Numerical simulations and measurements of a heart pump

Song, Guo Liang

2008

Song, G. L. (2008). Numerical simulations and measurements of a heart pump. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/41533>

<https://doi.org/10.32657/10356/41533>

Numerical Simulations and Measurements of a Heart Pump

Song Guoliang

School of Mechanical & Aerospace Engineering

A thesis submitted to the Nanyang Technological University
in fulfilment of the requirement for the degree of
Doctor of Philosophy

2008

ABSTRACT

Artificial heart pumps have been developed and advanced swiftly in recent years for their successful clinical applications as heart assisting devices or replacements of the malfunctioning hearts. There are many design criteria that have to be satisfied in order to develop a clinically effective and safe cardiac prosthesis, of which antitraumatic and antithrombogenicity are the two most important criteria which are closely related to the flow characteristics within the pump. Therefore, hemodynamics of the Kyoto-NTN magnetically suspended centrifugal blood pump is investigated in this study. The present investigation aims to understand the flow field in the pump with three different impeller blade designs, namely, 16 forward-bending blades (16FB), 16 straight blades (16SB) and 8 backward-bending blades (8BB). Numerical simulations and flow measurements were performed using the commercial software package, FLUENT, and laser Doppler anemometer (LDA) respectively under the operating condition of a 1:1 pump model. This is the first time that the LDA is applied to measure the flow of a pump model with the same size of the prototype. The close agreement between the measured and numerical results shows that the measurements have validated the numerical simulation model which has revealed faithfully the inner flow pattern of the blood pump.

The results show that the double volute design of the Kyoto-NTN magnetically suspended centrifugal blood pump has dominant effects on the flow in the blade channel and results in axis symmetrical pressure distribution in the volute as well as the axis symmetrical flow patterns in the impeller blade channels of all the three pump models investigated.

Meanwhile, the different impeller blade profiles also have significant effects on the flow in the pump. Under the same flow rate and impeller rotating speed, the 16SB impeller

generates the highest pressure head among the three pump models and the 8BB impeller produced the lowest. Circulating flow is identified in the 16SB and 16FB blade channels, while the flow in the 8BB blade channel is much smoother. Furthermore, the different impeller blade profiles have induced different levels of shear stress in the flow. The pump with 16FB impeller generates the highest shear stress to the blood among the three pump models, while the 8BB impeller produces the lowest shear stress. Through the comparison of the three impeller blade profiles studied, it is found that there is no a single impeller among the three models that is superior to the others in all aspects. The 16SB impeller, which produces relatively low shear stress and high pressure head, could be a compromising choice.

The comparison between the measurement results of the present 1:1 and the 5:1 pump models reveals that the 5:1 pump model, which was built according to the flow similarity law, has caused notable differences to the flow patterns in the blade channels from those of the 1:1 model. The difference could be due to the incomplete similarity of the enlarged pump model to the prototype. Although the enlarged pump model may have some similar dimensionless coefficients as those of the prototype, the incomplete similarity would render the enlarged model a failure in producing a well analogous inner flow with respect to those of prototype. It is therefore important to have the study on the 1:1 pump model if complete similarity cannot be totally attained in the enlarged pump model.

ACKNOWLEDGEMENTS

I would firstly express my deep appreciation to my supervisor, A/P Chua Leok Poh, for his invaluable advice and guidance throughout this work.

I would also express my thanks to Associate Professor Zhou Tongming for his enlightening comments and discussions. Thanks also to Associate Professor Yu, Ching Man Simen for his invaluable professional advices. My thanks also to Mr. Eric Yap of Fluid Mechanic Laboratory for his professional technical support, without which my experiment could never be carried out smoothly. Thanks also to Mr. Yuan Kee Hock for his efficient help in preparing the experimental materials and constructive ideas. I must also thank the computing staff, Mr. Teo Hai Beng, of Center for Advanced Numerical Engineering Simulations of MAE for sorting out all my computing problems.

My gratitude must also be given to many close friends, who have given me many supports and encouragements throughout the work. Special thanks must go to Dr. Jia Qipeng for his precious assistance in preparing the chemical solution and Mr. Wong Junjie for his valuable help in getting rid of my grammatical mistakes. Sincere gratitude must also go to Dr. Zhang Junmei for her precious advices in the numerical simulation, as well as Dr. Zhang Dongsheng and Zhou Wenxiu, whose suggestions in the experiment are much appreciated.

Last but not least, I would like to express my eternal gratitude to my family. Their morale and consistent supports have always inspired me to do my best.

TABLE OF CONTENTS

COVER PAGE	i
TITLE PAGE	ii
ABSTRACT	iii
ACKNOWLEDGEMENTS	v
TABLE OF CONTENTS	vi
NOMENCLATURE	x
LIST OF FIGURES	xiii
LIST OF TABLES	xix
CHAPTER 1 INTRODUCTION	01
1.1 Clinical Significance of Artificial Heart Pumps	01
1.2 The Centrifugal Blood Pump in the Present Study	03
1.3 Objectives and Scopes	06
1.4 Thesis Outline	09
CHAPTER 2 LITERATURE REVIEW	10
2.1 The Development of Mechanical Cardiac Support Devices	10
2.2 Ventricular Assist Devices and Total Artificial Hearts	11
2.2.1 Total Artificial Heart	12
2.2.2 Ventricular Assist Devices	15
2.2.2.1 Intra-Aortic Balloon Pump	16
2.2.2.2 Pulsatile Ventricular Assist Devices	17
2.2.2.3 Non-pulsatile Ventricle Assist Devices	19
a) Roller Pumps	20
b) Axial Blood Pumps	20
c) Centrifugal Blood Pumps	22
2.3 Design Considerations	25
2.3.1 Centrifugal Pumps	25
2.3.2 Flow Similarity	28
2.3.3 Blood Traumatic	31

Table of Contents

2.3.4	Thrombogenicity	34
2.3.5	Infection and Contacting Surfaces	36
2.4	Non-pulsatile Blood Pump as a Substitution of Pulsatile Pump	38
2.4.1	Comparison between Pulsatile and Non-pulsatile Blood Pumps	39
2.4.2	Physiological Considerations	41
2.5	Study Approaches in Rotary Blood Pumps	44
2.5.1	Hydraulic Performance Test	44
2.5.2	Hemolysis Test	46
2.5.3	Animal Test	47
2.5.4	Flow Visualization and Computational Fluid Dynamics	47
2.6	Previous Studies on the Kyoto-NTN Centrifugal Pump	53
CHAPTER 3	NUMERICAL METHODS AND THEORY	57
3.1	Physical Models	57
3.2	Governing Equations (Mathematical Model)	62
3.3	Numerical Calculation Method	67
3.3.1	Mesh Generation	67
3.3.2	Using Finite Volume Method to Solve Governing Equations	72
3.4	Boundary Conditions and Calculation Configurations	75
3.4.1	Boundary Conditions and Assumptions	75
3.4.2	Configurations of Calculation	78
3.5	Numerical Validation	80
3.6	Post-processing	82
CHAPTER 4	LDA MEASUREMENT METHODOLOGY	84
4.1	Experimental Pump Model	84
4.1.1	Experimental Flow Circuit	84
4.1.2	The Pump Unit	85
4.2	Measurements with Laser Doppler Anemometer (LDA)	89
4.2.1	Basic Theory of LDA	89
4.2.2	Description of LDA System	91

4.3	Working Fluid	94
4.4	Experimental Arrangement and Procedure	96
4.5	Experimental Uncertainties	102
CHAPTER 5	SIMULATIONS OF FLOW IN BETWEEN THE IMPELLER BLADES FOR THE THREE PUMP MODELS	107
5.1	Flow in the 16FB Pump Model	107
5.2	Flow in the 16SB Pump Model	123
5.3	Flow in the 8BB Pump Model	137
5.4	Comparison of the Shear Stress of the Pump Models	149
5.5	A Brief Summary	162
CHAPTER 6	NUMERICAL RESULTS OF FLOW IN THE GAP	166
6.1	Numerical Results of Flow in the Gap	167
6.2	A Brief Summary	191
CHAPTER 7	MEASUREMENTS OF FLOW IN BETWEEN THE IMPELLER BLADES FOR THE THREE PUMP MODELS	192
7.1	Flow in the 16FB Impeller Passages	193
7.2	Flow in the 16SB Impeller Passages	199
7.3	Flow in the 8BB Impeller Passages	204
7.4	Limitation of Dimensional Analysis	208
7.5	Particle Flow Angle at the Inlet and Outlet of Impeller	213
7.5.1	Flow Angle of Impeller 16FB	216
7.5.2	Flow Angle of Impeller 16SB	217
7.5.3	Flow Angle of Impeller 8BB	218
7.5.4	Comparison of Flow Angle of the Different Impellers	219
7.6	A Brief Summary	221
CHAPTER 8	CONCLUSIONS AND FUTURE WORKS	228
8.1	Conclusions	228
8.2	Future Works	232

REFERENCES	R-1
GLOSSARY	G-1

NOMENCLATURE

a	Local speed of sound (m/s)
a_{nb}	Influence coefficients for neighboring control volumes
a_p	Coefficient value at the center of the control volume
A	Cross sectional area (m^2)
b	The contribution of constant part from source term and boundary conditions
b	The height of the impeller blades (m)
C_p	Isotonic specific heat of the fluid ($kJ / kg \cdot K$)
d	Thickness of the pump gaps (m)
d_l	Diameter of the laser beam (m)
d_m	Width of the laser measurement volume (m)
D	Impeller diameter (m)
D_H	Hydraulic diameter (m)
E	The Euler's head (m)
E_u, E_v, E_R	Error in the measured velocity component in X direction, Y direction and resultant velocity
f	Incident laser light frequency (Hz)
f_r	Roughness function
F_i	Body force exerted on the working fluid (N / kg)
H	Pump pressure head ($mmHg$)
i, j, k	Coordinate index
k	Turbulence kinetic energy
L	Characteristic length (m)
l_m	The length of the laser measurement volume (m)
N	Impeller rotating speed ($1/s$)
N_s	Number of the data sample

p	Static pressure (Pa)
p_0	Total pressure (Pa)
p_e	Wetted perimeter of duct (m)
P	Pump driving power (W)
Q	Pump volume flow rate ($liter/min$)
r	Radius vector with respect to the center axis of the impeller
Re	Reynolds number, defined as $Re = \rho u d / \mu$
R^Φ	Unscaled residual
R^{Φ_s}	Scaled residual
S_c	Constant part of the source term
S_m	The source mass added to the continuous phase
S_p	Part of the source term that correlated with Φ_p
S_{scalar}	Scalar shear stress (Pa)
t	Time (s)
T	Temperature (K)
T	Torque (Nm)
u	Measured velocity component in X direction (m/s); Tangential velocity of the rotating impeller (m/s)
u_i	Velocity in i direction (m/s)
$\bar{u}'_i$	Time averaged velocity fluctuations (m/s)
u_t	Pump tip speed (m/s)
$\bar{u}, \bar{v}$	Sample mean of velocity component in X and Y direction respectively in measurement (m/s)
v	Measured velocity component in Y direction (m/s); Absolute velocity of flow (m/s)
v_r	radial velocity of flow (m/s)
v_w	tangential velocity of flow, swirl velocity (m/s)
$\bar{U}$	Velocity vector
V	Flow velocity (m/s)

W	Relative velocity of flow (m/s)
x_i	The Cartesian coordinates
X, Y, Z	Orientations in the measurement (m)
y	Normal distance between the computational node and the flow wall boundary (m)
z	The distance from the lower shroud to the cutting plane of the flow field in the impeller (m)
β	Fluid flow angle relative to the impeller tangential line (in degree)
β_1	Blade leading edge angle (in degree)
δ	Fringe spacing (m)
δ_{ij}	Kronecker Delta, which equals to 1 for $i = j$ and 0 for $i \neq j$
δ_r	Absolute roughness of the pump inner surface (m)
ε	Turbulence kinetic energy dissipation rate
η_{ij}	Reynolds stress tensor (Pa)
θ_i	Incident angle between the two laser beams (in radian)
θ_R	Velocity angle (in radian)
λ	Incident laser light wavelength (m)
μ	Viscosity for the working fluid ($Pa \cdot s$)
μ_t	Turbulence viscosity ($Pa \cdot s$)
ν	Kinematic viscosity of the working fluid (m^2 / s)
ξ	Distance measured from the pump housing to the cutting plane in the gap (m)
π_H	Head coefficient
π_P	Power coefficient
π_Q	Flow coefficient
π_{Re}	Reynolds number
ρ	Density of the working fluid (kg/m^3)
σ	Surface tension force (N/m)

τ_{ij}	Molecular stress tensor (Pa)
Φ_{nb}	Variable values for neighboring cells
Φ_p	Variable values for the center cell
ω	Rotating speed (rad/s)

LIST OF FIGURES

Figures	Descriptions	Page
Figure 1.1	Magnetically suspended centrifugal pump. (a) A schematic cross sectional view of the pump. (b) Impeller and configuration of magnetic bearing with permanent magnets (left), impeller and pump housing (middle), and electro- and permanent sensors (right). 1, Impeller; 2, electro-magnets; 3, permanent magnets; 4, gap sensor; 5, motor; 6, partition; 7, spiral casing; 8, pump casing	4
Figure 2.1	Classifications of Mechanical Circulatory Support Systems	11
Figure 2.2	Jarvik-7 Artificial Heart	13
Figure 2.3	AbioCor Artificial Heart	14
Figure 2.4	Schemes of Intra-Aortic Balloon Pump	16
Figure 2.5	HeartMate I Ventricular Assist Devices	18
Figure 2.6	Novacor Ventricular Assist Devices	19
Figure 2.7	MicroMed DeBakey axial blood pump	21
Figure 2.8	Medtronic centrifugal blood pump	23
Figure 2.9	Gyro Centrifugal blood pump	23
Figure 2.10	HeartMate III Centrifugal blood pump	24
Figure 2.11	Schematic view of centrifugal pump	26
Figure 2.12	The velocity triangles at the inlet and outlet of impeller	27
Figure 2.13	Illustration of flow induced blood damage by shear stress and exposure time (Paul et al., 2003)	33
Figure 2.14	Different levels of implantable cardiac prosthesis (Nosé et al., 2000)	38
Figure 2.15	Schematic diagram of the mock-loop circuit used in the hemolysis test	47
Figure 3.1	Basic structures of Akamatsu blood pump	57
Figure 3.2	A schematic view of pump configuration	58
Figure 3.3 (a)	Profiles and dimensions of the 16FB impeller	59

List of Figures

Figure 3.3 (b)	Profiles and dimensions of the 16SB impeller	60
Figure 3.3 (c)	Profiles and dimensions of the 8BB impeller	60
Figure 3.4	Physical model of the pump	61
Figure 3.5	An inner view of the model structure	62
Figure 3.6 (a)	Grid generation for the 16FB pump model	70
Figure 3.6 (b)	Grid generation for the 16SB pump model	70
Figure 3.6 (c)	Grid generation for the 8BB pump model	71
Figure 3.7	Overview of the segregated solution method	74
Figure 3.8	Mass flow rate at the pump inlet under different mesh schemes	81
Figure 4.1	Schematic view of the pump flow circuit of the experiment	84
Figure 4.2	Schematic of the acrylic casing of the pump model	86
Figure 4.3	Driving mechanism of the experimental pump model	88
Figure 4.4	Detail sectional driving of the pump unit, hollow shaft and center shaft	89
Figure 4.5	A schematic of the laser Doppler anemometer system layout	91
Figure 4.6	A schematic view of the laser transmitting system and the measurement volume	93
Figure 4.7	A schematic view of measurement windows for the rotating impeller passages	98
Figure 5.1	Distributions of static pressure in the volute and the impeller	108
Figure 5.2	Velocity distribution at the pump inlet of the 16FB pump model	109
Figure 5.3	Absolute velocity distribution at center plane of the 16FB pump	110
Figure 5.4	3-D view of the relative velocity distribution in the 16FB impeller channels	113
Figure 5.5 (a)	Positions of the view planes	114
Figure 5.5 (b)-(m)	Sectional views of the flow at the outlet of the blade channels and volute of the 16FB pump model	115-116

Figure 5.6	Static pressure distributions at the center plane of the 16SB pump model	123
Figure 5.7	Velocity distribution at the inlet of the 16SB pump model	124
Figure 5.8	Velocity distribution at center plane of the 16SB pump	125
Figure 5.9	3-D view of the relative velocity distribution in the 16SB impeller channels	127
Figure 5.10 (a)	Positions of the view planes	129
Figure 5.10 (b)-(m)	Sectional views of the flow at the outlet of the blade channels and volute of the 16SB pump model	129-131
Figure 5.11	Static pressure distributions at the center plane of the 8BB pump model	137
Figure 5.12	Velocity distribution at the pump inlet of the 8BB pump model	138
Figure 5.13	Velocity distribution at center plane of the 8BB pump	139
Figure 5.14	3-D view of the relative velocity distribution in the 8BB impeller channels	141
Figure 5.15 (a)	Positions of the view planes	142
Figure 5.15 (b)-(m)	Sectional views of the flow at the outlet of the blade channels and volute of the 8BB pump model	142-144
Figure 5.16 (a), (b)	Scalar shear stress distributions in the 16FB pump model with different impeller locations, note that impeller position in (a) is 11.25° ahead of (b)	151
Figure 5.17 (a), (b)	Mass weighted distribution of scaled shear stress within the 16FB pump model. The impeller position in (a) is 11.25° ahead of (b)	152
Figure 5.18 (a), (b)	Scalar shear stress distributions in the 16SB pump model with different impeller locations, note that impeller position in (a) is 11.25° ahead of (b)	155
Figure 5.19 (a), (b)	Mass weighted distribution of scaled shear stress within the 16SB pump model. The impeller position in (a) is 11.25° ahead of (b)	156
Figure 5.20 (a), (b)	Scalar shear stress distributions in the 8BB pump model with different impeller locations, note that impeller position in (a) is 22.5° ahead of (b)	158

List of Figures

Figure 5.21(a), (b)	Mass weighted distribution of scaled shear stress within the 8BB pump model. The impeller position in (a) is 22.5° ahead of (b)	159
Figure 5.22	Comparison of mass weighted distributions of the scaled shear stress of different pump models	161
Figure 5.23	Head capacity curves of the 16FB, 16SB and 8BB pump models	163
Figure 6.1	A schematic view of pump configuration	167
Figure 6.2	Static pressure distributions at the exit of the 16FB, 8BB and 16SB impellers	168
Figure 6.3 (a)	The static pressure distributions in the gaps of the 16FB pump model	169
Figure 6.3 (b)	The static pressure distributions in the gaps of the 8BB pump model	169
Figure 6.3 (c)	The static pressure distributions in the gaps of the 16SB pump model	170
Figure 6.4	Tangential velocity distributions at radius of 20 mm but at two different angular locations in the gap of the 16FB pump model	171
Figure 6.5 (a)	Radial velocity profiles at outer radii of the gap at angular position of 0° of the 16FB pump model	173
Figure 6.5 (b)	Radial velocity profiles at inner radii of the gap at angular position of 0° of the 16FB pump model	173
Figure 6.5 (c)	Analytical result of the radial flow in the parallel facing gap [Chan et al., 2000]	173
Figure 6.6 (a)	Velocity distributions in the gap of the 16FB pump model at view planes of $\xi/d=0.25$	175
Figure 6.6 (b)	Velocity distributions in the gap of the 16FB pump model at view planes of $\xi/d=0.5$	176
Figure 6.6 (c)	Velocity distributions in the gap of the 16FB pump model at view planes of $\xi/d=0.75$	176
Figure 6.7 (a)	The radial velocity distributions at different radii at center plane ($\xi/d=0.5$) of the gaps of 16FB pump model	178
Figure 6.7 (b)	The radial velocity distributions at different radii at center plane ($\xi/d=0.5$) of the gaps of 8BB and pump model	178

List of Figures

Figure 6.7 (c)	The radial velocity distributions at different radii at center plane ($\xi / d = 0.5$) of the gaps of 16SB pump model	179
Figure 6.8 (a)	Measurements of velocity vector distributions at center plane ($\xi / d = 0.5$) of the gap [Chua et al. (2002)]	180
Figure 6.8 (b)	Measurements of the radial velocity distributions at different radii at center plane ($\xi / d = 0.5$) of the gap [Chua et al. (2002)]	181
Figure 6.9 (a)	The radial velocity distributions at different radii at the plane of $\xi / d = 0.25$ of the gaps of 16FB pump model	182
Figure 6.9 (b)	The radial velocity distributions at different radii at the plane of $\xi / d = 0.25$ of the gaps of 8BB pump model	183
Figure 6.9 (c)	The radial velocity distributions at different radii at the plane of $\xi / d = 0.25$ of the gaps of 16SB pump model	183
Figure 6.10 (a)	The radial velocity distributions at different radii at the plane of $\xi / d = 0.75$ of the gaps of 16FBpump model	184
Figure 6.10 (b)	The radial velocity distributions at different radii at the plane of $\xi / d = 0.75$ of the gaps of 8BB pump model	184
Figure 6.10 (c)	The radial velocity distributions at different radii at the plane of $\xi / d = 0.75$ of the gaps of 16SB pump model	185
Figure 6.11 (a)	WSS on the impeller shroud surfaces of the gaps of the 16FB pump model	186
Figure 6.11 (b)	WSS on the impeller shroud surfaces of the gaps of the 8BB pump model	187
Figure 6.11 (c)	WSS on the impeller shroud surfaces of the gaps of the 16SB pump model	187
Figure 6.12 (a)	WSS on the inner pump housing surfaces of the gaps of the 16FB pump model	189
Figure 6.12 (b)	WSS on the inner pump housing surfaces of the gaps of the 8BB pump model	190
Figure 6.12 (c)	WSS on the inner pump housing surfaces of the gaps of the 16SB pump model	190
Figure 7.1 (a)	The measured velocity distributions at the middle plane	194
Figure 7.1 (b)	The numerical simulated 3-D velocity distribution of the 16FB impeller blade channels	194

List of Figures

Figure 7.2	Velocity distribution in the blade channels of the 16FB impeller on a 5:1 pump model [Ong, 2004]	198
Figure 7.3 (a)	The measured velocity distributions at the middle plane	200
Figure 7.3 (b)	The numerical simulated 3-D velocity distribution of the 16SB impeller blade channels	200
Figure 7.4	Velocity distribution in the blade channels of the 16SB impeller on a 5:1 pump model [Ong, 2004]	203
Figure 7.5 (a)	The measured velocity distributions at the middle plane	205
Figure 7.5 (b)	The numerical simulated 3-D velocity distribution of the 8BB impeller blade channels	205
Figure 7.6	Velocity distribution in the blade channels of the 8BB impeller on a 5:1 pump model [Ong, 2004]	207
Figure 7.7	The velocity triangles at the enter of impeller with the flow enter the impeller at different angle	214
Figure 7.8	The definition of the fluid particle flow angle, β	215
Figure 7.9 (a)	Inlet particle flow angle distributions of the 16FB impeller	222
Figure 7.9 (b)	Outlet particle flow angle distributions of the 16FB impeller	223
Figure 7.10 (a)	Inlet particle flow angle distributions of the 16SB impeller	224
Figure 7.10 (b)	Outlet particle flow angle distributions of the 16SB impeller	225
Figure 7.11 (a)	Inlet particle flow angle distributions of the 8BB impeller	226
Figure 7.11 (b)	Outlet particle flow angle distributions of the 8BB impeller	227

LIST OF TABLES

Tables	Descriptions	Page
Table 4.1	The estimated uncertainty of the parameters in the measurement	106
Table 5.1	The comparison of flow characteristics of the 16FB, 16SB and 8BB pump models	165
Table 6.1	Gap mass flow rates and percentages relative to the pump flow rate of the 16FB, 8BB and 16SB pump models	174
Table 7.1	Operating parameters of the prototype pump and the 5:1 enlarged model	212
Table 7.2	Dimensionless coefficients of the prototype pump and the 5:1 enlarged model	212

CHAPTER 1

INTRODUCTION

1.1 Clinical Significance of Artificial Heart Pumps

A normal heart is a hollow, strong and intermittent muscular pump. It supplies blood continuously through the cardiovascular system to bring oxygen and nutrition to all the organs and tissues in the body and to pick up the metabolic waste products from the body cells. A diseased or faulty heart can cause serious medical problems. In 2004, heart disease and stroke caused about 22.1% of all deaths in Singapore and this percentage has not varied much in recent years. According to the World Health Organization (WHO), the rate of heart disease is declining worldwide in general and people nowadays have a much better chance of surviving a heart attack than in the past. However, it estimates that by the year 2020, up to 40% of all deaths will be related to cardiovascular diseases.

