SIMULATION OF MIXED CONVECTION IN A CHANNEL WITH A CAVITY HEATED FROM DIFFERENT SIDE USING FLUENT

MOHAMMAD HAIQAL BIN SUPAAT

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

SUPERVISOR DECLARATION

"I hereby declare that I hacve read this thesis and in my opinion this report is sufficient in terms of scope and quality for award of degree of Bachelor of Mechanical Engineering (Thermal -Fluid)

Signature	:	....
Supervisor	:	En Mohd Noor Asril B. Saadun
Date	:	28 June 2013

SIMULATION OF MIXED CONVECTION IN A CHANNEL WITH A CAVITY HEATED FROM DIFFERENT SIDE USING FLUENT

MOHAMMAD HAIQAL B. SUPAAT

This report is submitted in partial fulfillment of the requirement for the degree of Bachelor Mechanical Engineering with Honors' (Thermal-Fluids)

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

June 2013

DECLARATION

"I hereby declare that the work in this report is my own expect for summaries and quotations which have been duly acknowledge."

Signature:	…………………………………
Author:	Mohammad Haiqal B. Supaat
Date:	28 June 2013

Dedicated to my beloved parents and all technical support team

in completing this thesis

ACKNOWLEDGEMENT

I am grateful to be able completing this thesis title “ Simulation of mixed convection in a channel with a cavity heated from different side using Fluent “ as a requirement in completing my first degree. This program taught me lot of things with many added value gained in this period.

I would like to show my appreciation toward my supervisor, En Mohd Noor Asril Saadun, panel 1, Dr. Cheng See Yuan during FYP 1, Dr. Faiz Redza B. Ramli during FTP 2 for giving a frequent technical advice in completing this assessment. Without cooperation from many parties, this project was impossible to be done on time and the objective couldn’t be achieved. My supervisor has guide in contributing the idea and procedure to hands this project from day one until the end.

Meanwhile, cooperation from UTeM technical staff and lectures also contribute into completing this project. Staff really cooperates in giving advice and information that needed during completing this project especially in technical support and formal dealing since this project require additional knowledge by visiting and learning from outside rather that UTeM only.

Last but not least, friends are the backbone for me on sprit supply and support. Commitment with lack of mis-judge attitude shown by project members really deserves appreciation. I wish, this project will be useful for further reference in the future.

ABSTRACT

This project requires a simulation to obtain the data in animation or drawing graphs. Computational Fluid Dynamic (CFD) is one of the methods used in industry and in education to solve problems related to fluid flow in closed channel. Through simulation, the type of fluid flow, streamlines and vortices can be seen to determine the ability of the airway momentum that produced through a rectangular shape to remove contaminants that are trapped in it. In addition, the project is also looking at the effects of conductive heat transfer that being supply at the sided wall to the water flow and it will be seen whether able to help the flow of air to remove pollutants altogether. High skills in understanding CFD with commands validation were begin with the meshing selection based on dependency test, journal study on initialization method, analysis from the defined problem in properties and physical value of the element that being part of the problem. Simulation shown that the left sided heat convective region of the cavity wall provide higher pressure gradient drop which contribute into removal rate of the contaminants. Aspect ratio of the cavity towards the removal rate were one of the interest in this study and as the AR increase, the removal rate also increase and this project subject to left-sided heat source with Aspect Ratio of 4 as the best design in term of removal, number of vortices form and streamlines pattern for steady flow simulation.

ABSTRAK

Projek ini memerlukan simulasi untuk mendapatkan data dalam animasi atau lukisan graf. Permodelan Bendalir Berkomputer (CFD) adalah salah satu kaedah yang digunakan dalam industri dan dalam pendidikan untuk menyelesaikan masalah yang berkaitan dengan aliran bendalir dalam saluran tertutup. Melalui simulasi, jenis aliran cecair, garis alir, dan vorteks boleh dilihat untuk menentukan keupayaan momentum air yang dihasilkan melalui bentuk segi empat tepat untuk membuang bahan cemar yang terperangkap di dalamnya. Di samping itu, projek ini juga melihat kesan pemindahan haba konduktif yang disalurkan pada dinding kaviti kepada aliran air dan ia akan dilihat sama ada dapat membantu aliran udara untuk mengeluarkan benda asing yang terdapat di dalamnya. Kemahiran yang tinggi dalam memahami CFD dengan pengesahan arahan telah bermula dengan bersirat pemilihan berdasarkan ujian pergantungan, kajian jurnal mengenai kaedah pengawalan, analisis dari masalah yang ditakrifkan dalam hartanah dan nilai fizikal elemen yang menjadi sebahagian daripada masalah. Simulasi menunjukkan bahawa kesan konduktif terhadap dinding kaviti sebelah kiri memberikan tekanan yang lebih tinggi terhadap kecerunan perubahan tekanan yang menyumbang kepada kadar penyingkiran bahan cemar. Nisbah aspek rongga terhadap kadar penyingkiran adalah salah satu kepentingan dalam kajian ini dan selari dengan peningkatan AR, kadar penyingkiran juga meningkat dan projek ini mendapati bahawa sumber haba sebelah kiri dengan Nisbah Aspek 4 sebagai reka bentuk yang terbaik dari segi penyingkiran, beberapa bentuk vorteks dan memperkemaskan corak simulasi aliran mantap.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dyanamic
NSE	Navier Stoke Equation
PIV	Particle Image Velocimetry
AR	Aspect Ratio
Ra	Rayleigh Number
Nu	Nusselt Number
Re	Reynold Number
Pr	Prantl Number

