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

Signature : ....

Name of Supervisor : Mr. Cheng See Yuan

Date : 30/5/06

FLOW OPTIMIZATION OF KOI POND USING CFD METHOD

MUHAMAD ZAMRI JAMALUDDIN

This thesis is submitted to Mechanical Engineering Faculty in partial fulfillment of the requirements for the award of Bachelor Degree in Mechanical Engineering (Thermal-Fluid)

Faculty of Mechanical Engineering
Kolej Universiti Teknikal Kebangsaan Malaysia

May 2006

"I hereby declare that this thesis entitled "Flow Optimization of Koi Pond using CFD Method" is the result of my own research except as cited in the references"

Signature : ....
Name : Muhamad Zamri bin Jamaluddin
Date : ....30/5/2006........

A special thanks to my project supervisor Mr. Cheng See Yuan. Not to forget my beloved family and all my beloved friends. Thanks for your guidance, strong support and lifelong encouragement...

Faculty of Mechanical Engineering, KUTKM

ACKNOWLEDGEMENT

First, I would like to express my greatest gratitude to Almighty Allah for giving me a chance to complete my thesis with all His blessings. I would like to acknowledge my project supervisor, Mr. Cheng See Yuan. The fundamental idea behind this project is his and he was a valuable source of information about this project. I am very thankful that he supervised my work and provided me with the much needed assistance in understanding the project. Without his guidance and support, I would not have been able to achieve the goals of this project.

To my lecturers who had thought me this far, a very special thanks to them. To all my friends especially Mr. Mohd Sharedzal Jonit and Mr. Faizul Ahmad who gave their support and encouragement throughout this whole project, a million thanks to them as well.

Finally, I would also thank both my parents, Jamaluddin Abdullah and Siti Fauziah Abdullah also my brother, Muhamad Nasri Jamaluddin for their lifelong encouragement. Without their support and confidence, I would not have made it this far.

ABSTRACT

The design of koi pond models using CFD software simulation is to determine the effect of pond geometry to the flow characteristic of water. The goal of this project is to design a pond without stagnant areas. A design guideline on how to build an optimize koi pond is revealed. A pond model is constructed in a few different geometry shapes using Ansys CFX 10.0 software with different parameters to study on how the flow will react with the change of these parameters. The listed parameters are pond depth, bottom contour, angle, radius and also the location of inlet (waterfall) and outlet (bottom drain) of the pond. All these are the possible parameters that may affect the flow characteristics of water in the pond. A 5 feet pond depth proves to be the most suitable depth for a koi pond. A bottom contour of 100 degrees is proven in the simulation to provide the self cleaning feature. Angle wider than 90 degrees and pond radius larger than 2 feet provides a smooth and better flow with no stagnation area.

ABSTRAK

Rekabentuk kolam ikan koi dengan menggunakan perisian simulasi 'CFD' dibuat untuk menentukan kesan bentuk kolam terhadap pengaliran air di dalam kolam tersebut. Satu rujukan atau peraturan untuk merekabentuk sebuah kolam telah diwujudkan. Model kolam telah direkabentuk dalam beberapa variasi bentuk dengan menggunakan perisian Ansys CFX 10.0. Ini bertujuan untuk mengkaji pengaliran air terhadap parameter-parameter yang berbeza. Parameter yang dikaji adalah seperti kedalaman kolam, kontur di dasar kolam, sudut kolam, jejari kolam dan yang terakhir sekali merupakan lokasi air masuk dan air keluar dari kolam tersebut. Semua parameter ini dipilih kerana ia mungkin akan memberi perbezaan dalam pengaliran air di dalam kolam. Kolam dengan kedalaman 5 kaki adalah sesuai untuk dibina kerana ia terbukti menghasilkan pengaliran yang baik. Tiada kawasan takungan yang berlaku. Kontur 100 darjah pada dasar kolam telah telah membuktikan pengaliran yang baik di dasar kolam. Sudut yang luas dan jejari yang besar iaitu melebihi 2 kaki dapat menghasilkan pengaliran air yang optimum di dalam kolam.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF SYMBOL

SYMBOL	DEFINITION
v	Velocity
Q	Flow Rate
A	Area
ρ	Density
g	Gravity
h	Height

LIST OF ABBREVIATION

ABBREVIATION	DEFINITION
CFD	Computational Fluid Dynamics

LIST OF APPENDIX

CHAPTER I

INTRODUCTION

1.1 Koi Pond Overview

What is koi pond? A Koi pond is an enclosed, recirculating, freshwater system for keeping Koi.

