SUPERVISOR DECLARATION

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

Signature:	…………………………….…...
Supervisor:	Mohd Noor Asril Bin Saadun
Date:	28 June 2013

CONTAMINANT REMOVAL FOR DIFFERENT GEOMETRY OF CAVITIES WITH HIGH REYNOLDS NUMBER USING FLUENT

MUHAMMAD ZULHAKIM BIN SHARUDIN

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

MAY 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: Muhammad Zulhakim Bin Sharudin
Date: 28 June 2013

Special for my beloved

Father and Mother,

Sharudin bin Che Ya and Zainab binti Razali

Family and Friends

ACKNOWLEDGEMENT

I would like to express my sincere gratitude to my loving parents, Mr. Sharudin bin Che' Ya and Mrs. Zainab binti Razali. Not to forget are my siblings and relatives who have given me morale and economic supports during the research. My parents have given their love, dream and sacrifice throughout my life. I am really thankful for their sacrifice, patience, and understanding that were much-needed to make this work possible. Their sacrifice had inspired me from the day I learned to read and write until what I have become now. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to achieve my dreams

A great thanks are due to my supervisor, Mr. Mohd Noor Asril bin Saadun and my panel. Dr. Shamsul Anuar bin Shamsudin for his invaluable guidance, continuous encouragement and constant support in making this research possible. I really appreciate his guidance from the initial to the final level that enabled me to develop an understanding of this research thoroughly. Without his advice and assistance it would be a lot tougher to complete. I also sincerely thanks for the time spent proofreading and correcting my mistakes. I must acknowledge as well the many friends, lectures, technicians, and many others who assisted, advised, and supported me.

Lastly I would like to thank any person who contributes to my final year project directly or indirectly. Their comments and suggestions were crucial for the successful completion of the studies.

ABSTRACT

A preliminary Computational Fluid Dynamics (CFD) study on the effect of high Reynolds number flows toward the contaminant removal in the cavity was carried out. Based on the simulation of the CFD, the two dimensional (2D) model analyses of the flow characteristics was done using the numerical solution of the Navier-Stokes equations based on the finite difference method. The flow characteristics in the cavity and the driven flow were modeled via turbulence model. The attention of this study focuses on the effect of different high Reynolds number flow in removing the contaminant in the cavity. As the parameter of the cavity, the different type of geometry and different aspect ratio of the geometry was used. The result shows the comparison in term of the flow visualization between each model with the different parameter used.

ABSTRAK

Kajian awal Bendalir Dinamik Berkomputer (CFD) yang dijalankan mengenai kesan aliran cecair dengan nombor Reynolds tinggi terhadap penyingkiran bahan cemar dalam ruang rongga. Berdasarkan simulasi CFD, model analisis dua dimensi dengan ciri-ciri aliran yang telah dilakukan dengan menggunakan penyelesaian berangka Navier-Stokes berdasarkan kaedah pembahagian takterhingga. Ciri-ciri aliran dalam rongga dan aliran terdorong dimodelkan melalui model persamaan gelora. Skop kajian ini tertumpukepada kesan perbezaan aliran nombor Reynolds tinggi dalam penyingkiran bahan cemar dalam rongga. Sebagai parameter rongga, jenis geometri yang berbeza dan aspek nisbah geometri yang berbeza telah digunakan dalam kajian ini. Keputusan menunjukkan perbandingan dari segi visualisasi aliran antara setiap model digunakan.dengan parameter yang berbeza

TABLE OF CONTENTS

CHAPTER	CONTENTS	PAGES

CHAPTER	CONTENTS	PAGES

LIST OF TABLES

LIST OF FIGURES

NO.	TITLE	PAGES

LIST OF APPENDICES

CHAPTER I

INTRODUCTION

1.1 BACKGROUND

In the natural environment such as lakes, rivers and drains, the same situation as the presence of cavity can clearly be seen. The points of interest are at the flows deal with the contaminant problem that arises in the cavities. The toxic, the suspended sediment and pollutant may accumulate which can the problem the flow and harmful to the environment. For the instant, the ejection of the contaminant from the cavity need to be done and can bring the solution to the problems.

