**SUPERVISOR'S DECLARATION**

I hereby admit that have read this report and from my point of view this report is enough in term of scope and quality for purpose for awarding Bachelor of Degree in Mechanical Engineering (Thermal-Fluid)

Signature : …...............................

Supervisor's name : Mohd Irwan bin Mohd Azmi

Date : …...............................

NATURAL CONVECTIVE HEAT TRANSFER IN SQUARE ENCLOSURES HEATED FROM BELOW

AHMAD SYAUQI BIN AHMAD SHUKRI

This report is presented to fulfill the requirement to be awarded with

Bachelor of Degree in Mechanical Engineering (Thermal-Fluid)

Faculty of Mechanical Engineering

MAY 2012

**DECLARATION**

"I admit this report has been written by me myself except for some quotation that has been noted well for each of them"

Signature : …..............................

Student's name : Ahmad Syauqi bin Ahmad Shukri

Date : …..............................

## DEDICATION

This report is dedicated to my beloved parents

Ahmad Shukri bin Ghazali and Noor Pida binti Raimi.

## ACKNOWLEDGEMENT

Thanks to Allah swt upon the completion of this report. Not forgetting my parents who always support me no matter what obstacles came through. Also special thanks to my supervisor, Mr. Irwan in guiding me along this report writing and project computation.

Last but not least, all friends especially my classmates, Banun, Nuratiqah and Fahdli who had taught me how to use ANSYS-Fluent software and not forgetting Mr. Fudhail as my Computational Fluid Dynamic (CFD)'s lecturer and my team who never give up in supporting me toward the completion of this report, Ali Ilman, Zamir, Ho Bee Kim and Siti Khadijah. Really appreciate all the helps and supports.

# ABSTRACT

Natural convective heat transfer is common nowadays. This study focuses on square enclosure cavity where the boundaries have been fixed to certain condition. The objectives are; i) to solve natural convection problem numerically by predicting the streamlines and isothermlines; ii) to compare the numerical and existing experimental result for natural convection and; iii) to predict natural convection phenomena at multiple adiabatic walls. The square enclosure has cold wall temperature on the left and right side and adiabatic wall on the top of it. From below, heat was supplied. All these boundary conditions have been set up in ANSYS-Fluent in 'meshing' and 'setup' section. All other steps have been well explained from a flowchart provided. Results were obtained in terms of images and Rayleigh Numbers. Isothermlines and streamlines are different with variation of heat source length, $\varepsilon$. Similarity between experimental and numerical result were almost 80%. Next in the simulation, it has been proved that multiple adiabatic walls generate more heat without any changes in pressure of container. All three objectives achieved well and validated by supervisor and panels.

# *ABSTRAK*

Pemindahan haba perolakan secara semulajadi telah menjadi lumrah di dalam kehidupan. Kajian ini tertumpu kepada ruang segi empat sama di mana keadaan sempadannya telah ditetapkan. Objektif kajian adalah seperti berikut; i) untuk menyelesaikan masalah pemindahan haba secara semulajadi dengan melihat perubahan bentuk aliran haba; ii) untuk membezakan keputusan kaedah berangka dengan keputusan eksperimen yang sedia ada dan akhir sekali; iii) untuk menganggar fenomena pemindahan haba pada keadaan diniding yang tidak membebaskan haba. Ruang segi empat tersebut mempunyai dinding yang bersuhu rendah pada keadaan menegak, dinding atas yang tidak memindahkan haba dan punca haba dari dinding bawah. Keadaan ini telah ditetapkan pada peringkat 'meshing' dan 'setup' di dalam program ANSYS-Fluent. Lain-lain langkah kaedah berangka telah dijelaskan dalam carta alir yang disediakan. Keputusan diperoleh dalam bentuk imej dan Rayleigh Number. Perubahan bentuk aliran haba berbeza selari dengan perbezaan panjang pemanas. Keputusan eksperimen dan kaedah berangka diperoleh hampir sama 80%. Juga telah dibuktikan bahawa ruang segi empat yang berdinding tidak me,mbebaskan haba manghasilakan haba yang lebih tanpa sebarang perubahan pada tekanan ruang dalaman. Kesemua tiga objectif telah dicapai dan disahkan oleh tuan penyelia dan para panel penilai.