A Koi pond should serve two main functions:

- One is to provide a healthy home for your Koi.
- The other is to provide "clear-water" so that we can enjoy the beauty.

The shape of the pond is much more important than most people realize. Many times people will get very creative with the pond shape so that it rambles all around the landscape and ends up looking like a piece of a puzzle. Only after the pond is built and filled with water the problem become apparent. It's difficult to keep it clean. If the pond is shaped so that it has dead areas where the water is not moving, leaves and debris will collect there and eventually sink. Additionally, the bottom contour of the pond is equally important. Flat bottom ponds require constant vacuuming to keep them clean. On the other hand, to build a pond that is almost "self cleaning" pay close attention to both the shape of the pond and the inside contour leading from the edges of the pond all the way to the bottom drains. A well designed pond shape and contour can save many hours of back breaking work and help provide a healthier environment for the koi.

The flow optimization is also another important factor in order to obtain a good and healthy koi pond. Stagnant or dead area in a pond can affect the circulation of water through the filter system. The aim of this project is to study how various geometry of pond can contribute the flow of water in the pond.

Basically in this project, a CFD simulation software will be utilize to determine the flow characteristic of water in a pond. This is to observe whether there are any stagnant areas in the pond.

1.2 Problem Statement

Today there is a serious problem when it comes to designing and building a quality koi pond. The problem is that few people can agree on how to do it. Most people who decide to build a koi pond start out very excited and determined to build it right the first time; then the problem starts. When they start researching and asking their friends for advice they soon get overwhelmed with conflicting advice. Much of the advice comes from individuals who, though meaning well, have limited experience in building a quality koi pond. They may have built a pond that turned out very successful but unless you build a pond exactly the same size and shape, what worked for them may not work for you.

For a koi pond to really be successful it needs to meet two basic requirements. First, it needs to provide excellent water quality. Not just clear water, but water that is healthy for the fish to live in, Without healthy water the koi experience could be one of constantly treating sick koi. Secondly, a successful koi pond should be easy to maintain. Many ponds, due to poor design, are extremely difficult to take care of.

1.3 Objectives of Project

- To determine the effect of pond geometry to the flow characteristic of water.

- Flow optimization in koi pond using a software method known as Ansys CFX 5.7.

- To develop koi pond design guidelines which reveal the optimum flow characteristic of water throughout the pond.

1.4 Scope of Study

- Literature study on overview of koi pond design, capacity, requirement and CFD modeling and simulation.

- Construction of koi pond models using CFX software.
  - Design existing koi pond for comparison studies.
  - Design other geometries with different parameters (radius, angle, depth, and inlet & outlet location) to observe the reaction flow.
- CFD simulation and predictions.
  - Collect results and data from actual pond.
  - Observe the simulation of water flow.

1.5 Project Significance

The aim of this study is to provide a guideline in creating a best perfect koi pond for all users. This project will focus more on geometry shape of a pond. Each of the pond design was developed to basically take all the guess work out of building a quality koi pond. So it is guaranteed that if everything has been followed accordingly, you will end up with a successful pond. That is, a pond that will provide excellent water quality and will be the easiest pond to maintain possible. This study will ensure that the water has to be able to circulate evenly without any stagnant or dead areas.

1.6 Expected Result

By the end of this project, effect of pond geometry to the flow characteristic of water can be determined. By observing the simulation done using Ansys CFX software, results can be obtained and validate. Then, a good design guideline can be reveal to develop an optimized koi pond which possessed the optimum flow throughout the pond.

