"I hereby declared that I have read this thesis an in my opinion this thesis is sufficient in terms of scope and qualify for the award of the Degree of Bachelor Mechanical Engineering (Design & Innovation)"

Signature	:	………………………
Name of Supervisor	:	………………………
Date	:	………………………

SIMULATION OF CAVITATING FLOW IN CIRCULATING WATER PUMPS IN A POWER GENERATION INDUSTRY

LEE JIA CHANG

A project report submitted in partial
fulfillment of the requirements for the award of
the Degree of Bachelor Mechanical Engineering (Design & Innovation)

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

May 2008

"I hereby declared that this is my own work except the ideas and summaries which I have clarified their sources"

Signature	:	………………………
Author	:	………………………
Date	:	………………………

Special dedicate to my family, supervisor, my friends, and all that help me to finish my thesis.

ACKNOWLEDGEMENT

Vision, values, and courage are the main gift of this thesis. I am grateful for the inspiration and wisdom of many thoughts that have been instrumental in its formulation.

First of all, I wish to express my sincere appreciation to my supervisor, Mr. Lee Yuk Choi, for his guidance, advices, motivation, critics, and friendship. His time spent on teaching me and shared with me based on his professional knowledge and experience of centrifugal pump. Without his help and support, this PSM would not have been the same as presented here. I would also extend my thanks to Independent Power Producer's Maintenance Manager, Mr. Chew. Thanks you of your permission to let us visit to plant and sharing with me the manufacturer data sheet. Then to Mr. Kamal and Mr. Afzanizan, your helps and the computational ideal sharing are really benefits to me, thanks.

I would also like to thanks to a group of ANASYS CFX simulation team which is Shirley Ng, Rina, Farib, Rina, Ros, and Dud. Without you all, I have not that much initiative to run my impeller simulation. I am very appreciate of you guy guidance and all the time we been discuss and running simulation until late night. Although project was tough but the computational knowledge that we had develop together impresses me. Special thanks to Shirley, when I give up to the simulation project but you always encourage me to move forward.

Last but not least, I want to thank to my Mama and Papa, for their affection, prayer, and support throughout my study. Finally, I want to thank my lovely Ang Bee Luan, for his encouragement, care, and love. Without them, this work of mine will never be a success.

ABSTRACT

This thesis presents a study in simulating impeller in relation with the cavitating flow in circulating water pumps in a power generation industry by using ANSYS CFX. This study mainly focuses on the concept of cavitations in pump, pump performance curve, system pump curve, and net positive suction head (NPSH). The ultimate goal of this project is to determine the best operating pump range. It is interesting to examine the system pump curve prediction to identify the inception cavitation zone. Basically the major problem in cavitation can be avoided by providing higher net positive suction head requirement. However, there are still a lot of issues in designing a water pumps. Therefore, a theoretical system pump curve was generated using Microsoft Excel 2007. In addition, Catia V5 and ANSYS CFX were applied to create computational fluid dynamic model. From simulation results, a decrease of NPSHa values produces the onset of cavitations. The major findings of this thesis present the theoretical and numerical results concerning the pump characteristics and performance breakdown at different flow conditions and the theoretical pump curve at fluctuating sea water were compared, and therefore the best operating pump range is identified between a flow rate of 12,430 m^3/hr and 14,470 m^3/hr to avoid the occurrence of cavitation in pump. As a result, Independent Power Producer (IPP) can always control their pump at best efficiency point and retain operating pump at a good condition.

