LAMINAR NATURAL CONVECTION IN ENCLOSURE CONTAINING FLUID

AIDIL AZLAN BIN AHMAD ZAMRI

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

‘Saya/Kami* akui bahawa telah membaca
karya ini dan pada pandangan saya/kami* karya ini
adalah memadai dari segi skop dan kualiti untuk tujuan penganugerahan
Ijazah Sarjana Muda Kejuruteraan Mekanikal (Termal-Bendalir)’

Tandatangan :…………………………………..
Nama Penyelia I:………………………………..
Tarikh :………………………………………….

Tandatangan:……………………………………
Nama Penyelia II:………………………………
Tarikh:………………………………………….

"Saya akui laporan ini adalah hasil kerja saya sendiri kecuali ringkasan dan petikan yang tiap-tiap satunya saya telah jelaskan sumbernya"

Tandatangan: ………………………..

Nama Penulis : ……………………….

Tarikh : ………………………

Buat ayahanda, bonda dan keluarga tercinta,
Insan tersayang, serta rakan-rakan seperjuangan,
Jasa dan Ingatan dari kalian tidak akan dilupakan.

ACKNOWLEDGEMENT

Alhamdulillah with His Mercy and Blessings, my greatest attitude to Allah the Almighty for the chances given to me to complete this project. My first acknowledgement goes to Mr. Shamsul Bahari bin Azraai, my project supervisor. Thanks for his tremendous help, advice, inspiration and unending guidance to me until completing this thesis. Without him, this project will not exist. The fundamental idea behind this project is his and he never stop from giving me support and encouragement in completing this project. Without his valuable guidance, I would not have been able to achieve the objectivity of this project.

Not forgotten to my beloved family especially my parents, Ahmad Zamri Bin Mohd Bak and Maimunah Bte Ibrahim. Your love and caring are really touch me. Thank you so much for giving me lifelong encouragement, inspiration, bravery, financial and also moral support. Last but not least, to my best friend, Nurul Akma Binti Mahpot that always there to give all the supports and ideas to complete this project. Hope that they get the best in life.

Lastly, my acknowledgment goes to my friend who really understand and willing to cooperate with me in this process of completing this project. Without your support I will never go this far. Thank you.

ABSTRACT

Computational Fluid Dynamics method was used to simulate the heat flow analysis of enclosure container. The simulation was base on a steady state formulation with the focus to assess the capabilities of Computational Fluid Dynamics tool for heat flow development of an enclosure situation. The comparison between the container is when the enclosure with vary situation, i.e. one heat source, two heat source and two heat source which is bottom and top. This suggests that the Computational Fluid Dynamics simulation can be used to boost the heat flow and heat transfer studies. Three cases are mainly discussed. First case is about the heat source from the top of the container, then from its bottom, and the third one from both places. The results get shows the profile of the flow. Convection is the main part of the profile. Gravitational effect is the important part of the heat distribution.

ABSTRAK

Kaedah menggunakan “Computational Fluid Dynamics” di dalam projek ini adalah bertujuan mendapatkan hasil simulasi pergerakan haba di dalam sebuah bekas segi empat yang tertutup. Simulasi ini berdasarkan situasi keadaan stabilnya dengan mengambil perhatian tentang kebolehan alat-alatan di dalam perisian itu sendiri dalam membina batas-batas bagi bekas yang tertutup tersebut. Perbezaan dan perbandingan dalam projek ini adalah di mana keadaan ataupun situasi dinding bekas itu berbeza seperti mana contohnya satu sumber haba iaitu diatas ataupun di bawah dan terakhir dua sumber haba. Selain itu, kaedah penggunaan perisian ini dapat meningkatkan daya pembelajaran mengenai pergerakan haba dan juga pemindahan haba dengan lebih jelas. Tiga kes utama akan dibincangkan iaitu punca haba dari atas bekas yang digunakan, kemudiannya sumber haba dari bawah dan terakhir adalah dari bawahnya. Keputusan diperoleh darinya adalah dalam bentuk alur aliran. Perolakan adalah pekara penting dalam hal ini disebabkan oleh penglibatan gravity yang membezakan antaranya.

