THERMAL PERFORMANCE ANALYSIS AND CHARACTERIZATION FOR MOBILE RADIO

By

CHEE JIUNN SHYONG

Thesis submitted in fulfillment of the requirement for the degree of Master of Science

November 2013

ACKNOWLEDGEMENT

I would like to address my appreciation to my supervisor Prof. Dr. Mohd. Zulkifly B. Abdullah, for spending his time to provide tremendous guidance, advises and encouragements in motivating me to complete my MSc research and also this thesis writing.

My sincere appreciation to the examiners, for helping me to check this thesis, providing useful feedback, valuable comments and correcting the technical errors which are crucial in determining the excellent outcome of this thesis.

Also, a millions thanks to my ex-colleagues at Motorola Penang, who were helping me a lot in compiling data, reviewing results and providing technical advice during the research phase of this research.

Last but not least, thanks to my beloved wife, Jennifer Ng Lee Ting and my family members, for consistently giving me supports in making this research a successful one.

TABLE OF CONTENTS

LIST OF TABLES

PAGE

LIST OF FIGURES

PAGE

LIST OF SYMBOLS

k	Thermal conductivity, $W/m.K$	25
k_t	Eddy conductivity, $W/m.K$	25
q_V	Volumetric heat source, W/m^3	25
$R_{specific}$	Specific gas constant, $J/kg^{-1}.K^{-1}$	25
l_m	Mixing length, m	26
G	Nusselt number	26
$\dot{\varphi}$	Velocity shear rate, $1/s$	26
κ	Von Karman constant, $\kappa = 0.41$	26
s	Normal distance from wall, m	26
σ_t	turbulent Prandtl number, usually $\sigma_t = 1$	26
L	Length of the cavity/enclosure	44
H	Height of the cavity/enclosure	44
T_H	temperature of the heat source, K	44
T_C	temperature of the cold plates, K	44
l	Length of the heat source, m	44
PA	Power Amplifier	46
DSP	Digital Signal Processor	47
T_a	Ambient temperature, K	50

ANALISIS PRESTASI DAN PENCIRIAN HABA UNTUK RADIO MUDAH ALIH

ABSTRAK

Aliran olakan tabii mantap dalam pemindahan haba radio mudah alih adalah sangat kompleks dan tiga dimensi. Penyelesaian berangka yang tepat dan dipercayai untuk kelajuan rendah (pada magnitud 1 x 10^{-1} m/s), aliran separa bergelora disebabkan oleh interaksi antara daya keapungan dan graviti masih adalah cabaran kepada kebanyakan perisian komersial dinamik bendalir pengiraan (CFD) yang sedia ada, di mana kod CFD terutamanya dibangunkan untuk menyelesaikan pemindahan haba olakan paksa analisis. Dalam kajian ini, syarat sempadan yang sesuai, model gelora, isipadu udara luaran dan kaedah bersirat yang menyumbang kepada ramalan pemindahan haba CFD untuk radio mudah alih telah dikenal pasti dengan menggunakan CFDesign, perisian kaedah unsur terhingga CFD.

Sebelum menjalankan simulasi haba olakan tabii pada radio mudah alih, usaha untuk mengesahkan ketepatan ramalan menggunakan CFDesign dengan model pergolakan yang berbeza, seperti zero-equation mixing-length, two-equation κ-ε and Low-Reynolds κ-ε dijalankan dengan menggunakan rongga segiempat persegi di mana sumber haba yang terletak di tengah-tengah pada permukaan bawah. Isoterma keputusan yang diperolehi diukur dengan keputusan Calcagni, Marsili et al. (2005) yang diramalkan oleh Fluent, kod kaedah isipadu terhingga, dan juga penemuan Aydin and Yang (2000) yang menggunakan kod kaedah perbezaan terhingga. Tujuan pengesahan ini adalah untuk mengenal pasti model yang paling sesuai untuk pergolakan yang menyampaikan taburan suhu dan medan aliran olakan yang realistic untuk simulasi CFD pemindahan haba dalam CFDesign.

Kemudian, satu seri simulasi pemindahan haba olakan tabii radio mudah alih telah dijalankan pada suhu ambien, 26^{o}C. Siasatan dan beberapa penalaan halus untuk isipadu luaran, syarat sempadan dan kaedah bersirat telah dijalankan untuk meningkatkan korelasi antara keputusan eksperimen & simulasi. Keputusannya dibandingkan dengan hasil eksperimen yang diperbetulkan dari imbasan haba Infra-Merah dan pengukuran termogandingan.

Kajian ini telah membuat kesimpulan bahawa 4 peraturan, iaitu model gelora, nisbah aspek udar luaran, syarat sempadan dan kaedah bersirat memainkan peranan yang penting dalam menyampaikan keputusan simulasi yang tepat untuk olakan pemindahan haba radio mudah alih. Dengan pelaksanaan yang betul untuk 4 peraturan yang dinyatakan di atas, rujukan simulasi persekitaran yang boleh dipercayai dan tepat untuk fenomena perolakan semulajadi radio mudah alih telah ditubuhkan dalam kajian ini. Kesilapan simulasi untuk suhu komponen-komponen kritikal adalah di antara -1,69% kepada 6.7% berbandingkan dengan eksperimen. Dengan menggunakan rujukan simulasi persekitaran yang dipercayai ini, kesan kepada komponen-komponene kritikal radio mudah alih dalam ambien suhu yang berbeza telah disiasat. Selain itu, pengaruh kuantiti sirip kepada peningkatan prestasi penyejukan telah difahami juga.