The most common cause of heart failure is heart muscle malfunction, which is a consequence of myocardial infarction or cardiomyopathy [Souhami and Moxham, 1994]. Other causes of the heart failure include ischemic heart disease, hypertension, and myocarditis. Heart diseases have a wide clinical spectrum ranging from the mild heart failure that has little effect on a person's day-to-day life to severe heart failure with symptoms such as decompensation even at a rest.

Common managements of the heart diseases include medication and conventional cardiac surgeries such as angioplasty, valve replacement and bypass surgery. However, the natural course of heart failure is usually progressive, leading eventually to severe heart

failure. In addition, heart failure may be serious from the outset, for example, if it is a result of myocardial infarction. These severe heart failures usually cannot be treated by medication or less invasive surgical procedures. Heart transplantation, which takes a healthy heart from a donor and implanting it in a patient with end-stage heart failure, has therefore become the only accepted form of surgical treatment for patients at the risk of dying from severe heart failure [Massin, 1996]. However, the number of donors is always much smaller than the number of patients on the waiting list and a large number of patients would have no chance to receive heart transplants and die [Abouawdi and Frazier, 1992; Nosé and Furukawa, 2004].

Partly due to the problems with the availability of heart donors outlined above, mechanical cardiac support devices (MCSD) are being adopted increasingly for patients with severe heart failure. MCSD prevent the risks associated with immunosuppression and rejection and can be produced in large quantities to meet the number of patients who may otherwise die before receiving a donor heart [Goldstein et al., 1998]. These devices were originally developed to bridge the patients of final stage heart disease to heart transplantation, but accumulated experience indicated that these devices could also be used as a bridge to cardiac recovery, or even a long-term alternative to heart transplantation. In any case, the intended function is to pump blood at a rate, which is compatible with the circulatory requirement of the body without damaging the blood cells or adjacent organs during the operation [Hogness and Antwerp, 1991].

1.2 The Centrifugal Blood Pump in the Present Study

Various types of cardiac assist blood pump have been developed in recent years. A general introduction of blood pumps working on different principles will be given in

Chapter 2. Among these pumps, centrifugal blood pump has been one of the most promising types and has been emphasized in recent years. As a continuous rotary blood pump, the centrifugal blood pump utilizes an impeller to propel the blood and usually has preferable compact configurations. The centrifugal blood pump does not contain flexing sacs, diaphragms and valves, which will degrade with time and fail finally, thereby, limiting the potential duration of use [Schöb, 2002]. The centrifugal propelling of the pump permits the rotation of the impeller at lower speeds while still achieving the desired flow rates [Zhang et al., 2006]. These advantages have led to a wide acceptance of centrifugal pumps in artificial heart researches.

The centrifugal pump design for this study is based on the Kyoto-NTN magnetically suspended centrifugal pump. The pump is an improvement and modification of the Teaspoon Pump which was first conceived by Bauermeister et al. (1982). The major evolvement is that the impeller is magnetically levitated between the casings. The schematic view of the Kyoto-NTN centrifugal heart pump is shown in Figure 1.1 (a). The magnetic forces are produced by the permanent magnets and electromagnets on both the impeller and the casing respectively [Akamatsu and Tsukiya, 1997]. Figure 1.1 (b) shows the configuration of the magnetic bearing. In one of the side plates of the impeller, 24 pieces of samarium permanent magnets are embedded circumferentially at equal distance and provide the force for magnetic coupling with the same number of permanent magnets embedded in the opposite end of the driving motor. By the force of magnet coupling, the impeller rotates. The other end plate of the impeller is composed of ferrite stainless steel that is the target of four gap sensors within the magnetic bearing. The proportional, integral and differential processes of the signals from the sensors control the currents in the four electromagnets embedded within the magnetic bearing. Under the control of

(a)

(b)

Figure 1.1 Magnetically suspended centrifugal pump.

(a) A schematic sectional view of the pump. (b) Impeller and configuration of magnetic bearing with permanent magnets (left), impeller and pump housing (middle), and electro- and permanent sensors (right). 1, Impeller; 2, electro-magnets; 3, permanent magnets; 4, gap sensor; 5, motor; 6, partition; 7, spiral casing; 8, pump casing

these four electromagnets, the impeller is positioned centrally and rotates freely in the pump.

The major disadvantages of the centrifugal blood pump with the blood-immersed mechanical contacting bearings and seals are the wear and tear of materials as well as heat generation which can lead to device malfunction and subsequently cause serious infection due to leakage of lubricant as well as thrombus formation around the contact area. The magnet levitation configuration allows suspension of the moving component without any mechanical contact, thus eliminating the wear and tear that will take place at the contact surface and heat generation. It has therefore reduced the limitations on the reliability and the life span of the pump [Hoshi et. al., 2006].

Amid the success of the magnetically suspended impeller design, different aspects of investigation are required. Like all the centrifugal blood pumps, thrombosis generation and hemolysis are two of the major problems to be considered before success in the pump design. Blood hemolysis is due to the damage to the red blood cells, which can lead to renal dysfunction, anemia and other syndromes. Mechanical forces, i.e. the high molecular and turbulent shear stresses of the flow can be the major cause of blood hemolysis in the pump [Philips, 1993]. The level of hemolysis is determined by both the magnitude of the shear stress and the exposure time [Paul et. al., 2003]. It has been reported that hemolysis tends to occur at the edge of impeller and at the lip portion of the volute [Tamagawa et al., 1996]. In subsequent studies of centrifugal pumps, these regions were carefully considered. For example, a smaller radial distance between the impeller edge and the volute was reported to produce higher level of hemolysis [Yamane et al.,

1998; Nishida et al., 1999] while the results of the experiment by Masuzawa et al. (1999) revealed that as long as the radial distance was large enough so that the flow characteristics of the boundary layer near the wall, which was about 0.2 mm, was not changed, shear induced hemolysis between the impeller and the volute should not be significantly aggravated.

Thrombus formations have been found to be prevalent in the small gap between the impeller and pump casing through in-vivo testing [Yamada et al., 1997]. A larger dimension of the gap would induce a larger flushing flow through the gap, thus reduce the possibility of the thrombus formation in the gap, but it would lead to low pump efficiency. Besides thrombus generation, too small a gap will induce a higher shear stress in the gap, resulting in hemolysis. It has been reported that maximum pump efficiency for the pump occurs with the gap range between 0.15 and 0.25 mm, but this does not guarantee minimum blood trauma [Park et al., 1996]. The gap width of the Kyoto-NTN blood pump is 0.2mm, well within this range, however, the leakage flow in the gap still requires investigation.

1.3 Objectives and Scopes

It has been believed that flow characteristics in the blood pump have significant effects on thrombosis and hemolysis. Therefore, careful investigations of the inner flow characteristics of the Kyoto-NTN centrifugal blood pump by means of numerical simulation and flow measurement are carried out in the present study. Computational methods have been widely used as complementary tools for blood pump development as they offer a convenient and efficient way for analysis of flow patterns. Furthermore, computational methods can provide a much higher resolution of the flow field than

measurement approaches, therefore facilitating further analysis such as the shear stress distributions and the assessment of critical regions of the flow field. In this work, the Kyoto-NTN magnetically suspended centrifugal blood pump with three different impeller designs namely 16 forward-bending blades (16FB), 16 straight blades (16SB) and 8 backward-bending blades (8BB) will be studied. The objectives of the present numerical work are to obtain the detailed inner flow characteristics of the pump and to analyze the effects of the impeller blade profiles on the flow in the pump.

In order to validate the numerical simulation of the flow in the pump, measurements of the flow will be carried out using laser Doppler anemometer (LDA). LDA is commonly used in flow measurements but is seldom used in the investigation of blood pumps [Pinotti and Paone, 1996]. In previous studies, the flow fields in the impeller blade channels were measured using LDA [Yu et al. 2001; Ong, 2004] and the flow fields in the gap of the Kyoto-NTN blood pump were measured using Hot Wire [Chua and Akamatsu, 2000; Chua et al. 2002; 2003]. However, these measurements were performed on a pump model that was scaled up five times. In this study, measurements of the flow were carried out directly on the 1:1 pump model of the prototype using LDA. This is an important but difficult measurement. Besides, there are scarcely any reports in literature so far on the LDA measurements of the exact size blood pump model. The measurement is performed with delicate refraction index matching between the blood analog and the pump casing as well as the impeller, which are made of Perspex. The measurements will not only validate the numerical simulation but also evaluate the scaling effects on the flow field of the five times enlarged pump model.

The project is carried out in order to accomplish the following objectives:

1. To design and build numerical models of the Kyoto-NTN blood pump. The dimensions of the models are the same as those of the prototype pump.
2. To understand the inner flow of the pump model, static pressure distributions and blood velocity distributions within the pump. Three impellers will be involved in the simulation to reveal the effect of impeller blade profiles on the flow in the pump.
3. To investigate the leakage flow of the pump model numerically. Washout mechanism of the pump leakage flow will be studied and compared with the results of the 5:1 pump model.
4. To investigate the shear stress distributions of the blood at different regions of the pump, the shear stress distributions of the three different pump models are compared and discussed in order to recommend the best performance impeller blade design.
5. To measure the flow in the blade channels of the three different impeller blade designs of the exact size pump model using LDA, so as to compare the measurement results with the numerical solutions and to validate them. It is also used to compare with the measurement results of the 5:1 enlarged pump model in order to check the flow difference between the 1:1 and 5:1 pump models. The effects of scaling of the enlarged pump and the precautions in applying dimensional analysis in the enlarged pump model is discussed and highlighted.

1.4 Thesis Outline

The thesis consists of eight chapters. Chapter 2 introduces the various types of heart pumps developed and their characteristics. Literature review of the works that have been done on the heart pumps is also presented in this chapter.

Chapter 3 introduces the pump model for numerical simulation. The mesh generation, boundary conditions and assumptions used are presented in this chapter as well. Chapter 4 introduces the LDA measurement theory and methods, the experimental pump model and the arrangements of the test rig. Experimental procedure will also be included in this chapter.

The numerical simulation results of the flow in between the impeller blades for the three pump models are documented in Chapter 5. Static pressure, velocity distribution and shear stresses are also presented and discussed. The numerical results of the flow in the gap of the three pump models are shown in Chapter 6. Chapter 7 presents the LDA measurement results of the flow in the impeller blade channels of the three exact size pump models. The comparisons between the measurement and numerical results are also presented and discussed. Furthermore, the comparisons between the flows in the 1:1 and 5:1 pump models are highlighted in this chapter. More importantly, the limitations of dimensional analysis on producing faithful flow patterns of the prototype in the scaled up model are discussed in detail. It is thus recommended to have direct measurements on the exact size pump instead of using the scaled up pump.

Finally, concluding remarks and the future work with respect to the development of the pump are presented in Chapter 8.

CHAPTER 2

LITERATURE REVIEW

2.1 The Development of Mechanical Cardiac Support Devices

Mechanical cardiac support devices (MCSD) have been a standard health care for most potential heart transplant patients with end-stage heart failure refractory in medical treatments. The technology and clinical experiences mainly evolved over the past 50 years. In 1957, Akutsu and Kolff (1958) reported the development and application of the first totally artificial heart in an animal model at Cleveland. The authors implanted a totally artificial heart in a living dog which survived for 90 minutes with the mechanical heart. The first human application of a totally artificial heart was by Denton Cooley and colleagues as a bridge to transplantation [Cooley et al., 1969]. They implanted a total artificial heart in a patient who could not be weaned from cardiopulmonary bypass. After 64 hours of artificial heart support, heart transplantation was performed, but the patient died of pseudomonas pneumonia 32 hours after transplantation.

After decades of research, much progress has been made since the early efforts. In 1994, HeartMate implantable pneumatic left ventricular assist device was the first MCSD approved for general clinical use. From then on, higher requests for proposal were issued and resulted in newer generations of ventricular assist devices which now include axial flow pumps and centrifugal flow pumps characterized by small size, quiet continuous flow and total implantability [Pantalos, 1993]. The spectrum of applications for these MCSD range from heart-lung pumps to artificial hearts, each specifically designed to overcome different problems encountered in the patients' treatment. In this project, the

Kyoto-NTN centrifugal blood pump was studied. As mentioned in the previous chapter, the pump has a rotating impeller and works on centrifugal principles. A general introduction of the varieties of circulatory support devices will be given in the following section.

2.2 Ventricular Assist Devices and Total Artificial Hearts

MCSD can be divided into different groups according to the different aspects of their features. According to the type of assist, they can be categorized as the total artificial heart, bi-ventricular assist devices, left ventricular assist devices, and right ventricular assist devices; according to the type of the generated flow, they can be categorized as pulsatile and non-pulsatile; to the time limitation, they can be categorized as short-term, medium-term and long-term; to working principle, electrohydraulic and pneumatic. The easiest way to categorize MCSD is based on the need of native heart replacement. Therefore, the MCSD is divided into two categories, as shown in Figure 2.1, Total Artificial Heart (TAH) and Ventricle Assist Device (VAD).

Figure 2.1 Classifications of Mechanical Circulatory Support Systems

A total artificial heart (TAH) is exactly what the name has stated. It is placed in the body and imitates the function of a real human heart. It is designed to provide the same circulation, flow rates, and overall functions of the heart it replaces. A ventricular assist device (VAD) helps patients whose hearts are not pumping sufficient blood to maintain adequate blood flow while either recovering from heart surgery or waiting for a heart transplant. VAD has also been used as resuscitative devices in patients with postcardiotomy cardiogenic shock [Tominaga et al., 1993]. The requirement of cardiac replacement is the major difference between TAH and VAD. When a VAD fails, the limited functions of native heart can still keep the patients alive. However, when a TAH fails, the patients may die within minutes. Recent trails indicated that VADs might play a role as permanent or destination therapy for patients with final-stage heart disease and are not in the list of transplantation candidates.

In the following sections, a brief discussion for each type of MCSD will be given with a few examples.

2.2.1 Total Artificial Heart

The first generation of TAHs were all pneumatically driven and possessed similar design characteristics [Richenbacher and Pierce, 1997]. They consist of two blood pumps, each composed of a rigid blood chamber and an air chamber. A membrane, which is fixed to separate the blood chamber and air chamber, is moved by air derived from an external drive unit. The air pipes connecting the TAH with the drive unit pass through the skin, and valves were installed at the inflow and outflow tract to ensure the unidirectional flow. A well known pneumatic Total Artificial Heart, Jarvil-7, as shown in Figure 2.2, was widely used until 1990. However, due to mismanagement and a high incidence of

thromboembolic complications, this device was withdrawn from the market in the early 1990s [Pierce et al., 1996]. Presently CardioWest (CardioWest, Tucson, AZ, USA) has taken over the production of Jarvik-7. More than 400 patients worldwide have been implanted with this device (including the former Jarvik-7 and Symbion) with acceptable clinical outcomes [Copeland et al., 1998]. Now, it is still the only TAH that is being clinically used as a bridge to transplant [Mesana, 2004].

Figure 2.2 Jarvik-7 Artificial Heart

In general, pneumatic assist devices show a high incidence of infections. This is mainly because of the percutaneous drivelines, which make these devices unsuitable as permanent artificial hearts. Therefore, a focus of the second generation of TAH is the realization of a totally implantable, wireless electrically powered TAH. However, with existing techniques, the electric TAH requires a minimum energy supply of 14 Watts, which cannot be provided by implantable batteries [Richenbacher and Pierce, 1997]. For this reason, new developments are aimed at improvement of rechargeable battery

techniques and a wireless energy transmission. In this present way, the implanted rechargeable battery can only sustain the power source for 40-60 minutes.

Although most of the Total Artificial Hearts are still under investigation, there are several kinds of commercially available electrically driven TAHs in clinical use. The AbioCor TAH (Abiomed Cardiovascular, Inc, Danvers, MA), as shown in Figure 2.3, uses a high-speed brushless direct-current motor that drives a unidirectional centrifugal pump [Pierce et al., 1996]. The pump consists of two ventricles with their corresponding mechanical valves. A rotary valve ensures that the fluid of this electro-hydraulic device first actuates the right ventricle and then the left, thus generating an alternately emptying blood chambers. A small chamber placed in between the left atrial cuff and the inflow valve manages the physiological left/right flow difference. Its stroke volume is between 60 and 65 cm³ with an output of between 4 and 10 liters/min. External power is delivered via a transcutaneous energy transmission (TET) coil located on the chest wall. The thermal, physiological, and hematological compatibility of the systems were verified during long-term animal studies, which also demonstrated that the AbioCor TAH could provide cardiac output greater than 10 l/min [Kung et al., 1995].

Figure 2.3 AbioCor Artificial Heart

The Sarns/Pennsylvania State University TAH uses a low-speed and high-torque brushless direct-current motor and a dual pusher plate roller screw energy converter [Pierce et al., 1996]. The blood pump consists of highly smooth segmented polyurethane sacs with Björk-Shiley Delrin disk inlet and outlet valves. The gas-filled compliance chamber is similar to that used in the Nimbus/Cleveland Clinic Foundation TAH. Used as a complete heart replacement, the device was able to keep experimental animals alive for more than 13 months.

The Nimbus/Cleveland Clinic Foundation TAH also uses a high-speed brushless direct-current motor and hydraulic actuator to drive two diaphragm-type blood pumps. The space between the two artificial ventricles contains the pump control electronics. It is vented to an air-filled compliance chamber placed between the lung and the chest wall. Pericardial tissue valves and biolized blood-contacting surface potentially eliminates the need for anticoagulation [Massiello et al., 1994]. The device, tested in calves for a period of up to 120 days, has a maximum output of 9.5 l/min.

2.2.2 Ventricular Assist Devices

It is estimated that more than 35,000 patients a year could benefit from the VADs. Left Ventricular Assist Device (LVAD) is the most commonly used. Blood is withdrawn from either the pulmonary veins, left atrium or the apex of the left ventricle and returned to the ascending aorta. There is also Right Ventricular Assist Device (RVAD), through which blood is drawn from the right atrium and returned to the main pulmonary artery, the Biventricular Assist Device (BiVAD), which is a combination of LVAD and RVAD could also be seen as a functional heart replacement. The VADs can be categorized into Intra-Aortic Balloon Pump (IABP), pulsatile and non-pulsatile VADs.

2.2.2.1 Intra-Aortic Balloon Pump

The Intra-Aortic Balloon Pump (IABP) is the most commonly used assist device for temporary support of the failing left ventricle after cardiac surgery. The IABP consists of a catheter-mounted polyurethane balloon connected to a driving system. The device works on a principle of counterpulsation, i.e., assisting the heart in series synchronously with the patients' ECG. As shown in Figure 2.4, the balloon is automatically inflated at the beginning of diastole, increasing the diastolic aortic pressure which in turn increases the coronary flow. During the systolic phase, the balloon is deflated, decreasing the LV afterload, which in turn decreases the myocardial oxygen consumption and increases the cardiac output. IABP is indicated for treatment of cardiogenic shock in acute heart failure or after myocardial infarct [Kumbasar et al., 1998] and for treatment of heart failure after open-heart surgery.

Figure 2.4 Schemes of Intra-Aortic Balloon Pump

However, IABP has its disadvantages. Severe kinking or arteriosclerosis of the iliac or femoral artery could make the insertion of the balloon impossible. Infections and bleeding complications were reported as well [Kantrowitz et al., 1986]. Unlike true blood pumps, the IABP depends on residue LV functions and therefore has only minor effects in patients with profound hemodynamic compromise [Scholz et al., 1994]. Last but not least, IABP works properly only when the patient's ECG is optimal, therefore, the system fails when severe arrhythmia or fibrillation take place.

2.2.2.2 Pulsatile Ventricular Assist Devices

All pulsatile pumps are basically membrane pumps in which the membrane is moved by air, liquid or by a pusher-plate. The driving source (air, water or electricity) reaches the pump via a tube through the skin.

There are several most widely used pulsatile ventricular assist devices. The HeartMate (Thoratec Laboratories Corp, Pleasanton, CA) [Maher et al., 2001], as shown in Figure 2.5, is an implantable ventricular assist device that can be driven either pneumatically or by a low-speed torque motor. The drive-line containing the electric cable and an air vent exits the skin to attach to the external drive console. A maximum stroke volume of 83 ml allows a pump output of up to 10 l/min. The pump consists of titanium housing with a flexible polyurethane diaphragm inside bonded to a rigid pusher plate, as demonstrated in Figure 2.5 (b) and (c). Porcine valves are placed in the inflow and outflow conduits to ensure unidirectional blood flow. Blood is withdrawn from the left ventricle via the apex and returned to the ascending aorta. The device is controlled and powered by a tube connected to the external unit. The pump is implanted in a pocket in the left upper quadrant of the patient's abdomen or abdominal wall, as shown in Figure 2.5 (a).

Figure 2.5 HeartMate I Ventricular Assist Devices

The Novacor VAD (Baxter Healthcare Corp, Berkeley, CA), as shown in Figure 2.6, is an electrically powered implantable pump designed for left ventricular support only. The device contains a polyurethane blood sac that is compressed by a dual symmetrically opposed pusher plates. Bovine pericardial valve-prostheses are incorporated into the grafts to ensure unidirectional blood flow. The pump, implanted in a pocket in the left upper quadrant of abdominal wall, provides circulatory support by taking over most of the workload of the failing heart. A flow conduit directs blood from the left ventricle into the blood pump. The pump ejects blood through an outflow conduit to the ascending aorta with a flowrate up to 10 l/min. The system is operated and monitored by an electronic controller and powered by the primary and reserve batteries. The batteries are connected to the implanted pump by a percutaneous lead through the skin of patient.

Figure 2.6 Novacor
Ventricular Assist Devices

The Thoratec VAD is a pneumatically powered device that is placed on the anterior abdominal wall. The system consists of a prosthetic ventricle that has a blood-pumping chamber of Thoralon polyurethane, cannulas for ventricular support and either a hospital-based pneumatic drive console or a portable battery-powered drive unit. Blood can be taken from the left ventricle or atrium and pumped into the aorta. A pneumatic power unit is used to provide the air pulses.

2.2.2.3 Non-pulsatile Ventricle Assist Devices

The non-pulsatile ventricular assist devices consist of blood pumps which generate blood flow continuously. These continuous blood pumps can be indicated as rotary blood pump because they all have a basic structure that includes rotating components and stationary components. The recent emphasis on the development of rotary blood pump should be a logical step. The rotary blood pumps generally have a size advantage over standard

pulsatile blood pumps. In many designs, the small size of the rotary blood pumps makes them less intrusive for implantable cardiac support applications. In recent years, non-pulsatile blood pumps have been broadly investigated and have been taken as a development tendency of long-term implantable blood pumps.

