A STUDY ON VELOCITY PROFILE IN PUMP IMPELLER AND ITS CASING

NOR MERLISA BINTI ALI

A project report submitted in partial fulfillment of the
requirements for the award of the degree of
Master of Mechanical Engineering.

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

DISEMBER 2008

To my beloved papa and mama

ACKNOWLEDGEMENTS

Very special thanks to my project advisor, Dr. Jamaluddin Md. Sheriff for sharing with me his profound insights and give me guidance, ideas and support in doing this project.

ABSTRACT

This project is to study velocity profile in pump impeller and its casing using numerical simulations in commercial software COSMOSFloWorks. The pump has different number of blades of impeller and a circular casing. The numerical simulations are carried out with a multiple frame of reference to predict the velocity distribution around the pump impeller and the casing. The impeller is solved in a rotating frame and the casing is solved in a stationary frame. This study also shows that the number of blades in impeller, flowrate and the impeller speed give a great influence in velocity distribution in pump. The turbulence model used in this study is standard $k-\varepsilon$ Turbulence Model. The velocity distribution is observed by varying the impeller speed and the flowrate for every number of blades in impeller.

ABSTRAK

Projek ini adalah untuk mengkaji bentuk halaju di dalam penolak pam dan sarungnya dengan menggunakan simulasi penomboran dalam perisian komersial COSMOSFloWorks. Pam ini mempunyai nombor bilah yang berlainan pada penolaknya dan satu sarung yang berpusing bentuknya. Simulasi penomboran yang dijalankan menggunakan rujukan bingkai yang pelbagai untuk mengjangka taburan halaju di sekeliling penolak pam sarungnya. Penolak pam diselesaikan di dalam bingkai yang berputar manakala sarung pam diselesaikan menggunakan bingkai yang pegun. Kajian ini juga menunjukkan bahawa bilangan bilah pada penolak pam, kadar aliran dan kelajuan penolak pam memberi pengaruh yang besar dalam taburan halaju di dalam pam. Model Golakan (Turbulence Model) yang digunakan di dalam kajian ini ialah 'standard $k-\varepsilon$ Turbulence Model'. Taburan halaju dikaji dengan mengubah halaju penolak pam and kadar aliran bagi setiap penolak pam yang mempunyai bilah yang berbeza- beza.

TABLE OF CONTENTS

II LITERATURE REVIEW

III RESEARCH METHODOLOGY

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

Q	-	Flowrate
n	-	Speed
H	-	Discharge head
P	-	Pressure
ρ	-	Density
V	-	Volume
T	-	Temperature
t	-	Time
τ	-	Shear stress
k	-	Turbulent kinetic energy
ε	-	Turbulent dissipation
μ_e	-	Effective viscosity
ℓ	-	Turbulent length scale
υ_t	-	Turbulent viscosity
$C_1, C_1, C_2, \sigma_k, \sigma_t$	-	Constants
K	-	Bulk Modulus of elasticity
υ	-	Kinematic viscosity
μ	-	Dynamic viscosity

LIST OF APPENDICES

CHAPTER I

INTRODUCTION

1.1 Background of Project

This project is to study the velocity profile in pump impeller and its casing using numerical methods. The method is using Computer Fluid Dynamics (CFD) by numerical simulations in Solidworks (COSMOSFloWorks). A selected built-in model of centrifugal pump is used in the numerical simulations. Centrifugal pump is basically used to convert mechanical energy into hydraulic energy by centrifugal force on the liquid used as a device to increase the pressure. Many researches have been done to optimize the design of centrifugal pump by studying on its velocity profile either at the pump impeller or in the casing. Some of the researches are based on a purpose to eliminate the noise from the pump while it is working which produced from the passing blade. But in this study, it is purposely to determine the velocity profile at the pump impeller and its casing. Besides that, it is also to gain knowledge about the flow distribution around the impeller and the casing of centrifugal pump by using the well-known numerical method, CFD. By running several steps of simulations in the commercial code COSMOSFloWorks, the flow distribution around the impeller and the casing can be visualized. Some important parameters such as velocity, pressure and the

turbulent energy are discussed further in the next chapter. The turbulent model which is used in this study is the standard $K-\varepsilon$ (two equations) turbulence model. The numerical simulations by using the commercial code COSMOSFloWorks are used in this study because it provides a faster way of simulation in order to study the flow analysis especially in rotating components such as centrifugal pump.