CHAPTER II

LITERATURE REVIEW

2.1 Analysis of sediment transport modeling using CFD for aquaculture raceways

A simulation model to analyze the water flow and sediment transport in aquaculture raceways was developed using a computational fluid dynamics (CFD) software package. The simulation was used to evaluate the efficiency of solids settling in the quiescent zone of existing trout raceways. This efficiency was based on the percentage of solids removed, which corresponds to the percentage of solids introduced into the raceway that settle in it, with settling taking place primarily in the quiescent zone. The raceway selected for model validation was a rectangular concrete raceway 30.0m long, 3.0m wide, 0.9mdeep, with a slope of 0.01. The raceway included a quiescent zone of approximately 5.3m in length, which was separated from the rearing area by a screen. The water flow rate through the raceway was approximately 0.058m3/s. Velocity measurements were recorded at 230 stations along the raceway using an acoustic Doppler velocimeter, for comparison with the results obtained from the simulations. For the purpose of simulating sediment transport, six groups of particles were used to account for the total suspended solids. The sizes of the particles selected were based on an experimentally determined distribution for solids from a similar raceway, and were 692, 532, 350, 204, 61, and 35_m for Groups 1–6, respectively. The particle density for each size was assumed to be 1150 kg/m3. Values of the percentage of solids removed for the different particle sizes were 100.0% for the largest particles, and 54.7, 0.9, and 0.1% for the three smallest particles, respectively. This methodology of analyzing the raceway sediment transport in terms of its percentage of solids removed based on CFD simulations can also be used to examine raceway design alternatives for improving the particle removal efficiency. (Dania L. Huggins, Raul H. Piedrahita, Tom Rumsey 2004).

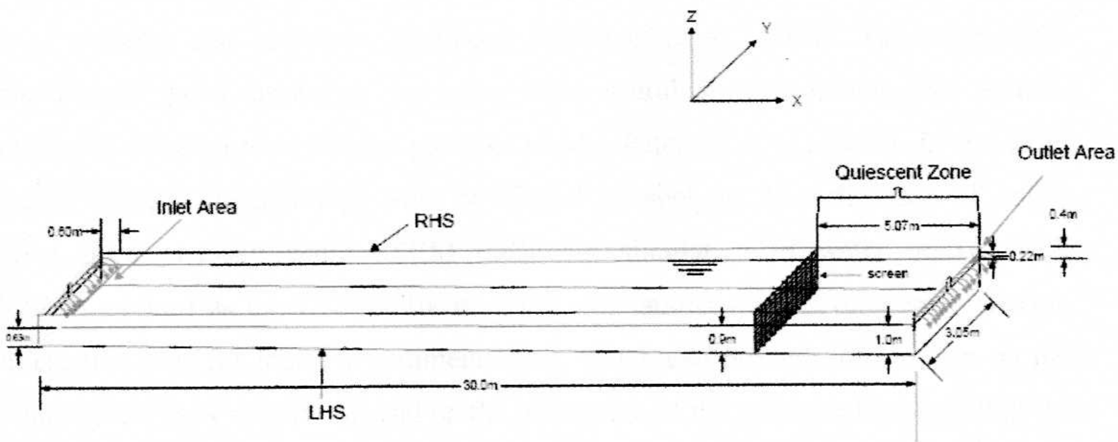

Figure 2.0 – Model Raceway

Given the large size of some trout rearing facilities, there is a potential for significant nutrient discharges (IWMGAO, 1999; NASS, 2001). There are strong economic, social, and regulatory pressures to reduce the release of nutrients from aquaculture operations. However, the large flows and very low constituent concentrations make it very difficult to treat effluents. A potentially effective strategy to reduce the release of solids, which contain nitrogen, phosphorus, and organic matter, is to increase the proportion that settles within a raceway's quiescent zone. Understanding how water and sediment particles move in the quiescent zone provides an opportunity to suggest raceway design modifications aimed at improving particle settling and improving the quality of the effluents. Therefore, the focus of this study was to develop and validate a computation fluid dynamics (CFD) model for a trout production raceway that can be used to evaluate the efficiency of solids settling for a particular raceway design. (Dania L. Huggins, Raul H. Piedrahita, Tom Rumsey 2004)

CFD simulations have been used in aquaculture to describe water flow and solids removal in circular tanks (Montas et al., 2000; Veerapen et al., 2002). In their 2002 study, Veerapen and coworkers studied the factors influencing the removal efficiency of swirl separators and double-drain fish tanks. Rasmussen (2002) used CFD modeling to determine the mixing characteristics of a turbot rearing tank and the transport of particulate organic material and the oxygen distribution in rectangular tanks. There are also some studies applying CFD modeling to aquaculture ponds to simulate water flow velocity patterns and sediment conditions (Peterson et al., 2000). The information obtained from pond modeling has also been useful in determining the optimal arrangement of aerators in shrimp growout ponds (Peterson et al., 2001). In the work presented here, a methodology was developed to analyze the efficiency of solids removal in a raceway using SSIIM (sediment simulation in water intakes with multiblock option) as the CFD software. CFD simulations were developed to analyze sediment transport for multiple sediment sizes, which can provide information on the distribution and flow of particles and on the proportion of the solids that settle within the quiescent zone, expressed as the percentage of solids removed.