According to the mechanical characteristics, the non-pulsatile VADs can be divided into 3 categories: roller pumps, centrifugal pumps and axial pumps.

a) Roller Pumps

The roller pumps are similar to the pumps used for cardiopulmonary bypass. The inflow and outflow cannula are connected by tubing made partially of silicone rubber. The silicone part is placed in a head with rotating occlusive rollers. During the head rotation, the tubing is compressed repeatedly by a roller and a non-pulsatile blood flow is generated. The roller pumps are simple to use and are relatively inexpensive devices. However, the roller pumps have some disadvantages: requirement of systemic anticoagulation, blood trauma leading to hemolysis, tubing spallation, fatigue, and etc.. These limitations preclude use of the roller pumps beyond hours.

b) Axial Blood Pumps

The axial flow motor is normally smaller in size than other kinds of blood pumps and contain rotary blades that spin at 6,000 to 20,000 rpm. Because of the continuous flow properties of axial flow pumps, there are no requirements for valves in the system.

The MicroMed DeBakey axial blood pump, as shown in Figure 2.7, is an electro-magnetically actuated, implantable titanium axial flow pump that connects to the LV apex

and ascending aorta. The pump is designed to produce flows of 5 l/min against 100 mmHg pressure with a rotor speed of 10,000 rpm [Wieselthaler et al., 2000]. The current design of this pump includes a fixed rpm rate that can be adjusted through an external device. During periods of patient mobilizations, power can be supplied by two 12-Volt DC batteries for several hours.

Figure 2.7 MicroMed DeBakey axial blood pump

The Jarvik 2000 Heart is a compact axial blood pump that receives inflow from the LV apex and outflow through a Dacron graft anastomosed to the descending thoracic aorta [Marlinski et al., 1998]. The rotor constitutes the only moving part of the device and is supported at each end by tiny blood-immersed ceramic bearings [Kaplon et al., 1996]. The current existing device is tethered to an external electrical power source through a percutaneous wire but a subsequent totally implantable version will contain a microprocessor-based controller that can sense and change pump speed according to the different phases of the cardiac cycle and receive power via a transcutaneous energy transfer system coil.

HeartMate II is a small axial pump that connects to the LV apex for inflow and the ascending aorta for outflow. An electro-magnetic motor (pump rotor) turns the turbine. Two cup-socket ruby bearings support the pump rotor. The outer boundary of the bearing's adjacent static and moving surfaces is washed directly by blood flow [Burke et al., 2001]. A first version of this device is powered through a percutaneous small-diameter electrical cable connected to system's external electrical controller [Siegel-Itzkovick, 2000]. A fully implantable system is under development.

c) Centrifugal Blood Pumps

Centrifugal blood pumps are somewhat larger than axial blood pumps but rotational speeds are much slower (about 1,500-4,000 vs. 6,000-20,000 rpm). Centrifugal pumps consist of non-occlusive pump head positioned within a rigid pump housing. The rotating head consists of a various number of impeller blades that generate a non-pulsatile unidirectional flow by creating a vortex. The centrifugal blood pumps do not require artificial valves.

Medtronic Biopump is an extracorporeal centrifugal device that can provide support for one or both ventricles. As shown in Figure 2.8, the pump consists of an acrylic pump head with inlet and outlet ports placed at right angle to each other. The transparent pump housing is shaped like a corn. The impeller, which is a stack of parallel cones, is driven by an external motor and power console. It was originally developed for cardiopulmonary bypass, but it can be used for short-term circulatory support beyond the surgical setting, as well as for both postcardiotomy cardiogenic shock patients and as a bridge to transplantation for patients who cannot be weaned from the devices [Oku et al., 1988]. The adult model pump can rotate up to 5000 rpm and can provide flow rates up to 10 l/min.

Figure 2.8 Medtronic centrifugal blood pump

Gyro pump (Baylor College of Medicine, Texas, USA), as shown in Figure 2.9, is a series of centrifugal pumps that are developed as an intermediate-term as well as long-term left ventricular assist devices. Inside the pumping chamber, an impeller is suspended by the top and bottom pivot bearings. An eccentric inflow effectively washes the top bearing regions [Ohara et al., 1994]. Secondary vanes at the bottom of the impeller effectively

Figure 2.9 Gyro Centrifugal blood pump

wash the bottom bearing regions, therefore, eliminating the blood stagnant region inside the pump and resulting in an antithrombogenic device. The normalized index of hemolysis (NIH) of this pump was comparable to that of the BioPump-80 [Nakazawa et al., 1998]. A late version of Gyro pump PI702 is made of titanium, and the priming volume of this pump is 20 cc with an impeller diameter of 50 mm.

The HeartMate III (Thermo Cardiosystems Inc., USA/Sulzer AG, Switzerland) centrifugal blood pump as shown in Figure 2.10 is a third generation LVAD after the reciprocating HeartMate I (pusher-plate) [Goldstein et al., 1998] and the axial flow HeartMate II [Butler et al., 1999]. Powered by magnetic levitation, this device combines the function of impeller rotation and levitation in a single magnetic structure. These features make the pump fit in acceptably small packages and can operate completely sheathed in titanium. The pump has a priming volume of 25 cc with a total mass of 474g [Loree et al., 2001]. Currently, this device is still under investigation.

Figure 2.10 HeartMate III Centrifugal blood pump

Although the centrifugal pumps are firstly designed for short-term mechanical cardiac assist, experimental studies demonstrated that some centrifugal pumps could be used beyond this period as well. For example, one of the most advanced centrifugal magnetically suspended impeller pump, the Terumo-Akamatsu centrifugal pump [Nojiri et al., 1997], has been confirmed to operate after 864 days of left ventricular bypass animal testing. The centrifugal pump is the main focus in this study and will be discussed later in more details.

2.3 Design Considerations

2.3.1 Centrifugal Blood Pump as a Turbo-Machinery

All turbo-machineries have a rotating part called the impeller, through which the fluid flow is continuous. The direction of fluid flow in relation to the plane of impeller rotation distinguishes different classes of turbo-machineries. One possibility is for the flow to be perpendicular to the impeller and, hence, along its axis of rotation. Machines of this kind are called axial flow machines. In centrifugal machines (sometimes called ‘radial flow’), although the fluid approaches the impeller axially, it turns 90° at the machine’s inlet so that the flow through the impeller is in the plane of impeller rotation. A third category is called mixed flow machines. They derive their name from the fact that the flow through their impellers is partly axial and partly radial. Both pump and turbines can be axial flow, mixed flow and radial flow. In the present study, the blood pump of radial flow is the main focus.

Centrifugal pumps consist basically of an impeller, which forces the fluid into the rotary motion by impeller action, and on spiral casing, which directs the fluid to flow and collects the fluid away from the impeller under a high pressure. A common example of centrifugal pump is shown in Figure 2.11. The fluid is first being sucked into the impeller from the inlet nozzle (location ① in Figure 2.11) through the impeller eye. As soon as the fluid encounters the leading edge of the impeller vanes, the fluid will diverge into different impeller passages and then follow the profile of the vanes and flow through the passage. When the fluid leaves the impeller, it flows directly into the volute. As a result of the impeller high-speed rotation, liquid leaves the pump (location ② in Figure 2.11) at a higher pressure than when it is at the inlet.

Figure 2.11 Schematic view of centrifugal pump

A study of the velocity components of flow through an impeller can be carried out graphically by means of velocity vectors. The shape of such vector diagrams is triangle and they are called velocity triangles. They are normally drawn at the inlet and outlet of an impeller and are named inlet and outlet triangles as illustrated in Figure 2.12, in which

the inlet is referenced as station 1 and the outlet is referenced as station 2. The symbols in the figure are defined as:

- W : the relative velocity of flow;
- v : the absolute velocity of flow;
- u : tangential velocity of the rotating impeller;
- v_r : radial velocity of flow;
- v_w : tangential velocity of flow, swirl velocity;
- β : flow angle, relative to the tangential line;
- D : impeller diameter;

Figure 2.12 The velocity triangles at the inlet and outlet of impeller

The input power to the pump is to change the fluid energy between the inlet and outlet of the impeller. The energy transfer between the impeller and the fluid, based on the 1-D Euler's equation, can be calculated by:

$$W_i = \dot{m}(u_2 v_{w2} - u_1 v_{w1}) \quad (2-1)$$

Since the work done per second by the impeller on the fluid is the rate of energy transfer, then rate of energy transfer per unit mass of fluid flowing should be:

$$gE = W_i / \dot{m} \quad (\text{Joules/kg}) \quad (2-2)$$

and

$$E = (u_2 v_{w2} - u_1 v_{w1}) / g \quad (2-3)$$

E is known as the Euler's head and expressed as meter. The Euler's head also represents the ideal theoretical head developed H_{th} .

Ideally, the inlet flow will have a zero swirl velocity (preswirl) such that $v_{w1} = 0$. Therefore, the inlet velocity triangle will be a right triangle and the fluid will enter the impeller with a purely radial component. Equation (2-3) can then be simplified as:

$$gE = u_2 v_{w2} = u_2^2 - \frac{Q \times u_2 \cot \beta_2}{\pi D_2 b} \quad (2-4)$$

In Equation (2-4) the radial velocity is replaced by the combination of volumetric flow rate, Q and the height of the impeller blade, b . Therefore, the ideal head rise is proportional to u_2^2 and thus, ω^2 , at shut off condition and varies linearly with flow rates. Based on the equations, the pump performance in producing pressure head and flow rate can be approximately calculated.

However, for the centrifugal blood pump as in the present study, it is not sufficient to concern only the overall pump performances of power efficiency, pressure head and flow rate that can be achieved by the pump. Due to the pump is using blood as working fluid, much more detailed information of the inner flow pattern is yet to be acquired so as to facilitate achieving the required clinical applicability during development of the pump.

2.3.2 Flow Similarity

Practically, the actual performance characteristics of centrifugal pumps have to be determined through experimental testing. A powerful mathematical tool in experimental work is dimensional analysis which replaces the variables by dimensionless groups so as to help to carry out the experiment more efficiently. In addition, the dimensionless groups provide the similarity laws governing the relationships between the variables within one family of geometrically similar machines.

Over the years, several hundred different dimensionless groups have been identified. Forces encountered in flowing fluids include those due to inertia, viscosity, pressure, gravity, surface tension and compressibility. Among them, the ratios of the inertia force to the forces of viscosity, pressure gravity, surface tension and compressibility lead to fundamental dimensionless groups in fluid mechanics.

Among them, the Re is the ratio of inertial forces to viscous forces. It is closely related to the flow characteristics in centrifugal pump and generally expressed as

$$Re = \frac{\rho VL}{\mu} \quad (2-5)$$

where L is a characteristic length descriptive of the flow field. In the 1880s, Reynolds studied the transition between laminar and turbulent flow in a tube. He discovered the Re is a criterion by which the state of a flow may be determined. Later experiments have shown that the Reynolds number is a key parameter for other flow cases to indicate the effects of viscosity and inertia on flows. “Large” Re flows generally are turbulent. Flows in which the inertia forces are “small” compared to viscous force are laminar flows.

If an experimental model achieves a complete similarity with the prototype of a physical system, it is required that there is a complete geometric similarity between the model and prototype, and the model must be a scaled version of the prototype. Furthermore, the dimensionless groups of the model and prototype must be the same, i.e.

$$\Pi_{\text{prototype}} = \Pi_{\text{model}} \quad (2-6)$$

where Π represents the dimensionless groups derived from the physical system. The flow similarities can be used to predict the performance of a prototype from the experimental model. They are based on the concept that the velocity diagram and streamline patterns in the prototype and experimental model with similarity are geometrically homologous, or the flow behavior has a resemblance to one another [Barenblatt, 1996].

For a given physical system, dimensionless groups can be achieved according to Buckingham Π -theorem. The Buckingham Π -theorem states that: Given a relation among n parameters of the form

$$g(q_1, q_2, q_3, \dots, q_n) = 0 \quad (2-7)$$

Then the n parameters may be grouped into $n-m$ independent dimensionless ratios, or Π parameters, expressible in function form by

$$G(\Pi_1, \Pi_2, \dots, \Pi_{n-m}) = 0 \quad (2-8)$$

The number m is usually, but not always, equal to the minimum number of independent dimensions required to specify the dimensions of all the parameters $q_1, q_2, q_3, \dots, q_n$.

For the centrifugal pump concerned in the present study, the dimensionless groups which are often employed include the Reynolds number, the flow coefficient, the head coefficient, the power coefficient and so on. However, it is worth to note that, the dimensionless groups derived could not necessarily be the same. It is common that different dimensionless groups are used for different physical problem or different aspects of behavior of a specific physical problem. Another fact is that, complete similarity between different physical systems has rarely been achieved even though the physical models are geometric similar. Practically, equalities of the dimensionless parameters that have dominative influences on the investigated physical problem are usually preferentially satisfied.

2.3.3 Blood Traumatic

To develop a clinically effective and safe cardiac prosthesis, it is essential to satisfy many design criteria, which include optimal size, satisfactory endurance characteristics, efficient operation, antitraumatic features for the blood, antithrombogenicity and biocompatibility. Unfortunately, these design criteria have opposing features. For example, size versus endurance, efficiency versus thrombogenicity and antitraumatic features versus antithrombogenicity features are particular opposing characteristics [Nosé, 1998]. Thus, the main concern in properly designing a cardiac prosthesis is how to

optimize the opposing design parameters, and the opposing design requirements. For totally implantable cardiac assist pumps, antitraumatic one of the most important indicators for pump performance.

Blood trauma refers to the blood damage occurring in interactions of blood with foreign surface as well as damage to blood corpuscles, which include red blood cells, white blood cells and platelets, at non-physiological flow conditions. The causes of the pump-induced trauma are not yet fully understood. It is believed that chemical ingredients, heat generated within the flow field, hemodynamics, and etc., will all have effects on the extent of hemolysis. Among these factors, fluid forces exerted on the blood cells could be a major cause of the hemolysis in rotating blood pumps. A good number of researches have shown that improving on the flow conditions within the pump would greatly reduce the level of hemolysis. It is unlikely and unnecessary to reduce the hemolysis to zero. For instance, Nosé (1998) has suggested that any pump exhibiting a NIH level less than 0.02g per 100 liter at 500 mmHg or less than 0.004g per 100 liter at 100 mmHg is considered acceptable. However, some designers aim at one-half of these hemolysis rates for safety.

Blood damage by the fluid forces primarily depends on magnitude and duration of the loading. Especially, flow induced shear forces within the flow field have damaging effects on the blood corpuscles. Figure 2.13 shows a relationship between blood damage and both shear stress and exposure time according to the experimental results on the laminar flow of Paul et al. (2003). The bold lines indicate the critical threshold levels of exposure time and shear stress. In the figure, index of hemolysis (IH), which expresses the hemoglobin that is set free by the damaged red blood cells, is used to indicate the

extent of blood damage. In other literatures, the nomalized index of hemolysis (NIH) or modified index of hemolysis (MIH) may be used as well.

Figure 2.13 Illustration of flow induced blood damage by shear stress and exposure time (Paul et al., 2003)

The studies of the relationship between the values of fluid shear stresses and hemolysis are not consistent among different investigators. Sutura et al. [1972] suggested that exposure time of 60 minutes under shear stress of 250 Pa is required for erythrocytes rupture. Sallam and Hwang [1984] pioneered a sampling technique in which erythrocyte samples were collected from selected locations in the flow field so that the relationship between local shear stress and cell damage could be determined. They suggested the

threshold level for erythrocyte damage is 400 Pa. Tamagawa et al. [1996] studied the turbulent shear orifice flow and suggested a hemolysis threshold shear stress of 1,800 Pa. The critical threshold shear stress level is defined as the amount of shear, which causes incipient hemolysis. Below the threshold, there is little damage due to shear stress on the erythrocytes. There is a large difference in the threshold values of shear stress for erythrocyte damage reported by various studies. It is thus difficult to get a practical shear stress threshold value of hemolysis from published papers. However, it is well agreed that the degree of blood cells damage not only depends on the shear stress level, but also highly relies on the history of the cell (accumulation of previous damage) and the exposure time within the region of fluid shear [Giersiepen et al., 1990; Kunov et al., 1996; De Wachter and Verdonck, 2002]. Based on this assumption, a few theoretical models were developed to estimate blood trauma [Giersiepen et al., 1990; Bludszuweit, 1994; Yeleswarapu et al., 1995]. However, none of these models have been fully validated [Arvand et al., 2005].

On the other hand, the roughness of the blood-contacting surface was also found to be related to the hemolysis. Takami et al. (1997(c); 1996) revealed that the surface roughness in the blood pump had different effects on hemolysis. Umezu et al. (1996) reported that under the same geometry of the flow channel, smooth blood contacting surface can remarkably reduce blood hemolysis rate. In another study, the degree of hemolysis as a function surface roughness was tested by Maruyama et al. (2006) using a rotational shear stressor. They suggested that red blood cells were destroyed not by fatigue failure caused by rolling on the roughened surface but due to the high shear stress generated by surface roughness.

2.3.4 Thrombogenicity

Anticoagulation is a quality for a cardiac assist pump to prevent blood clotting. Thrombus produced in a patient with an implantable blood pump would cause a stroke in the patient. It is a consequence of blood material interaction on the foreign surface of the blood pump. The interactions may consist of protein absorption, platelet adhesion, and platelet activation [Whicher and Brash, 1978]. Within seconds of blood contact, the artificial surface becomes coated with a layer of plasma proteins, the configuration of which is determined by the surface properties including roughness, charges, free energy, and chemical composition [Takami et al., 1998].

The thrombus generation in a cardiac assist pump is closely relevant to the thrombocytes lysis in the blood. Thrombogenesis is activated when thrombocytes are damaged. Thrombocytes are more sensitive to high shear stress than erythrocytes [Treichler et al., 1993]. Brown et al. [1975(a); 1975(b)] reported a threshold of 150 dynes/cm^2 (15 Pa) that caused thrombocytes lysis for long exposure time of 2 – 24 minutes. They also concluded that shear stress lower than 50 dynes/cm^2 (5 Pa) could cause thrombocytes aggregation. Colantuoni and Hellums [1977] investigated the response of thrombocytes to shear stress at very short exposure time which was similar to physiological time scale. No significant changes were seen in the thrombocytes exposed to a shear stress of 700 Pa for a period of 1 millisecond. When the shear stress was increased to between 700 and 1,500 Pa, chemical changes were found in blood specimen. They observed a significant reduction in the number of thrombocytes when the shear stress was above 1,500 Pa. Ramstack et al. [1978] were the first to report the relationship between applied shear rate and time of exposure with the damage potential. They found that the thrombocytes were activated and

resulting in hypercoagulability when platelet-rich plasma (PRP) was subjected to shear stresses of 30, 75 and 100 Pa, for exposure time of 25 – 165 milliseconds.

Sutera and Nowak [1988] showed that pulsed exposure resulted in more thrombocyte aggregation than continuous exposure for the same exposure time to the same maximum stress. This indicated that shear and velocity gradients enhanced thrombocyte activation, though these tests were done in a viscometer, with constant stress level and exposure time far above physiological level.

Besides the activation of thrombocytes, the generation of thrombus within a cardiac assist pump is relative to many factors, which include the chemical ingredient of blood, fluid dynamics of blood flow, material, roughness and moving mode of the inner surface of the pump and so on. However, experiences accumulated in the previous researches indicate that thrombus is formulated in areas of persistent stagnation in the flow field. Therefore, pump components must be designed to avoid large stagnant flow zones.

2.3.5 Infection and Contacting Surfaces

There are some other considerations that go into the building of an implantable cardiac assist device. The biological compatibilities are of great importance in the construction of heart pumps. As reported by Shumakov (1993), most causes in patients' death using VAD and TAH can be attributed to heart failure, multiple organ failure, bleeding and infection. The problem of infection is a recurring theme in implantable heart pump research. Several studies have shown that infection could be related to the placement and design of the heart assist devices [Nosé, 1991(a)]. Infection in patients after the heart pump transplantation can be as high as 60-80% [Shumakov, 1993]. Basically, infection can

either be patient-related or device-related. However patient-related infection such as pneumonia is of greater concern because it tends to linger and then worsen after the transplantation.

Several studies had shown that infection could be related to the placement and design of the heart assisting devices [Nosé, 1991(b)]. A study had shown that between 1982 and 1983, of the 167 patients who were implanted with Jarvik-7 TAH, only 44% of them survived the implantation. Almost all of the patients, who received the implants after one month or more, died due directly or indirectly to infection. In another clinical trial, electrically powered Norvacor VAD HeartMate that was implanted in the intra-abdominal region demonstrated over 90% survival rate. However, based on experimental studies conducted by Nosé group, it was shown experimentally that VADs and TAHs have the same infection rate. Nosé (1993) attributes the early high failure rate of TAHs to the induced mechanical stress imposed upon the system by semi-rigid-construction (artificial heart valves) of the device.

A good number of device-related infections are caused by percutaneous implantations. As shown in Figure 2.14, although totally implantable cardiac assist devices or artificial hearts are most admirable, nearly all assist pumps have percutaneous drive lines or external tether that stick out from the body. These lines often cause infection [Nosé et al., 1997]. Transcutaneous energy transfer to implanted rechargeable batteries can only allow the patients to be “untethered” for 30-60 minutes. So the battery can only be used as backup power supply for emergencies.

The material of blood contacting surface has also some effects on the performance on the cardiac assist blood pumps. Badly chosen material will lead to thrombus formation on its surface, subsequent thromboembolism and infection in the bloodstream [Nosé, 1991(b)]. When contacting with surfaces of different materials, platelet appears to have different properties of adhesion and activation, which are closely related to the thrombus formation [Chiang et al., 1991; Chatel, 1996; Ishitoya et al., 2002]. It has been reported that Al_2O_3 , polyethylene and titanium alloy (Ti-6Al-4V) have satisfactory antithrombogenicity in terms of platelet adhesion [Takami et al., 1998].

Figure 2.14 Different levels of implantable cardiac prosthesis (Nosé et al., 2000)

On the other hand, Yasuda (2000) reported that wall hardness has affected the hemolysis. The blood flow impact of the collision against the wall resulted in serious damage to red blood cells. In his report, when the wall hardness was decreased by applying a layer of silicone rubber, the rate of hemolysis was reduced. Wall roughness also plays an important role. Originally, very smooth surfaces were tried, but these caused the blood to clot. It was found that the internal surface could be applied with a coating on it to provide a relatively rough surface for the blood to contact. After a short period of time the blood forms a thin layer coating over the surface and this protects the rest of the blood passing through the artificial heart.

2.4 Non-pulsatile Blood Pump as a Substitution of Pulsatile Pump

In this project, a continuous blood pump based on the original design of the Terumo-Akamatsu centrifugal pump is mainly considered. Continuous blood pumps, which are taken as a trend in the development of ventricular assist devices, have been emphasized in recent years. Unlike the pulsatile blood pumps, continuous pumps do not produce pressure fluctuation during work. In past decades, the physiological effects of the non-pulsatile cardiac assist devices are also cordially investigated.

2.4.1 Comparison between pulsatile and non-pulsatile blood pumps

The technological requirements for implantable artificial heart are compact in size for implantation, biocompatibility and durability for long-term use [Allen et al., 1997]. The size of clinically useful VADs and TAHs is an important design consideration. Ideally, a TAH device should fit the anatomic space occupied by excised native heart while a VAD must displace organs without producing injuries directly or indirectly. A non-pulsatile cardiac assist device is designed remarkably more compact than a pulsatile one. In

contrast to the pulsatile device, the continuous pump does not require a ventricular reservoir with its inherent dead space and large internal volume necessary to accommodate the excursion of the flexing diaphragm. The size reduction in non-pulsatile pump allows for its implantation in a smaller space thus minimizing any detrimental effects on adjacent organs and tissue.