1.2 Problem Statement

This study looks ahead to determine the velocity profile at the impeller and its casing of selected centrifugal pump using numerical simulation. The centrifugal pump will use variable speed and flow rate and water as its working fluid. The results obtained will be analyzed to see the difference of velocity profile after applying variable speed and flow rate to the pump model which has different number of blades.

1.3 Objectives of the Project

The objectives of this project include:

i. To study the velocity profile at the pump impeller and the casing by using Computational Fluid Dynamics (CFD).
ii. To study the circulating flow around the impeller and the casing.
iii. To observe the velocity distribution at the impeller and the casing for different impeller speed and flowrate.

1.4 Scope of Project

The major focus of this project is:

i. Literature review on numerical approach using CFD especially on determination of the velocity profile in rotating impeller and the casing of centrifugal pump.
ii. This project will be using the selected centrifugal pump model with variable motor speed and flowrate.
iii. Water will be used its working fluid in the centrifugal pump.

1.5 Methodology

i. Collecting information from journals and books about the related researches of determination of velocity profile in pump impeller and the casing of centrifugal pump.
ii. Run the numerical simulation in Solidworks (COSMOS FloWorks) using the model in the software.
iii. Analyzing the results of the numerical simulations.
iv. Developing conclusions, recommendations and implications of this study.

1.6 Significance of Project

i. Encouragement for lots more studies and researches on the determination of flow behavior in pump impeller and the casing by using other CFD software and flow visualization methods such as PIV.

ii. An idea to other student or new researcher to do research project on flow behavior (velocity profile) or other parameter such as pressure in pump impeller and other component in centrifugal pump.

iii. An idea of determination of other flow parameters using other turbulence model like RNG- $K-\varepsilon$ turbulence.

1.7 Thesis Outline

The thesis is organized into five chapters. Chapter I contains of introduction, objectives, methodologies used, scope and the significance of this study. The idea of this study is explained in the introduction. Besides that, the problem statement of this case study is also stated as the reason this project is launched. Chapter II explains about the three keywords in this study: the centrifugal pump, velocity profile, and CFD. All the governing equations, the Navier-Stokes equation are also explained in this chapter. The related researches in other journals and books are reviewed in this second chapter. Chapter III explains about the methodology of this study which has been used to complete the project. The procedures of numerical simulation which is carried out by using the commercial software COSMOSFloWorks are also discussed in this chapter. The methods are discussed deeper including the initial and boundary conditions used in this numerical simulation. Chapter IV discusses about the results of the numerical simulation and the analysis of the result. Some figures of the velocity vectors and the

velocity contours around the impeller are showed and explained. The last chapter concludes the study, including with the recommendations for the future use and research that might be used for others.

REFERENCES

1. Reza Aghaei tog, A. Mesgharpoor Tousi, M. Soltani. "*Design and CFD Analysis of Centrifugal Compressor for a Microgasturbine"*. Aircraft Engineering and Aerospace Technology: An International Journal, Vol.79/2. 2007, 137 – 143.
2. John D. Anderson,Jr. (1995). *"Computational Fluid Dynamic: The Basics with Applications*". McGraw-Hill,Inc.
3. Li Xian-hua, Zhang Shu-jia, Zhu Bao-lin, Hu Qing-bo. "*The Study of $k-\varepsilon$ Turbulence Model for Numerical Simulation of Centrifugal Pump"*. The MOE Key Laboratory of Mechanical Manufacture and Automation, Zhejiang University of Technology.
4. S.M. Miner et al. *"Centrifugal Pumps: Two Dimensional Flow Analysis of a Laboratory Centrifugal Pump"*. Turbomachinery Digest. Vol.12, No. 1.1991, 6-7.
5. C.H.Liu, C. Vafidis, J.H. Whitelaw. *"Flow Characteristics of a Centrifugal Pump"*. ASME Journal of Fluids Engineering, Vol. 116. June 1994, 303-309.
6. L.P.Chua, S.C.M. Yu, H.L. Leo, W.K. Chan. "*Comparison of Flow Characteristics of Enlarged Blood Pump Models with Different Impeller Design"*. Int. Com. Heat Mass Transfer, Vol. 26, No.3. 1999, 369-378.

