2) The right hand wall is adiabatic (heat = 0 or constant T) and we observe that the blue color along the wall from inlet to out which indicate the adiabatic condition the other   left hand side is heat wall with heat flux of 50 W/m2 and from the above result the affects of heat flux is very clear and indicated in red color, therefore we conclude that this result is satisfied since it shows the adiabatic on the right wall and increasing in temperature on the left wall.

24

4.4 Velocity Contour

|	
3.00e-04	
2.85e-04	
2.70e-04	
2.55e-04	
2.41e-04	
2.26e-04	
2.11e-04	
1.96e-04	
1.81e-04	
1.66e-04	
1.52e-04	
1.37e-04	
1.22e-04	
1.07e-04	
9.21e-05	
7.73e-05	
6.24e-05	
4.76e-05	
3.27e-05	
1.79e-05	
3.00e-06	

Contours of Velocity Magnitude (m/s)

Jul 04, 2011
ANSYS FLUENT 12.0 (2d, pbns, lam)

Figure 9: velocity contour

As mentioned before the simulation has been for certain time and the velocity contour has been obtained and from the above results it showed that the fluid velocity is very slow as indicated in blue color in both sides (right hand side and left hand side) how the fluid is very fast at the centre of the channel as indicated in red color which gives the maximum velocity (0.0003m/s) more ever this velocity (maximum velocity) is still considered slow as mentioned in ref[6] assumed that Darcy flow is valid when the flow is not too fast and we can conclude that this result is very much convenience and can be validate by this assumption in ref[6] and it would be clear when we discuss the velocity magnitude verse position in figure10.

4.5 The velocity magnitude verse position (width H)
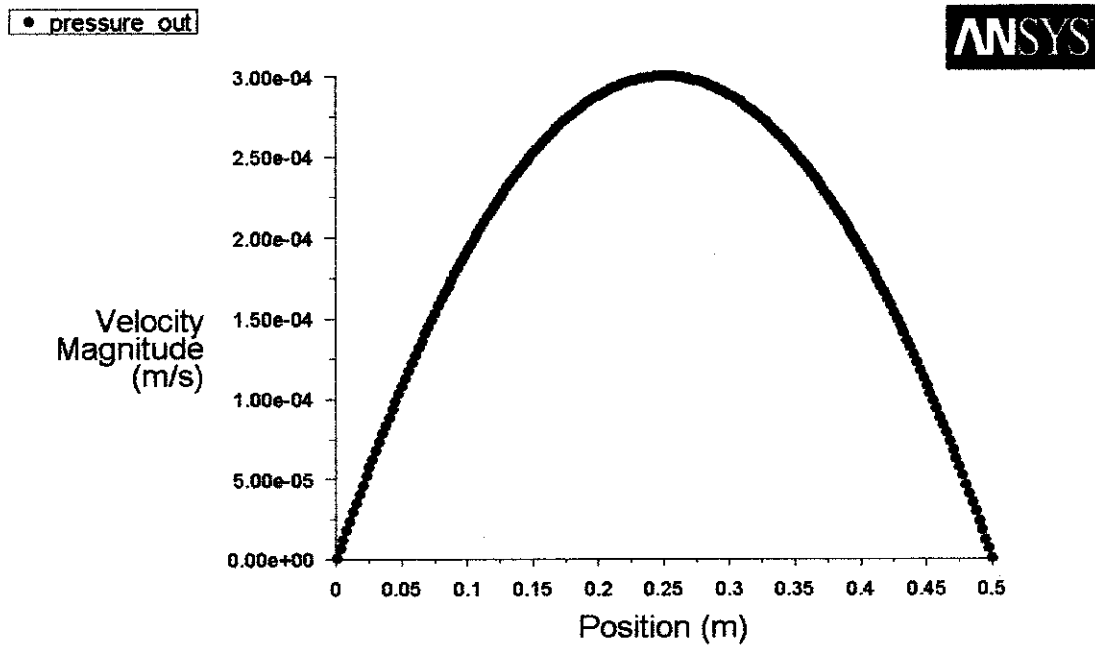
Figure 10 velocity profile

In this figure after we have the result for the velocity contour we have plotted the velocity magnitude verse the distance between the two walls and it indicated that the velocity at near to the wall is very small and it increases as the fluid gets far from the wall and it reaches the maximum value at the centre of the channel therefore if the width of the channel is (H= 0.5m) then the maximum velocity is at H= 0.25m but since it's natural convection the maximum velocity should be near the heated wall and this explained in figure 11 the maximum velocity is along the heated wall.

