KESAN HABA TERHADAP MENDAKAN DALAM RONGGA YANG BEREBEZA BENTUK
(THE EFFECTS OF HEAT FOR CONTAMINANT REMOVAL FROM DIFFERENT SHAPES OF CAVITY)

MOHD NOOR ASRIL BIN SAADUN

RESEARCH VOTE NO:
PJP/2012/FKM(9A)/S01084

JABATAN TERMAL BENDALIR
FAKULTI KEJURUTERAAN MEKANIKAL
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2014

ACKNOWLEDGEMENT

First of all, I express my gratitude to Allah Almighty on the completion of this project. A great appreciation to research members and also to final year student and thanks to Universiti Teknikal Malaysia Melaka for financial support under Short Term Grant vote no: PJP/2012/FKM(9A)/S01084.

KESAN HABA TERHADAP MENDAKAN DALAM RONGGA YANG BEREBEZA BENTUK

(Kata kunci: Mendapan, CFD, aliran salur)

Pencemaran yang baru ditemui pada sendi sistem perpaipan besar yang boleh menyebabkan kecacatan kepada produk perindustrian. Satu analisis berkomputer dilakukan untuk mengesahkan masa lampau struktur aliran rongga dengan menggunakan simulasi perisian FLUENT. Penyingkiran pencemaranberkonsepkan hidrodinamik dengan tambahan haba dan eksperimen dilakukan untuk bentuk segi empat tepat, segitiga dan separuh bulat panjang dengan nisbah kedalaman 2:1. Analisis berkomputer dijalankan untuk menganalisis pemindahan haba dan medan aliran di satu saluran di sebelah bawah rongga. Aliran tak mantap digunakan dalam penyelidikan ini untuk menganalisis aliran dan corak pusaran. Proses gerakan bendalir di sebuah lubang dikira melalui satu penyelesaian berangka persamaan Navier Stokes dengan persamaan tenaga untuk aliran laminar. Penyelidikan ini mula dengan penghasilan geometri manakala pencemaran ialah di dalam rongga. Bagi membuang kesemua pencemaran dalam rongga, keperluan hidrodinamik digunakan di dalam simulasi ini. Kesan hidrodinamik itu akan mengalir dari serokan saluran dan membuat pusaran untuk membuang pencemaran dari rongga. Kehangatan permukaan telah diaplikasikan untuk menetukan kadar pemindahan mempengaruhi penegluaran pencemaran dengan lebih tinggi atau sebaliknya.

Penyelidik:

Mohd Noor Asril Bin Saadun (Ketua)

Mohamad Shukri Bin Zakaria

Muhammad Zaidan Bin Abdul Manaf

Nur Hazwani Binti Mokhtar

Mohd Nazmin Bin Maslan

Email: asril@utem.edu.my

Vote No.: PJP/2012/FKM(9A)/S01084

THE EFFECTS OF HEAT FOR CONTAMINANT REMOVAL FROM DIFFERENT SHAPES OF CAVITY

(Keywords: Contaminant removal, CFD, flow channel)

Contaminant that are recently discovered at the joint of large piping system and can caused defect to industrial product. A computational analysis using simulation of FLUENT software of the hydrodynamic contaminant removal with heat effect is executed and experiments are carried out in order to validate the flow structure past rectangular, triangular and semicircular cavity of length-to-depth ratio 2:1. Computational analysis is performed to analyze the heat transfer and flow field in a channel with cavity heated only at the bottom sides. The cavity that needed to test comes with three (3) types that are square, triangle and semicircle. Unsteady flow are using in this research to analyze the flow of streamlines and vortices pattern. The process of fluid dynamic in a cavity is modeled via a numerical solution of the Navier–Stokes equations with the energy equation for laminar flows. This research starts with the geometry creation. Contaminant was inside the cavities. In order to remove all of the cavities, hydrodynamic need to take part in this simulation. The hydrodynamic will flow from the inlet of the channel and make vortices to remove the contaminant from the cavities. The heat was applied at the bottom of the cavities to analyze if there had present of heat, the contaminant removal rate will be higher or vice versa.

