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

Signature : ....

Name : Mr. Cheng See Yuan

Date : 15/05/07

EVALUATION OF THE CAPABILITY OF THE CFX COMPUTATIONAL FLUID DYNAMIC IN THE PREDICTION OF KOI POND FLOW

NASRUL BIN HARUN

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

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

April 2007

"I hereby declare that this thesis entitled "Evaluation of the capability of the CFX computational fluid dynamic in the prediction of Koi pond flow" is the result of my own research except as cited in the references"

Signature : ....

Name : Nasrul Bin Harun

Date : 15 - 05 - 07

ACKNOWLEDGEMENT

Firstly, I would like to express my deepest gratitude to Allah the Almighty for giving me chance to complete my thesis. I also would like to acknowledge my project supervisor, Mr. Cheng See Yuan. The fundamental for this project is his idea and he was a valuable source of information in this project. I am thankful for his assistance and guidance along this thesis completion process. Without his guidance and support, I would not have been able to achieve the goals of this project.

To my lectures who had thought me this far, a very special thanks to all my friend for their support and encouragement throughout this whole project, a million thanks to them as well.

Finally, to my family that encouraging and support me to fulfil this project. A thousand of gratitude to my parents, Harun Pasima and Norma Nole also to my brother Amiruddin Harun for his lifelong encouragement. I have been able to complete this project with their support and confidence.

ABSTRACT

Evaluation of the capability of the CFX computational fluid dynamic code in the prediction of Koi pond flow is to determine the precision of water flow in Koi pond water circulating system. The goal of this project is to determine the capabilities of CFD/CFX code in producing a likely real pond flow and analysis of the flowing water. The evaluations are started by determining the capabilities that will change the evaluation of precision such as the meshing and advance CFD process. This whole study will prove the capabilities of this software in variant of flexible ways. Within the different approach used, the capabilities of this software will be proven by supporting data. Though, the main objective of this study that is to predict the capabilities of this software achieved.

ABSTRAK

Penilaian tentang kemampuan kod komersil CFD didalam menjangkakan aliran air didalam kolam ikan Koi adalah bertujuan untuk menentukan kebolehan kod komersil ini dalam menunjukkan ketepatannya mengukur dan mejangka kadar alir air dalam sistem kitaran air kola ikan Koi. Matlamat utama projek ini adalah untuk menentukan kebolehan kod komersil CFD didalam menentu ukur keadaan sebenar kadar aliran air dalm sesebuah kolam. Penilaian in bermula dengan mengenal pasti faktor - faktor pemboleh ubah yang boleh meningkatkan kepersisan dan kejituan bacaan seperti jaringan-jaringan komponen dalam proses "meshing" dan "advance CFX process". Kajian ini akan cuba membuktikan kebolehan perisian komputer ini dengan meggunakan pelbagai kaedah. Melalui pendekatan-pendekatan yang digunakan dalam kajian, data-data yang boleh diterima pakai digunakan sebagai bukti kesahihan tentang kebolehan perisian ini dalam memberi data-data yang persis dan jitu.

TABLE OF CONTENTS

CHAPTER	CONTENTS	PAGE
	DECLARATION	iii
	ACKNOWLEDGEMENT	iv
	ABSTRACT	v
	ABSTRAK	vi
	TABLE OF CONTENTS	vii
	LIST OF FIGURES	x
	LIST OF TABLES	xii
	LIST OF SYMBOLS	xiv
	LIST OF ABBREVIATION	xv
	LIST OF APPENDIX	
1	**INTRODUCTION**	1
	1.1 Research Introduction	1
	1.2 Problem Statement	2
	1.3 Objective	3
	1.4 Scope	3
	1.5 Expected Result	4
2	**LITERATURE REVIEW**	5
	2.1 Introduction	5
	2.2 Computational Fluid Dynamics (CFD) Review	5
	2.2.1 Related elements in CFD designing	7