# TABLE OF CONTENT

**TITLE** **PAGE**

| TITLE | PAGE |
|---|---|

# LIST OF TABLES

# LIST OF FIGURES

**NO** **TITLE** **PAGE**

# LIST OF SYMBOLS

| | | |
|---|---|---|
| $a$ | = | Acceleration, ($m/s^2$) |
| $\phi$ | = | Angle from wall to wall |
| $T_C$ | = | Cold Temperature, ($°C$) |
| $\rho$ | = | Density, ($kg/m^3$) |
| $F$ | = | Force, ($N$) |
| $H_s$ | = | Heat source length, ($m$) |
| $\varepsilon$ | = | Heat source length, ($\varepsilon = 1/H$) (dimensionless) |
| $T_H$ | = | Hot Temperature, ($°C$) |
| $\upsilon$ | = | Kinematic viscosity, ($m^2/s$) |
| $m$ | = | Mass, ($kg$) |
| $g$ | = | Modulus of the gravity vector, ($g = 9.807ms^{-2}$) |
| Pr | = | Prandtl Number of Fluid |
| $p$ | = | Pressure, ($N/m^2$) |
| $Ra$ | = | Rayleigh Number |
| $H$ | = | Square cavity side, ($m$) |
| $\Delta T$ | = | Temperature difference, ($K$) |
| $c_p$ | = | Thermal Capacity, ($J/kg.K$) |
| $k$ | = | Thermal Conductivity, ($W/m.K$) |
| $\alpha$ | = | Thermal diffusivity, ($m^2/s$) |
| $\beta$ | = | Thermal expansion coefficient, ($1/K$) |
| $u$ | = | Velocity in x-direction, ($m/s$) |
| $v$ | = | Velocity in y-direction, ($m/s$) |

# CHAPTER I

# INTRODUCTION

## 1.1 Background

Convective heat transfer or also known as heat convection is a process involve when energy is transferred from a surface to fluid that flow over it because of the temperature gradient existence between surface and fluid. It does not matter whether the fluid is liquid or vapor there still will be energy transfer but with concern of rate of heat transfer that differ from one to another surface and fluid (Qi, 2007).

Convective heat transfer applied in many things around us and one very simple example is cooking. While cooking, the heat from stove is convectively transferred to pot and again from pot to food. Some more examples as such industrial involved metal cutting, cooling system of a building, cooling system in Central Processing Unit (CPU) and notebooks and last but not least, typical human habit that is to be near a fast blower after doing active activity to comfort their body. It looks simple but yet there are heat convection process involve in the routine.

### 1.1.1 Natural Convection

Natural convective flow can be either laminar or turbulent same like all other viscous flow. Due to low velocity in natural convective flows, laminar natural convection occurs more frequently compared to laminar forced convective flows. Increment in natural convective flows is due to density changes in the presence of the gravitational forced field (Yu, et al. 2011).

The term natural convection is being applied to flows resulting due to the gravitational force field and the term free convection being applied to flows arising due to the presence of any force field. Both are use to describe any flow changing due to temperature-induced density changes in a force field (Shang, 2006).

## 1.2 Numerical Method

The governing equations of fluid dynamics and heat transfer form the basis of numerical methods in fluid flow and heat transfer problems. The Navier Stokes equation coupled with energy equation only partially address the complexity of most fluids of interest in engineering applications. Thus, most problems that are tougher than we thought almost cannot be solved. The most reliable information pertaining to a physical process of fluid dynamics is usually given by an actual experiment using full scale equipments. However, in most cases, such experiment would be very costly and often impossible to be done (Azwadi, 2007).

With the recent computer technology, the Navier-stokes equation is able to be solved numerically. In order to simulate fluid flows on a computer, continuity equation, Navier-Stokes equation coupled with energy equation need to be solved with acceptable accuracy. Researchers and engineers need to discretize the problem by using a specific method before they can solve the problem. The numerical simulation begins with creating a computational grid. Grid is the arrangement of these discrete points throughout the flow field (Anderson, 1995). Depending on the method used for the numerical calculation, the flow variables are either calculated at the node points of the grid or at some intermediate points.

Common numerical methods are finite difference, finite volume and finite element method. In recent years, researchers have also developed other methods apart from the three aforementioned methods. However in this project, the numerical method that will be use is Taylor Series Expansion.

### 1.2.1 Taylor Series Expansion

In mathematics, a Taylor series is a representation of a function as an infinite sum of terms that are calculated from the values of the function's derivatives at a single point.

It is common practice to approximate a function by using a finite number of terms of its Taylor series. Taylor's theorem gives quantitative estimates on the error in this approximation. Any finite number of initial terms of the Taylor series of a function is called a Taylor polynomial. The Taylor series of a function is the limit of that function's Taylor polynomials, provided that the limit exists. A function may not be equal to its Taylor series, even if its Taylor series converges at every point. A function that is equal to its Taylor series in an open interval (or a disc in the complex plane) is known as an analytic function.