Key-researchers:

Mohd Noor Asril Bin Saadun (Principal)

Mohamad Shukri Bin Zakaria

Muhammad Zaidan Bin Abdul Manaf

Nur Hazwani Binti Mokhtar

Mohd Nazmin Bin Maslan

Email: asril@utem.edu.my

Vote No.: PJP/2012/FKM(9A)/S01084

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

1.1 Introduction

Fluids engineering is relatively related to the motion of fluids, behavior of fluids and more. The most commonly application of this fluids engineering in industry is about the usage of piping system, ducting system and more. In industry, piping system plays a main role as the important of backbone towards human body where it's become the medium of transporting, transmitting, transferring of system from one point to the another point. Thus the critical perception towards this indicator is very important in order to develop a good engineering environment. Flow past cavities have received great attention in the past decades in both experimental and numerical investigation due to its relevance to many practical engineering applications. High speed example of this flow type are flows over aircraft hulls, weapons bays, wheel wells and gas turbine channels while flow over surfaces with ribbing in heat exchangers and microelectronic chips, electronic devices on printed circuit boards are some low speed examples. The shape of cavities used in power engineering equipment varies vastly. In industry nowadays, the applications of flow past cavities being relatively related to the usage of piping system being apply. Considering the term of loses and the other aspects, the application of piping system need to be maintenance efficiently in order to provide the good system. Commonly the most critical problems due to this piping system is the present of contaminants that clearly reduces the efficiency of the piping system. This happen because as the number of contaminants increase it will reduces the surface area of the piping system and will reduce the total power being transmitted along the piping system towards its destinations. In real case of engineering world the term of removing the present of contaminants is being called as flushing. Flushing is the process was the piping system is being flush by greater water pressure by using water jet with the present of removal material such as sodium and more. By this process, the greater impact of pressure will slightly remove the present of contaminants in piping system. From the other term, in different usage of piping system such as piping system in cold area, the usage of cavities with thermal heater is very important in order to prevent the flow of fluids from stuck or freeze. Besides plays a main role in preventing flow from stuck, the present of cavities in piping system also can be used as the medium in process of contaminants removal as well. In the term of fluids dynamic, flow past an open cavity is known to give rise to self-sustained oscillations in a wide variety of configurations and these cavity oscillations are the origin of coherent and broad band sources of noise and if the flow is sufficiently flexible, flow induced vibration as well. Due to this research the main indicator is to determine the rate of contaminants removal for different shapes of cavities. Three shapes of cavity were prepared, semi-circle, triangular and rectangle with respective dimension. The influence of cavity shape on the flow structure within and immediate neighborhood of the cavity is the focal point of this paper as well. The behavior of fluid mechanism such as contour of stream wise, components of velocity, average vortices, Reynolds stress, streamline plots, intensity values are the important parameter for this research. The application of Particle Image Velocimetry (PIV) was used as the medium to analyze the behavior of flow measurement over different shape of cavities.

1.2 Objective

The objective for this research is:

1. To investigate the flow structure over different shapes of cavity of rectangular, triangular and semicircular with heat addition.

2. To determine the effects of cavities shapes on rate of particle removal.

1.3 Scope

The scopes of this research are:

1. To study the influence of cavity shape on the flow structure within and immediate neighborhood of the cavity in analysis.

2. To analyze the behavior of low and high Reynolds numbers flow in particle removal phenomenon using Particle Imaging instruments.

CHAPTER 2

2.1 Introduction

This research focused on the heat convection effect on fluid flow in the different cavity with interest geometry on vertical and horizontal plate heat source. Shang, 1993 studied on the aminar free con ec tion with consideration of the thermo physica properties and found that with constant ambient temperature it resu ting the descending order on rant a ue as the heat source being increase [1]. 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 behavior that with lower Prantl number, the rate were faster compare to high Prantl value [2 - 5]. 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 [6]. The finding provides comparison with the other study made by them by using of rectangular shape with effect on the cavity provide different flow pattern. This research hence to determine the flow pattern in the different side heat of cavity, investigate the macroscopic rigid particles removal in term of percentage due to heat effect, differentiate the outcome result using of different cavity ratio and determine the most affected situated heat source for rectangular shape on the contaminant removal.