In the term of flow, the incoming boundary layer of the flow in the leading edge of the cavity may be in the laminar or turbulent flow. But most of the time, real situations deal with turbulence flows. The Reynolds number will determine the kind of flow that creates in the streamline.

Most engineering applications dealing with the flow in a channel are encountered with the contaminant in the cavity. An example is the pipeline of fluid. The intersection of each joint of the pipeline especially at welded area of the connection is the location of the cavity. The particles of contaminant contained in the fluid flow will stuck and accumulate in this cavity. By then, the contaminant had to be removed because it can create a problem in the pureness of the fluid and change the flow of the fluid in terms of pressure and velocity.

The solution to the contaminant removal can be done by using pigging technique in the pipeline but it cannot be used in small diameter pipelines and in the long continuous pipelines. The next alternative is the hydrodynamic cleaning technique that uses the flow itself as the medium to remove the contaminant in the specific flow conditions. This project tries to study the contaminant removal using specific flow conditions by the Computational Fluid Dynamics (CFD) analysis.

1.2 PROBLEM STATEMENT

One of the alternative methods of cleaning process of contaminant in pipelines that has widely used this day is by hydrodynamics cleaning. The formation of re-circulating vortices in the cavities is different and it leads to either enhancing or preventing the removal of contaminant based on the geometry and aspect ratio of the cavities[1]. It also depends on the type of flow and Reynolds numbers of the flow[2]. Many researchers are investigating this kind of hydrodynamic contaminant removal but most of them are reported to use the laminar boundary layer upstream or low Reynolds number flow [3]. In a real situation, the process involves a turbulent incoming flow boundary condition in the leading edge of the stream. The experiment and simulation studies for flows with high Reynolds number are still lacking. The work will study the effects of the hydrodynamic removal contaminant using high Reynolds number of flow based on three different geometry of cavity and each of the geometry have three different aspect ratios. As the alternative study approach to the high cost experimental study, this project will be done on fully computational scheme of CFD using ANSYS FLUENT software to provides the extensive numerical study and result.

1.3 OBJECTIVE

The objectives of this study are:

1. To investigate the effect of high Reynolds number flow on hydrodynamic removal of contaminants.
2. To understand the flow characteristics of high Reynolds number in the different geometry of cavities.
3. To determine the relationship between the effect of high Reynolds numbers with different aspect ratio of the cavities.

1.4 SCOPE OF STUDY

The scopes of this study are:

1. Analysis of the effect of flow characteristic by using FLUENT.
2. Study on three different geometry of cavities which is semi-circular, triangular, and rectangular.
3. Study on the three different aspect ratio of each geometry cavity which is 2, 3, and 4.
4. Concentrate on the two dimensional (2D) analysis with the incompressible and the high Reynolds number of flow.

CHAPTER II

LITERATURE REVIEW

2.1 INTRODUCTION

The literature review searches for existing information of the related topics that have been discovered by other researchers to be used as a guide and reference for the current work. All the information and reference that used in this study was to be used as working steps in order to provide the added value and produce good additive to this study. This study sources also based on the same aspects or analogous of the previous study that taken the information from the example of journals, internet resources, experiments report and references book.

In this chapter, the study of contaminant removal for the different geometry of cavity used, attached and elaborated all the information to this chapter by regarding the history of the Computational Fluid Dynamics (CFD), high Reynolds number of flow, removing the contaminant, the geometry of cavities and the aspect ratio of the cavities.

2.2 HISTORY OF THE COMPUTATIONAL FLUID DYNAMICS (CFD)

From the early 1970's, CFD was started during the time. The complex combination between the mathematical calculation and physic analysis, create the simulation of fluid flows with the extension of computer sciences. The beginning of CFD was triggered by the sources availability of computer. In the advance of CFD, it increased with the connection of the evolution of the supercomputer technology.

The transonic flow has become among the first application used CFD as the method based on the non-linear potential equation solution. On the early 1980's, the first study of two dimensional (2D) analysis has been solved and then the study of three dimensional (3D) equation analysis started to be introduced in CFD. The rapid increasing of the computer technology created the faster supercomputer that can be possibly calculated and computed the numerical equation of the flows.