CONTENTS

CHAPTER	TITLE	PAGE
	PENGAKUAN	i
	DEDIKASI	ii
	ACKNOWLEDGEMENT	iii
	ABSTRACT	iv
	ABSTRAK	v
	CONTENTS	vi
	LIST OF FIGURES	ix
	LIST OF TABLES	xi
	LIST OF SYMBOLS	xii
	ABBREVIATIONS	xiii
CHAPTER I	**INTRODUCTION**	
	1.1 Overview	1
	1.2 Problem Statement	2
	1.3 Objective	3
	1.4 Scope	3
CHAPTER II	**LITERATURE RIVIEW**	
	2.1 Introduction	4
	2.2 Empirical Correlation for	

CHAPTER	TITLE		PAGE

		Rectangular Enclosure in Natural Convection	6
	2.3	Computational Fluid Dynamics	8
	2.4	ANSYS-CFX	9
	2.5	Previous Studies	9
CHAPTER III	**METHODOLOGY**		
	3.1	Introduction	14
	3.2	Computational Fluid Dynamics	16
		3.2.1 ANSYS-CFX	16
		3.2.1.1 Geometry/Mesh	16
		3.2.2 ANSYS CFX-Solver	18
		3.2.2.1 Physics Definition	19
		3.2.2.2 Solver	19
		3.2.3 ANSYS CFX-Post	20
		3.2.3.1 Post-processor	20
CHAPTER IV	**RESULTS AND DISCUSSION**		
	4.1	Overview	22
	4.2	Result and Discussion	22
		4.2.1 Contour	23
		4.2.1.1 Temperature Contour	23
		4.2.1.1.1 Case One (Top Heated)	23
		4.2.1.1.2 Case Two (Bottom Heated)	26

		4.2.1.1.3 Case Three (Both Heated)	28
		4.2.1.2 Velocity Contour	30
		4.2.1.2.1 Case One (Top Heated)	30
		4.2.1.2.2 Case Two (Bottom Heated)	31
		4.2.1.2.3 Case Three (Both Heated)	31
		4.2.2 Vector	32
		4.2.1.1 Velocity Vector	33
		4.2.1.1.1 Case One (Top Heated)	33
		4.2.1.1.2 Case Two (Bottom Heated)	33
		4.2.1.1.3 Case Three (Both Heated)	34
CHAPTER V	**CONCLUSION AND RECOMMENDATION**		36
	REFERENCES		37
	BIBLIOGRAPHY		40
	APPENDIX		42

LIST OF FIGURES

FIGURE NO.	TITLE	PAGE NUMBER
1.1	Rectangular enclosure with heat source at the bottom and top walls	2
1.2	Rectangular enclosure viewed by Solid Works	3
2.1	Boundary layer flow near a vertical plate (one side); notice the choice of *x* and y coordinates.	6
3.1	Flow chart of process to obtain the result using CFX	16
3.2	Creating the physical with two heat sources using Solid Works.	18
3.3	Meshing Process	19
4.1	Graph Temperature against elevation for top heat Source	25
4.2	Heat Contour in mid plane of an isometric container	26
4.3	Heat Contour from Front View	26
4.4	Graph Temperature against for heat source from bottom	27
4.5	Figure shows that temperature contour heated from its bottom in isometric view	28
4.6	Temperature Contour from front view	28
4.7	Graph Temperature against Elevation for Heat Source from above and below	29
4.8	Temperature Contour for both top and bottom heated plate in isometric view	30

FIGURE NO.	TITLE	PAGE NUMBER
4.9	Figure of heat contour front view	30
4.10	Velocity Contour For case one	31
4.11	Velocity Contour For case two	32
4.12	Velocity Contour for Case Three	33
4.13	Velocity Vector for Case One	34
4.14	Velocity vector for case Two from Front view	35
4.15	Velocity Vector for Case Three	35

LIST OF TABLES

TABLE NO.	TITLE	PAGE NUMBER
2.1	Convective Heat Correlation for Natural Convection	7

LIST OF SYMBOLS

Vs	= Mean fluid velocity, [ms^{-1}]
L	= Characteristic length, [m^2]
μ	= Dynamic fluid viscosity, [Nsm^{-2}]
ν	= Kinematic fluid viscosity: ν = μ / ρ, [$m^2 s^{-1}$]
ρ	= Fluid density, [kgm^{-3}]
Re	= Reynolds Number
u	= Velocity, [m/s]
d_h	= Hydraulic diameter, [m]
q_w	= Mean local heat flux at the heated walls, [W]
T_b	= Bulk temperature, [^0C]
T_w	= Wall temperature, [^0C]
Nu	= Nusselt numbers
h	= Mean heat transfer coefficient, [$W / m^2 K$]
D_h	= Hydraulic diameter [m]
k	= Thermal conductivity, [$W/(mK)$]
Q	=The Volumetric flow rate, [m^3 / s]
A	=Area, [m^2]
V	=Velocity, [m/s]

LIST OF ABBREVIATIONS

CFD = Computational fluids dynamics

CMC = Carboxymethyllcelulose

CHAPTER I

INTRODUCTION

1.1 Overview

Free convection in enclosure has attracted many researcher and also interest of investigators. Various applications in this term that really attractive such as electronic devices, furnaces, power plant and heat transfer through pipes. This result is relevant due to environmental science, where the heat transfer take part of responsibility for oceanic and atmosphere motion.