This result showed the identical result between the velocity contour and velocity profile which both showed the maximum velocity is 0.0003m/s. and we can conclude that solution is satisfactory in term of velocity magnitude ref [6].
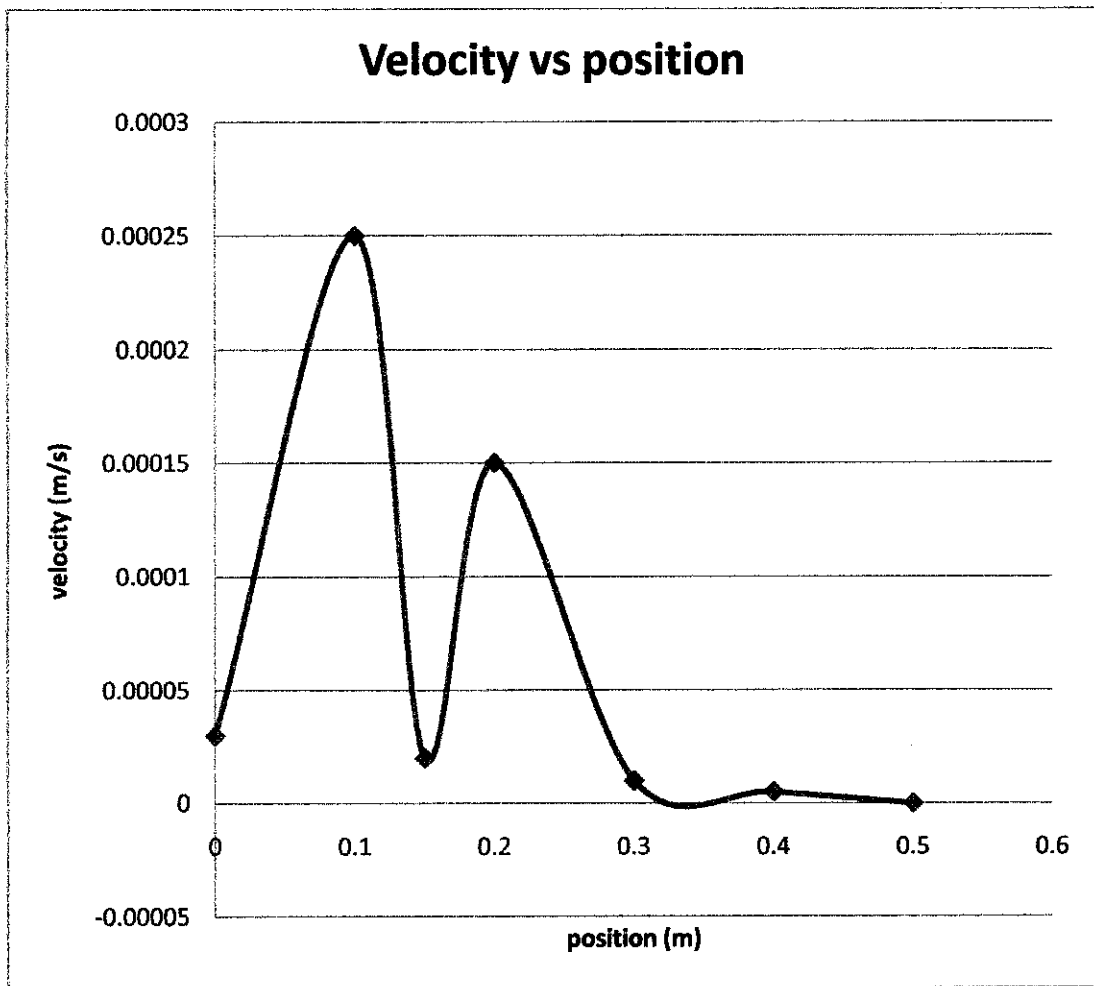
Velocity verse position