CHAPTER 3

3.1 Introduction

To study the flow characteristic of water with seeding particle along the different shapes of cavities, the proper and complete experimental apparatus need to be set up in order to get the effective result. In this experiment, the water should be flow in fully direction from input of cavity until the output of the cavities, thus the tank were used to provide the sufficient water. Besides that, the pump were used to provide greater fluid flow in this water channel and also to determine the suitable low Reynolds number flow of the water. In order to prevent the seeding particle to enter the pump after being remove from the cavity, the systematic filter were design inside the tank to filter out the seeding particle from enter the pump. The frame of this system was designed from the steel to support the weight of the cavity with a specific dimension relatively related to the specification of PIV instruments as well. Below are the specifications of the full experimental design for this experiment:

Table 3:1 Part Description

Parts	Description
Cavity	Rectangular Triangular Semi circular
Piping system	Simple piping system by using PVC material Dimension Height 87.5 cm Diameter of pipe 12.5 mm
Pump	Centrifugal pump Specifications :

	15 kw 0.5 hp 2.4 liter/minute flow rate
Tank	To provide sufficient water towards the system. Specifications : Made from steel coated with paint Capacity of 10 liter of water Simple filter at the tank inlet to filter out the seeding particle
Frame	To support the cavity shapes and all the structures in the system. Description : Made from steel (angle bar) Fabricated by welding technique Dimensions of 90 cm height and 80 cm wide Coated with black paint to prevent rusting.

3.2 Design

Diagram below show the suggested cavity shape for this analysis.