CHAPTER 1

INTRODUCTION

1.0 Introduction

This project aim to hands the best possibility on contaminant removal in cavity which involving the operation of fluid dynamic solving problem. Interest in fluid dynamic study embark into widely application solver such piping, air conditioning system, oil and gas, aeronautic and medicine technology development. Deep interest applied fluid dynamic as a control measurement in nuclear reactor which in the early stages with continuous research and development. Fluid acts as the working fluid in nuclear reactor which probable for future operation addition of goods [1]. Heat transfer on the meanwhile, was subjected as the addition heat addition to the system in order to determine the effect which helps for more productive operation involving fluid flow to removing contaminants. Contaminants hence, being produces in some application of closed fluid system and produced either by the friction effect by fluid towards walls or the content of excessive material transported by the working fluid itself. In some application such piping transport for food and gasoline, inert substance are considered as the unnecessary content which need to be avoid from mixing with the required material. Hence, the idea of forming a cavity mostly at the joint of the piping system became region of study interest in maintenance development nowadays. This project hence, focus to only one type of cavity only and being differentiate by different parameter such cavity ratio and degree of temperature supplied. Problem involving fluid dynamics can be solved in three methods which is theoretical, experimental and computational approach. This project focuses on the simulation method and being compared to experimental approach which refer to other paper similar to the problem. Computational fluid dynamic being chosen as the method to solve the problem due to it's flexibility on problem solving involving fluid dynamics and mathematical calculation as defined earlier. Solver that being used in this project was FLUENT inside ANSYS software that helps to make iteration in finding the solution. Element that consist in FLUENT that need to be shown in this project is forming streamlines and vorticity of the fluid flow to seen the pattern and the contaminant rate of removal. Shape of the cavity that being used is fixed on one shape only which is rectangular and heat sources situated at the side of the cavity's wall. Advance understanding in computational of fluid dynamic proved by Nursalasawati which study a different method on mathematical approach that applied by CFD and the improved mathematical modelling capable finds alternative ways on pressure handling in flow [2].which means that future development will produce accuracy on apply CFD and practical for fluid solver approach rather than experimental and theoretical which contribute to high cost spending and time taken. Validation was the most important part that should be consider in using of CFD for simulation so that the analysis will be based on the real stated problem statement.

1.1 Problem Statement

Field of study in fluid flow was widely being applied in industry especially in maintenance and innovation section. Application involving fluid flow as the medium such air conditioning and piping system needs this pattern study in order to identify the pattern effect that suitable to be use in any application. Mixed convection flow and heat transfer embark lots of interest in engineering study either for cooling and heating flow. Study in fluid with convective effect had been done by Oranzio which study the heated wall position on mixed convection in a channel with an open cavity. From the study, it concluded that the cavity ratio plays a main role in determining the streamlines and isotherm pattern with the ascending order of Nusselt number. Plus, this project determined that the temperature of source will decrease as the Reynold and Richardson number increase [3]. Time was the main parameter that needs to be considered in this project since the maintenance either in small or heavy industry required precision and efficiency in operation. Air conditioning bulking system for example, need a fast maintenance to avoid injury time while maintenance on going in order to fulfil continuous human comfort requirement. Heat convection heat is the innovative method to help the removal process produced in short time as possible. Convection effect and drag force towards contaminants determine the direction of the removal due to the streamlines of the flow. Working fluid also needs to be consider because different type embark different result and effect. In this project, liquid were selected as the working medium to satisfy the properties of contaminant because, if gas are preferred, the density different between gas and contaminant will be too low and contribute difficulty in determining the pattern result because of the time taken that will be too short in simulation. This simulation will be compare to experimental approach and due to the limitation in procedure set up for experiment, the analysis were subjected to 2-D, steady, incompressible and low Reynold number so that the result can be collinear compare to experimental.