THERMAL PERFORMANCE ANALYSIS AND CHARACTERIZATION FOR MOBILE RADIO

ABSTRACT

Steady natural convection flow field and its heat transfer of a mobile radio are very complex and three dimensional. Accurate and reliable numerical solution for low speed (at the magnitude of 1 x 10^{-1} m/s), transitional flow induced by the interaction between buoyancy force and gravity is yet a challenge on many commercially available computational fluid dynamic (CFD) codes, as these CFD codes mainly developed for solving forced convection heat transfer analysis. In this study, the appropriate boundary conditions, turbulence model, external volume and meshing methods that contribute to a reliable CFD heat transfer prediction of a mobile radio were identified by utilizing CFDesign, a commercial finite element method CFD tool.

Prior to carry out a thermal simulation of natural convection on a mobile radio, efforts to validate the prediction accuracy of CFDesign for different turbulence models, which are zero-equation mixing-length, two-equation κ-ε and Low-Reynolds κ-ε were carried using a simple rectangular square enclosure with heat source located centrally on the bottom surface. Isotherm results obtained are correlated with results published by Calcagni, Marsili et al. (2005), predicted by means of Fluent, a finite volume method code, and also results from Aydin and Yang (2000) by utilizing finite difference method code. The purpose of this prediction accuracy validation is to identify the most suitable turbulence model that deliver realistic temperature distribution and flow field of natural convection heat transfer CFD simulation in CFDesign.

Then, a series of natural convection heat transfer CFD simulations of mobile radio were performed at ambient temperature, 26^{o}C. Investigations on the effect of external air volume, boundary conditions and meshing methods had been carried out to improve the correlation between experimental & simulation results. Results obtained are compared with experimental results that collected from Infra-Red thermal imaging and thermocouple measurements.

This study has concluded that 4 rules, which are turbulence model, aspect ratio of external air volume, boundary conditions and mesh refinement technique play an important role in delivering accurate simulation results for natural convection heat transfer of mobile radio. With a proper implementation of the aforementioned 4 rules, a reliable and accurate simulation environment reference of natural convection phenomenon for mobile radio is established in this study. Simulation errors of critical components temperature are in between -1.69% to 6.7% comparing with experiments. Furthermore, by adopting this reliable simulation environment reference, thermal behavior of mobile radio under different elevated ambient temperature was studies. Additionally, influence of fins array design on heat dissipation performance was validated.

CHAPTER 1

Introduction

1.1 Natural Convection

Natural convection is a heat transport mechanism induced by density difference in the fluid occurring due to the temperature gradients. No external source is involved in generating the fluid motion. The only driving force for natural convection is buoyancy, which is resulted from the reaction of density difference of fluid and a reverse acceleration force (gravity, in most of the cases). In nature phenomena and most of the engineering applications, natural convection is the major mode of heat transport vehicle. For natural phenomena, convection cells formed from air rising above warmed land or water due to sunlight radiation are major part of all weather systems. Rising plume of hot air from fire, oceanic currents and sea wind formation are also phenomena of natural convection. In engineering application, natural convection is commonly visualized in rising smoke at top of chimney, fluid flows around shrouded heat dissipating fins. Natural convection is widely utilized in free air cooling without the aid of fans/pumps, from small scales such as IC package up to large scale process equipment.

Although natural convection plays a very important role on heat transfer, as mentioned by Ostrach (1988), modern research on it was started pathetically by German and Japanese scientists at early 1940s, because it was not considered to be of any practical importance. Fortunately, interest on natural convection is increasing since then.

Phenomenon of natural convection is ubiquitous in our daily experience, however, the first comprehensive review on this subject only revealed by Ostrach (1988) in 1960s. In his review, he concluded that basically there are 2 basic modes of flow generated by buoyancy. The first is referred as conventional convection, occurs whenever a density gradient is normal to the gravity vector. For the second mode, it is called unstable convection, occurs when the density gradient is parallel but opposed to the gravity vector.

Over the last two decades, the technology advancement in PC industry drives the ever-increasing power density of IC package. In order to cool the IC package effectively, numerous researches have been carried to understand the cooling limits of natural convection. At the early stage, experimental studies were done on confined natural convection in a cavity with different boundary condition configuration. It was observed that natural convection is very sensitive to changes in cavity geometry and the imposed boundary conditions. Compare to external (free) convection where the external region to the boundary layer is unaffected by the boundary layer, which able to simplify the fluid-flow problem by relying on Rayleigh number. Formation of flow in natural convection is more intricate because the external region is enclosed by the boundary layer form near the walls and generating a circulating core flows. Adding to complexity, the core flows behavior is not solely influenced by the boundary conditions, but also affected by the boundary layer. Hence, phenomenon of natural convection is coupling between fluid-flow, heat transport, boundary layer and core flows. These incur a mathematical challenge where coupling of the mass continuity equation, fluid-flow momentum equations and energy conservation equations are essential. Additionally, proper characterization of boundary layer is critical in determine the core flows.