Free convection fluid motion is due to buoyancy forces within the fluid. Now we consider situation which there is no forced velocity, yet convection current exit within the fluid. Such situation is referred as free or natural convection, and they originate when a body force act on a fluid in which there are density gradients. The net effect is a buoyancy force, which induced free convection currents. In the most common case, the density gradient is due to a temperature gradient, and the body force is due to the gravitational field.

In many systems involving multimedia heat transfer effect, free convection provides the largest resistance to heat transfer and therefore plays an important role in the design or performance the system. Moreover, when it is desirable to minimize heat transfer rates or to minimize operating cost, free convection is often preferred to forced convection.

1.2 Problem Statement

Problems occurred when there is a rectangular enclosure container with heat source at the top and bottom walls shown in figure 1.1 that heat source from supply from bottom and top of the enclosure. The heat source are then heated into a certain temperature so that's the fluid within the container shall have a convection condition. So, a software is use to obtain the result (heat flow). The software is important due to simulate the heat flow in order to see which part will be in high temperature and how does the heat moves. Eventually, from this problem, another problem can be issued, whereas in case of partition or without any partition. Figure 1.2 shows the container form in three-dimensional or in isometric view.

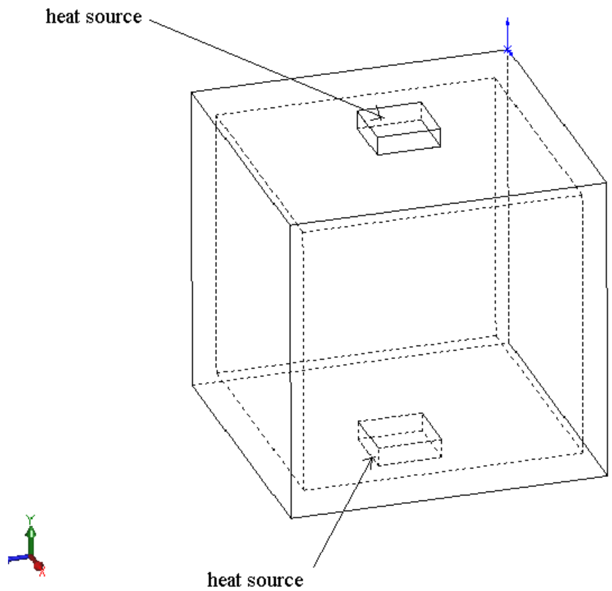

Figure 1.1: Rectangular enclosure with heat source at the bottom and top walls

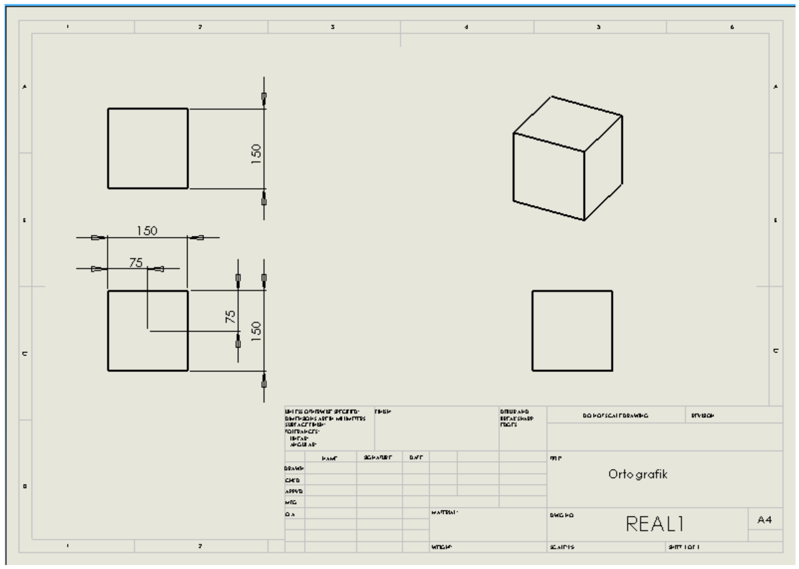

Figure 1.2: Rectangular enclosure viewed by Solid Works.