In the middle of 1980's, CFD started to shift the focus to the significant simulation of viscous flows by the Navier-Stokes Equations that highly demanding at that time. The evolutions of CFD also take the turbulence model into the new chapter of study with the different degree of numerical complexity and accuracy. By the time the leading edge in turbulence modeling was represented by the Large Eddy Simulation (LES) and the Direct Numerical Simulation (DNS), the usable of those approach is still less in the engineering application.

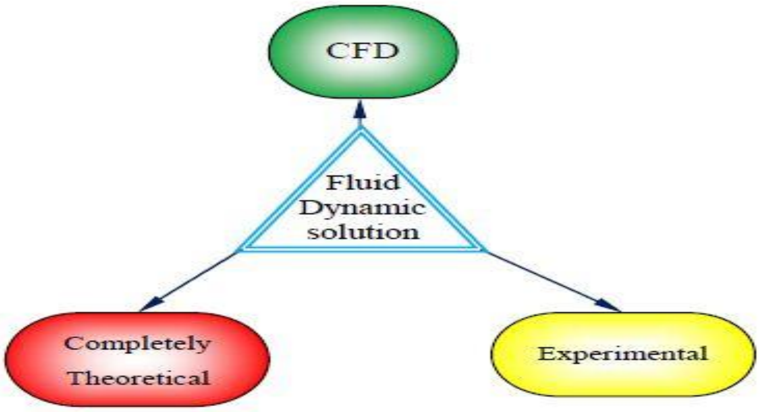

Figure 2.1: The relation in between the Computational Fluid Dynamics.

For this present day, many numerical techniques were developed for CFD is used in solving the study in the fields of aircraft, turbomachinery, car, and ship design. CFD is also applied in, astrophysics, meteorology, and oceanography in oil recovery. Thus, CFD becoming the important design and substantial research tool in engineering. The complex geometry cases can be treating due to the advance of numerical method solution and computer technology. By taken account those thing, the simulation of viscous flows with large scale size can be completed in the short period of time.

CFD is method that related to the solution of the equation of motion of the fluid and also concerned with the interaction of the fluid with the solid bodies. The equation involved the Euler equation that is the equation of motion of inviscid fluid and the Navier-Stokes equation that is for the viscous fluid as it well-known as the governing equations. There are three fundamental principles that need to be followed to govern the physical aspect of any fluid flows. These are that the:

1. Mass is conserved.
2. Energy is conserved.
3. Newton’s second Law is observed.

All the principle is expressed in terms of partial differential equations of mathematical modeling. In the other meaning, CFD is the part of replacing partial differential governing equations of the fluid flow with the numbers and specific these numbers in the space and time to obtain the complete numerical description of the flow field. The objective of most engineering analysis in the close form or open form is in quantitative description of the problem such as numbers.

2.3 HIGH REYNOLDS NUMBER FLOW

Most flows experience some turbulence flow. Some of them were exhibiting with very high Reynolds numbers and some of them were not high. Such as the atmosphere of earth has a condition of extremely high Reynolds number. It is the same with many engineering applications on the aircraft, car, ship and also the pipeline flow. In order to study the effect of the high Reynolds number toward the contaminant removal, the high Reynolds numbers needs to be better understood[6].

Nowadays, a lot of interest in the study have push CFD to used the higher Reynolds number in the studies involve with the flow such as the wind loading on structures, controlling the dispersion of the contaminant inside a compound and characterizing the flow in the engineering application environment. Many classic works demonstrated the separation of the flow, stationary vortices structures, and the vortex shedding. All of these in were difficult to model well[6][7][9][10].

For creating a higher Reynolds number flow, it means that there is need to have larger ratios between the smallest scale in the flow and the higher scale of the flows. Classically, there have some limitation in creating a quality high Reynolds number flow in the experiment such as the lack of the apparatus that can generate the turbulence flows and the low availability of measurement techniques. By then, using the simulation of CFD is the better option for the study of high Reynolds number compared to the conduct the experiments.

In the previous case study of contaminant, the removal of the contaminant, the rapid washout is highly desirable. It has achieved the high flow rates and resulting in high Reynolds number flow. Reynolds number linearly increased with the increasing of the flow rate. Principally, the effect of turbulent flow patterns for high Reynolds number flows, remain similar for different flow rates leading to similar transport pattern in example of the air flow characteristics. At the level of turbulence, airflow profiles of the turbulent flow will only change in their magnitude not in the term of shapes[11].