Figure 11 velocity verse position

The above figure showed the maximum velocity is near to left hand side wall which is the heated wall and since this results based on the natural convection therefore the maximum velocity in natural convection is on heated wall side ref [2] which is left hand side wall and the minimum velocity on the right hand side wall and from the figure11 it's indicated clearly the maximum velocity is on the left heated wall.
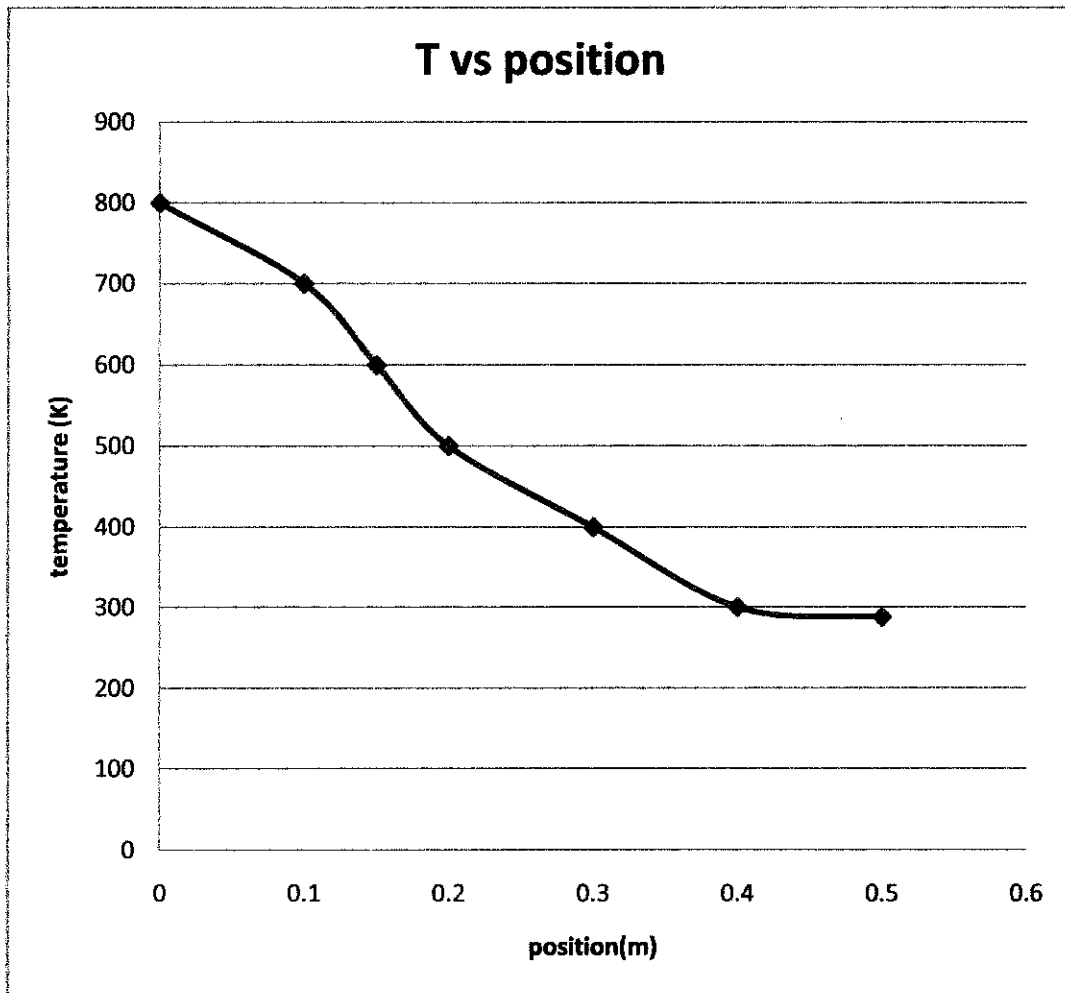
Temperature verse position

Figure 12 temperature verse position

After we have run the simulation and obtained the result then plotted the temperature verse the position we found that the temperature decreases as the fluid is getting far from the heated wall (left hand side wall) and increase as the fluid near the heated wall therefore the maximum temperature is on the left hand side wall as it's been indicated on the figure12 and it mentioned before in literature review the right wall is adiabatic (temperature is constant) ref [7] or the heat is equal zero and it's very clear from the result above the temperature near to right is constant and that indicates the adiabatic condition hence the result is acceptable.