7. Jose Gonzalez, Joaquin Fernandez, Eduardo Blanco, Carlos Santolaria. *"Numerical Simulation of the Dynamic Effects Due to Impeller-Volute Interaction in a Centrifugal Pump"*. ASME Journal of Fluid Engineering, Vol. 124. June 2002, 348-355.

8. Per Peterson, Magnus Larson, Lennart Jonsson. "*Measurement of the Velocity Field Downstream of an Impeller"*. ASME Journal of Fluids Engineering, Vol.118. September 1996, 602-610.

9. Shuliang Cao, Guoyi Pang, Zhiyi Yu. *"Hydrodynamic Design of Rotodynamic Pump Impeller for Multiphase Pumping by Combined Approach of Inverse Design and CFD Analysis"*. ASME Journals of Fluid Engineering, Vol 127. March 2005, 330-338.

10. Tarek Meakhail, Seung O-Pak. *"A Study of Impeller- Diffuser- Volute Intreaction in a Centrifugal Fan"*. ASME Journals of Turbomachinery, Vol. 127. January 2005, 84-90.

11. M. Zangeneh, A. Goto, T. Takema. "*Suppression of Secondary Flows in a Mixed- Flow Pump Impeller by Application of Three- Dimensional Inverse Design Method, Part 1- Design and Numerical Validation"*. ASME Journals of Turbomachinery, Vol. 118. July 1996, 536-543.

12. Deepthi Duddempudi, Yufeng Yao, David Edmondson, Jun Yao, Andy Curley. "*Computational Study of Flow over Generic Fan-Wing Airfoil"*. Aircraft Engineering and Aerospace Technology: An International Journal, Vol.79/3. 2007, 238 – 244.

13. A. Akhras, M. El Hajem, R. Morel, and J.-Y. Champagne, *"Internal flow investigation of a centrifugal pump at the design point"*. Journal of Visualization, Vol. 4. 2001, 91–98.

14. N. Pedersen, P. S. Larsen, and C. B. Jacobsen. *"Flow in a centrifugal pump impeller at design and off-design conditions—part I: particle image velocimetry (PIV) and laser Doppler velocimetry (LDV) measurements"*. ASME Journal of Fluids Engineering, Vol. 125. 2003, 61–72.

15. K.U.Ziegler, H. E. Gallus, and R. Niehuis, *"A study on impeller-diffuser interaction—parts I: influence on the performance"*. *ASME* Journal of Turbomachinery, Vol. 125. 2003, 173–182.

16. F. Shi and H. Tsukamoto, *"Numerical study of pressure fluctuations caused by impeller-diffuser interaction in a diffuser pump stage,"* ASME Journal of Fluids Engineering, Vol. 123. 2001, 466–474.

17. Y. K. P. Shum, C. S. Tan, and N. A. Cumpsty, *"Impeller-diffuser interaction in a centrifugal compressor"*. ASME Journal of Turbomachinery, Vol. 122. 2000, 777–786.

18. S. S. Hong and S.-H. Kang, *"Flow at the centrifugal pump impeller exit with circumferential distortion of the outlet static pressure"*. ASME Journal of Fluids Engineering, Vol. 126. 2004, 81–86.

19. D. Hagelstein, K. Hillewaert, R. A. Van den Braembussche, A. Engeda, R. Keiper, and M. Rautenberg. *"Experimental and numerical investigation of the flow in a centrifugal compressor volute"*. ASME Journal of Turbomachinery, Vol. 122. 2000, 22–31.