ABSTRAK

Tesis ini menerangkan tentang kajian simulasi pendesak yang berkaitan dengan pengaliran perongga dalam aliran pam air di dalam industri penjana kuasa dengan menggunakan ANYSY CFX. Tujuan utama kajian ini adalah untuk menentukan konsep perongga di dalam pam air, prestasi pam berlengkungan, sistem pam berlengkungan dan ‘Net Positive Suction Head’ (NPSH). Tujuan utama untuk project ini adalah mengenai penentuan jarak operasi terbaik bagi sebuah pam. Sistem pam berlengkungan ramalan diuji dan corak kavitasi diramal dan dikaji untuk mengenal pasti poin-poin dari pembentukan perongga awal dan perkembangannya.Secara asasnya, masalah utama perongga boleh dielakkan dengan mengekalkan keperluan ‘Net Positive Suction Head’ di tahap lebih tinggi.Walau bagaimanapun, masih terdapat banyak lagi isu dalam mereka bentuk pam air. Oleh sebab itu, Catia V5 dan ANSYS CFX telah diaplikasikan untuk membentuk ‘computational fluid dynamic’ model.Daripada keputusan simulasi, penurunan nilai NPSHa akan mengakibatkan pembentukan peronggaan. Penemuan utama dalam tesis ini menerangkan keputusan teori dan simulasi mengenai sifat-sifat pam dan prestasi pam di keadaan pengaliran yang berlainan telah dibanding. Maka, penentuan jarak operasi terbaik bagi sebuah pam ialah diantara 12,430 m^3/hr dan 14,470 m^3/hr supaya pam dapat mengelakkan pembentukan peronggaan. Kesimpulannya, Independent Power Producer boleh mengoperasikan pam dalam keadaan yang sempurna.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURE

LIST OF NOMENCLATURE

CAD	Computer-Aid Design
CFD	Computational Fluid Dynamic
p_o	static pressure, Pa
ρ_D	vapor pressure, Pa
H_v	Velovity Head,m
H_f	Friction Head, m
H_d	Total Discharge Head, m
H_s	Total Suction Head, m
H_{tot}	Total Head, m
NPSH	Net Positive Suction Head, m
NPSHr	Net Positive Suction Head Required, m
TDH	Total Deliver Head, m
P	static pressure
Pv	vapor pressure
3%	3% pump head drop
IPP	Independent Power Producer
BEP	Best Efficiency Point
h_p	Horse Power
Sp.gr	Specific Gravity

LIST OF APPENDICES

CHAPTER I

INTRODUCTION

1.1 Overview

A centrifugal pump is a simple device consisting of only a few parts that is designed to move liquids. All centrifugal pumps use an impeller and a stationary volute, also referred to as the casing or diffuser that houses the impeller (Sahdey, 2007). The pump casing provides a pressure boundary for the pump and contains channels to properly direct the suction and discharge flow. The pump casing has suction and discharge penetrations for the main flow path of the pump and normally has small drain and vent fittings to remove gases trapped in the pump casing or to drain the pump casing for maintenance.

Cavitating flow in circulating water pump is still continues to be the major problem concern for power plant industry. Cavitations are generally considered as a harmful phenomenon in the hydraulic transmission system, the efficiency of the system is reduced due to cavitations, and especially dynamic performances are deteriorated. When a pump cavitates, the vapour bubbles form in a low pressure region directly behind the rotating impeller vanes. These vapour bubbles then move toward the oncoming impeller vane, where they collapse and lead to physical shock to leading edge of the impeller vane. As the fluid becomes pressurized again in the pump, these bubbles implode, leading to pitting of the impeller and other pump components. In addition, since vapour has a lower density than liquid, cavitation leads to a reduction in the pump capacity and efficiency. Accordingly, a numbers of researches have been performed towards on designing of accommodating the effects of cavitations.

The net positive suction head (*NPSH*) is a measure of the proximity of a liquid to its bubble point (or vapour pressure). There are a few options available to increase *NPSHA* so that we could always control it below the *NPSHR*. Increasing the source pressure or reducing the fluid vapour pressure (by cooling) is rarely feasible. Therefore, there are two process variables remaining that can be adjusted which are the static head and friction losses. Static head can be raised by increase the elevation of the source inlet, lower the elevation of the pump inlet, and raise the level of fluid in the suction vessel. While for friction losses can be reduced either by increasing the diameter of the pump suction piping and reducing the equivalent length of suction line.