Producing pulsatile flow with a VAD or TAH tremendously increases the engineering complexity of the device. Most available blood pumps produce pulsatile flow by utilizing a flexible diaphragm that intermittently ejects blood. The repetitive motion of the flexible diaphragm causes significant stress to the material used, which can eventually lead to the degeneration of the blood contacting surface, resulting in thrombosis, calcification, decreased compliance and ultimately device failure. Additional components necessary for pulsatile pumps include unidirectional valves and seams where the valves and the pump housing intersect. These features of pulsatile blood pumps are subject to complications that can cause morbidity and mortality for the recipient [McCarthy and Sabi, 1994]. Pumps producing non-pulsatile flow generally have a single moving part, which is sufficient to sustain the flow, eliminating the need for flexing diaphragms, unidirectional valves, and seams. These characteristics of non-pulsatile flow pumps greatly reduce the risks of altered pump function, thrombosis and component failure. The potential for enhanced durability of non-pulsatile pumps is also an attractive feature when considering them for chronic circulatory support.

High efficiency power systems are a vital requirement for implantable blood pumps. Devices that produce pulsatile flow nominally require larger amounts of energy with low efficiency. The relative inefficiency of pulsatile flow originates from the multiple moving

parts and the necessary mechanical deformations of the pump components. In contrast, blood pumps producing non-pulsatile flow can deliver volumes of fluid equal to those delivered by pulsatile devices with less energy and acceptable efficiency. Non-pulsatile systems require a range of 2-20W of power with an efficiency of 25-60% [Akamatsu and Frazier, 1992].

Hemolysis, the destruction of red blood cell, is a feature of all mechanical blood pumps. No evidence proves that the presence or absence of pulsatility influence the development of hemolysis. The pulsatile devices have the unavoidable characteristics of turbulent flow and associated shear forces [Butler et al., 1991]. These forces are the primary cause of blood cell destruction. Advancements in pulsatile pump design, with emphasis on improved flow dynamics and alternation of ejection mechanics, have lessened the magnitude of inherent hemolysis. However, potentially severe hemolysis will always remain a component of a flexing diaphragm system with its unidirectional valves, dynamic internal components, and rapid acceleration of blood at high pressure through a restricted outflow orifice. Constant flow pumps are also not free of hemolysis. The design of continuous blood pumps and their mechanism of blood pumping impact the level of hemolysis. However, the inherent advantages of non-pulsatile blood pumps may make them more likely to avoid severe hemolysis [Araki et al., 1993].

The use of anticoagulation agents for the prevention of thrombus formation has been an unalterable requirement of pulsatile blood pumps. However, this method has many complications, including life-threatening problems. Additionally, there are patients who cannot tolerate anticoagulation agents because of a high risk of fatal hemorrhage for them. Improvement in biomaterials and fabrication techniques used for pulsatile blood pumps

has successfully lowered the risk for thrombus formation [McCarthy and Sabi, 1994]. Unfortunately, they have not eliminated the need for anticoagulation agents. Non-pulsatile blood pumps have avoided many of the thrombus generating components and design features of the pulsatile devices. However, some spots on axis of rotor, blood contacting impeller and casing remain potential sites for thrombus formation in continuous devices. In recent years, innovative techniques may decrease, if not eliminate, the formation of thrombus. For example, technique of using magnetic bearing in continuous blood pumps has been confirmed to be effective in reducing the thrombus formation [Nojiri et al., 1997].

2.4.2 Physiological Considerations

The traditional approach of total artificial heart and ventricular assist device development has been the mimicking of the native heart. However, with the increasing interest in continuous flow devices, a number of works have been done to challenge the prerequisite feature of blood pump functioning as a mimicking of natural heart beating. The absolute need for pulsatile blood flow and the maintenance of normal physiology was challenged in 1953. Wesolowski et al. (1953) used continuous non-pulsatile blood flow for a brief period of time to successfully maintain the circulation and normal organ function. Although man-made continuous flow pumps are unphysiological in nature, they could maintain circulation of humans for a prolonged duration. A growing volume of research data has accumulated during the past years, challenging the need for pulsatile flow in both systematic and pulmonary circulation [Dapper et al., 1992].

Support for the use of non-pulsatile flow was initially provided by observations and investigations performed during cardiopulmonary bypass (CPB) used for routine cardiac

surgical procedures. Although, the data has some limitations due to its short-term use as a CPB, it still has some relevance to non-pulsatile VAD and TAH investigations. The data indicated that non-pulsatile flow was less effective than pulsatile flow in maintaining homeostasis. However, more recent work, employing more sophisticated monitoring of these same systems demonstrated no advantage of pulsatile over non-pulsatile flow [Taguchi et al., 1988]. The non-pulsatile flow of CPB causes the renal blood flow to change. However, some works have disclosed that despite any changes in total renal blood flow and segmental distribution, there is no clinically discernable alternation of renal function when the circulation is supported by CPB non-pulsatile flow [Badner et al., 1992]. Additional researches have demonstrated that CPB non-pulsatile flow also maintains normal clinical functions of the brain, lung, and liver [Sakaki et al., 1992].

The chronic effects of non-pulsatile TAH or VAD have not been thoroughly investigated because the clinical application of continuous VADs and TAHs remains a recent advancement in cardiac surgery. Although small in number, the trials serve as the foundation for future evaluation of non-pulsatile blood pumps. The pioneering work was done in late 1970s at the Cleveland Clinic Foundation [Golding et al., 1980; 1982]. The study carried out biventricular bypass in calves using two centrifugal blood pumps. It is the first chronic non-pulsatile flow study conducted in mammals. The epoch made the first survivor lived for 34 days in 1979 with a completely de-pulse state. First implantation of continuous flow pump into human being was reported in 2000. Two male patients suffering from end-stage left heart failure were implanted with a DeBakey VAD axial-flow pump for use as a bridge to transplant. In both patients, the early postoperative phase was characterized by a completely non-pulsatile flow profile till the recovery of heart function took place 8 to 12 days after implantation [Weiselthaler et al., 2000].

The report of Golding (1980) documents a period of adaptation to non-pulsatile flow. During the first 7-10 postoperative days, calves receiving continuous flow devices demonstrated exaggerated levels of catecholamines, electrolyte imbalance, fluid overload, and anemia. These findings parallel the data accumulated from CPB experiments evaluating non-pulsatile flow. However, after this initial transition period, the animals reestablished near normal physiology. Daitoh (1978) has extensively evaluated the long-term effects of non-pulsatile perfusion in kidney physiology. His observations include the altered distribution of blood flow, decreased oxygen metabolism, etc. and there was no associated gross renal dysfunction or decrease in animal survival. In 1984, Sugita et al. demonstrated a need for more flow from a non-pulsatile blood pump than a pulsatile blood pump.

Chronic animal studies reported earlier and current clinical trials demonstrated that mammals including human could tolerate non-pulsatile flow for a prolonged duration [Weiselthaler et al., 2000]. Recovery of heart function was also reported even with the non-pulsatile flow by the appearance of the pulse. This evidence supports the idea that the mammals could adapt to a non-pulsatile blood flow. Pulse may not be a necessity from the point of peripheral organ perfusion. However, a large body of scientific data still needs to be accumulated regarding the engineering aspects of non-pulsatile blood pump and the physiological response to continuous flow.

2.5 Approaches in Studying Rotary Blood Pumps

Over the last decades, rapid advances have occurred in the development of rotary blood pumps. These have been fueled by a vast amount of work that have been done on it. The

methods used in these studies and developments varied from case to case. These methods are usually used as an evaluation of the qualities of a new pump design or improvements on it. It's hard to say which approach is more representative over the others as they are all standing for their individual principle of evaluations. However, these approaches can compensate each other to give us a more comprehensive guidance in the development of clinically reliable rotary blood pumps. Generally, there were several categories of approaches that were used in the investigations of rotary blood pumps in the past years:

2.5.1 Hydraulic Performance Test

A key issue in the design of rotary blood pumps intended for chronic use is to maintain a physiologically suitable output. A rotary dynamic pump produces flow in response to the difference between inlet and outlet. It is relatively common to design a rotary pump to produce a nearly constant pressure difference at a given rpm but the result is a very large swing in flow with small movements in pressure difference. For clinical use, however, a pump that could sustain the blood flow steadily under a wider range of pressure variation is preferred.

The fluid used in the hydraulic test is not necessary blood, a fluid such as glycerin water or other analogous bloods can be used. The total pressure head (outlet pressure less inlet pressure) and flow rate are measured at fixed rotational speeds and pressure-flow curves are generally plotted based on the results of measurement. Nondimensional parameters are usually used in the measured values. The coefficient relationships can be derived by dimensional analysis techniques from the basic equations governing fluid flow and are very useful in pump design.

The pump hydraulic efficiency (PE) can be calculated as following:

$$PE = Ph / Ps \quad (2-9)$$

The hydraulic output power of the pump (Ph in Watt) and the actuator input power (Ps in Watt) are calculated by the following equations respectively:

$$Ph = 0.002 \cdot \Delta P \cdot Q \quad (2-10)$$

$$Ps = 0.00034 \cdot RPM \cdot (T - T_0) \quad (2-11)$$

where ΔP is the total pressure head (mmHg), Q is the flow rate (l/min), RPM is the rotational speed (revolution per minute). T is the motor torque value when the motor is driving a pump, and T_0 is the torque value when the motor is operated without a pump head.

2.5.2 Hemolysis Test

Hemolysis test has been taken as a routine way in the evaluation of the antitraumatic features in the in-vitro study of rotary blood pumps. A mock-loop circuit is usually set up for the measurement, as shown in Figure 2.15. The circuit consists of tubes, blood reservoir, variable resistance and test pump. Flow meter and pressure indicators are also mounted in the loop to monitor the flow. The hemolysis level is tested by analyzing the blood samples extracted from the circuit in fixed time intervals. The blood used in the hemolysis test are usually fresh bovine blood, goat blood or human blood. The blood used in the test should be carefully kept so that it has very close properties as in the living body.

The normalized index of hemolysis (NIH) can be calculated according to the equation of Koller and Hawrylenko (1967):

$$NIH(g/100L) = \frac{\Delta FHb \times Volume \times [1 - (Hct / 100)] \times 100}{\Delta T \times Flow \ rate} \tag{2-12}$$

where ΔFHb is the increase in free plasma hemoglobin concentration (g/l) during the testing period, ΔT is the testing time (min), Hct is the hematocrit (%), Volume is the blood volume of each circuit (l), and flow rate is expressed as l/min. Besides the testing of hemolysis, which indicates the destruction level of red blood cells, other characteristics of blood trauma such as the destruction of white blood cells and the platelet activation and damage induced by rotary blood pumps can also be tested. The mock-loop circuit used in these tests are same as the one used in the test of hemolysis. However the methods used in analyzing the blood samples are different from each other for their own purpose of test [Takami et al., 1997(a); Kawahito and Nosé, 1997; Kawahito et al., 2000].

Figure 2.15 Schematic diagram of the mock-loop circuit used in the hemolysis test

2.5.3 Animal Test

Although costly and painstaking, animal tests are indispensable for any artificial heart before it can be used for clinical purposes. Healthy goats or bovines were usually selected

for the tests. Surgeries are required to be carefully conducted to exclude other factors that could impair the reflections of the animals. The animal survival and physiological responses of the animals after the pump implant are monitored and recorded for later study or clinical applications.

2.5.4 Flow Visualization and Computational Fluid Dynamics

The methods discussed in the previous sections are only used in the measurement of the overall performance of blood pumps. During the development process of the rotary blood pumps, however, large numbers of such tests are required even if one desires only small design improvements. As the blood trauma and thrombi generation as well as the pump efficiency are closely related to the inner flow dynamics of blood pump, flow visualization and computational fluid dynamics (CFD) analysis would be useful in evaluating the test results and reducing the number of the tests. In fact, the correlation studies between the pump tests and flow visualizations as well as CFD have been more and more enthusiastically conducted in recent years to establish the design methods of rotary blood pumps.

There are plenty of techniques available for experimental study of fluid flows. The major flow study techniques are CTA (constant temperature anemometer), LDA (laser Doppler anemometer), UDA (ultrasound Doppler anemometer), PTV (particle tracking velocimetry) and PIV (particle image velocimetry). Each has its own advantages and disadvantages for a particular study. Though the number of flow study within the blood pump has been increasing for the past decade, the results are still sketchy and mostly qualitative due to limitations during in vitro physiological condition simulation and application of flow study techniques.

There are a good number of cases that successfully used the flow visualization technique in the study of rotary blood pumps. In order to investigate the adequacy of the vertical angle of the eccentric inlet port of the C1E3 Gyro centrifugal pump, a flow visualization study was carried out by Takami et al. (1997(b)). Four pumps with different angles of eccentric inlet ports (0° , 30° , 60° and 90°) were studied and compared. The flow visualization study utilized a tracer method focused on the flow pattern just distal to the inlet port of each pump. The results of the flow visualization showed that, in the pumps with angles of 60° and 90° , the flow direction changed horizontally, causing a vortex formation; In the pump with the 0° angle, the inflow collided with the pump housing, resulting in a small vortex formation along the housing surface; in the pump with 30° angle, the inflow did not change its course, resulting in minimal space for vortex formation along the housing surface and the least hemolysis. The results suggested that the Gyro pump should have a 30° inlet port to have less vortex formation and less blood trauma.

Using flow the visualization technique, a study was performed by Asztalos et al. (1999) to correlate the areas of high shear velocity and stagnation with the possible sites of hemolysis and thrombus formation in a new closed-type centrifugal blood pump. The pump was a monopivot, magnetic-suspension blood pump with a normal rotational speed of 1900 rpm. Due to the insufficiency of the frame speed of the video camera, a scaled-up model was fabricated while maintaining flow similarly based on Reynolds law. The results showed that the flow in the front gap of the impeller was approximately about 30% of the main flow; the areas in the volute and around the washout holes were high shear locations and quantified with the highest shear velocity.

The mean velocity and Reynolds stress fields in the inner channels of Bio-pump were investigated by Piotti and Paone (1996). The velocity vector field was obtained using laser Doppler anemometer (LDA). A test circuit was designed to reproduce the flow resistance of a cardiopulmonary bypass circuit and to allow measurement of the velocity field in the centrifugal vaneless pump. The results revealed the influence of the inner pump geometry on the axial and radial velocity profiles and revealed that the pumping action of the vaneless centrifugal pump is restricted to a region in which the constrained forced vortex is established. The flow characteristics at the outlet of the rotating cones were also observed. Reynolds shear stresses were obtained based on the measurements of the velocity distributions. They found that flow disturbance caused by the cone tip and the fluid rearrangement at the initial portion of the channel increase the Reynolds normal and shear stresses respectively. LDA was proven to provide a suitable measurement methodology and its potential use with CFD techniques offers a powerful tool for developing new devices.

Yamane et al. (2004) carried out a flow visualization on a compact centrifugal blood pump developed for an implantable biventricular assist system using particle image velocimetry (PIV). The experiment was performed on a 250% scale-up acrylic pump model and a 64 wt% NaI solution was used as working fluid to avoid image deformation. Similarity law was applied to match the Reynolds number and specific speed between the acrylic model/working fluid and the actual device/blood. The experimental result indicated a similar flow pattern with an eccentric vortex in both the upper and lower gaps of the impeller. Based on the measurements of the inner flow field, oscillating shear

stresses were found around the pivot. The authors suggested that blood clotting would occur for a rotating speed less than 1,200 rpm at the bottom contact bearing.

Measurement of the inner flow fields of the blood pumps has some restrictions. Due to the compactness of the pump volume and the complexity of the inner structure of rotary blood pump, it is hard to get clear flow pictures in some strait locations in the pump. However, these locations are often of more interest to be studied. For some regions that are continuously swept by the high-speed rotor such as in the blade channels of the centrifugal pump, it is also difficult to get the flow images [Chua et al., 2006]. Using scaled-up model is a common treatment to solve the problem. However, it is still suspected that there might be some differences between the scaled-up model and the prototype.

In recent years, CFD has been typically used to assist the judgment in designing blood pumps. CFD-based design has been regarded as the best means to balance the often conflicting hydraulic and hematologic requirements in blood pump design [Burgreen et al., 2001]. A major benefit offered by CFD is the ability to evaluate designs quickly at an early stage, before committing to the expense of prototype fabrication and testing. At the detailed design stage, CFD can quickly investigate the effects of design changes on the blood flow. This would reduce the risk of unexpected knock-on effects that otherwise would become apparent only at a later stage. When a final design has been reached, CFD analysis can be used to confirm that design goals have been achieved. The detailed CFD picture of the flow field can be used often to support and explain experimental results, potentially strengthening regulatory submissions and providing a scientific base for clinical use.

There is a strong relationship between the mechanical loading on blood cells and their trauma. In-vitro hemolysis test requires experimental effort and quite a number of repetitions due to the large statistical variations. A computational assessment or quantification of shear-induced hemolysis in the design phase of artificial organs reduces the effort and cost of design. The choice of an appropriate and validated CFD solver is essential: the selected CFD solver must incorporate rotating frames of reference and acceptable boundary conditions for the rotor-stator frame interface. Other desirable solver features would include laminar and turbulent flow options for the incompressible Navier-Stokes equations, options for the non-Newtonian constitutive models for viscosity, and robust and rapid convergence characteristics [Burgreen et al., 2001]. The commercial CFD packages Fluent (Fluent, Inc., Lebanon, NH, USA) and CFX (AEA Technology plc., Oxfordshire, UK) are both excellent choices for rotary blood pump analysis [Burgreen et al., 2001].

Using the CFD technique, Miyazoe et al. (1998) studied the flow field within Nikkiso centrifugal blood pump numerically. TASCflow software was used for CFD analysis. In the simulation, standard $k - \varepsilon$ model was adopted and the number of nodes was about 260,000. The computational results were compared to the results of hemolysis test. The results of hemolysis test agreed with those predicted from the numerical solutions. CFD analysis revealed that the shear stress increased with the decrease of the gap dimension. They suggested that CFD analysis in comparison with hemolysis test could be a useful index for developing blood pumps.

In another example, computational fluid dynamics analysis was applied to investigate the flow within a microaxial blood pump [Apel et al., 2001]. Software package TASCflow (AEA, Otterfing, Germany) was selected as the solver, and the numerical model of pump was made by a three-dimensional grid generating software (Hexa, IcemCFD, Berkeley, CA, USA). Based on the numerical solution, flow induced shear stress was assessed by both the Lagrangian and mass statistical methods. They found that, for the microaxial blood pump, the viscous stress was of high importance compared to the turbulent stress and cannot be neglected. They also indicated that the turbulent model and near wall treatment have great effects on the accuracy of the prediction.

From above literature reviews, it can be seen that various means have been explored by researchers to understand in depth the effect of fluid hemodynamics on the performance of ventricular assist blood pumps. Combining with these techniques, detailed information of velocity and pressure distributions within the blood pumps, and subsequently, the locally high shear stress levels and/or flow stasis, which have the potential to invalidate a design due to adverse hematologic consequence, can be broadly investigated.

2.6 Previous Studies on the Kyoto-NTN Centrifugal Pump

For the Kyoto-NTN Magnetically Suspended Centrifugal Pumps (MSCP) developed by Terumo-Akamatsu group, the impeller is suspended and rotates freely at the center of the pump casing with the groups of magnet and the control mechanism. The schematic of the configuration of the magnets has been shown earlier in Figure 1.1 of Chapter 1. By the magnetically suspension design of the impeller, the shaft and seal are eliminated. However, the design configuration of the impeller itself can also affect the flow within

the pump. Sakuma et al. (1996) suggested that the recirculating flow within the impeller of centrifugal blood pumps would affect the rate of hemolysis. Furthermore, they demonstrated that the existence of irregular flow pattern within the impeller of blood pump would deteriorate its hemolytic properties. Three kinds of impeller configurations were tested by the Kyoto-NTN group. The impeller with 16 forward bending blades with an outlet angle of 90° , the impeller with 4, 6 and 7 backward bending blades with outlet angles of 30° , 40° and 40° respectively and the impeller with 7 straight blades with an outlet angle of 90° were tested. It was demonstrated that backward impellers are less efficient due to large hydraulic losses based on the profile design. Among the various configurations, the forward impeller produced the highest pressure and efficiency [Akamatsu et al., 1992; Akamatsu and Tsukiya, 1998].

In the hemolysis study, the index of hemolysis and temperature change in the Kyoto-NTN pump is better than those of the BIO-pump. However, it is still plagued by a layer of white thrombus, which adheres to the rough surface of the gap between the impeller and casing [Akamatsu et al., 1995]. A larger gap to a certain value is advantageous in the avoidance of thrombus formation [Ikeda et al., 1996], but it results in lower pump efficiency because of an increase in the regurgitant flow through the gap [Yamada et al., 1997]. The Kyoto-NTN MSCP model has a gap size of 0.2mm and a corresponding regurgitant flow in the range of 1-2 l/min. According to the study, at this regurgitant flow rate, no further thrombus was observed. Flow visualization of the Kyoto-NTN centrifugal pump was performed by Tsukiya et al. (1997). The results indicated that the volute tongue has remarkable effect on the flow in the blade channels. The flow in the blade channel was separated and unsteady even at the best efficiency point due to the interaction between the impeller and volute.

To further investigate the details of the flow in the pump, a 5 times scaled up pump model with similar geometry to that of the prototype had been fabricated to facilitate the study. The investigation of velocity in the impeller passages was performed at NTU, the models with impellers of 16 forward bending blades, 16 straight blades and 8 backward bending blades respectively had been measured using Laser Doppler anemometer (LDA), blood analog was used as the pumping fluid [Yu et al., 2001]. The main difference between the models and prototype is that the model was not coupled magnetically but was coupled mechanically and driven by motor. This is due to the fact that the 5 times scaled up model requires a strong and complicated magnetic suspension mechanism to suspend the heavy impeller. Therefore, DC motor and shaft were used as the support of the driving mechanism of pump model.

The measurement of the leakage flow of the Kyoto-NTN MSCP was performed by Chua and Akamatsu (2000) and Chua et al. (2002; 2003) on the 5:1 enlarged pump model. Both the radial and tangential velocities at the gap were measured under different flow rate. No cross flow in both the radial and tangential velocity distributions was found at the seven radial locations. The resulted vector plot showed that the double volute design of the pump has created a washout mechanism in the clearance gap. There is no vortex found at different flow rates of the pump. At the low flow rate, a more obvious washout mechanism was observed from the measurement results.

It should be reminded that published literature on the flow characteristics in centrifugal blood pump is indeed insufficient. For the Kyoto-NTN Magnetically Suspended Centrifugal Pump (MSCP), which is the main concern in this study, the leakage flow and

main flow in the pump were usually treated separately in the previous CFD works. However, the relationships between the leakage flow and main flow should not be neglected since the gap flow would take a remarkable portion of the total flow in the pump. As reported by Yamada et al. (1997), under the operating condition, the leakage flow rate is about 20-40% of the inlet flow. Therefore, a numerical simulation that studies the inner flow of the rotary pump as a whole would be supposed more contiguous to the real flow in the pump.

Furthermore, for the Kyoto-NTN pump, experiments have been carried out including the measurement of the flow in between the forward-bending, straight and backward-bending blades and in the gaps based on the 5:1 enlarged pump model. However, further studies are still required on the inner flow characteristics of 1:1 pump model. This is essential for the pump design study and also to understand the scale effect of the enlarged pump model. Therefore, in the present work, three-dimensional numerical simulations were carried out on the entire flow field including both gap flow and the flow in between the impeller blades in the 1:1 model of Kyoto-NTN MSCP with different impeller blade profiles. The measurements on the inner flow of the 1:1 pump model using LDA were carried out to validate the numerical solutions. This is of significance as the measurements using LDA on the exact size pump was seldom done as far as can be found in the literature.