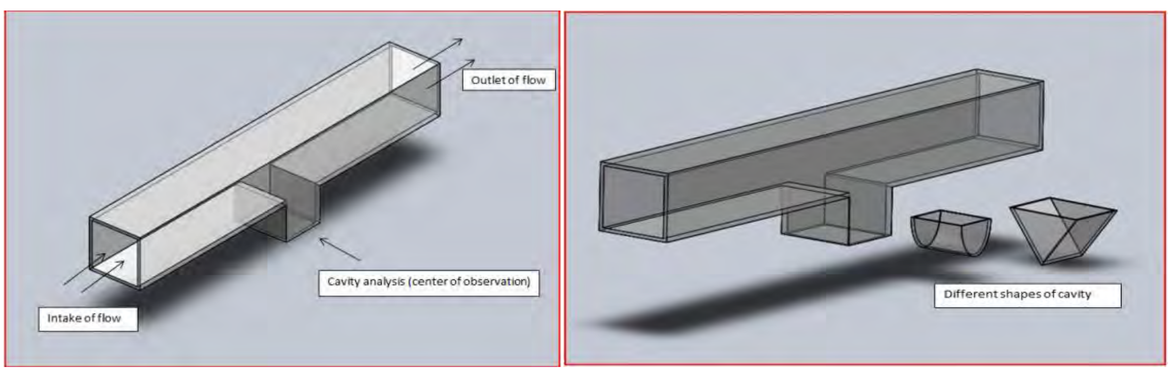

Figure 3:1 Geometry for the cavity and flow channel

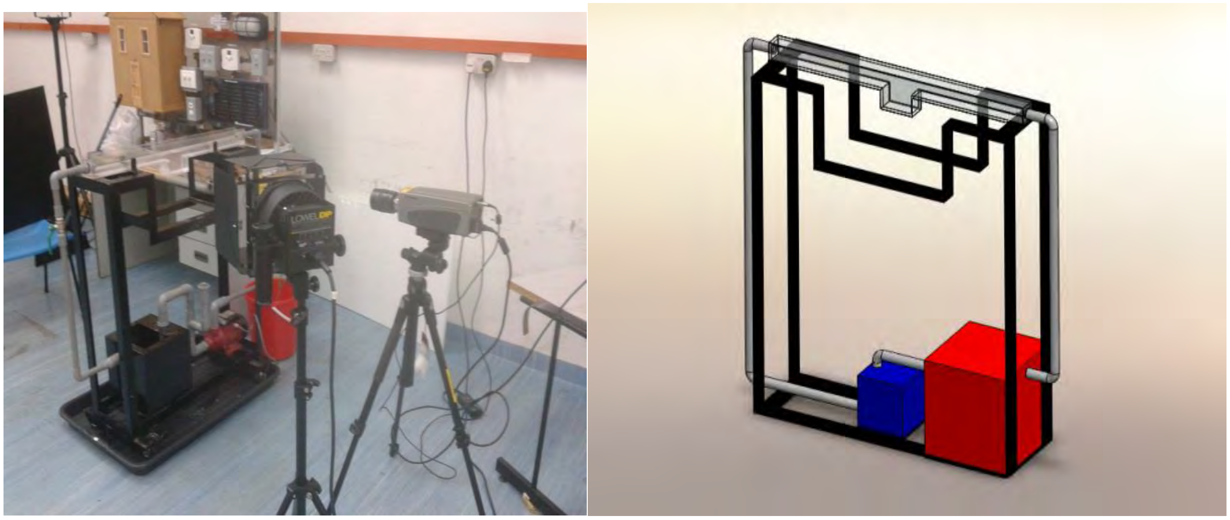

Figure 3:2 Experiment setup

In this research there are several steps need to be followed to get the good result. The procedure for this experiment is:

a) Set up the orientation of camera, lighting and cavity.
b) Fill up the cavity with water and seeding particle.
c) Turn on the apparatus and also water pump.
d) Control the valve to get the specific Reynolds number flow by using Flow Meter device.
e) For flow imaging experiment, as the water fill the cavity start inject the liquid powder constantly in the cavity.
f) For particle rate removal, as the water fill the cavity start analyze the removal rate of the cavity with stopwatch.
g) Take the image of the cavity by using CDU device with respective frame per second.
h) Record the flow process within a specific time.
i) Transfer the data from High Speed Camera to the computer.
j) Analyze the data by using 1P200 software.

3.3 Mathematical Modeling

Vibrations Navier-Stokes equation is a compulsory equation in fluid dynamic fields. The equation is usually apply to the fluid dynamic modeling and this equation usually considered as incompressible. Most fluids flow application was using this equation because in explained the conservation of mass, momentum, and energy the equation almost perfect. In addition, there is no analytical solution to this equation as there are too many Partial Difference Equation (PDE) term in the equation. Now, the research for methodology were wrote down, the equation still not solved by all people in the world. Many type of numerical method were tried out by scientist and engineer, but all of them failed. This equation is unsolvable. It cannot be solved but can be simplified. The dimensionalised governing equations of the fluid flow are given respectively by the equation:

Continuity

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad (1)$$

where the mass doesn't changing in value for the steady flow.

Momentum

x direction

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

y direction

$$u\ \frac{\partial v}{\partial x} + v\ \frac{\partial v}{\partial y} = -\frac{1}{\rho}\ \frac{\partial p}{\partial x} + v\left(\left(\frac{\partial^2 v}{\partial x^2}\right) + \left(\frac{\partial^2 v}{\partial y^2}\right)\right) \quad (3)$$

where u and v are the velocity components in the x and y directions respectively, p is the pressure, ρ is the constant density, and   is the   iscosity. Using the dimensionless definitions,

$$t = \frac{tU}{h}, x = \frac{x}{h}, y = \frac{y}{h}, u = \frac{u}{U}, p = \frac{p}{\rho u^2} \quad (4)$$

the governing equation (1) to (3) become

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (5)$$

Momentum

x direction

$$u\ \frac{\partial u}{\partial x} + v\ \frac{\partial u}{\partial y} = -\frac{\partial \rho}{\partial x} + \frac{1}{Re}\left(\left(\frac{\partial^2 u}{\partial x^2}\right) + \left(\frac{\partial^2 u}{\partial y^2}\right)\right) \quad (6)$$

y direction

$$u\ \frac{\partial v}{\partial x} + v\ \frac{\partial v}{\partial y} = -\frac{\partial \rho}{\partial x} + \frac{1}{Re}\left(\left(\frac{\partial^2 v}{\partial x^2}\right) + \left(\frac{\partial^2 v}{\partial y^2}\right)\right) \quad (7)$$