Although the application of natural convection cooling is widely used in numerous engineering applications, it does not receive much attention from the researcher until recent years. There are several reasons behind this; one of the most significant is the high level of complexity on analyzing natural convection phenomenon experimentally and numerically.

Natural convection is easily noticeable in almost everywhere in daily life. However, detailed analysis on the physic of the fluid-flow and heat transport is challenging due to observation have to be done up to fluid particle level. In experimental study, advance techniques such as Particle Image Velocimetry (PIV) can be utilized to obtain instantaneous velocity measurements and related properties in fluids. The drawback of these methods are expensive investment and complicated apparatus setup.

The inherent coupling of continuity, momentum & energy equations of natural convection incurs a mathematical challenge on solving five Partial Differential Equations (PDE) simultaneously. In order to solve these numbers of PDEs for thermal problem involves complex three-dimensional geometries, high level of computational difficulty is anticipated. Moreover, turbulence modeling on natural convection still limited to algebraic (zero-equation) model where eddy viscosity and eddy conductive are calculated from flow variables. Unlike forced convection, a handful of well-defined turbulence modeling is available to tackle respective application. For natural convection, energy exchange is happen within the inner layer of boundary layer, thus, solving it by turbulence modeling that require wall function to model the turbulent flow next to the wall is inappropriate.

1.2 Application of Natural Convection on Electronic Devices

There are several advantages on cooling by natural convection. One of the most important and obvious benefits is natural convection requires no external source (e.g. pump, fan, compressor) to initiate the cooling process. Heat transfer in natural convection is purely a consequence of physic, where heat energy is transfer from heat source surface to surrounding fluid creates a density different between heated fluids and quiescent fluids. With accelerating factor which is normally gravity force, heated fluids with lower density is moving upward and leftover volume is replaced by cooler quiescent fluid. As long as the heat source remains, this buoyancy effect continues.

Since there is no external source required, natural convection is the most effective greenest method for electronic cooling. Additionally, environmental pollutions such as noise that comes as side effects in most of the forced convection applications are an exception on natural convection. In brief, natural convection is the best cooling solution for system that designed low maintenance, long life and low acoustic noise. However, ever increasing of heat density of IC on electronic device is pushing the envelope of cooling limit provided by natural convection, Since the day it existed, theoretical limit of heat transfer coefficient for natural convection by air remains unchanged at 10 W/m^2/K, (Whitelaw 2011).

Another benefit of natural convection is its extreme long-term reliability against forced convection. The wear & tear effect due to the fluid motion is negligible. A well design natural convection system is maintenance free and able to last for decades.

Recent awareness on energy efficiency brings natural convection back into stage after being ignored for over many years. A good example is the solar panel application, to optimize the energy conversion efficiency of photovoltaic (PV) cells; temperature of the solar panel is one of the factors. Contrary to popular belief, the efficiency of a solar cell decreases with increasing of temperature. General rule of thumb is that the efficiency of a solar cell decreases with 0.5% for every 1 °C above 25 °C. Hence, proper cooling design via natural convection has to be implemented to keep the solar panel temperature low. In recent research, attention is given to investigate the critical tile angle of solar panel at different heat flux condition that triggers unsteady natural convection.

Adoption of natural convection cooling on LED lighting is also getting serious attention as the rapid growth of LED fabrication technology in recent years. Stringent lighting requirements especially on noise pose a great challenge on cooling. Current cooling design for conventional fluorescent and incandescent light no longer works on LED lighting. In convection lighting, majority of the heat dissipated is through radiation. Whereas in LED lighting, the dominant heat transfer mode is conduction, heat generated from the LED chip is dissipated to environment by heatsink attached directly to LED die. Hence, existing thermal solutions no longer works. Furthermore, small surface area of LED die incurs high heat density; this generates another thermal challenge on improving heat spreading performance. Operating environment for lighting applications also prohibits other means of active cooling methods. To enable rapid adoption of LED lighting, the need to modify existing lighting infrastructure shall be reduced to minimum. Hence, currently a natural convection cooling method shall be used.

1.3 A Brief Intro of Mobile radio

Mobile radio in this study is referring to wireless communication device which is transmitting and receiving signals via radio frequencies. It is mounted to a motor vehicle usually with microphone and control panel in reach of the driver. Typically, it is powered by the host vehicle's 12 volt electrical system. Depends on the frequency band, power level of a mobile radio can up to 110 Watt (see Appendix A.), with average efficiency of a typical Power Amplifier (PA) at 80%, the heat dissipated from the PA is at around 20 Watt.