CHAPTER 5

5.0 CONCLUSION AND RECOMMENDATIONS

5.1 CONCLUSION

We have described in this work/ project the behavior of a vertical open ended channel filled with porous medium such as aluminum as porous medium and we have studied the natural convection behavior in a vertical channel and since the channel is being heated in the left hand side wall with constant heat flux and adiabatic in the right hand side wall therefore the results showed that on left wall the temperature is increasing along the channel while in the other wall the temperature is constant($T_1 = T_2$) see temperature contour which is led us that its adiabatic from equation of heat $(Q = C_p A \Delta T)$.

Furthermore the maximum temperature in the channel we can manipulate it by increasing or decreasing the heat flux in order to get the desire temperature (the appendices 1 and 3) for temperature contour for different heat flux.

For the velocity according to the Darcy flow and local thermal equilibrium we have made assumptions that Darcy flow is valid when the flow is not too fast and from the velocity result is showed that the velocity is very small it's about (0.0003m/s) and this indicate that the result we obtained from the simulation is satisfactory and further more in natural convection the velocity near the heated wall is higher and from the figure11 of velocity verse position(width of channel) it showed the velocity near the hot wall is higher.

However the potential application of vertical the channel include solar chimney and solar wall used for ventilation and preheating of Air In the buildings further more for natural convection it's very common to used in computer chips as cooling system as well as in mine industries.

6. REFERENCES

1) Bejan(2004.) A, convection heat transfer 3rdedwiley .

2) Binay K.Dutta (2006) heat transfer principle and applications, university of Calcutta

3) Donald A Nield (2006) Convection In porous Media third edition, University of Auckland, New Zealand.

4) Eduardo Cao (2010)Heat transfer in process engineering , chemical engineering department of Buenos Aires University

5) Pop and Ingham (1998) convective heat transfer. Mathematical and computational modeling of viscous fluid and porous media.

6) Andreozzi AManaca. Thermal and fluid dynamics behavior of symmetrically heated vertical channel with auxiliary plate ,international journal of heat and fluid dynamics fluid ,2001

7) Yanik. Boutin et al (2009) , Journal .Optimal mixed convection for maximal energy recovery with vertical porous channel (solar wall)

8) B.Sunden, computational analysis of convection heat transfer, University of Udine, Italy.

9) R.Byro Bird et al(2007) Transport Phenomena second edition.

10) W.M.kays , Convective Heat Transfer and Mass Transfer ,4th edition.

11) T. Hayat et al (2008) Journal, analytical solution for MHD flow for a third order fluid in a porous channel.

12) ANSYS FLUENT 12 User's guide (2009)