1.2 Objective of the Project

This project is about modeling and simulating impeller in relation with the cavitating flow in circulating water pumps in a power generation industry. The objectives are as follow:-

1. To understand the concept of cavitations in pump, pump performance curve, Net Positive Suction Head.
2. To model the impeller based on the actual dimensions used in the power generation industry.
3. To simulate the impeller under different flow rate conditions.

1.3 Scopes of the Project

The scope of the project is generally are as below:-

1. Literature study of impeller used in the industry and other related sources.
2. Calculate the performance characteristics of the pump over the range of flow rates.
3. Simulate the impeller using ANSYS CFX to obtain the results by consequently the NPSH to produce the onset of cavitations.
4. Examine the system pump curve prediction to identify the inception cavitation zone.

1.4 Problem Statement

The overall aim of this project is to investigate the cavity pattern using ANSYS CFX, computational fluid dynamics (CFD) software package. It is a finite-volume method to solve the governing equations for a fluid. By understanding the impeller used in current power plant industry and water pump characteristic, it will be an useful tool to assist in pump designing which user can also monitor the pump performance for necessary modifications of existing problem to avoid cavitations.

Besides, FLUENT is the alternative solution for cavity patterns analysis. FLUENT is a highly automated flow modeling tool that allows design and process engineers to rapidly and accurately validate their designs. Using FLUENT to rapidly analyze the impeller fluid flow and cavity problems throughout development happened during its operation in different condition. From existing impeller used in power generation industry, the 3-dimension of model is created using 3D-scanner and exported to ANSYS CFX or FLUENT to obtain the performance flow pattern to study and simulated the pump performance upon which the impeller is operating under cavitating and non-cavitating conditions to show the effect of cavitations mode as the source of impeller contribution.

Ever since the pump replacement in a power plant, minor pitting on the pump blades (impeller) was noted during periodic maintenances and inspections. These had been unscheduled breakdown failures, as a result of catastrophic failure of the pump arising from a broken impeller. Continuing operation in such a manner cannot warrant trouble free operations, improved pumps reliability and plant availability. Cavitation and flow excited vibration on the pump with frequent unscheduled failures was successfully detected with vibration based investigations. The investigations strongly implied a system design problem and pump operating conditions that had resulted failure of the impeller. Therefore, there is a need to overcome such unscheduled failure by performing a thorough study in the cavitation on the impeller.

CHAPTER II

LITERATURE REVIEW

In this chapter, I will present a brief overview review of impeller and cavitating flow in circulating water pumps literature which consists of several numerous studies from the past and presents. All the studies are features the theories that are explained cavity damage and phenomena. Background review on theoretical modal analysis is briefly explained with description on related studies in the area power plant industry. The final part of this chapter describes the fundamental theory of cavitating flow in circulating water pumps.

2.1 Basic of Cavitations

Cavitation is usually found in a fluid flows where the local pressure drops below saturation pressure of liquid, therefore liquid evaporation and generation of vapour bubbles in the low pressure region. Existence of cavitation can degrade the device performance, produce undesirable noise, lead to physical damage to the device and affect the structural integrity (Sahdey, 2007). According to Athavale finding (2000), after he applied a number of rotational cavitation problem encountered in turbo machines to assess its robustness and capability of predicting the occurrence, extent and strength of cavitaiton zones. When cavitation occurred is typically alters the pressure field on the working surfaces of the turbo machines which is charges in the blade pressure and charges in the blade loading. In addition, the predicted cavitation zone is located in the leading portion of the suction side. Besides, he simulated on various specific speed on a same centrifugal water pump, greater the specific speed can produce a bigger cavitation zone.

Kelecy (2003) defined the cavitation as the generation of vapour bubbles in a liquid due to a local reduction in pressure below the vapour pressure of the liquid at a given temperature.