CHAPTER 3

NUMERICAL METHODS AND THEORY

3.1 Physical Models

In this study, a blood pump model which is based on the prototype of the Kyoto-NTN centrifugal blood pump is the mainly focused. The basic structures and dimensions of the blood pump model are shown in Figure 3.1, there is only one inlet and one outlet in the pump. The outlet is a double volute design which situated at 180 ° apart, to direct the blood flow into two separate paths in the casing so as to prevent any unbalanced radial thrust on the magnetically suspended impeller. The impeller, which has a diameter of 50mm, is magnetically suspended at the middle of the pump casing and rotating in anticlockwise direction during pump operation. The inner surfaces of the pump casings are parallel to each other, which form the two gaps respectively with the two side faces of

Figure 3.1 Basic structures of Kyoto-NTN blood pump

the impeller shrouds. The inlet and outlet of the two pump gaps are located at the inner and outer brims of the impeller respectively. For the pump model in this study, the width of the pump gap is 0.2mm.

In order to show the components of the centrifugal blood pump more clearly, a schematic view of the pump is demonstrated in Figure 3.2. The pump has a unique moving component, the impeller, which has been totally suspended in the casing. The flow within a centrifugal blood pump is rather complicated. Blood enters the pump along the axial direction through the inlet pipe, after passing the stationary cap, the flow turn to the radial direction and then the impeller passages. In the blade passage, the centrifugal force of the rotating impeller drives the fluid radially outward. At the outlet edge of the impeller, a major part of blood goes into the volute and the remaining minor part enters the gaps between the pump casings and the impeller shrouds as shown in Figure 3.2. Then this portion of blood reenters the impeller blade channels through the eye and eventually leaves the pump through the pump volute.

Figure 3.2 A schematic view of pump configuration

Figures 3.3 (a), (b) and (c) demonstrate the profiles and primary dimensions of the three impellers, namely 16 forward bending blades (16FB), 16 straight blades (16SB) and 8 backward bending blades (8BB) respectively, in the study. The first prototype has an impeller diameter of 40mm but it was increased to 50mm to obtain a stronger magnetic force for more stable rotation of the impeller [Akamatsu, 1995]. In each of the impellers, four of the impeller blades are thicker than the others to accommodate the screws. The thicker blade would cause slight differences in the geometry of the blade channel of the impeller. As shown in the figures, this difference is more remarkable for the 8BB impeller and it is possible that this slight difference of the geometry would affect the flow patterns in the blade channel.

Figure 3.3 Profiles and dimensions of the (a) 16FB (b) 16SB and (c) 8BB impellers

The model generated for the numerical simulation has the same geometry as the Kyoto-NTN blood pump model. As shown in Figure 3.4, the model coincides in shape with the blood flow field region within the entire centrifugal pump. In order to achieve a steadier inlet flow, a prolonged inlet pipe, which is 5 times of the pipe diameter, was adopted in the inlet port of the simulation model.

Figure 3.4 Physical model of the pump

Figure 3.5 gives a detached view of the numerical simulation model. The inner geometry of the pump is quite complex. According to the flow characteristics within the pump, the simulation domain was divided into four subzones: they are inlet zone, impeller zone, volute zone and two gap zones. The impeller, which is the only moving component of the simulation model, is rotating around the Z-axis. During the steady working phase of the pump, the rotating speed of the impeller is fixed. The rotation of the impeller in the pump casing causes the pressure head to increase from the inlet to the outlet of the pump. During the solving of the numerical model, these zones were linked up and solved simultaneously.

Figure 3.5 Inner view of the model structure

3.2 Governing Equations (Mathematical Model)

In fluid flows, conservation equations for mass and momentum are used as governing equations. Under the Cartesian coordinates, the governing equations for the flow of Newtonian fluid are presented as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = S_m \quad (3-1)$$

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho F_i \quad (3-2)$$

$$\frac{\partial}{\partial t} (\rho I - p) + \frac{\partial}{\partial x_j} (\rho I u_j) = \frac{\partial}{\partial x_j} (u_i \tau_{ij}) + \frac{\partial}{\partial x_j} \left(\lambda \frac{\partial T}{\partial x_j} \right) \quad (3-3)$$

where:

$$I = C_p T + \frac{\vec{U} \cdot \vec{U} - \omega^2 r^2}{2}$$

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \mu \frac{\partial u_l}{\partial x_l}$$

$$\delta_{ij} = \begin{cases} 1 & i = j \\ 0 & i \neq j \end{cases}$$

$$i, j = 1, 2, 3$$

p : The static pressure

r : Radius

t : Time

u_i : The Cartesian components of $\vec{U}$

x_i : The Cartesian coordinates

ρ : Density of the fluid

μ : The molecular viscosity

ω : Rotating speed

C_p : Isotonic specific heat of the fluid

F_i : The body force

S_m : The source mass added to the continuous phase

T : Temperature

$\vec{U}$: Velocity vector

The body force F_i is the combination of gravitational body force ρg_i and external body force E_i . In a rotating reference frame, the body force F_i is expressed as:

$$F_i = g_i - 2(\vec{\omega} \times \vec{U})_i - [\vec{\omega} \times (\vec{\omega} \times \vec{r})]_i + E_i \quad (3-4)$$

where $\vec{\omega}$ is the rotation velocity and $\vec{r}$ is the radius vector.

For turbulent flow, it is impractical to resolve all the scales of motion. It is assumed that velocity fluctuations can be statistically averaged (a process called Reynolds-averaging). Reynolds stress was added to the stress term. For the incompressible flow, the stress term can be expressed as:

$$\tau_{ij}^T = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j} \quad (3-5)$$

where $\overline{u'_i u'_j}$ denotes the time average of velocity fluctuation at each vector component.

According to the Boussinesq hypothesis, the Reynolds stress is proportional to the mean velocity gradients:

$$\rho \overline{u'_i u'_j} = -\mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \frac{2}{3} \delta_{ij} \left(\rho k + \mu_t \frac{\partial u_l}{\partial x_l} \right) \quad (3-6)$$

Therefore, the Navier-Stokes equations still apply with the molecular viscosity μ replaced by the spatial variable viscosity μ_{eff} :

$$\mu_{eff} = \mu + \mu_t$$

The modified equations contain additional unknown variable μ_t . Turbulent models are needed to determine the variable in terms of known quantities. For the standard $k - \varepsilon$ model, the turbulent (eddy) viscosity, μ_t , is computed by combining k (turbulent kinetic energy) and ε (turbulent kinetic energy dissipation rate) as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

In order to solve all the variables, k equation and ε equation are added to the equation group. Therefore, the governing equations of the turbulent incompressible flow can be written as:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (3-7)$$

$$\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu_{eff} \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \delta_{ij} \rho k \right) + \rho F_i \quad (3-8)$$

$$\frac{\partial}{\partial t} (\rho I - p) + \rho u_j \frac{\partial I}{\partial x_j} = \frac{\partial}{\partial x_j} \left[C_p \left(\frac{\mu}{P_r} + \frac{\mu_t}{P_{rt}} \right) \frac{\partial T}{\partial x_j} \right] + \frac{\partial}{\partial x_j} (u_i \tau_{ij}^T) \quad (3-9)$$

$$\rho \frac{\partial k}{\partial t} + \rho u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P - \rho \varepsilon \quad (3-10)$$

$$\rho \frac{\partial \varepsilon}{\partial t} + \rho u_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_1 \frac{\varepsilon}{k} P - C_2 \rho \frac{\varepsilon^2}{k} \quad (3-11)$$

where:

$$\tau_{ij}^T = \mu_{eff} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k$$

$$I = C_p T + \frac{\bar{U} \cdot \bar{U} - \omega^2 r^2}{2} + k$$

The variables in above equations are time-averaged values. Equations (3-7) to (3-11) can be expressed as the following uniform formula, which is independent of coordinate system:

$$\frac{\partial}{\partial t} (\rho \Phi) + \nabla \cdot \bar{F} = S^\Phi \quad (3-12)$$

where:

$$\vec{F} = \rho \vec{U} \Phi - G^\Phi \nabla \Phi$$

$$\Phi = \begin{bmatrix} 1 \\ u_i \\ k \\ \varepsilon \\ T \end{bmatrix} \quad G^\Phi = \begin{bmatrix} 1 \\ \mu_{eff} \\ \mu + \frac{\mu_t}{\sigma_k} \\ \mu + \frac{\mu_t}{\sigma_\varepsilon} \\ \frac{\mu}{P_r} + \frac{\mu_t}{P_{rt}} \end{bmatrix}$$

$$S^\Phi = \begin{bmatrix} 0 \\ -\frac{\partial p}{\partial x_i} + F_i \\ P - \rho \varepsilon \\ (C_1 P - C_2 \rho \varepsilon) \varepsilon / k \\ \frac{1}{C_p} \left\{ \frac{\partial}{\partial x_i} \left[u_i \left(\mu_{eff} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k \right) \right] - \frac{\partial}{\partial x_i} (\rho u_i k) \right. \\ \left. - \frac{\partial}{\partial x_i} \left(\rho u_i \frac{\vec{U} \cdot \vec{U} - \omega^2 r^2}{2} \right) + \frac{\partial}{\partial t} \left(p - \rho k - \rho \frac{\vec{U} \cdot \vec{U} - \omega^2 r^2}{2} \right) \right\} \end{bmatrix}$$

$$P = \left[\mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k \right] \frac{\partial u_i}{\partial x_j}$$

$$\mu_t = C_\mu \rho k^2 / \varepsilon, \quad \mu_{eff} = \mu + \mu_t$$

In the above equations, $C_\mu, C_1, C_2, \sigma_k, \sigma_\varepsilon, P_r$ and P_{rt} are model constants with the following values [Launder and Spalding, 1974; Rodi, 1984; Markatos, 1986].

$$C_\mu = 0.09, \quad C_1 = 1.44, \quad C_2 = 1.92, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3,$$

$$P_r = 0.9, \quad P_{rt} = 0.72$$

The uniformity of the governing equations brings some advantages in the coding of the CFD software, for one does not have to program all the equations involved. In solving the flow problem, equation (3-12) is discretized on each control volume or element, which is defined based on the grid generated for the whole simulation zone. For convenience, equation (3-12) is also written in its integral form:

$$\frac{\partial}{\partial t} \int_{\Omega} \rho \Phi d\Omega + \int_S \vec{F} \cdot \vec{n} dS = \int_{\Omega} S^{\Phi} d\Omega \quad (3-13)$$

where Ω denotes the control volume or element, S denotes the surfaces of each control volume or element and $\vec{n}$, the surface vector.

3.3 Numerical Calculation Method

In this study, the numerical simulation was conducted by a commercial CFD package, which is FLUENT (Version 6.1.24). It provides two numerical methods: segregated solver and coupled solver. The two numerical methods employ a similar discretization process (finite-volume), but the approach used to linearize and solve the discretized equations is different. The segregated solver has traditionally been used for incompressible and mildly compressible flows while the coupled approach may give a performance advantage on solving high-speed compressible flows. In this study, segregated solver was selected for the incompressible flow employed.

3.3.1 Mesh Generation

As stated earlier, the first step in the modeling procedure is to generate grids for the flow geometry. FLUENT can use grids which comprise of triangular or quadrilateral cells (or a combination of the two) in two dimensions, as well as tetrahedral, hexahedral, pyramid, and wedge cells (or a combination of these) in three dimensions. The choice of mesh type

will depend on the application. When choosing the mesh type, the following issues were considered:

- Setup time

Many flow problems solved in engineering practice involve complex geometry. The creation of the structured or block-structured grids (consisting of quadrilateral or hexahedral elements) for such problems can be extremely time-consuming, if not impossible. Setup time for complex geometry is therefore the major motivation for using unstructured grids employing triangular or tetrahedral cells.

- Computational expense

When geometry is complex or the range of length scales of the flow is large, a triangular or tetrahedral mesh can often be created with far fewer cells than the equivalent mesh consisting of quadrilateral or hexahedral elements. This is because a triangular or tetrahedral mesh allows cells to be clustered in selected regions of the flow domain, whereas structured quadrilateral and hexahedral meshes will generally force cells to be placed in regions where they are not needed. Unstructured triangular and tetrahedral meshes offer many advantages for moderately complex geometry. However, quadrilateral and hexahedral elements are also economical in some situations for they permit a much larger aspect ratio than triangular and tetrahedral cells. For relatively simple geometry, in which the flow conforms well to the shape of the geometry, such as a long thin duct, high-aspect-ratio quadrilateral or hexahedral cells may be a better choice.

- Numerical diffusion

A dominant source of error in multi-dimensional situations is numerical diffusion, also termed false diffusions. All practical numerical schemes for solving fluid flow contain a finite amount of numerical diffusion. This is because numerical diffusion arises from truncation error that is a consequence of representing the fluid flow equations in discrete form. Higher order discretization scheme used in FLUENT can help in reducing the effects of numerical diffusion on the solution. Note that the amount of numerical diffusion is inversely related to the resolution of the mesh. One way of dealing with numerical diffusion is to refine the mesh. Furthermore, numerical diffusion is minimized when the flow is aligned with the mesh.

In regions where the flow pattern is expected to be complicated, especially the region near the sharp or abrupt features, a greater nodal density is required in order to provide a given level of approximation and hence accuracy. An efficient way is to refine only the areas where the geometry or solution is expected to be more complex (rapid variations) or at regions of interest where greater accuracy is required.

Based on the disciplines discussed above, grids were generated. Figures 3.6 (a), (b) and (c) show the meshed simulation model of the pump with the 16FB, 16SB and 8BB impellers respectively. Element density and element shape are primary factors that affect the grid quality, which needs to be combined with the concerns of geometry and anticipated flow characteristics. The shapes of the grid cells of the simulation models are mainly hexahedron, tetrahedron, wedge and pyramid. The grid was most dense at the regions around the impeller blades and coarser at the inlet and outlet of the pump as the flow at the impeller

(a)

(b)

(c)

Figure 3.6 Grid generation for the (a)16FB (b)16SB and (c)8BB pump models

regions was anticipated to be most complicated while at the inlet and outlet of the pump, the flow was anticipated to be smoother. At different zones of the pump model, the meshes were generated independently.

Different components in this model are connected by interfaces. The interface means a pair of faces lapping over each other, between which the information about the flow can be exchanged. To compute the interface flux, the intersection between the interface zones is determined at each new time step. The adoption of interface brings some flexibility in the grid generation because the interface does not require the grid faces to be aligned on it. For the models with complex geometry, this feature is very useful in overcoming the over skewing of the cells, thereby, resulting in a good mesh quality. Interfaces are also the

boundaries of the rotating and stationary reference frames. Sliding mesh was used to solve the interactions between the rotational and stationary parts in this study.

Since the quality of mesh plays a significant role in the accuracy and stability of the numerical computation. The attributions associated with mesh quality such as smoothness (a measure of changes in cell volume between adjacent cells), skew (which can be defined as the difference between the cell shape and the shape of an equilateral cell of equivalent volume, highly skewed cells can decrease accuracy and destabilize the solution) and face ratio (which is a measure of the stretching of the cell) are examined, before reading into FLUENT.

3.3.2 Using Finite Volume Method to Solve Governing Equations

In this study, segregated solver was used to solve the governing equations sequentially (i.e., segregated from one another). Because the governing equations are non-linear and coupled, several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the steps illustrated in Figure 3.7 and outlined below:

1. Fluid properties are updated based on the current solution. (If the calculation has just begun, the fluid properties will be updated based on the initialized solution.)
2. The u , v and w momentum equations are each solved in turn using current values for pressure and face mass fluxes in order to update the velocity field.

3. Since the velocities obtained in Step 1 may not satisfy the continuity equation locally, a “Poisson-type” equation for the pressure correction is derived from the continuity equation and the linearized momentum equations. This pressure correction equation is then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass flux such that continuity is satisfied.
4. Where appropriate, equations for scalars such as turbulence, energy, species (including the properties of the fluid) and radiation are solved using the previously updated values of the other variables.
5. When interphase coupling is to be included, the source terms in the appropriate continuous phase equations may be updated with a discrete phase trajectory calculation.
6. A check for convergence of the equation set is made.

FLUENT uses a control-volume-based technique to convert the governing equations to algebraic equations that can be solved numerically. The performance of finite volume method starts from the integral form of the conservation equations, which are applied to each Control Volume (CV) and discretized in order to obtain one algebraic equation per CV. Each equation involves the unknown from the CV-center and from a certain number of neighboring CVs.

Figure 3.7 Overview of the segregated solution method

After discretization, the conservation equation for a general variable at a cell P can be written as:

$$a_p \Phi_p = \sum_{nb} a_{nb} \Phi_{nb} + b \quad (3.14)$$

where a_p and Φ_p are the coefficients and variable values at the center of cell, a_{nb} and Φ_{nb} are the influence coefficients and the variable values for neighboring cells. Meanwhile, b is the contribution of the constant part of the source term S_C in $S = S_C + S_p \Phi$ and of the boundary conditions.

3.4 Boundary Conditions and Calculation Configurations

3.4.1 Boundary Conditions and Assumptions

The boundary conditions are set according to the pump operating state with an impeller rotating speed of 2000 rpm and pump flow rate of 5 liters/min. At the inlet port, the static pressure was set as boundary condition. In using FLUENT, the total inlet gauge pressure was required as an input and was set to ‘zero’ in the present study. The static pressure was calculated according to the following equation:

$$p = p_0 - \frac{1}{2} \rho |V|^2 \quad (3.15)$$

where V is the inlet flow velocity. As the flow is incompressible, the ambient pressure has no effect on the flow. In this simulation, the ambient pressure was set as standard atmospheric pressure.

The turbulent kinetic energy (k) and turbulent dissipation rate (ε) of the incoming flow were estimated according to the empirical equations of fully developed flow in duct [FLUENT Manual 6.2, 2005]:

$$k = \frac{3}{2} \left[0.16 V_{avg} \left(\text{Re}_{D_H} \right)^{-\frac{1}{8}} \right]^2 \quad (3.16)$$

and

$$\varepsilon = 2.35 \frac{k^{\frac{3}{2}}}{D_H} \quad (3.17)$$

where $\text{Re}_{D_H} (= \frac{\rho V_{avg} D_H}{\mu})$ is the Reynolds number of the flow in duct with the hydraulic

diameter $D_H (= \frac{4A}{p_e})$ as the characteristic length.

where ρ is the fluid density, V_{avg} is the average velocity in the duct, μ is the dynamic viscosity, A is the duct cross sectional area and p_e is the wetted perimeter.

Static pressure was set at the inlet and outlet boundary. With the setting of pressure boundary conditions at the inlet and outlet, the required flow rate through pump model can be obtained through iteration. The pressure at the outlet could be adjusted during the simulation so as to achieve the pump flow rate of 5 liters/min. This pressure adjustment function is provided in FLUENT software and was activated during the simulation.

The flow in the pump model was assumed as unsteady. The fully implicit discretization scheme was adopted so that Courant stability restriction on the time-step size can be avoided. However, over-large time-step size was avoided to reduce the convergence difficulty. Due to the periodic property of the rotor/stator model, 20 time steps were set for each blade passing, i.e. 320 time steps per revolution of the impeller with each time step equals to 9.38×10^{-5} s. The cell Courant number of the whole field was ranged from 0.2 to 14. Sliding meshes were used in the coupling between the pump rotor and stator, the impeller orientations were calculated after each time step. No slip boundary condition was assumed at the solid wall of the model.

In this study, the fluid is considered as a Newtonian fluid, which was set to have the same viscosity and density as those of human blood. A Newtonian fluid is, by definition, one in which the coefficient of viscosity is constant at all shear rates (Nichols and O'Rourke, 1990). Most of the homogenous fluids can be closely approximated by this characteristic but suspensions of particles deviate from it and the deviation would be apparent as the particle size becomes appreciably large in comparison to the dimensions of the channel in

which it is flowing. Human blood is essentially a suspension of erythrocytes in plasma and shows anomalous viscous properties. However, the variability of blood viscosity is detectable only when the shear rates are very small and the diameter of the flow channel is less than 1 mm (i.e. about 100 times of the major diameter of the red blood cells). For the centrifugal blood pump models, the dimensions of the flow field in the pump are generally over 3 mm. Therefore, the Newtonian treatment of blood in this study should be a reasonable assumption.

The device Reynolds number ($Re = \rho u_t D / \mu$) of the pump is 78,500, where u_t and D are the impeller tip speed and impeller diameter respectively. In the model, a standard $k - \varepsilon$ model was adopted in the simulation. The $k - \varepsilon$ model has been successfully used in recent years in designing various types of blood pumps. However, the limitations of $k - \varepsilon$ model such as inaccuracy in simulating the low-Reynolds number turbulent flow or turbulent flow with swirl and separation near the solid boundary were also widely reported. In the simulation, an enhanced wall treatment has been adopted along with the $k - \varepsilon$ model in order to consider the effect of viscous flow below the fully turbulent region in the turbulent boundary layer. The enhanced wall treatment blends linear (laminar) u_{lam}^+ and logarithmic (turbulent) u_{turb}^+ laws-of-the-wall using a function suggested by Kader (1993),

$$u^+ = e^{\Gamma} u_{lam}^+ + e^{\frac{1}{\Gamma}} u_{turb}^+ \quad (3.18)$$

where, $\Gamma = -\frac{a(y^+)^4}{1 + by^+}$, $a = 0.01c$, $b = \frac{5}{c}$, $c = \exp(f_r - 1.0)$

where $y^+ = \frac{y(C_\mu^{1/4} k^{1/2})}{\nu}$, $C_\mu = 0.09$ and f_r is a roughness function.

The enhanced near-wall function extends the application of the standard logarithmic near wall function from fully turbulent region to the laminar sublayer, buffer region, and fully-turbulent outer region. The viscosity-affected near-wall region is thus resolved all the way to the viscous sublayer of the near wall boundary of the pump model.

For the blood flow within a centrifugal pump, there is definitely a slight temperature fluctuation while the blood flowing through the pump. However, because the fluid is totally incompressible and there is a relatively short staying time of the blood in the pump, the effect of the temperature fluctuations on the fluid density and viscosity are very small. Therefore, the change of the flow velocity distribution within the pump caused by the temperature fluctuation is not considered in the study.

3.4.2 Configurations of Calculation

During the solution, the second-order implicit scheme was used in the temporal discretization, the second-order Upwind discretization scheme is used in the k and ε equations and Quick scheme was used in the momentum equation.

Pressure-velocity coupling is achieved by deriving an equation for pressure from the discrete continuity equation. FLUENT provides three pressure-velocity coupling algorithms: SIMPLE, SIMPLEC and PISO. All these algorithms use a relationship between velocity and pressure correction to enforce mass conservation and to obtain the pressure field. In this study, SIMPLEC was found to have a better convergence rate with relax factors of 0.5-0.7 and 1 for momentum and pressure equations respectively.

At the end of solver iteration, the residual sum for each of the conserved variables is computed and stored, thus recording the convergence history. The criterion of convergence is explained as following.