A one-dimensional Taylor series is an expansion of a real function $f(x)$ about a point $x = a$ is given by

$$f(x) = f(a) + f^{'}(a)(x-a) + \frac{f^{''}(a)}{2!}(x-a)^2 + \frac{f^{'''}(a)}{3!}(x-a)^3 + ... + \frac{f^{n}(a)}{n!} \quad (1.0)$$

If $a = 0$, the expansion is known as a Maclaurin series.

Taylor's theorem (actually discovered first by Gregory) states that any function satisfying certain conditions can be expressed as a Taylor series.

The Taylor (or more general) series of a function $f(x)$ about a point $a$ up to order $n$ may be found using Series $[f\{x,a,n\}]$. The $n^{th}$ term of a Taylor series of a function $f$ can be computed in Mathematica using Series Coefficient $[f\{x,a,n\}]$ and is given by the inverse Z-transform.

### 1.2.2 Computational Fluid Dynamics (CFD)

The physical aspects of any fluid flow are governed by the following three fundamental principles:

1. Mass is conserved
2. $F = ma$ (Newton's second law)
3. Energy is conserved.

These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations. Computational fluid dynamics is, in part, the art of replacing the governing partial differential equations of fluid flow with numbers, and advancing these numbers in space and time to obtain a final numerical description of the complete flow field of interest (Anderson, et al. 2009).

This is not an all-inclusive definition of CFD; there are some problems which allow the immediate solution of the flow field without advancing in time or space, and there are some applications which involve integral equations rather than partial differential equations. In any event, all such problems involve the manipulation of, and the solution for, *numbers*. The end product of CFD is indeed a collection of numbers, in contrast to a closed-form analytical solution.

However, in the long run the objective of most engineering analyses, closed form or otherwise, is a quantitative description of the problem, i.e. numbers. Of course, the instrument which has allowed the practical growth of CFD is the high-speed digital computer. CFD solutions generally require the repetitive manipulation of thousands, or even millions, of numbers a task that is humanly impossible without the aid of a computer. Therefore, advances in CFD, and its application to problems of more and more detail and sophistication, are intimately related to advances in computer hardware, particularly in regard to storage and execution speed. This is why the strongest force driving the development of new supercomputers is coming from the CFD community.

## 1.3 Problem Statement

Natural convection is initiated by temperature difference which affects the density and relative buoyancy of a fluid. This phenomenon occurs without the assistances of any external force. Natural convection in a square enclosure problem is often used as a benchmark for numerical solutions. It is important to access the capability of a numerical progress to produce results. In this study, the capability of numerical method available shall be assessed with localized heating from the bottom wall. The validated model will then be used to predict results at different condition which is multiple adiabatic walls of square enclosure.

## 1.4 Objectives

Previously in 2004, an experiment has been compute by researcher from Italy about this problem in a way to obtain result and necessary observation for the situation. Now, along with advance in technology especially in terms of engineering software, numerical method can be use to run the experiment. This has influence me to do this final year project with three main objectives and they are:

1. To solve natural convection problem numerically by predicting the streamlines and isothermlines.
2. To compare the numerical and existing experimental result for natural convection.
3. To predict natural convection phenomena at multiple adiabatic walls.

## 1.5 Scope

Project scope is only subjected to laminar air flow in a 2D square enclosure. This assumption is somehow will help a lot in comparing experimental and numerical result of the project. Also in the numerical analysis, steady state condition will be taken in consideration. Out of all this natural convection heat transfer in square enclosure will be focused on condition where heat is supply from below.