Modern mobile radios consist of a radio transceiver, packaged inside an enclosure, and a microphone with push –to-talk button. For a very long period since mobile radio was invented, it is operating in analog protocol as it is mainly used for voice communication. In the decade after year 2000, signals transmission in digital protocol becomes viable due to the technology advancement of microprocessor. Hence, this transforms mobile radio into a feature-rich communication device. In additional to voice communication, digital mobile radio also capable to transmit any form of digital media such as photos and videos, as well as web access.

This expansion of functionality induced by digital protocol leads to a usage pattern changes on mobile radio. Compare to analog mobile radio where full performance is only needed when there is transmission (Tx), digital mobile radio is required to operate in continuous full performance condition. This difference of usage pattern leads to two different heat transfer scenarios. For the first usage pattern where full performance is only needed intermittently, it can be considered as a mode of transient heat transfer as the heat is only generated for a short period of time and then following by another period of heat dissipation. However for continuous full performance usage pattern, obviously it is steady state heat transfer mode. In this mode, heat generation and dissipation happen at the same time even equilibrium state is reached. The steady state heat transfer mode poses a more stringent challenge to the thermal management of mobile radio as there is no dedicated period for cooling.

Current major manufacturers of mobile radio are Motorola Solutions and Kenwood. Figure 1.1 shows the digital mobile radio from Motorola that being the focus of this steady natural convection heat transfer study.

Figure 1.1: Motorola Mobile Radio

1.4 Problem Statement

Mobile radio is a communication device that widely deployed in patrol cars, fire brigade, forwarding trucks, taxi and ambulance. Temperature in a confined vehicle environment can reach 60^{o}C (see Appendix A.) under direct hot sun during summer. In order to operate in this extreme temperature, proper heat transfer design is critical for mobile radio.

Due to the constraints of space, power, environment and reliability, cooling by mean of forced convection is not preferred. However, cooling a mobile radio by natural convection has been an engineering challenge for many years. In recent years, radio transmission technology is transitioning from analog to digital due to the increase of customer's demands on data transmission and video streaming. To enable this, more processing power is needed. With the form factor of the mobile radio remains unchanged and the ever shrinking processor package size, a detailed understanding on the heat transfer of mobile radio is essential.

Thermal management is vital in mobile radio development due to its high heat dissipation from the major heat source during radio signal transmission. For example, Power Amplifier (PA) has been identified as the component with highest heat dissipation in this study. If the heat dissipation path from PA to the heat sink design at the bottom chassis is inefficient, heat dissipated by the PA will be trapped inside the enclosure and then raising the temperature of the surrounding electronic components, such as ICs and capacitors. Over time, this phenomena tends to degrade the performance of the electronic components by reducing the component's MTBF (Mean Time between Failure) and finally contribute to failure of the system. In worst case condition, it can also cause active components exceed their maximum junction temperature and lead to immediate malfunction.

On the other end, heat dissipated to the external surfaces such as top chassis and bottom chassis can cause discomfort burning effect to user when it is being touched. In the industry, this is safety touch compliance listed under IEC/EN60590, allowable touch specification for metal surface is 70^{o}C.

With all constrains mentioned above, it is a challenge to optimize the thermal performance of mobile radio by completely depend on experimental analysis. Therefore, heat transfer simulation needs to be developed in order to validate various design concepts more quickly and effectively. To ensure high accuracy simulation results, proper setup of the simulation parameters such as boundary condition, external volume, turbulence model and meshing is crucial.

1.5 Objectives

Several objectives have been identified for this entire study, see following for the summary:

1. To understand the effect of external air volume ratio on buoyancy flows in order to identify the best aspect ratio of external air volume. Besides, this study also aims to determine the best boundary condition by evaluating influence of different boundary conditions based on velocity flow field induced by natural convection.
2. To establish an accurate and reliable simulation environment reference for natural convection heat transfer CFD analysis on mobile radio by correlating results of numerical analysis with experimental investigation. As operating ambient temperature of mobile radio is up to 60^{o}C (see Appendix A.), it is also crucial to verity

the natural convection behavior of mobile radio under different elevated ambient temperature.

3. To discover the optimum fins quantities on mobile radio that deliver maximum heat dissipation performance.

1.6 Scope of Work

This research is aimed to understand the heat transfer performance of on a naturally cooled mobile radio by means of CFD analysis. CFDesign, a finite element method (FEM) computational fluid dynamics (CFD) tool are used to perform the simulations of this study.

To establish a accurate simulation environment setup, a correlation analysis are conducted to access effects of boundary conditions, aspect ratio of the external air volume, mesh size, turbulence mode, ambient temperature and Rayleigh Number. Isotherm and streamline results were compared with research published by using finite element method (FDM) and finite volume method (FVM). Purpose of the comparison is to understand the behavior of the different parameters. What is the most suitable simulation environment setup that mimics the thermal test environment on mobile radio? In addition, advantages and disadvantages of FEM in CFD simulation was discussed.

With best simulation environment setup defined, CFD model of mobile radio was developed using Pro/Engineer and numeral analysis were executed by means of CFDesign.

In order to further validate the prediction accuracy of the simulated results, experimental studies were carried out on prototype mobile radio. Results were recorded in the form of Infra-Red thermal imaging and thermocouple measurement.