1.2 Objectives

The objectives of this project are:

1) To determine flow pattern in the different side heat of cavity.
2) To investigate the macroscopic rigid particles removal in term of percentage due to heat effect.
3) To differentiate the outcome result using of different cavity ratio.
4) To determine the most affected situated heat source for rectangular shape on the contaminant removal.

1.3 Scopes of Project

1) FLUENT will be used to simulate the effect of flow characteristic in the cavity.
2) The test case is flow in cavity heated from different side, focusing on streamline plots, isotherm and time taken for particle removal.
3) The type of flow that being use in analysis was laminar and the flow was assume as incompressible for analysis.
4) Graphical plotted should be done to analyse the display data generated from the software.
5) Experimental conducted to ensure simulation could be validate.

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

Previous study guides the finding of this project on understanding the terms of keyword and the flow in making a simulation. Parameters that consist in this project such flow properties, heat addition and boundary validation need a solid understanding and by previous research the capability of the FLUENT in solving problem will be synchronise to the problem statement and objectives that need to be carried out. Journal, books, article and internet was the source in finding the previous study for references in completing this project. From the research, it found that lot of study have been made before by other students to study the problem involving fluid dynamics. This study helps in this project to validate and justify any method or act that being taken during methodology. This review was taken with aims to understand:

a) The heat transfer principle to be added and used in this project with the parameters that need to be considered.
b) The elements consist in FLUENT for data validation and governing equation determination to state the problem.
c) The definition of the cavity and the contaminant that being used in this project.
d) The working fluid that being used in this project and it's properties with validation in FLUENT use purpose.
e) Deep geometry selection of the cavity and heated wall.

f) The previous limitation and plan to improve and not repeat the similar limitation.

2.1 Heat Convection on Flow

2.1.1 Introduction

Heat defines as form of energy that can be transferred from one system to another as a result of the temperature different. Interest in heat study, heat transfer hence being focused of the rate of heat that being transfer. In matter of time, this study explains on the behaviour of the heat while in the process. Heat can be transfer either by conduction, convection and radiation. The different among these three types of heat transfer is the medium of the process. Heat conduction could take place in solids, liquids, or gases and the principle stated that the heat being transferred from more energetic particles of a substance to the less region particles. Heat convection hence, transfer heat between solid and the adjacent liquid or gas that is in motion and involve combination of both conduction and fluid motion. Convection can be happened in two type of transfer which is natural convection and forced convection. The last type of heat transfer is radiation which classified as the fastest rate of transfer since it apply concept of electromagnetic wave emitter and do not require the present of intermediary medium such convection and conduction.

2.1.2 Heat Parameter

This paper focused on the heat convection effect on fluid flow in the cavity channel with interest geometry on vertical and horizontal plate heat source. Shang, 1993 studied on the laminar free convection with consideration of the thermophysical properties and found that with constant ambient temperature (T_{∞}) it resulting the descending order on Prantl value as the heat source being increase [4]. Prantl number was the parameter in determining the Nusselt number which rate the heat transferred. Hady then back in 1995 investigate the heat convection effect on the horizontal plate with heat source being manipulated with three different value of Prantl and found that the flow boundary layer shown the behaviour that with lower Prantl number, the rate were faster compare to high Prantl value [5]. Shi and Vafai then carried out a mixed convection in an obstructed cavity with heated horizontal walls. Water properties as an ambient medium had been used by Sinha and using working fluid of water at low temperature where the relation between density and temperature are nonlinear and other water properties such viscosity and thermal conductivity have been consider constant [6]. Vajravelu and Sastri then making reconsideration based on Sinha study by using accurate relation between water density and temperature and ignoring other variation [7].

Manca have studied the location of the heat source to the flowing fluid. It found that with a constant heat flux. Simulation gained was from the heat convective in open cavity with the horizontal plate being heated. The result determined that the source temperature experience the decreasing as Reynolds number and Richardson number increase and the recirculation pattern formed inside the cavity [8]. From this paper there is possibility either the contaminant can be remove faster or trapped due to the circular motion. Hence, this project will propose the best design of rectangular cavity with the appropriate length ratio of the sided wall.

2.2 Computational Fluid Dynamic

2.2.1 Introduction

Computational Fluid Dynamics (CFD) defines as the mathematical approach in defines the problems involving fluid dynamics. Figure 1 below illustrated the flow in solving problem by applying the Computational approach. The problem needs to be stated in mathematical modelling so that the computer capable to making analysis based on the iteration procedure. The governing equation then could be form from the descript assumption and the parameter consist in the problem and determine the character in form of sequence formulation [9]. The next step is the selection of the grid by discretisation. The mathematical formulation in finite different determines the analytical solution to the problem. In Order to be analyse by computational approach, the equation need to be convert to the algebraic partial different so that the numerical equation can be solve using the solver situated in CFD. The result can be represent in various form such plotting, animation, contour and simulation by using CFD.