	2.3 CFD Data process	10
	2.3.1 Pre Processing	10
	2.3.2 Solving	10
	2.3.3 Post Processing	11
	2.4 Applications of CFD in Studying Koi Pond Flow and Other Means	11
	2.4.1 CFD in Koi Pond Water Flow System	11
	2.4.2 Computational Fluid Dynamic Applications	12
3	**METHODOLOGY**	14
	3.1 Introduction	14
	3.2 Pond geometry category	15
	3.2.1 Pond Design	16
	3.2.1.1 Angled Shape Pond	16
	3.2.1.2 Simple Shape Pond	18
	3.3 ANSYS CFD/CFX Simulation Process	20
	3.3.1 CAD Model Preparation	20
	3.3.2 Mesh Generation	21
	3.3.3 Pre-processing	22
	3.3.4 Solver Stage	22
	3.3.5 Post-Processing	23
	3.4 Parameters that effect the flow results precisions in mesh generation process	24
	3.4.1 Spacing	24
	3.4.2 Inflation of Inflated Boundaries	25
	3.4.3 Stretch of Meshed Grid	25
	3.5 Parameters That Effect Flow Results Precision In Pre-processing Process	26
	3.5.1 Define The Boundary Condition	26
	3.5.2 Define The Solver Control Criteria	26
	3.8 Project requirements	27
	3.8.1 Software requirement	27
	3.8.2 Hardware requirement	27

4	**RESULT AND DISCUSSION**	28
	4.1 Calculation	28
	4.1.1 Calculation for Velocity Inlet Value for Angled Shape Pond	28
	4.1.2 Calculation for Velocity Inlet Value for Simple Shape Pond	29
	4.2 Parameters That Affect The Flow Results Precisions In Simple Shape Pond	30
	4.2.1 Face Spacing	31
	4.2.2 Edge Spacing	38
	4.2.3 Inflation of Inflated Boundaries	44
	4.2.4 Boundary Layer with Face Spacing	49
5	**CONCLUSION**	53
	REFERENCE	55
	APPENDICES	56
	Gantt Chart of Project Flow	

LIST OF FIGURES

FIGURE NO	CONTENT	PAGE
Figure 3.1	Angled shape isometric view of 7 foot depth pond	16
Figure 3.2	Angled shape top view	17
Figure 3.3	Angled shape front view	17
Figure 3.4	Angled shape side view	7
Figure 3.5	Simple shape isometric view of 7 foot depth pond	18
Figure 3.6	Simple shape top view	18
Figure 3.7	Simple shape front view	19
Figure 3.8	Simple shape side view	19
Figure 3.9	CAD pond model	20
Figure 3.10	Mesh generation	21
Figure 3.11	Pre-processing	22
Figure 3.12	Streamlines in post-processing	23
Figure 3.13	Features of meshing with (left) and without face spacing (right)	24
Figure 3.14	Features of meshing with (left) and without edge spacing (right)	24
Figure 3.15	Features of meshing with (left) and without inflated boundaries (right)	25
Figure 4.1	Isometric view simple shape koi pond	30

Figure 4.2	Graphical interpretation for meshing without and with face spacing	31
Figure 4.3	Coordinate step placement for face spacing	33
Figure 4.4	Face spacing factor of velocity versus coordinate step	34
Figure 4.5	Velocity versus coordinate of face spacing	34
Figure 4.6	Velocity versus time for face spacing factor	35
Figure 4.7	Time versus elements for face spacing	37
Figure 4.8	Graphical interpretation with edge spacing	38
Figure 4.9	Coordinate step placement for edge spacing	40
Figure 4.10	Edge spacing factor of velocity versus coordinate step	40
Figure 4.11	Velocity versus coordinate of edge spacing	41
Figure 4.12	Velocity versus time for edge spacing factor	42
Figure 4.13	Graphical interpretation with inflated boundaries	44
Figure 4.14	Coordinate step placement for inflated boundaries	45
Figure 4.9	Boundary layer effect on velocity	46
Figure 4.10	Velocity versus coordinate for inflation of inflated boundaries	46
Figure 4.11	Velocity versus time for inflation of inflated boundaries	48
Figure 4.12	Combination of Inflation boundaries and face spacing	49
Figure 4.13	Coordinate step placement for both conditions	51
Figure 4.14	Velocity versus coordinate step	51