The water flow velocity calculations are based on Navier–Stokes equation for turbulent flow in a general 3D geometry for non-compressible and constant density flow (Olsen, 1991). SSIIM uses the k–ε turbulence model to predict the shear stresses (Versteeg and Malalasekera, 1995). The k–ε model focuses on the mechanisms that affect the turbulent kinetic energy. More details on how the k–ε model is used for calculating the turbulent shear stress are given in Olsen's SSIIM manual (Olsen, 2002).

SSIIM has some limitations in terms of its capabilities with different geometries, hydraulic configurations, and the number of cells (control volumes or elements). Due to these limitations some assumptions had to be made and these are described as follows. (Dania L. Huggins, Raul H. Piedrahita□, Tom Rumsey 2004)

As a first assumption, the model considers that the influent flow is uniformly distributed along the full depth of the raceway at the first upstream cross-section. This assumption was made because simulating influent water splashing on the surface of the water (Fig. 4), as in a real raceway, made convergence in the model slow or unreachable in some cases. This difficulty in simulating a surface inlet has been reported by other authors (Bates, 2000) and it appears to be a problem not only with this particular software package, but with CFD algorithms in general (Montas et al., 2000; Bates, 2000). Partly as a result of this assumption, validation of the model was carried out for the downstream end of the raceway, especially in the quiescent zone.

As in other studies where CFD modeling has been applied for fish tank design, the presence of fish was not included in the model, due to the complexity that would be introduced and the lack of detailed information that would be needed for model development. For model validation purposes it would have been best to be able to measure velocities in an empty raceway but it was difficult to combine this requirement with the planning and daily activities at the farm where the raceway used to collect calibration data was located. However, this assumption does not weaken the model in the quiescent zone where particle settling analysis is performed, since the fish are excluded from this region (Fig. 2.0).

2.2 Conclusion

So from this research, it did not mention much about the shape of the model that can also contribute to the flow pattern of water. It is mainly focus on transportation of sediment in the model raceway along with water. However, the project that will be conduct here is going to focus on a different perspective which is koi pond as the model. This project will only simulate the flow of water from inlet to outlet without any fish presence due to its complexity but still using the same CFD software.

CHAPTER III

METHODOLOGY

3.1 Introduction

This chapter will brief about the method that will be conduct to observe the flow pattern of water. The project methodology has to be research to make sure the method is suitable and have optimum criteria which are low cost, precise and high efficiency. The method will be divided into a few stages and each stage will be described.

Firstly is to decide on what shape of pond to be design. After that, CAD drawing is done using Ansys CFX 10.0 software. CFX software will be explained more detail in this section later on.

Next is to construct a few different geometry shapes using Ansys CFX with different parameters to study on how the flow will react with the change of these parameters. The listed parameters are pond depth, bottom contour, angle, radius and also the placement of inlet (waterfall) and outlet (bottom drain) of the pond. All these are the possible parameters that may affect the flow characteristics of water in the pond.

3.2 Pond Geometry Category

There are a few common shapes which most people would do. It can be categorized into three main categories.

- Formal
- Semi-formal
- Informal

Before proceeding any further, let's find out what does it mean by shape. A shape is commonly known as polygon. A polygon, literally "many angle" is a closed planar path composed of a finite number of sequential line segments. The straight line segments that make up the polygon are called its sides or edges and the points where the sides meet are the polygon's vertices. If a polygon is simple, then its sides (and vertices) constitute the boundary of a polygonal region, and the term polygon sometimes also describes the interior of the polygonal region (the open area that this path encloses) or the union of both the region and its boundary.

3.2.1 Formal Shape

Formal shape is defined as a shape constructed only with linear lines regardless of the amount of lines. Let it be triangle with three lines, square with four or even pentagon with five lines total. All of them fall in the same category.

Radius or fillets is restricted in constructing a formal shape. If $r \geq 0$ it is no longer formal. That will fall in a different category which will be explained next. Of course it has to be closed contour or it can't contain water in it.

For angle wise, let's take an L shape for example. It is commonly known that L shape possessed a 90 degrees angle. What happens when the angle is more or less than 90 degrees? Will that still be considered as a formal shape? Yes, they are still formal.