The final outcome of this study is to provide a validated simulation environment setup that capable to deliver accurate thermal simulation results for mobile radio. With this setup, upfront thermal simulation at early concept design stage becomes viable, which eventually leads to shorter product development time and lesser prototype investment.

1.7 Summary of the Thesis

The high level outline of this dissertation is shown at Figure 1.2. The first chapter of this dissertation is defining problem statemen, follow by objectives and scope of work, which have been explained earlier prior to this chapter.

Second chapter is discussing about the literature research that related to this study. In this chapter, literature on numerical analysis on natural convection in a cavity with bottom heated element is studied. Furthermore, investigation on the effectiveness of different turbulence models in natural convection prediction is also carried out. A brief explanation of the mathematical formulation of Navier-Stokes equation and theory of Mixing Length Turbulence model are covered at the latter part of this chapter. Purpose of the literature study is to identify reference that can be used as comparison or supporting information for current study.

In the third chapter, different methods for fluid dynamics studies (AFD, EFD ad CFD) are explained. Subsequently, a more detailed explanation is provided on the CFD method and then follows by a short overview on those numerical techniques (FEM, FDM and FVM) that available in this method. Besides, a brief introduction of the commercial CFD numerical analysis software used in this study (CFDesign) is given. Lastly, an overview of the major stages of the software in solving a heat transfer problem is explained.

Chapter 4 outlines the simulation methodology that being practiced in this study. At first, setup configuration and procedures of experimental analysis are illustrated. Then, method to evaluate the prediction accuracy of CFDesign software is explained. Afterwards, approaches to achieve a simulation environment reference are disclosed.

In Chapter 5, comparison study on the results obtained from the experimental analysis and simulation analysis is carried out in order to discover the simulation environmental reference. Finally, with the simulation environment reference identified, further investigations are conducted to understand the influence of different ambient temperature and the effect of fins quantities on mobile radio.

In the last chapter, conclusion of this study on natural convection heat transfer of mobile radios is published as well as recommendations for future potential research is proposed.

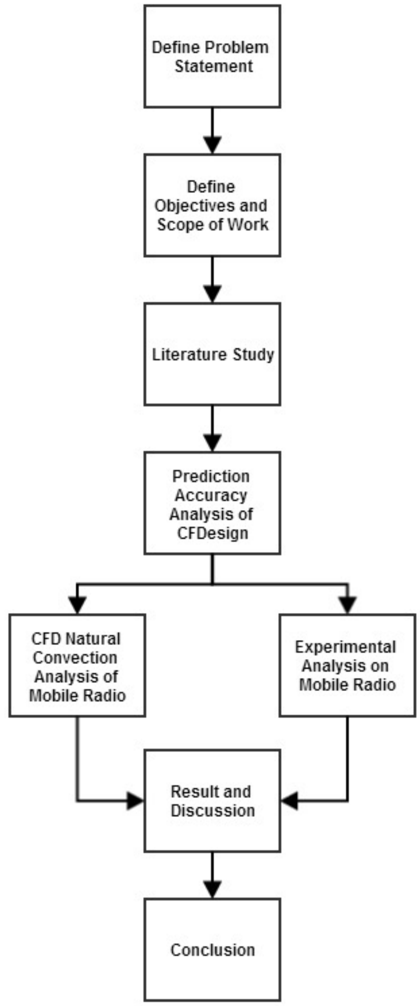

Figure 1.2: Flow Chart of Major Activities

CHAPTER 2

Literature Study

2.1 Numerical Analysis on Natural Convection

Numerical analysis on Natural convection of air (*Pr* = 0.71) in square cavities with heated elements at bottom or heated bottom surface, which is also called as unstable/unsteady convection, has been extensively studied for different Rayleigh number range between 10^3 and 10^7. As the primitive heat transfer mode for many engineering & industrial applications, such as passive cooling of electronic devices, energy exchange of building structure, solar energy collector, cooling of nuclear reactor and etc., various studies were published by scientists and researchers from academics background, as well as engineers from industrial sectors. A collection of literature on the natural convection heat transfer in a square cavity with heated elements at bottom is listed at Bibliography. Some of the important & relevant work are referred and serve as a background for this study.

From the literature available, most of the studies were focusing on the numerical analysis based on Finite Difference Method (FDM) & Finite Volume Method (FVM) from the year 2000 until 2011. The effect of different boundary condition on top & lateral walls in a square cavity with heated from bottom are studied numerically by commercial code and self-developed algorithm.

2.1.1 Natural Convection in a Square Cavity Heated Locally From Below and Cooled from Above

Cheikh, Beya et al. (2007) investigated numerically the natural convection in a square cavity heated locally from below and cooled from above with several specified top & sidewalls boundary conditions, the results are compared with published numerical solution by de Vahl Davis (1983), Xin and Quéré (2002), Sharif and Mohammad (2005). All comparisons show good agreement. Similar configurations but with completely heated bottom surface instead of partial heated strip was studied by Corcione (2003) for different width-to-length ratio.