LIST OF TABLES

TABLE NO	CONTENT	PAGE
Table 4.1	Velocity of different minimum edge length for face spacing factor	32
Table 4.2	Coordinate step for velocity measuring in pond system	33
Table 4.3	Numbers of element and time taken for face spacing	36
Table 4.4	Velocity of different constant edge length for edge spacing factor	38
Table 4.5	Coordinate step for measuring velocity in pond system	39
Table 4.6	Numbers of element and time taken for edge spacing	42
Table 4.7	Velocity of different minimum edge length for inflated boundaries factor	44
Table 4.8	Coordinate step for measuring velocity in pond system	45
Table 4.9	Total velocity for each inflation factor	47
Table 4.10	Numbers of element and time taken for inflated layer	47
Table 4.11	Velocity of combination of inflate layer with face spacing and face spacing	50
Table 4.12	Coordinate time step for boundary layer with face spacing	50

Table 4.13	Total velocity for two different mesh conditions	52
Table 4.14	Numbers of elements and total wall clock time	52

LIST OF SYMBOL

SYMBOL	DEFINITION
Q	Flowrate
V	Velocity
A	Surface area
d	Diameter
p	Pressure
ρ	Density
g	Gravitational velocity
h	Height
U_i	Velocity components
f_i	Body force
t	Time
δ	Changes in motion distance

LIST OF ABBREVIATION

ABBREVIATION	DEFINITION
CFD	Computational fluid dynamic
Re	Reynolds number
CAD	Computer aided design
GUI	Graphical user interface
BGK	Bhatnagar-Gross-Krook

CHAPTER 1

INTRODUCTION

1.1 Research Introduction

This research is conducted to evaluate commercial computational fluid dynamics CFD/CFX code capabilities of prediction of Koi Pond water flow. The goal of this study is to find the capabilities of CFD/CFX computational technology on the effect of pond geometry and boundary condition to the flow characteristic of water in the pond.

A computational fluid dynamics CFD/CFX code are an engine software of determining and demonstrate a number of capabilities of flow characteristics and condition of deferent variables and environment of a Koi pond. It will test the pond in different geometries and variables with different independent verification in order to obtain a same result but within different range of values which depends to the references.

A Koi Pond is an enclosed, recalculating, freshwater system for keeping Koi that has two main functions that is to provide a healthier environment and gives clean water to the contained fish. Shape of the pond is important in designing a Koi Pond. This is because, the design will take effect on the water flow of the pond and may effect on the fish health. It is caused by non-circulating water in the filtering system that contains dirty water and composed material that did not flow in the filtering system.

Instead of the design of the pond, the boundary condition and sizes may also take effects to the water flow and it characteristics. This is because, the flow in the pond are depends on the container where water motion are flows within the container and in this case the pond boundaries.

This study is concentrating on does the CFD/CFX code could generate and evaluate the flow of a Koi Pond with different geometries and boundaries. Results of this study may be useful in evaluating and considering the usage of this software in determining and evaluating fluid motion in a system used.

1.2 Problem Statement

In this study, the commercial CFD/CFX codes are tested by using a Koi pond as a medium of studies. The software tested and evaluated whether the results that obtained are fulfilling the objective of this studies. Research on the capabilities of predicting of Koi pond flow by commercial CFD/CFX software are to be tested, thus further investigations are required to answer some open questions regarding the capabilities of computational technology on determining the flow condition on Koi pond, such as:-

- How the boundary condition and size give effect to the pond flow?
- How far the CFD/CFX code can shows verification of the pond flow?
- Does the CFD/CFX code capable to show the prediction flow a Koi Pond?