APPENDICEs

Appendix 1
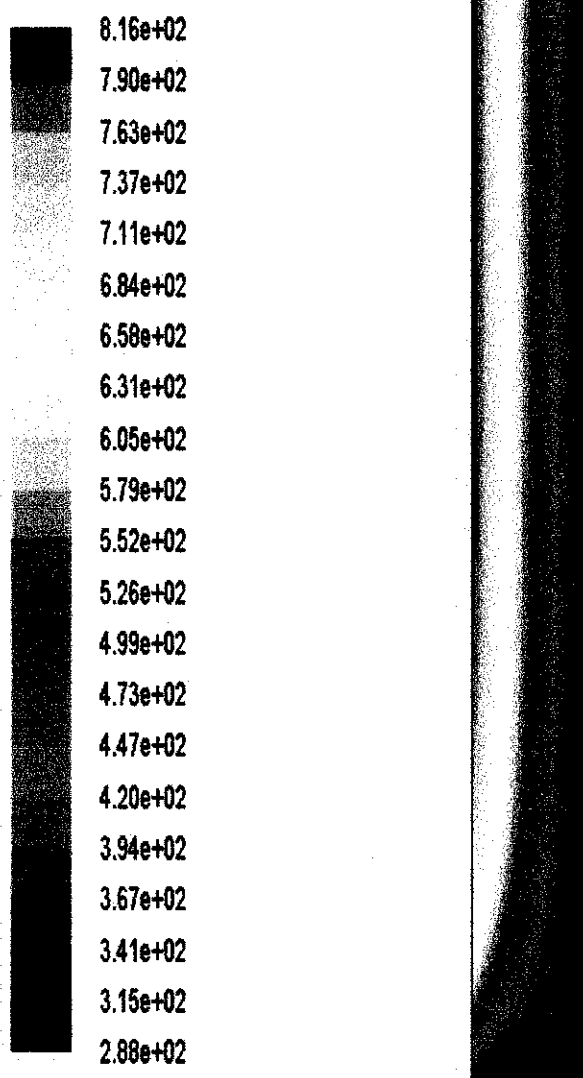
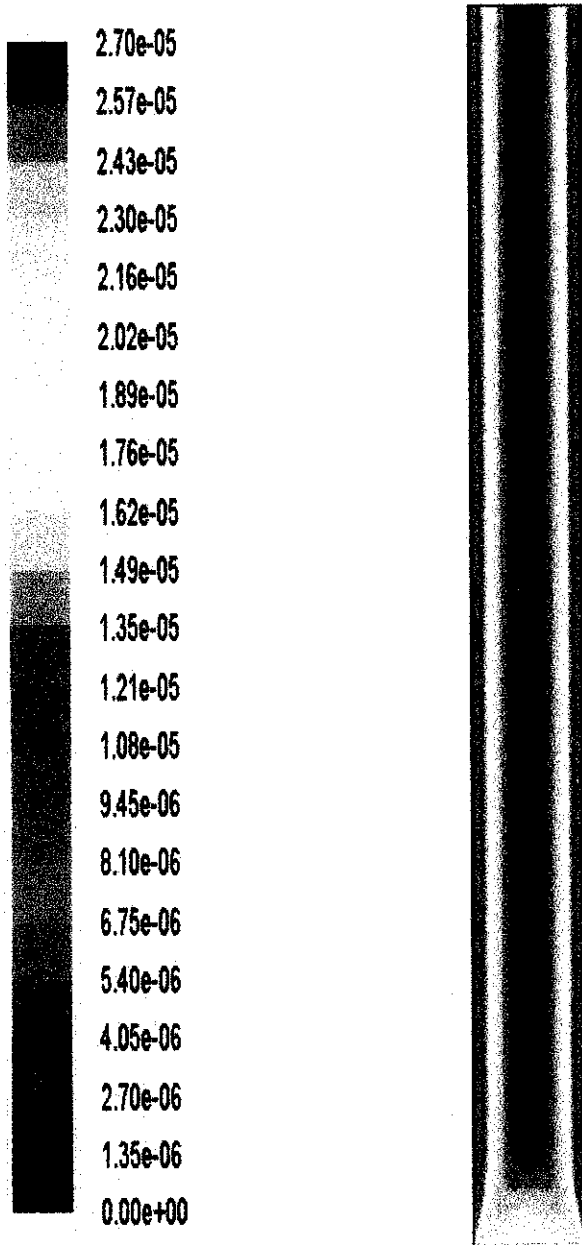Contours of Static Temperature (k)   Jul 31, 2011
                                                          ANSYS FLUENT 12.0 (2d, pbns, lam)
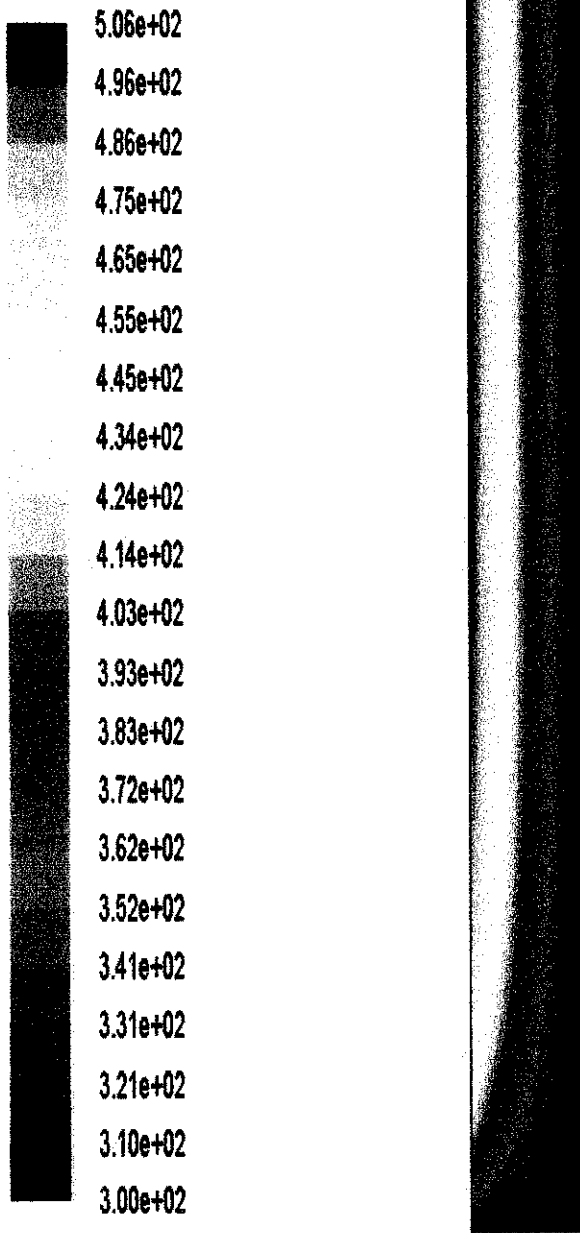
Temperature contour with q'' =25W/m2