Note that equation of (6) and (7) are determined the force act by the flow. In this project, 2D fluid flow being subjected to the convective effect caused by heated wall which stated at the y-component.

Energy Equation

$$\rho c \frac{DT}{Dt} = \frac{Dp}{Dt} + \Phi + \mathrm{k}\nabla^2\mathrm{T} \qquad (8)$$

and use the following non-dimensional forms of the temperature and dissipation function

$$T = \frac{T-T_0}{T_w-T_0}, \Phi = \frac{L^2}{U_0^2\mu}\Phi, P = \frac{(p+\rho g y)L^2}{\rho {u_I}^2} \qquad (9)$$

and this lead to

$$= \frac{DT}{Dt} = \frac{1}{RePr}\nabla^2\mathrm{T} \qquad (10)$$

$$= U\frac{\partial\theta}{\partial X} + V\frac{\partial\theta}{dY} = \frac{1}{RePr}\left(\frac{\partial^2\theta}{\partial X^2} + \frac{\partial^2\theta}{\partial Y^2}\right) \qquad (11)$$

where Re= $\frac{u_I L}{v}$; Pr= $\frac{v}{a}$; Ra= $\frac{g\beta(Th - TI)L3}{va}$Are Reynolds number, Prandtl number and Rayleigh number respectively. Dimensional quantity from the defined geometry and assumption made previously then being substituted into these equations for a non-dimensional parameter. From this mathematica mode ing, the re ation of parameters can be shown c ear y and it's important to define every properties involved in problem statement in order to ensure that solver making all of the consideration.