The residual R^Φ computed by FLUENT's segregated solver is the imbalance in equation (3-14) summed over all the computational cells. This is referred to as the “unscaled” residual. It can be written as:

$$R^\Phi = \sum_{cellsP} \left| \sum_{nb} a_{nb} \Phi_{nb} + b - a_p \Phi_p \right| \quad (3-19)$$

In general, it is difficult to judge convergence by examining the residuals defined by equation (3-19), since no scaling is employed. This is especially true in enclosed flows such as natural convections in a room where there is no inlet flow rate, which can be used to compare the residual. FLUENT scales the residual using a scaling factor representative of the flow rate through the domain. This “scaled” residual is defined as:

$$R^\Phi_s = \frac{\sum_{cellsP} \left| \sum_{nb} a_{nb} \Phi_{nb} + b - a_p \Phi_p \right|}{\sum_{cellP} |a_p \Phi_p|} \quad (3-20)$$

For the momentum equations the denominator term $a_p \Phi_p$ is replaced by $a_p V_p$, where V_p is the magnitude of the velocity at cell P. In this project, the scaled residual is used as a criterion, since it is a more appropriate indicator of convergence. The convergence criteria set for velocity is 10^{-5} in this study. All the calculations are carried out on the workstation (SGI Origin 2000) with the operation system of SGI IRIX.

3.5 Numerical Validation

Computational fluid dynamics offer much more versatility and resolution than in-vivo or in-vitro method, yet computations must be validated carefully to estimate the accuracy of result, the accuracy of solutions cannot be determined exactly but can only be estimated by comparing with some available data.

Prior to producing the final results, various aspects of the numerical model were tested. It included: grid independency, advection models and time-step size. The tests were carried out under the impeller rotating speed of 1920 rpm and the pressure difference of 10,000 Pa between the inlet and outlet ports. The fully implicit scheme is used in the transient flow study of this project, for it is unconditionally stable with respect to the time-step size. The time steps of 1/80, 1/160, 1/320 and 1/500 per time-period have been tried and examined by comparing the resulted mass flow rate. Note that the time-period is defined as the time for the impeller to complete a revolution. 1/320 (9.38×10^{-5} s) of the time period was chosen in the present simulation. To eliminate the starting effect of the transient flow, the computation is carried out over a relatively long time (4 seconds) to ensure that the flow in the pump is fully developed. The monitoring of mass flux at the pump outlet port is taken as a reference for judging the flow development. The resulted flow in the middle plane of the impeller and the mass flux at the inlet port were used as indicators of the effects of the various models.

Different grid schemes were used to check if the solution converges. When the meshes get finer, convergence towards some fixed values provides some level of confidence about the solution obtained. Figure 3.8 shows the comparison of resulted mass flow rates of the 16FB pump model under different grid schemes. As shown in the figure, the mass

flow rates under 282,000 nodes has a larger discrepancy with the other two mesh schemes, while the flow rate under 510,000 nodes (presently adopted scheme for the 16FB pump model) is acceptably close to the value under further densified scheme (650,000 nodes) with a maximum difference of 2%. Similarly, grid independence was also checked for the 16SB and 8BB pump models. Current grid schemes were found to be valid in solving the inner flow of the pump models.

Figure 3.8 Mass flow rate at the pump inlet under different mesh schemes

In all the results presented, the advection terms were modeled using a second-order accurate Upwind and Quick schemes. The first-order accurate Upwind Scheme did produce small difference in the velocity distribution, which may be due to the false diffusion errors associated with the lower-order discretion scheme.

In order to have a stable flow at the inlet, an extended inlet pipe was adopted. The length of inlet pipe in the present model is 5 times of the pipe diameter. A longer pipe has been

tested, the length of which is ten times of the pipe diameter, the velocity distributions were found to be unaffected by this change.

3.6 Post-processing

When the computation is converged, some further data can be calculated based on the solution of flow field. In assessing the severity of the shear stress that exerted on the blood in the flow field, a scalar shear stress value was adopted in this study according to the method proposed by Bludszuweit (1995).

$$S_{scalar} = \frac{1}{\sqrt{3}} \sqrt{\sigma_{ii}^2 + \sigma_{jj}^2 + \sigma_{kk}^2 - \sigma_{ii}\sigma_{jj} - \sigma_{jj}\sigma_{kk} - \sigma_{kk}\sigma_{ii} + 3(\sigma_{ij}^2 + \sigma_{jk}^2 + \sigma_{ki}^2)} \quad (3.21)$$

The shear stress in turbulent flow was obtained as follows,

$$\sigma_{ij} = \eta_{ij} + \tau_{ij} \quad (3.22)$$

where η_{ij} and τ_{ij} are molecular stress and Reynolds stress respectively, which are given as,

$$\eta_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3.23)$$

and

$$\tau_{ij} = -\frac{2}{3} \rho k \delta_{ij} + \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3.24)$$

where μ_t is the fluid turbulent viscosity and δ_{ij} is the Kronecker's symbol. FLUENT provides the components of the velocity gradient at each node. Therefore, the intensity of shear force of the blood was assessed by the scaled stress value in the whole flow field.

The resulted data was stored or exported as other format file that can be used for the further post-processing. Moreover, FLUENT also provides the graph tools to draw the vector and contour graphs. In this project, velocity vectors and profiles are captured by FLUENT, while contours and plot of shear stress distributions are mainly done using Tecplot and Excel.

CHAPTER 4

LDA MEASUREMENT METHODOLOGY

4.1 Experimental Pump Model

4.1.1 Experimental Flow Circuit

In this study, the flow in the blade channel of 1:1 Kyoto-NTN centrifugal blood pump model was measured using laser Doppler anemometer (LDA). In order to simulate the operating condition of the pump, a closed loop flow circuit was fabricated for the experiment as shown schematically in Figure 4.1. In the circuit, most of the working

Figure 4.1 Schematic view of the pump flow circuit of the experiment

fluid was contained in a reservoir, which is located 70cm above the pump model. The cylindrical reservoir was made of thick acrylic plate and has an internal diameter of 140mm and height of 330mm. Flexible silicon-platinum hoses with internal diameter of 12.7 mm were used to connect the pump model and reservoir. The mass flow rate is controlled by Valves A and B which are located at the inlet and outlet ports of the reservoir respectively. A flow meter was placed at the exit of the reservoir to measure the flow rate. The advantage of using soft hoses in the pipeline is that they provide more flexibility in arranging the different components in the flow circuit. Across the pump model, a pressure transducer was mounted on the pipeline to measure the pressure head generated between the inlet and outlet of the pump model. Valve C controls the overflow of the working fluid in the circuit and it will be closed during the operation of the pump model.

4.1.2 The Pump Unit

An acrylic model of the Kyoto-NTN centrifugal blood pump was fabricated for measuring the inner flow of the pump. The model pump consists of three main parts: the impeller, pump casing and driving mechanism. The pump has a double volute pump casing, which was manufactured in two separate parts: the lower and upper pump casings which are shown in Figure 4.2. The two pieces of the pump casing are assembled tightly into a single piece by the screws. A cable seal is used to prevent leakage through the seam between the casings. The cable seal was embedded in a way that following the contour of the outer most recess profile. Silicon grease was applied to all the seal contacting surfaces to enhance the sealing effect and to lubricate the contacting surface. There are eight drilled-through holes on the two pieces of the casing to accommodate the bolts and nuts that are used to tighten them up. Washers

were attached in between the contact surface of the casing and the bolts (and the nuts) to prevent over loading of local stress that could cause fracture and damage to the acrylic casing.

Figure 4.2 Schematic of the acrylic casing of the pump model

The pump casings were mounted on the alignment block with two steel plates. The alignment block is seated in the guiding block (which was fixed on the experimental rig platform) and is allowed to slide within it. The movement of the alignment block is made possible by loosening the screws on both sides. The moving of the alignment block is required for the adjustment of the axial clearance between the impeller and pump casing, which is crucial for correctly mimicking the inner flow of the pump. Both the alignment and guiding blocks are made of stainless steel and machined to a tight tolerance fit to maintain the axial clearance between casing and impeller.

The impeller was sandwiched between the pump casings by means of bolts and nuts. The impellers were of close type. Its dimensions were adopted from the prototype of the Kyoto-NTN blood pump [Akamatsu et al., 1992]. The impeller has the internal and external diameters of 26mm and 50mm respectively. The impeller blades and lower shroud of the impeller were machined from a solid acrylic block. The upper shroud, which was made of clear acrylic plate, was polished and mounted onto the lower shroud by means of four screws. The vanes of the impeller have uniform height of 3.5mm.

For the prototype of the Kyoto-NTN centrifugal blood pump, the impeller was magnetically suspended and rotated in blood between the pump casing. However, in the model pump of present study, the impeller was mounted onto a shaft to avoid the use of the magnets as the opaque magnets would block the laser beam during measurement. The shaft was driven by a servo motor unit that ensures constant speed of the centrifugal blood pump.

Figure 4.3 shows a schematic view of the driving mechanism of the pump model. The driving shaft was coupled to the servo motor at one end. The pinion on the driving shaft was attached to the meshing gear on the hollow shaft. The end of the hollow shaft was connected to the impeller. The center cone in the pump that directs the fluid from the inlet to the impeller was attached to a separated shaft, called center shaft, to avoid it from turning together with the impeller and to ensure that the flow condition of the experimental model and the prototype are the same. The center shaft was accommodated in the hollow shaft and the end was extended to the back of the supporting block where it can be fastened and held stationary by the back ring.

Figure 4.3 Driving mechanism of the experimental pump model

Figure 4.4 shows the detailed sectional drawing of the assembly of the center shaft and the hollow shaft. The circumference of the hollow shaft is positioned by two bearings mounted in the supporting blocks. The rotating parts consist of the hollow shaft, shaft head and the impeller. When the driving shaft rotates, it will turn the hollow shaft together with the head shaft and the impeller while the center shaft is kept stationary by the back ring.

A servo motor is selected for the pump model because it can maintain a constant rotating speed which is required to be maintained in the experiment. For the current servo motor in the pump driving mechanism, it has a rated torque of 0.64 Nm at a rated speed of 3000 rpm. The servo motor was controlled by a NPSA-Z series digital control unit (Nikki Denso Co., Ltd.), through which, all the motion control of the

servo motor can be achieved. In the control unit, a shaft encoder was built in. The shaft encoder signals can be sent out directly from the control unit. Therefore, an external shaft encoder device that needs to be attached to the moving shaft can be avoided in the experiment.

Figure 4.4 Detail sectional driving of the pump unit, hollow shaft and center shaft

4.2 Measurements with Laser Doppler Anemometer (LDA)

4.2.1 Basic Theory of LDA

Laser Doppler Anemometer (LDA) is a non-invasive instrument to measure the velocity of a flow. In LDA, a polarized laser beam is split into two parallel beams using a beam splitter. These parallel beams are then focused on a single point. Since the laser beam is polarized, an interference pattern is formed within the intersecting volume. The interference pattern is oriented so that the maxima and minima surfaces

lie within planes perpendicular to the plane in which the two beams lie and parallel to the central axis of the laser probe. Within the intersecting volume is a region called the measuring volume. The measuring volume is an ellipsoid whose boundaries are defined by the surface where the light intensity is equal to the intensity at the center of the intersecting volume multiplied by e^{-2} .

The interference pattern can be used to measure the velocity of a particle in the following way. Suppose a particle is passing through the center of the measuring volume and that its velocity vector lies within the plane of the split beams and is also perpendicular to the axis of the laser. As it passes through the interference pattern, it alternates between being in a maximum or a minimum until it exits the other end of the measuring volume. When in a maximum, the particle would reflect a great deal of light and when in a minimum it would reflect very little light. A photodetector could then be used to measure the amount of light reflected by the particle as it traveled through the measuring volume. The signal produced by such a detector would be similar to a sine wave whose frequency depends both on the velocity of the particle perpendicular to the interference pattern and on the fringe spacing within the measuring volume. The signal pulses will not be equal in amplitude but will have a bell shape as the intensity of the light is the greatest at the center of the measuring volume. The equation that relates the frequency of the signal produced by the photodetector to the velocity of the particle can be written as:

$$\text{velocity (m/s)} = \text{fringe spacing (m)} * \text{frequency (Hz)}.$$

4.2.2 Description of LDA System

In this study, LDA system was used to measure the flow within the blade channel of a centrifugal blood pump. Figure 4.5 shows the schematic view of the LDA layout. The LDA system used is manufactured by TSI Incorporated. The laser generator is a 300mW Argon-ion laser from Omnicrome™. The laser beam from the Argon-ion laser generator is input to the Colorburst multicolor beam separator. The laser is then separated into two visible green and blue lights at wavelength of 514.5nm and 488.0nm respectively.

Figure 4.5 A schematic of the laser Doppler anemometer system layout

In the Colorbust, the laser is converted from a single multilane laser into four laser beams, two green and two blue beams. One of the laser beams of each color has a 40MHz frequency shift added to allow for measurements in negative flows. Each of the green and blue colors is for measuring one of the velocity components. The laser beams are then transmitted to the fiber optic probe by a separate single-mode, polarization-preventing optical fiber coupler.

The fiber optic probe contains a transmitting lens, which has a focal length of 67.8mm and aperture of 10.5mm, to focus and intersect the laser beams to form an elliptical measurement volume. Figure 4.6 shows a schematic view of the laser transmitting system and the measurement volume formed. The resulting measurement probe volume of the green beam is 1.70mm (l_m) long and 100 μ m (d_m) wide, with 23 fringes of 4.36 μ m spacing between two fringes. The measurement volume formed by the blue laser beams has a length of 1.60mm and width of 95 μ m, with 23 fringes of 4.13 μ m spacing between two fringes. The dimensions of the measurement volume can be calculated from the incident light wavelength λ , frequency f , the laser beam diameter d_l , constant e^{-2} and inclined angle between the two beams θ_i . The width (d_m) and length (l_m) can be calculated as:

$$d_m = \frac{4f\lambda}{\pi d_l e^{-2}} \quad (4.1)$$

$$l_m = \frac{d_m}{\tan(\theta_i/2)} \quad (4.2)$$

The number of fringes, N_f along the x-axis is given by

$$N_f = \frac{d_m}{\delta} \quad (4.3)$$

where δ is the fringe spacing.

Figure 4.6 A schematic view of the laser transmitting system and the measurement volume

The optical receiving fiber, which receives the back-scattered light from the particles in the fluid, was integrated in the fiber optic probe. The light signal is transmitted to Colorlink multicolor receiver. It separates this light into green and blue lights of 514.5nm and 488.0nm wavelength by using dichroic mirrors and notch filters. The lights are then converted into electrical signals by the photomultiplier tube (PMT). The signal is high-pass filtered to remove the pedestal and then mixed with internal

frequency. The resulting frequency is then low-pass filtered and an analog signal is sent to the IFA755 signal processor that extracts frequency data from the signals.

The IFA755 digital burst Correlator is a digital signal processor that is specially designed to process signals produced by the LDA. It distinguishes the signal burst from the noise purely by the signal to noise ratio (SNR). It receives analog signals from the Colorlink multicolor receiver and extracts velocity information from these signals. Its main components are a bandpass filter, amplifier, burst detector, burst detector digitizer, sample digitizer and a digital burst autocorrelator.

The measurement of the flow in between the impeller blades requires a rotating machinery resolver module (RMR). The RMR contains a shaft encoder that is connected to the driving motor shaft. The shaft position signal is sent to the DataLink multichannel interface unit that allows data acquisition to include peripheral data, such as temperature and pressure, synchronous with velocity data from TSI LDA signal processor. The RMR module provides the ability to match individual velocity data points with angular position of rotating machinery. Every time a signal processor records velocity data from a particle in fluid flow, a signal is sent to the RMR module, causing it to latch the angular position of the rotary device at the same instant. An integer corresponding to the angular position information is sent with velocity data to the computer.

4.3 Working Fluid

The main difficulty in performing velocity measurements inside the centrifugal blood pump roots in the small dimensions and complex geometry of the pump. Only a

nonintrusive measurement technique, such as LDA, can be employed for velocity measurement. However, in the measurement of the flow in the blade channel, the laser beams would totally transmit through 6 interfaces between the acrylic plastic and working fluid. The refraction occurring at these interfaces would cause the deformation of the measurement volume and interference pattern. On the other hand, because of the small dimensions of the pump, the reflection of the laser light occurred at the interfaces is very close to the measurement volume. The scattered light from the particles in the fluid would then be overwhelmed by the reflected laser shine from the interfaces.

In order to overcome these difficulties in the measurement, a working fluid that matching the refractive index of the acrylic plastic ($RI=1.49$) was used. The working fluid comprised a mixture of Tin(II) Chloride, water and glycerin. Mixtures of various combinations in different proportion of Tin(II) Chloride were carried out and tested in order to obtain a mixture that has a refractive index which matches the pump casing material, and also has the same viscosity of the blood that is used to simulate the flow in the numerical pump model. The working fluid used in this study involved mixing about 25% of pure glycerin and 75% aqueous solution of Tin(II) chloride by weight. In order to maintain stability of the mixture, a very small amount of hydrochloric acid was added. The refractive index and the viscosity of the working fluid were measured using a commercial refractometer (model ATAGO 3T) and a controlled rate rheometer (model Contraves Low Shear 40) respectively. The measurements were performed under a temperature of 25°C , which is same as the temperature of the air-conditioned room where the LDA measurement was taken. The working fluid has a

refractive index of 1.49, dynamic viscosity (μ) of 5.4×10^{-3} Pa·s and density of 1.618×10^3 kg/m³, resulting a kinematic viscosity of 3.3×10^{-6} m²/s.

4.4 Experimental Arrangements and Procedure

The centrifugal blood pump model in the experiment was mounted on a steel platform with the pump front casing surface perpendicular to the platform. The steel platform and the pump flow circuit were located on a horizontal experimental table. Usually the measurements of liquid flow velocity using LDA do not need seeding as the particles carried by the impure liquid would be sufficient to meet the requirement of a LDA measurement. However, due to the high speed and small dimensions of the pump model, measurements of the flow in between the impeller blades using LDA would encounter remarkable noises, which were high enough to block the genuine signals of the flow. Seeding was thus required. The seeding selected was silver coated hollow glass sphere with an average diameter of 10 μ m. The following property of the seeding was tested by a sedimentation experiment: add the seeding to a working fluid sample and keep the working fluid still. No apparent sediment or floatation of the seeding particles was observed after fifty minutes.

In the measurement, the fiber laser probe was mounted on the precision automated three-degree freedom traversing system. The motions along the X, Y and Z axes of this software-controlled traversing mechanism were driven by step motor with an accuracy of ± 0.0127 mm (0.0005 inch). The center axis of the laser probe, which is parallel to the Z axis of the traverse mechanism, was perpendicular to the front casing surface of the pump model. The locations of the measurement points were defined by the coordinate data in a traverse grid map file, which is transferred to the LDA control

software before the measurement. In order to ensure the measurement points at the same location of the pump model, the coordinate data in the traverse grid map file was set relative to a starting point, which is marked on the inner surface of the pump casing. Therefore, matching of the measurement point on the pump model can be achieved by aligning the traverse mechanism at the starting point before the measurement with a maximum difference of 0.2mm.

Under operating condition, the impeller was rotating at a speed of 2000 rpm. Therefore, finding a measurement point in a specified blade channel cannot be fulfilled by the movement of the traversing mechanism only. Thus the shaft encoder and the RMR module were used to help locating the measurement point in the impeller blade channel. The basic functions of the RMR module have been introduced earlier in Section 4.2.2. The shaft encoder gives continuous information about the shaft position by providing a continuous string of 1000 TTL level pulses per revolution, as well as a separate once-per-rev (OPR) pulse. The OPR signal can be used as a reference point of the angular location of the rotating impeller. Before the measurement, the correlation between the OPR signal and the physical angular location of the impeller was achieved. This was done by monitoring the OPR signal through an oscilloscope and recording the physical angular location of the impeller when the OPR signal was triggered. Based on the physical angular position reference point, the measurement windows were created through the RMR module. "Window" is angular position district and is defined in the measurement. Only the velocity data that is within the district can be sent to the processor and recorded. Using the window defined, the laser probe can measure the fluid velocity under a freezing state of the impeller. In other words, measurements were carried out only when the impeller

moved to a specified angular position, while at the other angular locations, the LDA processor was screened from the velocity data signals.

Figure 4.7 shows a schematic view of the measurement windows for the rotating impeller passages. The figure takes the 16FB pump impeller as an example. In the figure, the black triangle at the outer brim of the impeller indicates the angular reference of the impeller and the boundaries of the RMR controlled measurement window were indicated by the lines in the figure. The measurement windows were set so that, when the angular reference of the impeller moved to the measurement window, the trailing edges of two of the thicker blades were at the tips of the splitter

Figure 4.7 A schematic view of measurement windows for the rotating impeller passages

plate and cutwater. The parameters that defined the measurement windows were set through the LDA system control software. As the profile of the impeller was axially symmetrical, four measurement windows were created to increase the data rate of the measurement. The width of each measurement window is 1° . As the impeller was rotating at a speed of 2000rpm, the time period of the LDA reading velocity data is 8.3×10^{-5} s for one measurement window and 3.3×10^{-4} s per revolution of the impeller.

In this study, flows in the blade channel of three impellers with 16 forward-bending blades, 8 backward-bending blades and 16 straight blades were measured using laser Doppler anemometer. The measurements on the three pump models were carried out at an impeller rotating speed of 2000rpm and flow rate of 5 liters per minute. The experimental procedure can be summarized as following:

- i. The cooling fan of the LDA system is turned on.
- ii. The power source for the LDA, traversing mechanism and the pump test rig are switched on.
- iii. All isolating valves of the tank are opened (they are to be closed completely at all the time when not running the experiment).
- iv. The Argon-ion laser unit is activated. The laser safety screen is opened and the lens protecting cover of the probe is taken off. Then the focusing and the crossing of the laser beams to form the measuring volume are checked. The

intensity of the laser is kept at low level until it is ready for taking measurement for safety reasons.

- v. The personal computer for the control software of the LDA system and data acquisition is switched on. The controller of the traversing mechanism is switched on. The LDA system control software, PACE is activated. The RMR module is activated and shaft encoder is linked up.
- vi. The laser probe is aligned precisely to a point that has been marked on the pump. This is achieved with the assistance of a microscope. The X, Y and Z coordinates of the traversing mechanism are set to zero. Therefore, this point will coincide with the origin of grid map to be traversed by the laser that has been created and transferred to the PACE earlier.
- vii. The laser probe is then moved forward in a fixed distance that has been calculated earlier, to place the measuring volume at a position which is half of the height of the impeller blades for velocity measurements, i.e., at the center plane of the impeller blade channel.
- viii. The pump is turned on and the speed is increased slowly to 2000rpm. Throttling the valves and adjust the flow rate of the pump model to 5 l/min. Then the pump is run for about 3 minutes to achieve steady state.
- ix. The parameters setting for the LDA system in PACE are adjusted based on the Doppler burst signal that was observed from the oscilloscope to obtain the

desired data rate. These adjustments have to cooperate with the adjustment of the laser intensity.

- x. The Window Selection menu is opened to adjust the window width, window location as well as window number during the measurement. Then check from monitor window of the LDA control software to ensure that the angular positions that latched on the velocity data were all in the range of the measurement window defined.
- xi. The traversing mechanism and measurement stop automatically when the probe is moved to the last point on the grid map. The pump speed is then decreased slowly until it stops totally. Then the pump motor is switched off and all the isolating valves are closed.
- xii. The intensity of the Argon-ion laser is decreased slowly and the laser unit is switched off. The cooling fan is shut off after 10-15 minutes when the system is cooled down.
- xiii. The measurement results are stored in RAW data files. Data from a single measurement point is stored in one file. The RAW data files are then processed and stored.