For high Reynolds number flow, the inertial effect of the flow will be the main factor in determine the contaminant removal. Therefore, the inlet velocity boundary conditions need to be accurate. It is necessary to carefully model the flow at atmospheric Reynolds number using higher accuracy but at the same time readily achievable to reproduce the desired features of the high Reynolds number flow. It also included the grid resolution, technique to systematically relaxing the grid and a few of modeling assumptions to characterize the fidelity[12].

In this contaminant removal for different geometry of cavity with high Reynolds number study, the high Reynolds number and turbulent flow is the most important since it will bring the effect to the modeling of the simulation and in the same time it will affect the result. The properties of high Reynolds number turbulent flow are connected with the transport dynamics of the large scales. Therefore it is important to focus on computing the resources on captured the scales accurately. In the perspective of application on the hydrodynamic contaminant removal, accurate and time dependent may be all it needed and by choosing the grid size for a uniform fashion using the boundary in between the large scales and the inertial range is necessary[13].

2.4 REMOVING THE CONTAMINANT

Drag force is the force component that resistant to the opposite direction of the flow velocity. In other meaning, the drag force is the force of flow friction over a body and it is increasing with the increase of speed. For the essential, the drag force on the any type of body is proportional to the square of relative speed in between the fluid and the body and also proportional to the fluid density.

The drag force coefficient in the curtain body depends on the direction of the fluid flow and the Reynolds number. The variation of the Reynolds number within the range of interest is usually small but depending on the type of study. For the high Reynolds number, the drag force coefficient cannot treat as the constant all the time because it is dependent to the direction of the flow as the high Reynolds number will create the turbulence flow and direction of the flow is not consistence. The streamline body and the boundary layer around the body that attached to the surface of the body will cause a low drag force coefficient. The high drag force will result the wake.

The drag force coefficient of the object such as particle and contaminant cannot be constant but it is a function of the Reynolds number. At the high Reynolds number, the flow around the object will change from the transition to the turbulent especially at the point where separation from the surface object. With this flow separation, the Reynolds number will be high for small object and low viscosity fluid. The drag force coefficient is different in between the type of the fluid.

There is study of the hydrodynamic resistance to particle transportation. It research for the axisymetric creeping flow caused by the particle migrating in the cavity. From the drag force on the particle was calculated, it found that the particle experiencing a local maximum drag force larger than the particle that located in the center of cavity when passing through an aperture. Other than that, the drag force also greater than the particle near the closed end of the cavity. Additional of the result, for connected cavities, the drag force is smaller than the corresponding system.[19]

Tsong et el. (2006) has study about the contaminant removal from the cavity. It has explaining the detail behavior of solid particles in lid-driven cavity flow in between the size of micro to macro of the particles in term of experimental result[20]. The connection and interaction between fluid and solid in the nature was complicated causes the study of it is still less. The formation of the vortices in the cavity by the flow was believed that it can maximizing or minimizing or also can preventing the contaminant removal depends on the parameter of flow and also the characteristics of the contaminant itself[3].

This work focuses on removing the contaminant by using the high Reynolds number water flows and study the relationship of the drag forces, particles that contain in the water flow and also how it link to the removing of the contaminant by the formation of vortices create by the flow at the cavity.

2.5 GEOMETRY OF CAVITY

Cavity is the hollow space in between the area within the body or surface[14]. In the channel or piping, the cavity can be observed in its curtain surface or body. The presence of cavity usually cannot be avoided because a cavity is created when it comes to the interconnection between two connecting pipes. Sometime, in welding joint also created the cavity

Studies have been conducted by used flow in the boundary layer as the study to the cavity problem case. The solutions to the problems can be implemented in the real world such as oil and gas piping system, aeronautical and automobile part, and building structural. Each of the cavity is different in shape and geometry and bring the different kind of problem occur based on the differential situation. Most of the past study and research, the rectangular geometry was the favorite kind of geometry shape for the cavity selection of studies[4][3][5][15].