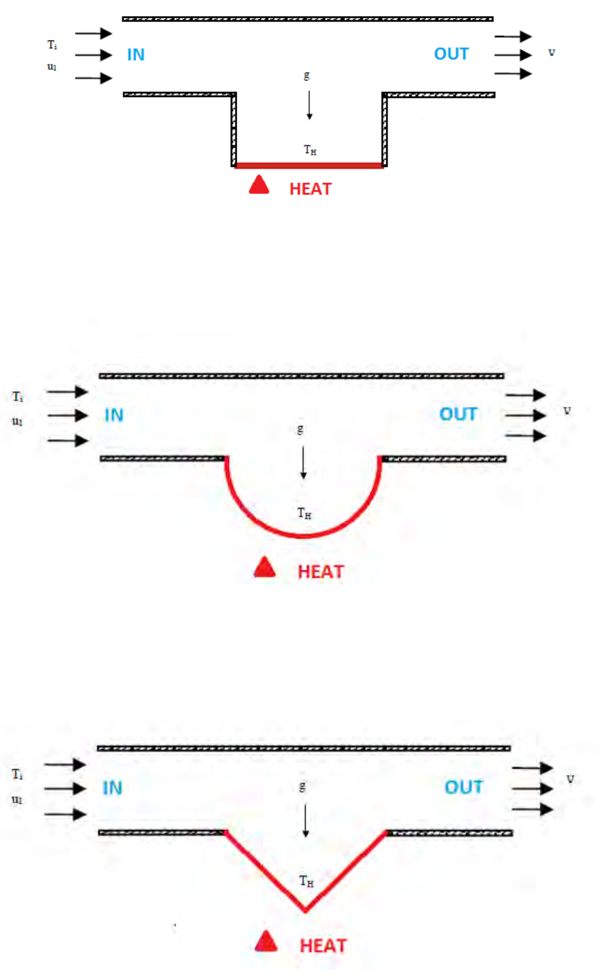

Figure 3:3 Different side of heated

CHAPTER 4

4.1 Experiment

In flow measurement experiment, the focal point of this study is to determine the rate of contaminant removal for each cavity shapes with constant flow of water with high Reynolds numbers flow and the flow pattern within the cavity with low Reynolds number flow. The rate of contaminant removal was observed by a several aspect. The usage of Particle Imaging Technique provides the measurement of the whole flow field in qualitative manner. The present of contours of constant averaged stream wise and transverse components of velocity, contours of constant averaged vortices, Reynolds stress and streamline plots for each cavity type. In addition, stream wise velocity, Reynolds stress is compared for all cavity types. Effect of cavity shapes on flow structure within the cavity is discussed in details.

In this experiment, generally discussing about two main results which are:

i. Flow pattern for low and high Reynolds number in each cavity.
ii. Particle rate removal for high Reynolds number in each cavity.

Rate of Particle Removal for Rectangular Cavity

To determine the rate of particle being removed in the specific cavities, the qualitative analysis is being used. The image of the result was compared from 0 seconds to 20 seconds. The comparison between two images will represent the rate of particle being removed. The images were divided into grid cube with 28 grid width and 16 grid height. From this, the total grid is about 448 grids. The indicator lines were stated to initiate the initial condition. After 20 seconds the quantity of particle were calculated from the indicator line as well. To calculate the percentage of particle being removed, the grid is assumed in three different condition which is full, half and quarter. Thus , to get the amount of particle being removed, the particle left after 20 seconds is calculated respect to the condition.

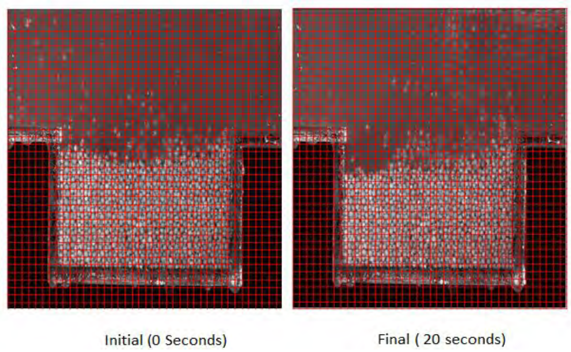

Figure 4:1 Rectangular Cavity

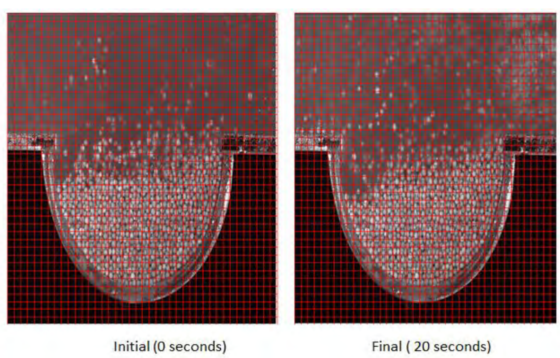

Figure 4:2 Semicircle cavity

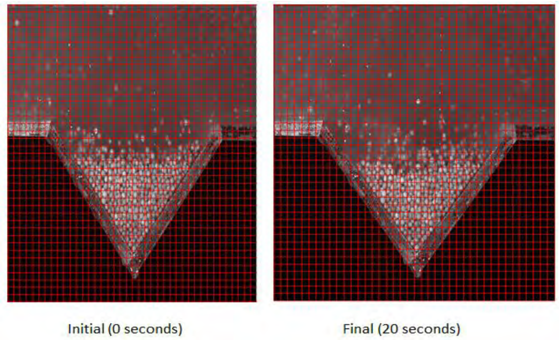

Figure 4:3 Triangular Cavity

4.1.2 Data Analysis

In this experiment, technique of qualitative analysis was used. Thus the behavior of the fluid particle along the cavity with respective flow of Reynolds number were observed and here are two types of Reynolds number being used which is low Reynolds number (Re<4000) and high Reynolds number (Re>4500). For determination of flow particle within the cavity for every shape, using the powder liquid which is in white color to determine the flow within the cavity. The liquid powder had been injected along the channel the as the flow enter the cavity area, the image were snap and recorded.