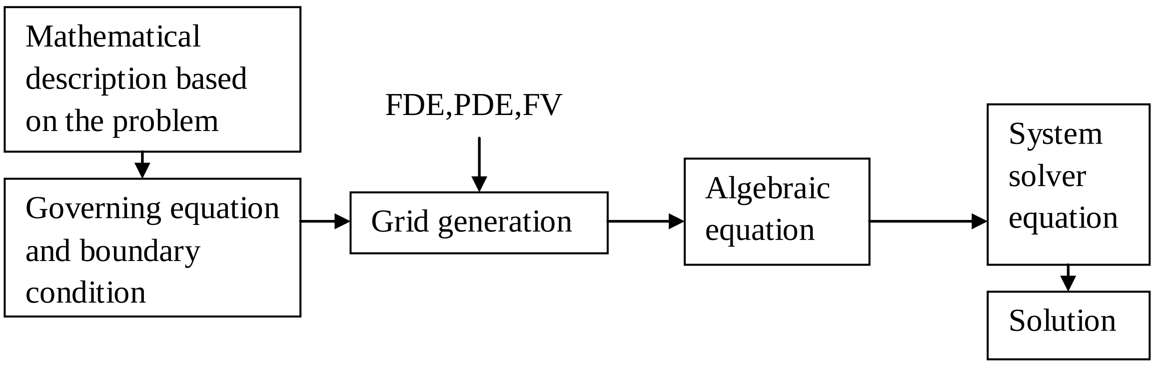

Figure 2.1: Mathematical conversion for CFD

From the previous study, variety numerical approach on defining the flow problem had been carried out. Approach such Galerkin Spectral method [9], high order Finite Difference [10] and Chebyshev schemes [11]. This development of define of flow method will embark on the positive acceptable method of using CFD as efficient as the real experimental approach in the future.

2.2.2 Navier Stokes Equation

Navier stokes equation (NSE) being used to represent the fluid properties based on the principle of mass conservation, Newton second law and energy conservation. By apply modification of NSE, the principle will be represent in form of continuity equation, momentum equation and energy equation. This equation defines the fluid properties consist on problem such velocity, pressure, fluid density, fluid viscosity, gravity and heat flux [2].

From NSE [2],

Continuity equation:

$$\bar{\upsilon} \cdot u = 0 \tag{2.1}$$

Conservation of momentum:

$$u_t + u \cdot \bar{\upsilon}u = \frac{1}{\rho}\bar{\upsilon} \cdot p + \nu\, \bar{\upsilon}^2 u + f \tag{2.2}$$

Conservation of energy:

$$E_t + \bar{\upsilon}(Eu) = q = p \,.\, \bar{\upsilon}u + f\,.u \tag{2.3}$$

This equation which generated by the NSE are differ due to the physical modelling of each defined problem and being simplified later until the algebraic numerical reach as simplest as possible for numerical order to CFD. According to the mathematical define of the problem, governing equation which formed due to the NSE then can be formulate. CFD however demand numerical solution which means that the assumption governing gained in analytical NSE method need to be modified into partial differential [10] which then numerically solve by the solver inside CFD (FLUENT).

2.2.3 Streamlines and vorticity

This project focused on one shape of cavity which is rectangular. From the previous study, many projects have been done with this type of shape and other shape to determine the removal rate. Kuo and Chang meanwhile, have made a study on contaminants removal through the circular cylinder by using various value of Reynolds number [12]. Their finding provides comparison with the other study made by them by using of rectangular shape with effect on the horizontal plate and the different was determined from the streamline behaviour inside the cavity [13]. Their finding found that there is a different among the shapes with conclusion that the cavity provide different flow pattern. Hence, this project determined to understand only one shape which is rectangular and make modification on cavity ratio effect to the streamlines produced.

Other study that supports Kuo and Chang finding is experimental approach made by C.Ozalp. His paper runs the experiment with rectangular, triangular and semicircular by using method of Particle Image Velocimetry (PIV) to capture the seeding removal inside the cavity [14]. This paper hence embark toward many more research with CFD base simulation to be compare with the pattern gained by Ozalp. Other research determined the solution with the involvement of the heat conduction effect. The streamlines and vorticity will be defines into mixed region of temperature and the effect on streamlines function gained. Avedissian carried out the paper that