1.3 Objective

The objectives of this proposed study project are based on:-

1. To develop pond model within several different boundary condition that can be express and simulated in a CFD code.
2. To construct and define the pond within different meshing features for every different boundary conditions.
3. To investigate the capabilities of CFD code in predicting and shows verifications in Koi pond flows within the different range.

1.4 Scope

The scopes of the proposed project are:

- Construction of Koi Pond models using CFX software
  - Designs CFD 3-dimensional Koi Pond model using commercial computational technology CFD software.
  - Design different boundary size and condition parameters to observe the reaction flow.

- CFD simulations and predictions.
  - Calculate flow at the pond systems and make evaluation on flow characteristic of different boundaries condition and geometries.
  - Study the software capabilities in determining water flow.

1.5 Expected Results

By the end this project, capabilities of commercial CFD code on determining and prediction of Koi Pond water flow characteristics achieve. It is expected that this computational technology could give precise simulation of pond flow. The comparison between several type of pond geometries can be expressed and evaluate using this software. The characteristic of each flow hopefully can be determine and differentiate with each changes in boundaries condition and characteristics and geometries.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This chapter conclude reviews on CFD/CFX code. Here, the software introduction will be stated. In stead of that, the application of CFD code in determining the Koi Pond flow will be stated and elaborate. This literature review continues with the discussion on the information related with CFD applications on the flow characteristics.

2.2 Computational Fluid Dynamics (CFD) Review

The key word in this software is Computational which means something related with mathematics and in modern applications, computing and Fluid dynamics are the condition and the dynamics of a flow. CFD/CFX is an insight, foresight and efficiency computational technology that enables to study the dynamics of thing that flow and its behaviour. This technology is a sophisticated computationally based design and analysis technique. It can be applied by applying the fluid flow physics and chemistry to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena such as flows of gasses and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid structure interaction and acoustics through computer modelling.

This method is a better solution on findings flow characteristics of body or container of a design rather than test it through experiment. This computational CFD technique enables to show virtually inside a design and how it performs even by experimentally, it is difficult to be seen or viewed through any other means.

CFD predict the fluid problem by describing the fluid flow case with a mathematics equation and by solving the mathematical equation. The mathematical technique includes finite deference methods, finite element methods and filter-scheme methods.

There are two major categories in CFD application that is the commercial code and research code. FLUENT, Multiphysics, CFX, STAR-CD, PHEONIX, FloWizard, and are the example of this code. This application is basically used in one specific reason of findings. It can be applied in several types of operations methods such as aerodynamics. Through CFD, the details about the flow field such as shear stresses, velocity and pressure profile and flow streamlines is obtained.

Basically, CFD solve three equations called the Navier-Strokes equation. These equations can be presented in the formulation and in Cartesian coordinates:

- Conservation of mass : $\frac{\partial \rho}{\partial t}+\frac{\partial(\rho U_i)}{\partial x_i}=0$
- Conservation of momentum : $\frac{\partial(\rho U_i)}{\partial t}+\frac{\partial(\rho U_i U_j)}{\partial x_j}=\frac{\partial p}{\partial x_i}-\frac{\partial \tau_{ij}}{\partial x_i}+\rho f_i$
- Conservation of energy : $\frac{\partial(\rho H)}{\partial t}+\frac{\partial(\rho U_j H)}{\partial x_j}=\frac{\partial \rho}{\partial t}-\frac{\partial}{\partial x_i}(U\tau_{ij}+Q_i)+\rho U_i f_i$

The first equation state that the mass fluid is conserved. This equation is often called the continuity equation which provides information on velocity due to changes in the geometry. The second equation, the conservation of moment, postulated that the change of linear momentum on a particle is proportional to the force acting upon it. Conservation of momentum is expressed in a manner similar to the continuity equation, with vector components of the momentum replacing density, and with a source of momentum to represent forces acting on the fluid. This equation serves to determine aerodynamic loads. The third equation describes the conservation of energy.