1.3 Objective

To predict numerically the heat distribution in an enclosure container with top and bottom heat source.

1.3 Scope

The scope of this project is to design the rectangular enclosure with wall heat source on top and bottom container and also simulate with Computational Fluid Dynamics method by using ANSYS-CFX software.

CHAPTER II

LITERATURE REVIEW

2.1 Introduction

Natural convection is the fluid flow originated by gravity forces acting on non-uniform-density fluids; the density changes may be due to thermal or to solutal gradients. Many different natural-convection configurations are of interest, from the simplest hot or cold vertical plate in a fluid medium, to external convection around hot or cold bodies, or internal convection within hot/cold enclosures (non-isothermal).

Laminar flow, sometimes known as streamline flow, occurs when a fluid flows in parallel layers, with no disruption between the layers. In fluid dynamics, laminar flow is a flow regime characterized by high momentum diffusion, low momentum convection, pressure and velocity independent from time. It is the opposite of turbulent flow.

The boundary layer on a hot vertical plate is a canonical thermo-fluid-mechanics problem. A semi-infinite vertical wall is shown in Fig. 2.1 (it must extend upwards if it is hot, or downwards if it is cold), maintained at constant temperature, T_w, and immersed in a dilatable fluid (with thermal expansion coefficient α), at temperature T_∞ far from the wall, in a gravity field, g, giving rise to a boundary-layer flow of thickness δ, which starts at the entry border and grows along the length of the plate, with the longitudinal velocity growing from u=0 at the wall, to a maximum value within the boundary layer, and finally decreasing to u=0 at the outer edge of the layer the outer fluid is at rest except for the very small entrainment flow implied by the boundary layer growth.

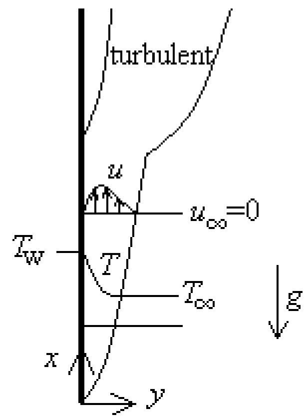

Figure 2.1: Boundary layer flow near a vertical plate (one side); notice the choice of x and y coordinates.

The terms governing the flow near the vertical plate, assumed steady and incompressible except for the momentum buoyancy term (Boussinesq model), are the following;

I. Mass balance (continuity):

The law of conservation of mass/matter, also known as law of mass/matter conservation states that the mass of a closed system of substances will remain constant, regardless of the processes acting inside the system. This means that matter or mass cannot be created or destroyed although it can change form.

II. Momentum Balance

The law of conservation of momentum is a fundamental law of nature, and it states that the total momentum of a closed system of objects (which has no interactions with external agents) is constant. In short when an object had a collision with another object, the momentum for both of them are the same.

III. Energy Balance

Conservation of energy states that the total amount of energy in any system remains constant, although it may change forms for example friction turns kinetic energy into thermal energy. Meaning that energy cannot be created or destroy but can be changed from one to another.

Fluids are also the main part of the convection movement. There is a great variety of heating fluids and cooling fluids, which can be grouped basically as gases, liquids, and phase-changing fluids.

2.2 Empirical Correlation for Rectangular Enclosure in Natural Convection

A small tilting in the hot vertical plate analysed before, already causes great changes in the flow, since the boundary layer detaches at several places along the upper side of the plate (if hot; the lower side in a cold plate), forming three-dimensional patches due to flow instabilities. That is why most heat and mass natural-convection correlations are empirical fittings from experimental data. Table 2.1 gives a compilation of convective heat correlation for natural convection.