2.70e-05	
2.57e-05	
2.43e-05	
2.30e-05	
2.16e-05	
2.02e-05	
1.89e-05	
1.76e-05	
1.62e-05	
1.49e-05	
1.35e-05	
1.21e-05	
1.08e-05	
9.45e-06	
8.10e-06	
6.75e-06	
5.40e-06	
4.05e-06	
2.70e-06	
1.35e-06	
0.00e+00	

Contours of Velocity Magnitude (m/s)

Jul 23, 2011
ANSYS FLUENT 12.0 (2d, pbns, lam)

Velocity contour with q'' =25W/m2

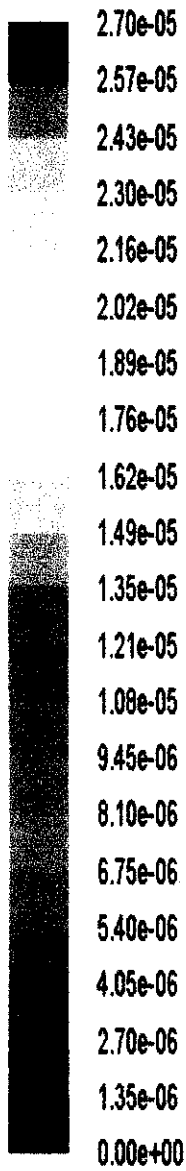|	
5.06e+02	
4.96e+02	
4.86e+02	
4.75e+02	
4.65e+02	
4.55e+02	
4.45e+02	
4.34e+02	
4.24e+02	
4.14e+02	
4.03e+02	
3.93e+02	
3.83e+02	
3.72e+02	
3.62e+02	
3.52e+02	
3.41e+02	
3.31e+02	
3.21e+02	
3.10e+02	
3.00e+02	

Contours of Static Temperature (k)

Aug 13, 2011
ANSYS FLUENT 12.0 (2d, pbns, lam)