2.2.1 Related Elements in CFD Designing

Navier – stroke equation

These equations are name after Claude-Louis Navier and George Gabriel Stroke. This equation consist set of equations that describe the motion of fluid substance such as liquid and gasses. In this equation, changes in momentum and pressure are related with each other which the pressure change is causing the changes of momentum in fluid particles. The Navier-stroke equations are dynamically statement of forces acting at any point in a fluid because of the molecular interactions and dictate of a fluid.

This equation forms also describe the relative physics of real life and the conservation of universe that conclude large number of phenomena. They are used to predicting the weather, ocean current, water flow in pipe, motion of stars in galaxy, and flow around an air foil. By this advantage and capabilities, these equation are widely used in predicting a flow related model and designs. In a higher level, this equation coupled with Maxwell's equations so it can be used in modelling and study on magneto hydrodynamics.

Turbulent flow

In designing a pond water flow, there will consist laminar and turbulent flow that create by the motion of a moving fluid. This type of flow is known as chaotic regime characterized or stochastic properties changes and may lead to uncertain flow of a fluid motion. The turbulence flow generally produce when the Re number reaching about $Re \leq 10^5$ 0. This dimensionless Reynolds number characterizes whether flow conditions lead to neither Laminar nor Turbulent flow. For instant, in a pipe line the turbulence flow will occur when Reynolds number is about 2300. As a fluid flow increase, at some point transition is made to turbulent flow that consist unsteady flows which interact with each other. This cause drag due to boundary layer skin friction increases. Turbulent causes the formation of eddies which are defined by the length scale and turbulent diffusion coefficient.

Laminar flow

The word Laminar comes from the movement of adjacent fluid particles together in "laminates". Laminar flow also known as streamline occurs a fluid flows in parallel layers with no disruption between the layers. In fluid dynamics, laminar flow regime characterized by high momentum diffusion, low momentum convection, and pressure and velocity independence from time.

Drag force

Drag force is a combination of aerodynamic or hydrodynamic forces which tends to reduce speed. The higher speed of a body, greater drag force produce to the body which means greater opposing force to the body occur which increase the Reynolds number and turbulent flow through the body occur. Though, the flow behaviour, transfer of heat, mass, phase change, chemical reaction, mechanical movement and stresses, deformation, pressure and shape effects of flow are changed due to these changes. Drag is made up by friction due to opposite direction of motion of a body and fluid surroundings. It acts in parallel direction to the object surface at the parallel surface and perpendicular at the surface that will create stagnant flow on a body.

Eddy

In fluid dynamics, eddy is defined as swirling fluid that creates reverse current when the fluid flows past an obstacle. Moving fluid creates a space devoid of downstream-flowing water on the side of the object. Fluid behind the obstacle flows into the void creating a swirl of fluid on each edge of the obstacle, followed by a short reverse flow of fluid behind the obstacle flowing upstream, toward the back of the obstacle.

Boundary layer flow

Boundary layer is a medium or body which contact with the fluid flow. On the surface of the boundary layer, there is friction or viscous forces that produce by the roughness or friction of molecule and the fluid molecule viscosity. This may caused the flow region adjacent to the wall in which the viscous effects are significant. Near the boundary layer, there is no velocity flow and it is gradually increase due to position changes from the surface.

Stream line

In CFD applications, the result of findings is shown in streamline graphical solutions. This is an important concept of showing the result of simulation of a design in proper way. Streamline is a path traced out by discrete particle as it moves with the flow. By this way, observation on fluid flow is shown neatly and less unnecessary figure confusing for results taking. Since there is no normal component of the velocity along the path, mass cannot cross a streamline. The mass contained between any two streamlines remains the same throughout the flow field.