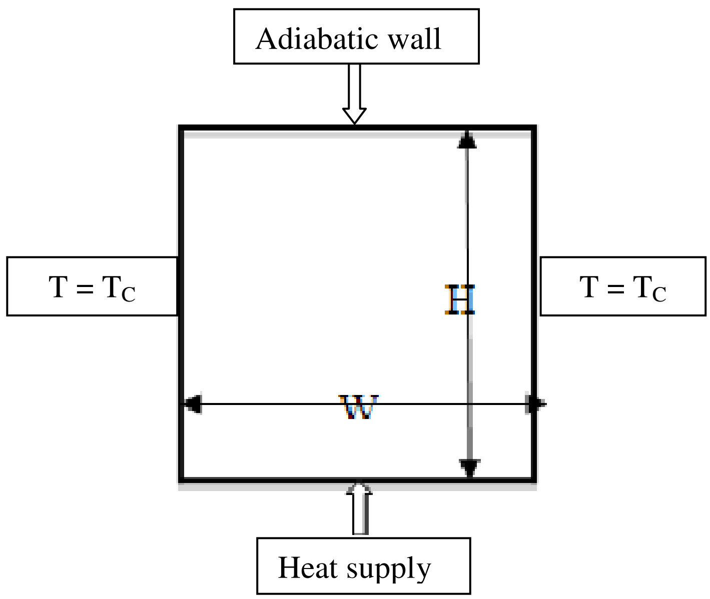


Figure 1.1 Heat transfer in square enclosure heated from below

# CHAPTER II

# LITERATURE REVIEW

## 2.1 Natural Convective Heat Transfer in Rectangular Cavity

Convection is an important type of heat transfer in technology and always occurs in nature. For years, it has been an area of considerable interest in many fields of study and research. There are two type of convection that is natural and force convection.

.

Natural convection heat transfer in an enclosed cavity has been a standard comparison problem for checking the computational methodologies related to general Navier-Stokes equations (Saitoh and Hirose, 1989). A rectangular shape would be a simple choice in simulating the problem since square shape did not have much vary in dimension, boundaries, and other properties related to it (Giri, et al. 2003). The condition for a benchmark numerical solution for a square cavity is considered as wall heated on the left side, wall cooled on the right side, and with adiabatic or perfectly conducting boundary conditions on the upper and lower walls (De Vahl, 1983).

In a different study to create bench mark solutions to natural convection in square cavity problems (Saitoh and Hirose, 1989), the Rayleigh number is varied while the aspect ratio is restricted to unity. The streamlines and isotherms at Rayleigh number of

$1x10^4$ and $1x10^6$ was presented. The observation can be used to validate an existing numerical model.

In this project, simulation for basic enclodure cavity will be done first before move to boudaries where heated from below.

## 2.2 Problem Physics and Boundary Condition for Square Cavity

The physical cavity is present in Figure 2.1. temperature gradient of the cavity will effect the generation of bouyancy. Resulting from that, the natural convection will happen where heat transfer from wall to another in certain streamline and isotherm line. Adiabatic wall is for controlling factor since it happen to be block all other heat or different kind of energy from outside that can affect the result.

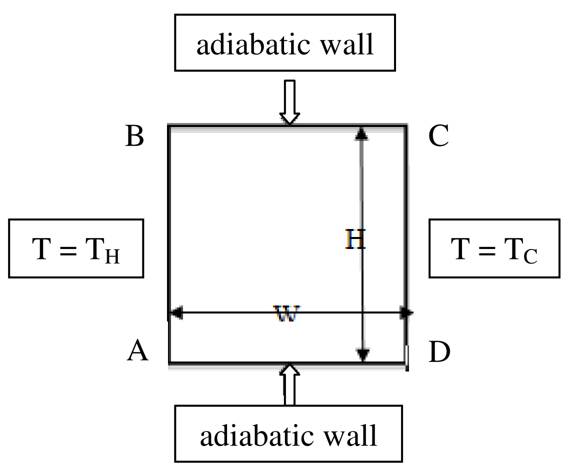


Figure 2.1 Problem physics and boundary condition for rectangular cavity

## 2.3 Governing Equations for Convective Heat Transfer

In order to predict convective heat transfer rates, the distribution through the flow field for pressure, velocity vector and temperature must be determined. Once the distributions of these quantities are determined, the variation of any other quantity can be obtained.

The distribution of these variables can be obtained by applying principles of conservation of mass, conservation of momentum (Newton's Law) and conservation of energy (First Law of Thermodynamics). These equations for steady laminar two-dimensional convective flow are given as follows (Cengel, 2003):

Continuity Equation :

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

Momentum Equation in x-direction :

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

Momentum Equation in y-direction :

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

Energy Equation:

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

## 2.4 Boussinesq Approximation

As discussed before, effect of temperature gradient will make changes in density of fluid. The changes then will affect the natural convection flows in the cavity. Since density is one of the properties of fluid, thus other fluid's properties that vary with the temperature effect must be take into consideration to obtain good desired result of analysis.

The only effect on the fluid is the generation of the buoyancy terms. Density effect in most free convective flow is relatively small and the buoyancy is the only effect left. It cause other properties that vary with the temperature gradient will all be neglected along with density.

$$\frac{\Delta\rho}{\rho}\leq 1$$