4.5 Experimental Uncertainties

The uncertainty sources of the LDA measurement stem mainly from velocity biasing, velocity gradient broadening, the accuracy of the signal processor and finite sampling size. The velocity biasing occurs because of the seeding particles over dense in the fluid. More than one particle passing through the measurement volume at the same time would cause more than one scattering center of laser and thus the velocity is faster than the mean. It is not possible to guarantee this condition is not occurring in the measurement but it can be manipulated to ensure that most of the time there is one and only one particle in the measuring volume. Another factor causing the velocity biasing is the rotation of the seeding particle in the fluid. The particle with larger size is more vulnerable to the unbalanced shear forces in the fluid and thus causing it to rotate. The rotating surface of the particle would subsequently cause an additive value to the fluid velocity. However, this biasing would not be severe until the size of the seeding particle is extremely large. MacLauhlin and Tiederman [1972] showed that the velocity biasing effect would cause an error to mean velocity measurement less than 2% based on a 95% confidence level.

Velocity gradient broadening arises when the finite sized control probe volume lay across a mean velocity gradient. Particles passing through the measurement volume would have a range of velocity. The resulting probability density function would be skewed and broadened. Durst et al. [1990] suggested correction for the mean velocity and mean velocity fluctuation which are the function of mean velocity gradient and the diameter of the measurement volume.

The accuracy of the TSI IFA 755 digital burst correlator stated by the manufacturer (TSI IFA 755 manual) has a maximum value of 1% measured in terms of signal to noise ratio (SNR).

As indicated by the sedimentation test mentioned earlier in Section 4.4, the error caused by seeding particle lagging is estimated to be lower than 0.1%.

In the measurement, the sample size of the flow in between the impeller blades was 200. The sample size is restricted by the Doppler signal data rate during the experiment. The data rate was not only affected by the conditions of the experimental test rig but also dramatically reduced by the width of the data windows set in the RMR module. Four windows covering one degree each have led to only 4/360 of the data being accepted during the measurement. The data rate in current measurement was about 5 to 40.

Micro-sized particles are required in the flow field when a LDA system is employed to measure velocity. Not only should the particles be able to follow the flow but they are also expected to produce continuous Doppler signals. This requires high particle concentration in the regions of interest all the time. However, the particle concentration could be low somewhere in the measured region. By assuming the behavior of turbulence to be Gaussian, the uncertainty of the root mean square is found to be dependent on the number of data points instead of the flow conditions [Yanta and Smith, 1985]. From the statistical analysis, the error function was (based on 95% confidence limit),

$$\text{Error} = \sqrt{\frac{2}{N_s}} \quad (4.1)$$

Where N_s is the number of data samples per measurement point.

By using the method of propagation of errors [Dieck, 1992], the experimental uncertainties of various parameters could be evaluated. In the determination of a quantity that is expressed by a functional relationship of one or more measured variables, the effect of the error in the measurement variables on the computed function is often needed. One common approach in evaluating the effects of the errors is Taylor's series error propagation. In this study, the magnitude of the resultant flow velocity is computed from the measured velocity components in X and Y directions. Therefore, the magnitude of the flow velocity error will be the combined error of the individual measurements.

The effect of the combination of errors from the sources can be computed with an expression,

$$E_R = \left[\left(\frac{\partial V}{\partial u} \right)^2 (E_u)^2 + \left(\frac{\partial V}{\partial v} \right)^2 (E_v)^2 \right]^{1/2} \quad (4.2)$$

where E_R is the error in the resultant flow velocity

E_u is the error in the measured velocity component in X direction

E_v is the error in the measured velocity component in Y direction

$\partial V / \partial u$ is the effect coefficient which expresses the effect of an error in u will have on the resultant velocity V

$\partial V / \partial v$ is the effect coefficient which expresses the effect of an error in v will have on the resultant velocity V

From Equation 4.2, E_R can be expressed as

$$E_R = \left[\left(\frac{\bar{u}}{\sqrt{\bar{u}^2 + \bar{v}^2}} \right)^2 (E_u)^2 + \left(\frac{\bar{v}}{\sqrt{\bar{u}^2 + \bar{v}^2}} \right)^2 (E_v)^2 \right]^{1/2} \quad (4.3)$$

where $\bar{u}$ and $\bar{v}$ are the mean of local X and Y velocity components respectively. The values of the error in the measured velocities were obtained from at least 21 independent sets of readings taken after separate calibration. As such, they provide an indication of the repeatability of measured data (Dieck, 1992). The estimated uncertainties of the parameters in the measurement are given in Table 4.1.

Parameter	Random Error	Random Error in Percentage
Position (X, Y)	$\pm 0.2 \text{ mm}$	--
Position (Z)	$\pm 0.5 \text{ mm}$	--
Impeller angular position, θ	$\pm 0.2 \text{ degree}$	--
Velocity biasing	--	2%
Velocity gradient broadening	--	1%
Seeding particle lagging	--	0.1%
Accuracy of signal processing	--	0.2%
Root mean square of the finite sample	--	10%
E_u	--	2.24%
E_v	--	2.24%
E_R	--	2.66%
Velocity angle, θ_R , (8BB)	--	3.90%
Velocity angle, θ_R , (16FB)	--	2.96%
Velocity angle, θ_R , (16SB)	--	3.43%

Table 4.1 The estimated uncertainty of the parameters in the measurement

CHAPTER 5

SIMULATIONS OF FLOW IN BETWEEN THE IMPELLER BLADES FOR THE THREE PUMP MODELS

In this chapter, the flow fields in the impeller blade channels of the 1:1 numerical blood pump model for the 16FB, 16SB and 8BB blade designs will be presented and discussed in detail. Due to the continuous movement of the impeller in the numerical model, the results shown in this chapter are the instantaneous flow fields at a given time step, which is about 4 to 5.6 seconds after the starting time to ensure that the flows are fully developed. The flow velocity distributions will be presented in several sectional planes due to the complexity of the pump model geometry. The 3-D geometry of the simulation model has been shown earlier in Chapter 3. Note that the flow presented is under pump operating conditions with the impeller rotating speed at 2000 rpm and flow rate at 5 l/min.

5.1 Flow in the 16FB Pump Model

The static pressure distribution at the center plane of the pump is shown in Figure 5.1. Note that the static pressure presented (and also for the 16SB and 8BB pump models in Sections 5.2 and 5.3 respectively) is relative to the pump inlet port, where the gauge pressure is set to zero. Under the operating conditions, the pump with the 16FB impeller produces a pressure head of 14550 Pa (109.4 mmHg) between the pump inlet and outlet. It can be observed in Figure 5.1 that the static pressure in the pump increases gradually with the increase of radial location along the blade channel and is further increased when the flow enters the volute. The static pressure in the volute passages rises in the tangential direction and the highest static pressure regions are observed around the tips of the splitter

Chapter 5 Simulations of Flow in between the Impeller Blades for the Three Pump Models

plate and the cutwater, which are indicated by A and B respectively in Figure 5.1. These high pressure regions are formed because of the flow blockage at the tips of the splitter plate and cutwater. This brings the flow to stop there and thus causes increase in pressure due to the stagnation effect. The building up of the high pressures is required not only for pushing the fluid against the frictional resistance through the double volute passages to the pump outlet but also for providing the required pressure to supply blood to the whole body of the patient. Because the outer volute passage is longer than the inner one, the static pressure built at A is higher than that at B to overcome the higher frictional resistance. It can be observed in Figure 5.1 that the static pressure distribution in the volute is approximately symmetrical, i.e. the static pressure distributions in the first quadrant (from $\theta=0^\circ$ to 90° , and so on for the second, third and fourth quadrants as usual convention for the Cartesian coordinate) are similar to those in the third quadrant, and the pressure distributions in the second and fourth quadrants are similar to each other.

Figure 5.1 Distributions of static pressure in the volute and the impeller

These symmetrical static pressure distributions in the volute are useful in creating a better balance of radial thrust force exerted on the impeller and subsequently ease the complication on the control and manipulation of the magnetically suspended high speed rotating impeller.

Figure 5.2 shows the flow at the pump inlet part. It can be observed that the flow from the pipe inlet to the impeller is quite smooth. But, the flow is not uniform before entering the impeller passage and the velocity magnitude decreases from the upper shroud to the lower shroud of the impeller. This is mainly due to inertia built up by the fluid when it flows through the bending conduit formed by the stationary cone and the curved pump casing wall. It should be noted that the geometry of the pump inlet, especially the shape of the stationary cone, should have been optimized. Improper inlet geometry, for example, the over rapid increase of the cross sectional area along the flow passage would aggravate the imbalance of the flow speed and eventually cause vortical flow before entering the impeller. For a centrifugal pump, the vortical flow at the inlet will extend into the impeller blade channels and impair the flow in the impeller.

Figure 5.2 Velocity distribution at the pump inlet of the 16FBs pump model

The flow at the center plane of the 16FB pump model is shown in Figure 5.3. Note that the vectors in Figure 5.3 indicate the absolute velocity of the flow. It can be seen that the flow at the pump inlet (the eye) is mainly in radial direction with much lower velocity magnitude than the other zones. Due to the high rotating speed of the impeller, the fluid velocities between the impeller blades are overwhelmed by the tangential velocity distribution. In the volute passages, the flow basically follows the passage direction and has a lower tangential velocity magnitude than those in the blade channels. As shown in Figure 5.3, the fluid velocities in the inner volute passage are higher than those in the outer volute passage since the outer volute channel is about 75% longer than the inner volute passage and thus will create greater resistance to the flow due to the friction caused by the four-solid-wall bounded conduit in the second half of the passage.

Figure 5.3 Absolute velocity distribution at center plane of the 16FB pump model

Figure 5.4 shows the instantaneous velocity distributions in 16FB impeller blade channels with a small diagram at the center indicating the impeller position relative to the cutwater lip (where the reference angular location $\theta=0^\circ$ is defined). Note that the velocities are relative to the rotating speed of the impeller with the direction and magnitude of the velocity indicated by the arrow and length of the vector respectively. Further note that the velocity vectors are in fact 3-D but they can only be presented in a two-dimensional plane. This is why some of the blades have double images. In order to prevent the overcrowding of the velocity vectors in the figure, only one out of every seven vectors is randomly elected and presented in the figure.

As shown in Figure 5.4, the flow pattern in the blade channel varies with its location relative to the volute during impeller rotation. As mentioned earlier, the velocity distributions presented in Figure 5.4 are instantaneous while all the blade channels are rotating about the pump axis. At different time steps, the blade channel would take a different angular position around the axis. Therefore, the flow pattern in different blade channels can be regarded as the flow evolution that occurs in a single blade channel, but at different angular position with the progress of time.

It can be observed in Figure 5.4 that the flow in different quadrants of the impeller is approximately symmetrical. In other words, the flow patterns in blade channels 1, 2, 3 and 4 are similar to those in blade channels 9, 10, 11 and 12, respectively, while flow patterns in blade channels 5, 6, 7 and 8 are similar to those in channels 13, 14, 15 and 16 respectively. This approximately symmetrical pattern of the flow in impeller is expected because the cutwater and splitter plate of the pump have divided the volute into two separate passages which are located 180° apart from each other around the impeller. This

geometrical feature of the volute will affect the flow in the impeller and result in an approximately symmetrical flow in the impeller. This symmetrical flow was intentionally designed to facilitate the ease of controlling the magnetically suspended impeller for more balanced radial thrust [Akamatsu and Nakazeki, 1994].

Some common flow characteristics in the blade channels can be observed in Figure 5.4. In each of the blade channel, there is a main stream in the positive radial direction along the pressure side of the blade channel. Near the suction side of the blade channel, flow in general has a lower velocity magnitude in the positive radial direction and some flows are even in the negative radial direction as could be found in blade channels 1, 2, 6, 7, 8, 9, 10, 15 and 16. It can be found in Figure 5.4 that the reverse flow mainly occurs along the suction side of the blade channels that are located near the tips of the splitter plate and cutwater. This is because, as shown earlier in Figure 5.1, the outlets of blade channels 6, 7, 8 and 15, 16 are facing high static pressure in the volute. These high static pressure regions will cause high resistance of the through-flow from the blade channels and thus induce reverse flow along the suction side. Meanwhile, the reverse flow along the suction side of blade channels 9, 10 and 1, 2 occurs because these blade channels sequentially follow blade channels 6, 7, 8 and 15, 16 respectively. Although the static pressure at the outlet of blade channels 9, 10, 1 and 2 is low, the strong reverse flow in blade channels 6, 7, 8, 15 and 16 would take some time to be dissipated and thus could be continued and be observed in the suction side of blade channels 9, 10, 1 and 2. Between the positive and negative flows in the blade channels, a vortex can be induced, in which a portion of the fluid will be trapped by the circulating flow and has a prolonged resident time in the blade channel.

As shown in Figure 5.4, there is another vortex induced at the exit of blade channels 1 and 9, one of which is indicated by the lightning arrow in Figure 5.4. This vortex spans the exiting edge of the blade channel and rotates in clockwise direction. This vortex is induced because the blade channels 1 and 9 have just passed by the tips of the cutwater and splitter plate respectively. The difference between the static pressure before and after the tips of the splitter plate and cutwater would induce the vortical flow at the exit of the blade channel. Furthermore, as the static pressure built up at the splitter plate tip is higher than that at the cutwater tip as shown earlier in Figure 5.1, the vortex induced at the exit of blade channel 9 would be stronger and thus would last longer than that of blade channel 1. This can explain, as observed in Figure 5.4, the vortex at the exit of blade channel 1 dissipating and disappearing in blade channel 2, while the vortex at the exit of blade channel 9 lasted from blade channels 10 to 11.

Figure 5.4 3-D view of the relative velocity distribution in the 16FB impeller channels

To further explore the flow patterns at the outlet of impeller passages and the volute closely, some detailed sectional views of the flow at different angular positions are shown in Figure 5.5. The positions of the sectional plane, which are indicated by the straight lines labeled from “b” to “m” in Figure 5.5 (a), are cutting through the blade channels 1, 3, 6 and 8 which are on the upper half of the impeller. As shown in Figure 5.5 (a), there are three sectional planes for each of the four blade channels, which are located near the pressure side, middle and suction side of the blade channel. In Figures 5.5 (b) to (m), the horizontal axis represents the radial (R) direction of the impeller and vertical axis represents the axial (Z) direction which is along the impeller blade height from the lower impeller shroud (LIS) to upper impeller shroud (UIS). Note that the blood velocity at these locations is mainly in the tangential direction (see Figure 5.4), which is normal to the sectional planes.

Chapter 5 Simulations of Flow in between the Impeller Blades for the Three Pump Models

(b)

(c)

(d)

(e)

(f)

(g)

Figure 5.5 (a) Positions of the view planes, (b) to (m) Sectional views of the flow at the outlet of the blade channels and volute of the 16FB pump model

Figure 5.5 (b) shows the flow near the pressure side of blade channel 1. For ease of presentation, the vertical wall at the right hand side of the cutting plane is referred to as sidewall, while the top and bottom walls of the volute are referred to as ceiling and floor respectively. It can be observed that the flow along the blade channel in cutting plane “b” to the volute has no clear direction at the beginning. This is consistent with those observed in blade channel 1 near the pressure side in Figure 5.4. This is because the flow is in a transient stage due to the interaction between the induced clockwise vortex at the outer rim of the blade channel and the flow along the pressure side as demonstrated in Figure 5.4. The blade channel outlet as shown in Figure 5.5 (b) is dominated by the 60° slanting downward flow from the volute sidewall back to the channel. This reverse slanting flow should be due to the residue effect of the high pressure at the cutwater tip and the bouncing effect caused by the blockage of volute sidewall which is very close to the impeller exit at the beginning of the volute and results in reverse flow near the UIS and positive flow near LIS.

Figure 5.5 (c) shows the flow at the middle cutting plane of blade channel 1. It can be seen that the flow is uniform and in the positive radial direction along the blade channel in cutting plane “c” towards the volute. A vortex is induced at the channel outer brim of the cutting plane “c” and causes an anticlockwise flow from the UIS to LIS. This vortex can be identified as the same vortex located at the outer brim of blade channel 1 shown earlier in Figure 5.4. This indicates that the vortex at the outer brim distributed at both the X-Y and R-Z planes of blade channel 1 and is three dimensional in nature. Blood in the volute and blade channels could be exchanged through the rotation of the vortex. Affected by the induced vortex, a secondary flow in the positive axial direction (Z

direction) along the volute vertical sidewall has occurred. This is together with the mainstream which is normal to the cutting plane and flow along the volute passage.

The sectional view of the flow near the suction side of blade channel 1 is shown in Figure 5.5 (d). At the outlet of the blade channel, the flow is uniform and going positively from the blade channel to the volute. As shown in Figure 5.5 (d), there is a region in the blade channel at the radius of about 0.0225m, where the flow is very small in magnitude. It should be noted that the magnitude of the flow is not really small as the velocity at this region is mainly in tangential direction, which is normal to the view plane as shown in Figure 5.4. At the radii smaller than 0.022m, the flow is in the negative radial direction. This is because the cutting plane at this particular region is nearly parallel to the suction side of the blade channel, where the reverse flow occurs (see Figure 5.4). In the middle section of the volute, the secondary flow is mainly in the positive radial direction and no vortex was formed.

Figure 5.5 (e) shows the sectional view of the flow at the pressure side of channel 3. It can be seen that there is no reverse flow occurring at the outlet of the blade channel and the flow in the sectional plane of the channel is all in the positive radial direction. There are two small vortices. One is rotating in anticlockwise direction near the ceiling and the other one is rotating in clockwise direction near the floor. Comparing with the flow at the pressure side of blade channel 1 shown in Figure 5.5 (b), it can be found that the flow at pressure side in blade channel 3 has recovered from the negative flow caused by the high static pressure at the tip of the cutwater. When the cutting plane is being shifted to the middle of blade channel 3 as shown in Figure 5.5 (f), it can be observed that the flow is

uniform and in the positive radial direction along the blade channel until the pump volute, no reverse flow from the volute to the blade channel is found at the channel outer brim.

The sectional view of the flow near the suction side of the blade channel 3 is shown in Figure 5.5 (g). It can be seen that the flow at the channel outlet is relatively uniform and in positive radial direction. Inside the blade channel, it can be observed that the velocity magnitude is very low. This is because in blade channel 3 as shown in Figure 5.4, the positive flow initiated along the pressure side is strong and occupied namely 70% of the space in blade channel 3. There is only a low velocity region close to the suction side of the blade channel. From Figures 5.5 (e) to (g), it can also be observed in the volute that the two vortices have increased in size from plane “e” to “g” with the increase of the volute spacing. The vortices are located near the ceiling and floor of the volute and rotate in anticlockwise and clockwise direction respectively. As shown in the figures, the vortex near the ceiling is stronger than that near the floor. This should be due to the non-uniform flow at the impeller inlet that the flow at the UIS is much stronger than that at the LIS as shown earlier in Figure 5.2. However, both the vortices have no apparently effect on the outflow of blade channel 3 as the vortices are overall still weak and small in size.

The flow pattern near the pressure side of blade channel 6 is demonstrated in Figure 5.5 (h). It can be seen that the positive flow from the blade channel to the pump volute has significantly increased compared to that at the pressure side of blade channel 3. Due to the significant increase of the volute spacing at this cutting plane, the vortex near the ceiling has increased in size and occupied the upper half of the volute channel. Meanwhile, the vortex near the floor has been weakened tremendously as compared to the

ceiling vortex. Affected by the ceiling vortex, the outflow from the blade channel is inclined about 45° downward to the right of radial axis.

The vortex in the volute has a more significant effect on the outflow at the middle cutting plane of blade channel 6 as shown in Figure 5.5 (i). Negative flow from the volute to the blade channel occurs near the UIS, while positive radial flow occurs near the LIS of the cutting plane “i”. The counter clockwise circulating flow around the vortex center is correlated to the positive and negative flow at the outlet of the blade channel and would thus remarkably aggravate the flow disturbance there. This is why significant negative flow occurs at the outer brim of the blade channel 6 center as shown in Figure 5.4. Due to the negative flow, blood would exchange in between the pump volute and the blade channel during the impeller rotation.

At the suction side of blade channel 6 as shown in Figure 5.5 (j), the velocity magnitudes of both the positive and negative flows at the outlet of the blade channel are reduced. However, the strong circulating flow around the ceiling vortex in the volute is still persisting and the center of the vortex is moving slightly towards the volute center with the increased volute space as angular location of the cutting plane advances, while the weak circulating flow can still be observed near the volute floor. The flow from radius of 0.019m to 0.023m in the blade channel is in negative direction. This is due to the reverse flow along the suction side of blade channel 6 as shown earlier in Figure 5.4. It is not surprising of the weak positive flow at the beginning of the cutting plane, which is in fact starting from the pressure side as indicated in Figure 5.5 (a).

Chapter 5 Simulations of Flow in between the Impeller Blades for the Three Pump Models

When we shift the cutting plane to blade channel 8, it can be seen in Figure 5.5 (k) that the positive flow at the blade channel pressure side is smaller than that at the pressure side of channel 6 shown earlier in Figure 5.5 (h). At this angular location, the vortex induced at the volute ceiling has further increased in size and progressively affected the flow at the outer brim of the blade channel. As shown in Figure 5.5 (k), the circulating flow induced by the vortex in the volute spans the outlet of the blade channel. The vortex has induced strong flow in negative axial (Z) direction (minus 'Y' direction in the figure) at the outer brim of the blade channel.

At the middle plane of blade channel 8 as shown in Figure 5.5 (l), the flow pattern between the blade channel and volute is similar to that of the blade channel 6 shown earlier in Figure 5.5 (i). At this moment, the center of the vortex in the volute has shifted to the middle of the volute passage and closer to the outer brim of the blade channel and the vortex has induced a stronger circulating flow from the UIS to LIS than those in cutting plane "i". Note that the flow in the blade channel maintains its positive flow direction, although there is a low velocity region due to the interaction between the positive flow and the circulating flow induced by the vortex in the volute.

At the suction side of blade channel 8 as shown in Figure 5.5 (m), the center of the vortex has shifted further downwards to the left hand corner of the volute floor and much closer to the channel outlet, the vortex has caused the flow at the outer brim of the blade channel to go from UIS to LIS and then out of it. Meanwhile, a new vortex begins to formulate at the lower right hand corner of the volute. Because the angular position of the cutting plane 'm' is very close to the tip of the splitter plate (see Figure 5.5 (a)), with the further increase of the angular position, the vortex in the volute would encounter the tip of the

Chapter 5 Simulations of Flow in between the Impeller Blades for the Three Pump Models

splitter plate and disappear rapidly. Due to the symmetrical flow of the impeller passages, the flow patterns in the blade channels at the lower half of the impeller are similar to those in the upper half and are not presented here.

From Figures 5.5 (b) to (m), it can be seen that with the increase of angular position from blade channel 1 to 6, the positive flow from the blade channel to the volute is increasing in size and intensity. Although the volute static pressure increases tangentially from blade channel 1 to 6, it is nevertheless lower than those near the tips of the splitter plate and cutwater and not high enough to block the outflow of the blade channel. Therefore, the positive flow from the blade channel to the volute keeps on increasing from blade channel 1 to 6. Furthermore, the forward blade profile design has increased the momentum of the fluids in the blade channel since they rotate at the same forward direction of the flow in volute. Except the flow at the outlet of the pressure side of blade channel 1 is negative, which should be due to the location of cutting plane “b” has just passed by the tip of the cutwater, where one of the highest static pressure is located. With the further increase of volute static pressure along the outer volute passage, the positive flows from the blade channel 6 to 8 are being suppressed. However, the positive flow would still persist till the middle section of blade channel 8.

Meanwhile, due to the increase of the volute passage space with the increase of the angular location, the vortex induced in the volute passage has increased tremendously in strength especially the one located near the volute ceiling, which has progressively strengthened its influence on the flow at the outer brim of the blade channel. Furthermore, from the flow at the cutting planes, it can also be observed the flow in the blade channel

is complicated and three dimensional in nature, such as the vortex induced at the outer brim of blade channel 1.