20. R. K. Byskov, C. B. Jacobsen, and N. Pedersen. *"Flow in a centrifugal pump impeller at design and off-design conditions—part II: large eddy simulations"*. ASME Journal of Fluids Engineering, Vol. 125. 2003, 73–83.

21. F. Gu, A. Engeda, M. Cave, and J.-L. Di Liberti, *"A numerical investigation on the volute/diffuser interaction due to the axial distortion at the impeller exit"*. ASME Journal of Fluids Engineering, Vol. 123. 2001, 475–483.

22. A. Goto, M. Nohmi, T. Sakurai, and Y. Sogawa, *"Hydrodynamic design system for pumps based on 3-D CAD, CFD, and inverse design method"*. ASME Journal of Fluids Engineering, Vol. 124. 2002, 329–335.

23. J. González, J. Fernández, E. Blanco, and C. Santolaria. *"Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump"*. ASME Journal of Fluids Engineering, Vol. 124. 2002, 348–355.

24. O. Coutier-Delgosha, R. Fortes-Patella, J. L. Reboud, M. Hofmann, and B. Stoffel. *"Experimental and numerical studies in a centrifugal pump with two-dimensional curved blades in cavitating condition"*. ASME Journal of Fluids Engineering, Vol. 125. 2003, 970–978.

25. S. Khelladi, S. Kouidri, F. Bakir, and R. Rey. *"Flow study in the impeller-diffuser interface of a vaned centrifugal fan"*. ASME Journal of Fluids Engineering, Vol. 127. 2005, 495–502.

26. J.M. Sivo, A.J. Acosta. C.E. Brennen, T.K.Cauhey. "*Laser Velocimeter Measurement in the Leakage Annulus of a Whirling Shrouded Centrifugal Pump"*. Laser Anemometry – 1994: Advances and Applications (ASME 1994), Vol. 191.111-117.

27. Ian C. Shepherd, R. Frank La Fontaine, Laurie W. Welch, Ron J.Downie. "*Velocity Measurement in Fan Rotors Using Particle Image Velocimetry"*. Laser Anemometry – 1994: Advances and Applications (ASME 1994), Vol. 191. 179-183.

28. Thomas C. Fu, Joshua Weltz. "*A High Spatial Resolution Particle Displacement Velocimetry Image Analysis Algorithm"*. Laser Anemometry – 1994: Advances and Applications (ASME 1994), Vol. 111.203-208.

29. Nahla M. Zaki, Jiin- Jen Lee. "*Laser Doppler Velocimeter Measurements in Separated Flows in Circular Pipe"*. Laser Anemometry – 1994: Advances and Applications (ASME 1994), Vol.111. 149 – 154.

30. William M. Humphreys, Jr., Scott O. Kjelgaard, Scott M. Bartam. "*A Time-Resolved Particle Image Velocimeter for Unsteady Flow Field Studies"*. Laser Anemometry – 1994: Advances and Applications (ASME 1994), Vol. 111. 161 – 166.

31. J. Wu, J. Sheridan, J. Soria, M.C. Welsh. "*An Investigation of Unsteady Flow Behind a Circular Cylinder Using a Digital PIV Method"*. Laser Anemometry – 1994: Advances and Applications (ASME 1994), Vol.111. 167 – 172.

32. A. Shekarriz, J. R. Philips, T.D. Weir. "*Experimental Study of a Pseudoplastic Jet Using Particle Image Velocimetry"*. Laser Anemometry – 1994: Advances and Applications (ASME 1994), Vol. 111. 191 – 201.

33. T.J McGregor, D.J.Spence, D.W. Coutts. "*Laser-based Volumetric Colour-Coded Three-Dimensional Particle Velocimetry"*. Optics and Lasers in Engineering 45, 2007, 882 – 889.

34. COSMOSFloWorks 2006 Tutorial Manual.

35. http://www.cfd-online.com

36. http://www.cheresources.com/centrifugalpumps3.shtml

37. http://www.engineersedge.com/pumps/centrifugal_pump.htm

38. John F. Douglas, Janusz M. Gasiorek, John A. Swaffield. (2001). *"Fluid Mechanics*". 4th Edition. Prentice Hall.