According to Friedrichs (2003), rotating cavitation as two-phase instability phenomenon means major impacts on performance and reliability of hydraulic pumps whenever it occurs.

Medvitz (2002) examined and founded that Cavitation physics play an important role in the design and operation of many liquid handling turbomachines. In particular, cavitation can give rise to erosion damage, noise, vibration and hydraulic performance deterioration. Accordingly, a large body of research has been performed toward understanding the physics of, designing away from, and designing to accommodate the effects of cavitation.

2.2.1 Type of Cavitation in Centrifugal Pumps

Cavity development in a centrifugal pump is fully controlled by the discharge coefficient according to the relative flow velocity incidence angle at the impeller inlet (Avellan, 2004) which strongly affects the pressure distribution on the blades at the inlet. At the rated discharge, traveling bubble cavitation can be observed on the suction side of the blades. This type of cavity corresponds to a low incidence angle of the flow and depending on the design of the impeller, the minimum of pressure being at the impeller throat.

For a lower discharge value, the flow incidence is increased and then, a leading edge cavity appears. For low values of σ, cavitation vortices can be observed at the inlet of the runner coming from the leakage flow through the shroud seal.

2.2 Axial Inducer Influence on the Performances in Cavitating Regime

Inducer is generally placed upstream of a centrifugal or mixed flow impeler in order to improve its cavitation resistance. According to Bakir et al., (2003), they conducted an experiment of the influence of the shape of the leading edge and its sharpening on the cavitating behavior of an inducer. For high flow rates the stable cavitaties developed on both sides of the blade. In order to reduce the critical cavitation number,the studied show to incline the leading edge towards the outlet in the meridian view and towards the back for the front view. In addition, sharpening improves the performance in the vicinity of the nominal flow rate.

2.3 Numerical Prediction of Cavitation in a Centrifugal Pump

According to Kelecy (2003), the results of a numerical study of cavitating flow within a centrifugal pump. By carefully controlling the cavitation process through the adjustment of the pump exit pressure, it was shown that the cavitation model in Fluent 6.1 could accurately predict both the inception and subsequent development of cavitation bubbles within the pump impeller. Specifically, the model was able to reproduce both the sudden drop in pump head rise and final level of NPSH. In addition, local effects such as blade leading edge separation and an inlet shroud cavity bubble were captured by the model.

The fact that the steady-state solutions became more difficult to converge as the cavitation increased within the impeller suggests that an unsteady approach is warranted for highly cavitating flows. Indeed, the existence of rotating cavitation would require not only an unsteady solution procedure but also the representation of the entire impeller, as the periodic boundary condition would not apply. It would be of interest to apply the present cavitation model to an unsteady simulation of the pump model to see if this phenomenon could be captured numerically.

Besides that, Vaughan (2007) simulated the pump performance using ANSYS CFX simulation software then compared the simulated result to real performance data. Vaughan has examined and founded that data point can be connected to show total dynamic head, horsepower and efficiency with respect to flow. The performance data then can be extracted via CFD post-processor and a virtual performance curve. The simulated pump performance accurately predicted the actual pump performance in all models completed to date.

2.4 Cavitation Flow Calculations

According to Dupont, Okamura (2003), a benchmark for commercial codes was established using the calculations made by code vendors. This benchmark has shown CFX-Tascflow to be the most appropriate code for predicting cavitation performance regarding accuracy and for computing time in the process of an impeller design. This is because the method used in this code corresponds to a steady approach and is therefore much faster then the other codes that are using methods with unsteady approaches. Contrary to CFX-Tascflow, which performed a flow analysis in the impeller alone, the other code vendors chose to perform a calculation in the whole pump. This partially explains the large differences in the calculation times.

The results obtained with CFX-Tascflow for two similar impellers were compared with those of a simple cavity prediction method. The latter method is based on the solving of the Rayleigh-Plesset equation using the noncavitating pressure distribution along grid lines obtained using a three-dimensional Navier-Stokes equation. This simple approach has been shown to provide, in a very short