Finding from these literatures provide a solid validation on the effectiveness of CFD numerical analysis in simulating natural convection with heated bottom surface. Regardless of completely or partially heated bottom, correct buoyancy phenomenon of natural convection always can be predicted by mean of CFD numerical analysis.

2.1.2 Natural Convection Heat Transfer Due to Influence of Asymmetrical and Symmetrical Heat Trip

Corvaro and Paroncini (2008) compared natural convection heat transfer results due to the influence of asymmetrical and symmetrical heat strip located on the bottom surface of a square cavity filled by air between experimental analysis and numerical analysis based on commercial finite volume code Fluent 6.2.16. The natural convection induced by a determined length of $\varepsilon = 1/5$ located at two different dimensionless distance, δ from the left side of the cavity: first one is $\delta = 0.4$ and the second one is $\delta = 0.5$. The experimental analysis was carried out through a holographic interferometer and through a 2D PIV system.

For Rayleigh number range from 6 x 10^4 to 3 x 10^5, good agreement was obtained between experimental result and simulated result. Backed by this excellent correlation, Corvaro and Paroncini (2008) concluded that simulated results were remaining accurate for Rayleigh number range from 1 x 10^4 to 8 x10^5.

2.1.3 Natural Convection of Air in Two-dimensional, Rectangular Enclosure with Localized Heating From Below and Asymmetrical Cooling from Sides

Aydin and Yang (2000) investigated numerically natural convection of air in a two-dimensional, rectangular enclosure with localized heating from below and asymmetrical cooling from the sides. Heat source is centrally located on the bottom wall, and four different values of the dimensionless heat sources length, 1/5, 2/5, 3/5 and 4/5 are considered. Solutions for Rayleigh number values from 1x 10^3 to 1x 10^6 are considered. The streamline & isotherm results obtained show excellent agreement with the results in Calcagni, Marsili et al. (2005) study. This validates that the existing commercial code (Fluent) able to deliver accurate results which as per observed in experimental analysis.

2.1.4 Natural Convection in A Cavity with Uniformly and Non-uniformly Heated Bottom Wall, Adiabatic Top Wall and Constant Temperature Cold Vertical Wall

Basak, Roy et al. (2006) performed a numerical study to investigate the steady natural convection flow in a square cavity with uniformly and non-uniformly heated bottom wall, and adiabatic top wall maintaining constant temperature of cold vertical walls. A penalty finite elements method with bi-quadratic rectangular elements has been used to solve the governing mass, momentum and energy equations. For Rayleigh number ranged from 1 x 10^3 to 1 x 10^5 and Prandtl number ranged from 0.7 to 10, consistent results were obtained with respect to continuous and discontinuous Dirichlet boundary conditions. Basak, Roy et al. (2006) concluded that conduction dominant heat transfer modes occurs at Ra ≤ 5 x 10^3 for uniform heating of bottom wall whereas it occurs at Ra ≤ 2 x 10^4 for non-uniform heating. Research carried out by Basak, Roy et al. (2006) is one of the very few investigations available to access the effectiveness of finite elements method numerical study on natural convection. His finding correlates well with simulated results predicted by finite volume method and finite difference method.

2.1.5 Combined Heat Transfer of Radiation and Natural Convection in a Square Cavity Containing Participating Gases

Lari, Baneshi et al. (2011) analyzed the effect of radiative heat transfer on natural convection heat transfer in a square cavity under normal room conditions. Finite Volume Method has been adopted to solve the governing equations of temperature, velocity and heat flux distribution. For radiative transfer prediction in absorbing-emitting media, the discrete ordinates method (DOM) is used. Their radiative-convection model is validated by comparison with test cases solutions from Tan and Howell (1991), Han and Baek (2000) and Mondal and Mishra (2009). Then, the effects of Rayleigh number from 10^2 to 10^6 and optical thickness in a range from 0 to 100 on temperature and velocity distributions and Nusselt numbers are investigated. The results show that radiation plays a significant role on temperature distribution and flow pattern in the cavity even under room conditions with a low temperature difference.

The finding provided by Lari proves that radiation impact should be included in the numerical analysis of natural convection heat transfer even the temperature difference is very small. Sometimes in the studies focused on natural convection, the radiative mode of heat transfer is neglected because of the overwhelming amount of computation resources it requires. However, as the computational resources is getting cheaper and easily available over the last 2 decades, in most of the natural convection heat transfer simulation on engineering problems, coupling radiative mode together other governing equations become viable.

2.2 Turbulence Model

2.2.1 Turbulence Models Research with One-Dimensional Heat Source

Turbulence models play an importance in predicting accurate natural convection phenomenon. Walsh and Leong (2003) assessed the performance of several commonly used numerical models such as k-ε, Renormalized Group k-ε and Reynolds stress model, in predicting heat transfer due to natural convection inside an air-filled cubic cavity. Commercial finite volume code Fluent was utilized in the assessment. In their study, it was found that accurate prediction was not able to be obtained with all three turbulence models for the unsteady convection case, where hot surface is on the bottom. Convectional instabilities lead to chaotic fluid motion making an unsteady numerical solution difficult to achieve.