Temperature contour with $q^{''}$ =10W/m2

Appendix 4

2.70e-05
2.57e-05
2.43e-05
2.30e-05
2.16e-05
2.02e-05
1.89e-05
1.76e-05
1.62e-05
1.49e-05
1.35e-05
1.21e-05
1.08e-05
9.45e-06
8.10e-06
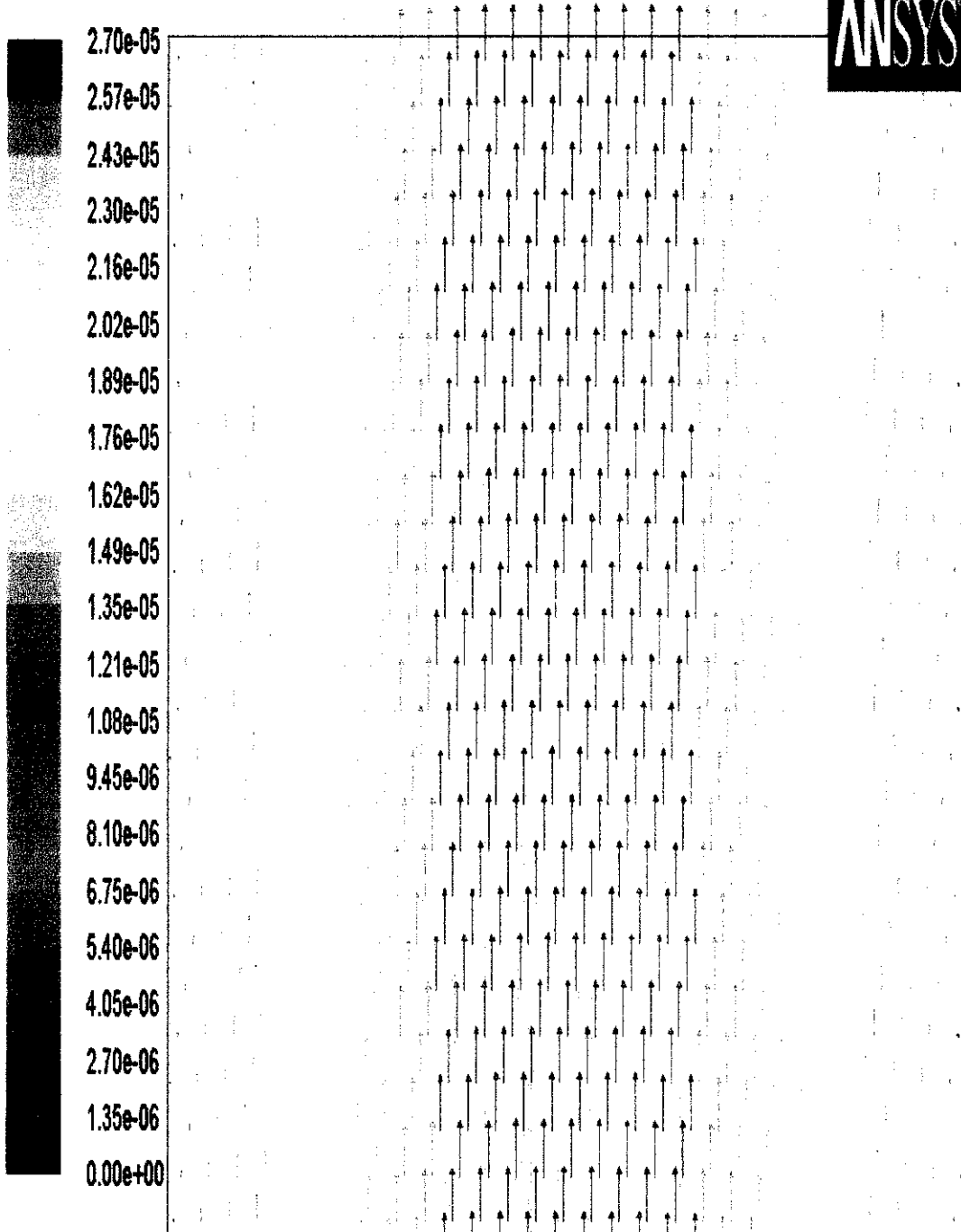6.75e-06
5.40e-06
4.05e-06
2.70e-06
1.35e-06
0.00e+00

Contours of Velocity Magnitude (m/s)

Aug 13, 2011
ANSYS FLUENT 12.0 (2d, pbns, lam)

Velocity contour with $q^{''}$ =10w/m2

Appendix 5

Velocity Vectors Colored By Velocity Magnitude (m/s)

Aug 13, 2011
ANSYS FLUENT 12.0 (2d, pbns, lam)

Velocity vectors with $q'' = 10W/m2$