Flow Pattern Analysis

In this analysis, the liquid powder had been used to determine the flow of fluid within the cavity. Thus to obtain a precise and consistent data, all the image were snap within initial (0 seconds) picture until final (70 seconds) picture. The entire test on every cavity is within this time interval. From this the flow properties of every cavity within respective of time were determined. The data gain from each experiment were clearly plotted by Red Spot indicate the direction of flow in the cavity. In each cavity there is the Red Spot along the eight images of data. It’s c ear y determined the disp acement of f uid particle in each cavity from initial to the final data analysis. From this method the differences between each cavity fluid flow with respect of fluid flow which is can be differentiate either High or Low Reynolds numbers.

Particle Removal Rate

In this analysis the rate of the seeding particle being removed will be determined with respective of time within each cavity. The High Reynolds numbers of flow were used to get the differences between each cavity. The images were snap from initial (0 seconds) and final (20 seconds). As the image being determined, the image is being divided into grid system about 448 grids for all shapes. To define the rate of particle being removed by calculating the percentage of seeding particle being remove after 20 seconds. The calculative data will represent the rate of removal. By this method, the ability of each cavity to remove the particle inside its will be determined.

Formation of Vortices structures within cavities

From the experiment, it's c ear y being determined that the al cavities form a single dominant vortex within the cavity with a little disturbance on average flow upstream portion of the cavity. In fluid flow analysis, for rectangular cavity we determine that recirculation formation is symmetric. In the others hand, the recirculation of triangular cavity is skewed to the upper right portion of the cavity. Besides that, the recirculation inside the semi-circle cavity was also form skewed to the right portion of the cavity. The low Reynolds numbers flow, the formation of recirculation inside rectangular is more towards the center of the cavity however the formation of recirculation inside triangular and semi-circle cavity is skewed towards the right portion. From the other points, in this experiment the fluid exerts some portion of downstream edge of the cavity and the shear layer has adequate time to develop instabilities the presence of downstream edge. By this, it will form the cavity resonance. Even though there are the formations of single vortex within each of the cavities, there is some fluid that did not flow according to the flow stream. For an example in rectangular cavity, most of the liquid powder make the recirculation during the flow but there are some particle were dropped at the bottom of the cavity and did not flow. This is clearly related to the phenomenon of cavity resonance.

Pattern of Streamlines

In this experiment, there are two types of Reynolds number flow being implemented thus it's gi e the ar iation of resu t from one to another within the ca i ty shapes. The most common differences when the Reynolds number flow is change is the streamlines flow within the cavity. As the Reynolds number increase the rate of fluid entrainment near the upstream cavity wall into the shear layer and forming the single focus. This clearly shown that as the Reynolds numbers increase the formation of a vortex is become more effective and bigger as well. In fluid mechanics terms, this phenomenon is called as dominant vortex.

In Rectangular cavity for small Reynolds numbers flow, the vortex more localized towards the trailing edge of the cavity and become expands and occupy the whole cavity as the Reynolds numbers flow is high. In Triangular cavity shape, the formation of vortex is near to upper right of the cavities in low Reynolds flow and becomes almost to the upper right as in the high Reynolds numbers flow. However in semi-circle cavity the vortex are formed much more near to the upper right of the cavity as the Reynolds number increase. From this, the outcome of the analysis is the increases of the Reynolds number flow resulted the formation of the vortex either is asymmetric or skewed structure within the cavity.

Vorticity Contours

From the result, the formation of vortices is resulted at the trailing and leading edge of the structure as well. It's happen because of the interactions between the wall and the shear layer. In rectangular cavity, the vortices is high both on the leading and trailing edge. Besides that in triangular cavity the vortices level are smaller at the leading corner compare to the trailing corner of the cavity. The formation of vortices contours in semi-circle is the lowest due to the smallest vortices level at the leading corner compare to the trailing corner.