Similar validation on the effectiveness of turbulence models also performed by Rundle and Lightstone (2007) using commercial CFD code, ANSYS CFX 5.7. Three turbulence models, standard k-ε, Wilcox k-ω and Shear Stress Transport Model (SST) were validated against the experimental data provided by Ampofo and Karayiannis (2003). Results showed that Wilcox k-ω model is the preferred turbulence model for calculating natural convection flows due to its superior accuracy. For standard k-ε model, convergence was unable to be reached with various attempts to aide convergence, including relaxation of time step loop, relaxation of mass equations and variation of grid size.

Sharma, Velusamy et al. (2007) published two literatures about their numerical studies on turbulence natural convection by means of finite volume method with standard k- ε turbulence model. On their first paper, they validated the effect of radiation in air filled rectangular enclosures with aspect ratio is varied from 0.5 to 2.0, heat source located on the entire bottom wall, cold surface for remaining vertical and top walls. Studies have been carried out for a wide range of influencing parameters such as Rayleigh number, enclosure aspect ratio, emissivity of the wall surface and external heat transfer coefficient. It was observed that the flow distribution in the enclosure is influenced by the external heat transfer coefficient and surface radiation for Rayleigh number $\geq 1 \times 10^{11}$. For Rayleigh number $\leq 1 \times 10^{8}$, the radiation reduces natural convection by reducing the effective driving temperature difference leading to a reduction of approximate 25% in the convection Nusselt number. However, the contribution of radiation is of similar order as that of natural convection. Therefore, the total contribution of natural convection and surface radiation is very close to the natural convection contribution with isothermal cold walls. Hence, for numerical analysis of natural convection in a square cavity with bottom heat source and isothermal cold walls, with Rayleigh number ≤ 1 x 10^8, although radiation heating is not enabled, the total Nusselt number is similar with the scenario that enable radiation heating.

In the second literature by Sharma, Velusamy et al. (2007), they presented results of numerical investigation of turbulent natural convection in a square enclosure with localized heating from below and symmetrical cooling from vertical sides, by means of finite volume method. Source length is varied from 20% to 80% of the total width of the bottom wall, with isothermal or isoflux as boundary condition. Top wall and unheated portion of the bottom are considered to be adiabatic, whereas sidewalls are isothermal. Standard k-ε model was used to model turbulence for Rayleigh number from 1 x 10^8 to 1 x 10^{12}. The working fluid is air with Prandtl number, Pr = 0.71. Different with conventional standard k-ε model that requires wall function to represent the viscous sub-layer, Sharma discretized the region close to the walls with sufficiently fine grid to enable integration up to the walls. On the walls, turbulent kinetic energy is set to zero and turbulent energy dissipation, ε is set to infinity. In order to enable numerical implementation where infinity value is not possible, ε is set to 1 x 10^{25} at the walls based on his observation on variation of the average Nusselt number with ε for a typical case of Ra = 1 x 10^{10}, the average Nusselt number remains constant for all the numerical values ≥ 1 x 10^{22}. Therefore, it was justifiable to use 1 x 10^{25} in the simulations. With this approach, the convergence difficulty issue faced by Rundle and Lightstone (2007) was resolved. This extent envelop of numerical analysis on natural convection up to Ra = 1 x 10^{12}. Based on the simulated results obtained and correlated with least square regression analysis, Sharma claimed that functional relationship between the

Nusselt and Rayleigh number is Nu ~ $Ra^{0.334}$ for isothermal heating, whereas for isoflux heating, Nu ~ $Ra^{0.233}$.

Most recently, Saravanan and Sivaraj (2011) further investigated natural convection in an air filled enclosure with a localized nonuniform heat source mounted centrally on the bottom wall. Except for the nonuniform heat source, all other geometries and boundary conditions are the same with Aydin and Yang (2000), Calcagni, Marsili et al. (2005) and Sharif and Mohammad (2005). Effects of the nonuniform heat source in different source nonunifomity parameter, λ and length, ε on heat transform, momentum and energy were studied for Grashof number Gr = 1 x 10^6 and 1 x 10^7. The symmetrical temperature contours and flow fields about the center of the enclosure for uniform heat source are distorted by nonuniform heating. It is found that for Gr = 1 x 10^6, nonuniform heating from the line source enhances the overall heat transfer rate significantly compared to uniform heating of same heat source length, whereas for Gr = 1 x 10^7, its effect is marginal.

2.2.2 Turbulence Model Research for Two/Three-Dimensional Heat Source

For all the aforementioned researches on natural convection, heat source was centrally located at the bottom wall with various length ratios along the horizontal axis. For the sake of numerical simplicity and stability, the geometry of the heat source is one-dimensional; height of the heat source in vertical axis was always remains zero. However, in engineering applications that rely on natural convection, such geometry of heat source is very rare. In most of the applications, the centrally located heat source is normally a three-dimensional

IC package or electronic device with enclosure. Hence, as the cold fluid moves from isothermal vertical walls towards center, some of the fluid will encounter the vertical surfaces of the heat source before reaching the top surface. Therefore, numerical analysis on one-dimensional heat source is not sufficient to simulate the flow physic along the vertical surfaces of a two/three-dimensional heat source, for example, the momentum lose that affect the buoyancy flow afterward.