Table 2.1: Convective Heat Correlation for Natural Convection

Configuration	*Heat convection*
<u>Horizontal rectangular enclosure</u> (Internal steady flow, with two opposite adiabatic walls, and a ΔT between the other two walls at separation L, and $Nu_L \equiv hL/k$): -Heated from above (stable gradient; no fluid-flow; $Nu_L=1$) -Heated from below with $Ra_D<1708$, (no fluid-flow; $Nu_L=1$) -Heated from below with $1708<Ra_D<10^8$ and $Pr>0.7$, (flow instability; onset of convection at $Ra_D=1708$).	$Nu_L = 1+1.44\left[1-\frac{1708}{Ra_L}\right]^{+} + \left[\frac{Ra_L^{1/3}}{18}-1\right]^{+}$ $[\,]^{+}$ means that these terms must only be accounted if positive (set to 0 if negative). Holland et al. (1976).
<u>Vertical rectangular enclosure</u> (Internal steady flow, with a ΔT between the two vertical walls at separation L, and the adiabatic walls at separation H), $Nu_L \equiv hL/k$.	$1<H/L<2$, any Pr, $Ra_L>10^3(0.2+Pr)/Pr$: $Nu_L = 0.18\left(\frac{Pr\,Ra_L}{0.2+Pr}\right)^{0.29}$ $2<H/L<10$, any Pr, $Ra_L<10^{10}$: $Nu_L = 0.22\left(\frac{Pr\,Ra_L}{0.2+Pr}\right)^{0.28}\left(\frac{L}{H}\right)^{0.25}$ $10<H/L<40$, any $Pr>1$, $10^4<Ra_L<10^7$: $Nu_L = 0.42Ra_L^{0.25}Pr^{0.012}\left(\frac{L}{H}\right)^{0.3}$

<table>
<tr><td></td><td></td></tr>
<tr><td>

<u>Inclined rectangular enclosure</u>

(Internal steady flow, with a ΔT between the two closest walls at separation L, tilted an angle θ from the vertical, and the adiabatic walls at separation H), $Nu_L \equiv hL/k$.

Fluid flow changes beyond a critical angle $\theta_{cr}(H/L)$:

<table>
<tr><td>H/L=</td><td>1</td><td>3</td><td>6</td><td>12</td><td>>12</td></tr>
<tr><td>θ_{cr}</td><td>25º</td><td>53º</td><td>60º</td><td>67º</td><td>70º</td></tr>
</table>

</td><td>

$0<\theta<\theta_{cr}$, $H/L<12$, $Pr>0.7$

$$Nu_\theta = Nu_{\theta=0}\left(\frac{Nu_{\theta=0}}{Nu_{\theta=90^\circ}}\right)^{\theta/\theta_{cr}}\left(\sin\theta_{cr}\right)^{\frac{\theta}{4\theta_{cr}}}$$

$0<\theta<\theta_{cr}$, $H/L\geq 12$, $Pr>0.7$

$$Nu_\theta = 1+1.44\left[1-\frac{1708}{Ra_L\cos\theta}\right]^{+}\left(1-\frac{1708\left(\sin(1.8\theta)\right)^{1.6}}{Ra_L\cos\theta}\right)+\left[\frac{\left(Ra_L\cos\theta\right)^{1/3}}{18}-1\right]^{+}$$

$\theta_{cr}<\theta<90^\circ$, any H/L, $Pr>0.7$

$$Nu_\theta = Nu_{\theta=90^\circ}\left(\sin\theta\right)^{1/4}$$

</td></tr>
</table>

2.3 Computational Fluid Dynamics

CFD is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. However, even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. More accurate software that can accurately and quickly simulate even complex scenarios such as transonic or turbulent flows are on going area of research. Validation of such software is often performed using a wind tunnel.

2.4 ANSYS-CFX

CFX is a commercial CFD program used to simulate fluid flow in a wide variety of applications. CFX allows engineers to test systems in a virtual environment. It has been applied to the simulation of water flowing past ship hulls, gas turbine engines (including the compressors, combustion chamber, turbines and afterburners), aircraft aerodynamics, pumps, fans, HVAC systems, mixing vessels, hydrocyclones, vacuum cleaners, and more. Pretty much anything that involves fluid flow can be simulated, if you have the expertise and computing power. Regarding computing power, CFX can give any cluster a run for its money. It is highly scalable and has been shown to maintain nearly linear scalability to as many as 500 processors.

2.5 Previous Studies

Natural convection in enclosures has attracted many of researchers in the world. The main ideas of heat transfer made a role to lead the research success. Through all applications such as furnace design, buildings, cooling of electronic equipment, nuclear reactor and others refer and studied on heat transfer buoyant flow in enclosures.

Results that come out are studied either by numerically or experimentally or even both. Generally, studies about convective in enclosures without partition convinced by Elder (1965a, b, 1966). Then, later results presented by Catton (1978) and Yang (1987), Ostrach (1972, 1982).

The results (natural convection) can be used as a bench mark data for validation to solve Navier-Stokes equations for various numerical methods. The bench mark numerical results were obtained with grids up to 81 x 81 points at Rayleigh numbers of