Kuznetsov and Sheremet (2010) had investigated the effect of finite thickness heat-conducting walls at local two-dimensional heating at bottom for turbulent natural convection in a rectangular enclosure; all walls were adiabatic except the left vertical wall. Similar with many numerical studies on one-dimensional heat source, standard two-equation k-ε model with wall functional, along with Boussinesq approximation was used. Attention was given to the effects of the Grashof number $1 \times 10^8 \leq Gr \leq 1 \times 10^{10}$, transient factor $0 < \tau < 1000$ and thermal conductivity ratio $k_{2,1} = 5.7 \times 10^{-4}$, 6.8×10^{-5}, where $k_{2,1}$ is ratio between thermal conductivity of gas and solid wall. From the simulation results, one convective cell and three recirculation zones are formed in the gas cavity. Focus has been given to the basic convective cell due to buoyancy force and the emerging of secondary circulation between the heat source and the bottom side of the right wall. As the Gr reach 5×10^9, an appearance of circulation is seen above the heat source. The increase of Grashof number is also reflected in less intensive heating of the bottom layers of the gas cavity and reduction of thermal boundary layer thickness along the left vertical wall of the heat source. Kuznetsov's study validated that heat source with finite thickness (a.k.a two/three-dimension geometry) can essentially modify the formation of convective circulation cells, temperature fields and flow conditions. Additionally, it leads to the development of thermal boundary layer along the vertical wall of the heat source. The mathematics formulation developed & validated on one-dimensional heat source are still applicable on predicting the turbulent natural convection on two/three-dimensional heat source.

2.3 Mathematical Formulation of Navier-Stokes

The steady conjugate heat transfer of natural convection and conduction of solid in an enclosure had been carried out. The flow regime is considered in transitional flow region; therefore Reynolds-Averaged Navier-Stokes (RANS) is adopted. Due to the density different induced by buoyancy flow is affecting both momentum and energy transport, the RANS equation and energy conservation equations have to be coupled and solved simultaneously. This requirement poses a great challenge on numerical methods in term of computational resource and also numerical stability.

2.3.1 Solving of RANS Equation, Energy Conservation Equation and Ideal Gas Law

Partial Different Equations (PDEs) listed below are RANS and energy conservation equations with Boussinesq approximation in CFD numerical analysis respectively.

Continuity Equation

$$\frac{\partial(\rho U)}{\partial x} + \frac{\partial(\rho V)}{\partial y} + \frac{\partial(\rho W)}{\partial z} = 0 \tag{1}$$

X-Momentum Equation

$$
\begin{aligned}
\rho U \frac{\partial U}{\partial x} + \rho V \frac{\partial U}{\partial y} + \rho W \frac{\partial U}{\partial z}
&= \rho g_x - \frac{\partial P^*}{\partial x} + S_{DR} + S_\omega + \frac{\partial}{\partial x}\left[(\mu + \mu_t)\frac{\partial U}{\partial x}\right] \\
&+ \frac{\partial}{\partial y}\left[(\mu + \mu_t)\left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x}\right)\right] + \frac{\partial}{\partial z}\left[(\mu + \mu_t)\left(\frac{\partial U}{\partial z} + \frac{\partial W}{\partial x}\right)\right]
\end{aligned} \tag{2}
$$

Y-Momentum Equation

$$
\begin{aligned}
\rho U \frac{\partial V}{\partial x} + \rho V \frac{\partial V}{\partial y} + \rho W \frac{\partial V}{\partial z}
&= \rho g_y - \frac{\partial P^*}{\partial y} + S_{DR} + S_\omega + \frac{\partial}{\partial x}\left[(\mu + \mu_t)\left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x}\right)\right] \\
&+ \frac{\partial}{\partial y}\left[2(\mu + \mu_t)\frac{\partial V}{\partial y}\right] + \frac{\partial}{\partial z}\left[(\mu + \mu_t)\left(\frac{\partial V}{\partial z} + \frac{\partial W}{\partial y}\right)\right]
\end{aligned} \tag{3}
$$

Z-Momentum Equation

$$
\begin{aligned}
\rho U \frac{\partial W}{\partial x} + \rho V \frac{\partial W}{\partial y} + \rho W \frac{\partial W}{\partial z}
&= \rho g_z - \frac{\partial P^*}{\partial z} + S_{DR} + S_\omega + \frac{\partial}{\partial x}\left[(\mu + \mu_t)\left(\frac{\partial U}{\partial z} + \frac{\partial W}{\partial x}\right)\right] \\
&+ \frac{\partial}{\partial y}\left[(\mu + \mu_t)\left(\frac{\partial V}{\partial z} + \frac{\partial W}{\partial y}\right)\right] + \frac{\partial}{\partial z}\left[2(\mu + \mu_t)\frac{\partial W}{\partial z}\right]
\end{aligned} \tag{4}
$$