5.2 Flow in the 16SB Pump Model

The contour map in Figure 5.6 shows the static pressure distribution at the center plane of the 16SB pump model. Under the flow rate of 5 l/min and impeller rotating speed of 2000 rpm, the 16SB pump model produces a pressure head of about 15100 Pa (113.5 mmHg). It can be observed in Figure 5.6 that there are similar pressure distribution characteristics of the 16SB model as those of the 16FB model shown earlier in Figure 5.1. The static pressure increases from the pump eye through the blade channel to the pump volute. In the two volute passages, the static pressure increases tangentially and is approximately symmetrical, i.e. the static pressure distribution in the first quadrant is similar to that in the third quadrant, and the pressure distribution in the second quadrant is similar to that in the

Figure 5.6 Static pressure distributions at the center plane of the 16SB pump model

fourth quadrant. As those happened to the 16FB model, the highest static pressure in the pump occurs at the regions around the tips of the splitter plate and cutwater, which are indicated by “A” and “B” respectively in Figure 5.6. These high static pressure regions are due to the stagnation effect caused by the tips of the cutwater and splitter plate in the flow channel. The lowest static pressure in the pump occurs at the leading edge of the impeller blade suction side, where the pressure could be 2000 Pa lower than that at the pump inlet.

Figure 5.7 shows the velocity distribution at the inlet of the 16SB pump model. It can be seen that there is no apparent difference of the flow in 16SB pump model from that of the 16FB model as shown earlier in Figure 5.2. The flow from the inlet pipe to the impeller is relatively smooth. As shown in Figure 5.7, due to the inertia of the fluid in the flow, the velocity magnitudes at the end of the pump inlet are not uniform and decrease from the upper impeller shroud to the lower impeller shroud. This non-uniformity of the flow at the end of the pump inlet could affect the flow in the blade channels.

Figure 5.7 Velocity distribution at the inlet of the 16SB pump model

Figure 5.8 shows the absolute velocity distributions at the center plane of the 16SB pump model. The flow at the pump inlet (the eye) has relatively low velocity magnitude and is basically in radial direction. The flow velocity increases drastically in tangential direction when it enters the impeller blade channels. The highest absolute fluid speed which is more than 5 m/s can be found at the outlet of the blade channel. In the volute, the tangential velocities gradually reduce along the two volute passages. Because the flow resistance at the outer volute passage is higher than that at the inner one, the flow velocity at the exit of the outer volute passage is lower than that of the inner volute passage. This has indicated that there is a difference in mass flow rate between the inner and outer volute passages and the mass flow rate through the inner volute passage is higher than that through the outer volute passage.

Figure 5.8 Absolute velocity distribution at center plane of the 16SB pump model

In order to facilitate the discussion of the flow in the impeller, a 3-D view of the flow field in the 16SB blade channels is given in Figure 5.9. The position of the blade channel relative to the tip of cutwater is numbered in the diagram at the center of the figure. Note that the flow fields shown in Figure 5.9 are instantaneous and the velocities indicated by the vectors are relative to the impeller rotating speed. In order to prevent the overcrowding of the vectors and to make the figure easy to read, only one of every seven vectors in the impeller is selected and presented.

As shown in Figure 5.9, the flow patterns in the blade channels of the straight blade impeller are approximately symmetrical, in which flow pattern in blade channel 1 is similar to that in blade channel 9 and the flow pattern in blade channel 2 is similar to that in blade channel 10 and so on. This symmetrical flow pattern in the blade channels can be attributed to the insertion of the splitter plate, which is located at 180° angular distance away from the cutwater. The splitter plate separated the volute passages into two symmetrical parts which subsequently induced the symmetrical flow patterns in the impeller blade channels.

Comparing with the flows in the forward bending blade channel shown earlier in Figure 5.4, it can be seen in Figure 5.9 that the flows in the straight blade channel have more variations, e.g. reverse flow is found in most of the straight blade channels. Because the straight blade has an inlet leading edge angle of 90° , blood collided with the pressure side of the blade at the entrance of the channel. Positive flow along the pressure side can be observed in all the straight blade channels, while near the suction side, reverse flow occurs in most of the blade channels except for channels 4, 5, 12 and 13. Between the positive flow and reverse flow, a vortex will be formulated and in blade channels 1 and 9,

the size of the vortex could be nearly as large as the width of the blade channel. It can be seen in Figure 5.9 that the vortex has extended to the inlet of the blade channel and has resulted in reverse flow at the channel inlet.

Figure 5.9 3-D view of the relative velocity distribution in the 16SB impeller channels

The induction of the vortices in the blade channel is correlated to the static pressure distribution in the pump volute. When the blade channel moves to the higher pressure regions of the volute, the reverse flow in the blade channel would increase significantly and the vortex in the blade channel would be induced subsequently. As shown in Figure 5.9, with the changing of the blade channel location relative to the volute, the vortex in the blade channels experiences the process, i.e., from the initiation, growing up, dissipating until vanishing. The vortex is formed initially in blade channel 14 and grows

to full size in blade channel 16, which is facing one of the highest static pressure at the cutwater tip. After that, the vortex persists in blade channels 1, 2 and 3. At this stage, the vortex is dissipating due to the low volute pressure facing these blade channels. When reaching blade channel 4, the vortex has vanished completely. Therefore, no vortex is found in the blade channel 4 and the following blade channel 5. When moving to blade channel 6, the vortex in the blade channel is initiated with the increase of the static pressure in the volute. Similar to the process discussed earlier, the vortex grows in blade channels 6, 7 and 8, then dissipates in blade channels 9, 10 and 11 and no vortex occurs in blade channels 12 and 13. It can be seen that, as the blade channel faces a changing static pressure in the volute during the rotation of the impeller, the flow patterns in the blade channel will vary accordingly with the vortex in the blade channel initiates, grows, dissipates and vanishes twice in a complete revolution of the impeller.

Another flow pattern in the straight blade channels is the negative flow from the volute to the blade channel. The negative flow interacts with the outgoing positive flow of the blade channel and causes circulating flow at the outlet rim of the blade channel. However, unlike those of the 16FB model, the negative flow from the volute would not induce a second vortex at the outlet of the straight blade channel. In order to have a more detailed view of the flow in the blade channels and the interaction between the outlet flow of the blade channel and the flow in the volute, some sectional planes were cut along the blade channel. Note that the chosen blade channels (1, 3, 6 and 8) and the three cutting planes (near the pressure side, at the middle and near the suction side) in each of the chosen blade channel of the 16SB model as demonstrated in Figure 5.10 (a) are the same as those of 16FB model.

Chapter 5 Simulations of Flow in between the Impeller Blades for the Three Pump Models

(a)

(b)

(c)

(d)

(e)

Chapter 5 Simulations of Flow in between the Impeller Blades for the Three Pump Models

(f)

(g)

(h)

(i)

(j)

(k)

Figure 5.10 (a) Positions of the view planes, **(b)** to **(m)** Sectional views of the flow at the outlet of the blade channels and volute of the 16SB pump model

Figure 5.10 (b) shows the sectional view of the flow near the pressure side of blade channel 1. It can be seen that the flow along the cutting plane “b” is strong and uniform initially. Since the volute width is small and narrow at this location, a vortex was thus induced with the positive radial flow going along the UIS side of the blade channel outlet, made a U-shape turn in the volute and back to the blade channel along the LIS and interacted with the outgoing flow. It can be observed in the figure that the vortex center is near to both the blade channel exit and the LIS.

The flow at the middle cutting plane of blade channel 1 is shown in Figure 5.10 (c). It can be seen that the initial flow is still positive and uniform but with much smaller magnitude than those in cutting plane “b”. The vortex near the pressure side in Figure 5.10 (b) still persists in the middle region of the blade channel. However, the intensity and size of the vortex is reduced as compared with that at the previous cutting plane. This indicates that

the vortex induced at the outlet of blade channel 1 spans from the pressure side to the middle of the blade channel and reduces its intensity and size tangentially from the pressure side to the middle of the blade channel. As what has happened at the pressure side, negative flow from volute to the blade channel occurs along the LIS side, and positive flow occurs near the UIS side at the middle of the blade channel.

At the suction side of the blade channel 1 as shown in Figure 5.10 (d), the vortex at the blade channel outlet has almost vanished. At the brim of the blade channel outlet, flow is positive from the blade channel to the volute, except for those fluids at the LIS side, the flow velocity is very low due to the effect of the circulating flow at the channel outlet. In the blade channel, there is a low velocity region at the radius of about 0.0225 m. This is because the flow is mainly in tangential direction and impinging on the suction side of the blade channel 1 at this location and the flow is then bifurcated into positive radial flow towards the volute and negative radial flow back towards the channel entrance (see Figure 5.9). Note that the flow as can be seen in Figure 5.10 (d) is uniform and negative towards the channel inlet from the radial location (0.225 m) in this cutting plane, this is consistent with the vortex induced in blade channel 1 as shown earlier in Figure 5.9 with the negative radial flow along the suction side. In the volute, the secondary flow is in positive radial direction and no vortex is observed.

Figures 5.10 (e) to (g) show the flow at the three cutting planes of blade channel 3. At the pressure side as shown in Figure 5.10 (e), the blood is flowing strongly and uniformly along the cutting plane “e” towards the volute. It can be found that the flow in the cutting plane is much higher in magnitude than those along the pressure side of blade channel 1. At the same time, the vortex seen earlier at the pressure side of blade channel 1 has

dissipated in blade channel 3, except a low velocity region occurs near the LIS as shown in Figure 5.10 (e). The secondary flow in the volute is mainly in positive direction and the secondary flow is weak at the regions near both the ceiling and floor of the volute.

At the middle plane of blade channel 3 as shown in Figure 5.10 (f), the outflow from the blade channel is weaker as compared with that at the pressure side and a weak negative radial flow is found at the LIS side of the cutting plane “f”. The weak negative radial flow can be due to the residue of a weak vortex with the center closed to the lower right corner of the blade channel outlet. This is consistent with the earlier observation that the vortex has dissipated and nearly disappeared in blade channel 3 as demonstrated earlier in Figure 5.9.

At the suction side of blade channel 3 as shown in Figure 5.10 (g), the flow at the outer brim of the blade channel is all positive but with lower velocity magnitude as compared with those at the pressure side and along the middle cutting plane of the blade channel. The low velocity region in Figure 5.10 (g) has extended from radial locations of 0.0215m to 0.0225m, varying across the cutting plane from LIS to UIS. This is consistent with those observed in blade channel 3 of Figure 5.9 where the flow is in tangential direction (normal to the cutting plane) and impinging on the suction side; the flow is then subsequently bifurcated into positive and negative radial flows, which can also be observed in the cutting plane “g”. In the volute, the secondary flow is still mainly in the positive radial direction. At the regions near the ceiling and floor of the volute, the secondary flow is weak. However, as shown in Figure 5.10 (g), a vortex has begun to formulate near the volute floor.

Figure 5.10 (h) shows the flow near the pressure side of blade channel 6. It can be seen that the flow along the pressure side is quite uniform and the velocity is much larger in magnitude than those of blade channels 1 and 3. At the outlet of the blade channel, blood flows from the blade channel to the pump volute and maintains at relatively high velocity. In the volute, with the increase in area at this angular location, two vortices have been induced and the vortex near the floor is larger in size than the one near the ceiling as shown in Figure 5.10 (h). However, the vortices in the volute do not affect the flow at the blade channel as the positive radial momentum of the flow along the pressure side is strong enough to overcome the circulating flow around the vortex in the volute.

At the middle plane of blade channel 6 as shown in Figure 5.10 (i), positive radial flow in the blade channel is much weaker than that at the pressure side. Meanwhile, the two vortices with one at the ceiling and the other one at the floor of the volute have further increased in size. The circulating flow around the volute floor vortex renders negative flow along the LIS side of the cutting plane “i”. It can be seen in Figure 5.10 (i) that negative radial flow at the outlet of the blade channel has a higher velocity magnitude than that occurring at the middle cutting plane of blade channel 3 as shown in Figure 5.10 (f). This should be attributed to the higher static pressure in the volute at this angular location.

Figure 5.10 (j) shows the flow at the suction side of blade channel 6. At the outer brim of the blade channel, the flow is basically positive, however, negative flow can be found at the LIS of the blade channel. Due to the higher volute static pressure, it can be seen that the low velocity region has expanded in size (ranging from 0.023m to 0.0235m) and the flow is bifurcated with the weaker positive flow going towards the volute and stronger negative flow going towards the channel inlet as shown in Figure 5.10 (j). In the volute,

two vortices maintain about the same size as in the cutting plane “i” and most of the flow is in positive radial direction at the center of the volute.

Figures 5.10 (k), (l) and (m) demonstrate the flow at the cutting plane at the pressure side, middle and suction side of blade channel 8 respectively. Note that the static pressure at the downstream of blade channel 8 is the highest among the blade channels. At the cutting plane near the pressure side of blade channel 8 shown in Figure 5.10 (k), the positive flow from the blade channel to the volute is lower in magnitude than those at the pressure side of blade channel 6 as shown earlier in Figure 5.10 (h). The center of the vortex near the floor of the volute has shifted towards the outer brim of the blade channel and caused negative flow along the LIS of the blade channel. As shown in Figure 5.10 (k), the vortex near the ceiling of the volute has dissipated but the flow in negative radial direction still exists there.

The flow patterns at the middle plane and suction sides of blade channel 8 are quite similar to each other. As shown in Figures 5.10 (l) and (m), at the outer brim of the blade channel, the flows are all in the negative direction from the volute to the blade channel. The negative flow was caused by the high static pressure in the volute at the location. Furthermore, as shown in Figures 5.10 (l) and (m), the flow within the blade channels are all in negative radial direction and the magnitudes of the negative flow are much higher in cutting plane “m” than those in cutting plane “l”. This is due to the cutting plane “m” being closer to the tip of the splitter plate than the cutting plane “l”. The vortices in the volute have dissipated at this location. At the center of the volute, the secondary flows are low in magnitude, while near both the volute ceiling and floor, the secondary flows are in negative radial direction.

There are similar flow characteristics from the blade channel to the volute of the 16SB pump model as compared to those of the 16FB pump model. Furthermore, since the cutting planes in the blade channels of the 16SB model are nearly parallel to the impeller blade, the flows in these cutting planes have provided a complete view from the inlet to the outlet as compared to those of the 16FB model. The positive flow from the blade channel to the volute is generally increasing from blade channel 1 to 6, although the static pressure in the volute increases tangentially. With the further increase of the static pressure, the positive flow from the blade channel to the volute is reduced from blade channel 6 to 8. With the increase of the volute passage spacing, vortices in the volute increase remarkably in size and strength which could tremendously affect the flow at the blade channel outlet.

There are, however, some differences between the flow of the 16FB and 16SB pump models. The vortices at the volute ceiling and floor are more balanced and the circulating flow in the volute is weaker in strength in the 16SB model than those in the 16FB model. In addition, the negative flow from the volute would penetrate deeper into the blade channel of the 16SB pump model than that of the 16FB model and this would in some circumstances induce a vortex near the LIS at the outlet of the blade channel. However, this vortex is not expected to produce extensive shear stress to the flow since the fluid velocity around the vortex is low in magnitude and the influence is restricted to the LIS region. On the contrary, the flows near the exit of the 16FB blade channels are more seriously affected by the vortex in the volute.

5.3 Flow in the 8BB Pump Model

Figure 5.11 shows the static pressure distributions at the center plane of the 8BB pump model. The lowest static pressure in the pump is located near the leading edge of the backward-bending impeller blade. In the impeller, the pressure gradually increases along the blade channel and the pressure is further elevated in the pump volute. As shown in Figure 5.11, the pressure is not uniform along the volute but increases tangentially in the two separated volute passages and attains the highest value at the tips of the splitter plate and cutwater. As shown in Figure 5.11, the static pressure distribution in the pump is approximately symmetrical, i.e., the pressure distribution at the first quadrant in the pump is similar to that at the third quadrant, and the pressure at the second quadrant is similar to that at the fourth quadrant. Same as the 16FB and 16SB models, the symmetrical pressure distribution is mainly due to the insertion of the splitter plate, which separates the volute

Figure 5.11 Static pressure distributions at the center plane of the 8BB pump model

into two symmetrical volute passages and is designed for the ease of controlling the rotating magnetically suspended impeller. Under the impeller rotating speed of 2000 rpm and pump flow rate of 5 l/min, the pump with the 8BB impeller produces a pressure head of 13100 Pa (98.5 mmHg), which is the lowest pressure generated among the three pump models.

Figure 5.12 shows the center sectional view of the flow at the inlet region of the 8BB pump model. It can be seen that the flow at the inlet of the 8BB model is similar to those of the 16FB and 16SB models. The non-uniformity of flow, which is due to the inertia of the fluid when it flows through the bending region, is observed at the end of the inlet conduit with the velocity magnitude decreasing from the upper impeller shroud to the lower impeller shroud. While at other regions of the pump inlet, the flow is quite smooth.

Figure 5.12 Velocity distribution at the pump inlet of the 8BB pump model

The absolute velocity distribution at the center plane of the 8BB pump model is shown in Figure 5.13. Because of the high rotating speed of the impeller, the absolute flow velocities are overwhelmed by the tangential velocity distribution. At the inlet zone of the pump, the tangential velocity is relatively low and mainly in radial direction. Then the velocity magnitude increases rapidly along the blade channel. In the volute, the flow velocity is generally lower than that at the blade channel outlet. Furthermore, the flow in the outer volute passage is lower in magnitude than that in the inner volute passage due to the higher resistance to the flow in the outer volute passage. These features are common in all the three pump models investigated.

Figure 5.13 Absolute velocity distribution at center plane of the 8BB pump model

Figure 5.14 shows the 3-D view of the flow field in the blade channels of the 8BB model with the position of the blade channels relative to the cutwater being indicated by the

diagram at the center of the figure. Note that the flow pattern shown in Figure 5.14 is instantaneous and the velocities, which are represented by vectors in the figure, are relative to the rotating speed of the impeller.

The flow in the backward-bending blade channels is much smoother than those in the forward-bending and straight blade channels. Furthermore, the static pressure which is increasing in the tangential direction along the volute does not affect significantly the flow patterns in the blade channels of the 8BB pump model. As shown in Figure 5.14, the flow in the blade channel basically follows the blade shape. There is no reverse flow occurring at either the pressure side or suction side and no apparent circulating flow is observed in the blade channels. It can be seen in Figure 5.14 that there are some differences of the flows in the blade channels 2, 4, 6 and 8 from those in blade channels 1, 3, 5 and 7. There are weak negative flows from the volute to the outlet of blade channels 2, 4, 6 and 8. In addition, a low velocity region is formed at the trailing edge of the suction side of each of these even number blade channels and reverse flow can even be seen at the trailing edge on the suction side of channel 4. This is because, comparing to the geometry of the odd number blade channel, the even number blade channel has narrow cross section between the inlet of the pressure side and at about 1/3 the location of the suction side, followed by a relative wide opening sectional area towards the channel outlet (see Figure 5.14). The geometry has created a blockage effect and resulted in less fluid flows into the even number blade channel at the inlet and subsequently lower positive flow momentum at the outlet of the blade channel. Since the low positive flow momentum is not sufficient to sustain the positive flow at the outer brim of the even number blade channels, negative flow is resulted at the trailing edge of the suction side of these blade channels.

Figure 5.14 3-D view of the relative velocity distribution in the 8BB impeller channels

In order to have a more detailed view of the flow along the blade channel and volute, similar cutting planes were made as before for those 16FB and 16SB pump models. The locations of the cutting planes are represented by the straight lines being labeled from “b” to “m” as shown in Figure 5.15 (a). The flows in the corresponding cutting planes are demonstrated in Figures 5.15 (b) to (m) respectively. As shown in Figure 5.15 (a), blade channels 1 to 4, which constitutes the upper half circle of the impeller, are selected to demonstrate the flow pattern and three cutting planes, which are located near the pressure side, at the middle and near the suction side of the blade channel outlet, are made for each selected blade channel. Due to the symmetrical flow, the flow patterns in the blade channels in the lower half of the impeller are similar to those in the upper half circle and are not shown here.

(a)

(b)

(c)

(d)

(e)

Chapter 5 Simulations of Flow in between the Impeller Blades for the Three Pump Models

Figure 5.15 (a) Positions of the view planes, **(b) to (m)** Sectional views of the flow at the outlet of the blade channels and volute of the 8BB pump model

Note that all the cutting planes are near the pressure and suction sides of the blade channel at the outer brim of the impeller only. Due to the backward-bending of the blades, the cutting planes are not following closely to the shapes of the pressure or suction sides in the blade channel as shown in Figure 5.15 (a). In Figures 5.15 (b) to (m), the horizontal axis represents the radial (R) direction and the vertical axis represents the axial (Z) direction of the pump and the velocity vectors shown in the figures are the local velocities on the cutting planes.

Figure 5.15 (b) shows the flow at the cutting plane near the pressure side of blade channel 1. Since the cutting plane “b” is nearly perpendicular to the tangent of pressure side and the flow as shown in Figure 5.14 is also nearly perpendicular to the cutting plane, it can be seen that the flows initially are very small. The flow at the outlet of the blade channel is in the positive radial direction with the velocity vectors located at the regions near the UIS and LIS are higher than those at the center. In the volute, the secondary flow is

smooth and the magnitude of the flow is slowly reduced due to the main flow direction as shown in Figure 5.14 is almost perpendicular to the cutting plane. Near the floor of the volute, it can be observed that a vortex is induced.

At the middle cutting plane of blade channel 1 as shown in Figure 5.15 (c), the flow is initially small in magnitude, however, the velocity increases rapidly and becomes uniform along the cutting plane. Note that the cutting plane ‘c’ is inclined with the tangent to pressure side blade at an angle smaller than that in Figure 5.15 (b) and the flow is thus inclined at a small angle to the cutting plane. At the outlet of the blade channel, the flow maintains in the positive radial direction and the velocity magnitude is higher than that of the cutting plane “b”. In the volute, the secondary flow is mainly in the positive radial direction but inclining at about positive 45° to the radial direction except for those in the regions near the floor and ceiling, where small circulating flows are observed.

The flow at the cutting plane near the suction side of blade channel 1 is shown in Figure 5.15 (d). It can be observed that the flow pattern along the blade channel is similar to that in the cutting plane “c”. At the outer brim of the blade channel, positive flow is maintained near the UIS, while negative flow occurs near the LIS due to the vortex induced near the volute floor. It can be observed that the circulating flow around the vortex has swept clockwise from the LIS to UIS along the outer brim of blade channel, causing the initial uniform positive radial flow skew to the UIS. At the other regions of the volute, the flow is basically in the positive radial direction. A smaller and much weaker vortex can be observed at the ceiling of the volute.

Figure 5.15 (e) shows the cutting plane view of the flow at the pressure side of blade channel 2. It can be seen that the flow in the blade channel is uniform and positive towards the volute and the velocity magnitude is much higher than that at the pressure side of channel 1 as shown earlier in Figure 5.15 (b). In the volute, the secondary flow is in the positive radial direction for most of the region, while near the floor, a vortex is induced. However, the vortex was small in size and confined in the region. Near the volute ceiling, disturbance in the secondary flow can also be observed but there is no vortex as shown in Figure 5.15 (e).

At the middle cutting plane of blade channel 2 as shown in Figure 5.15 (f), the velocity magnitude is smaller than those in cutting plane ‘e’. This is due to cutting plane “f” inclining at a larger angle to the tangent of the blade pressure side than the previous cutting plane. The flow is gradually reduced along the blade channel and the flow at the outer brim of the blade channel is even in the negative radial direction. This is consistent with the flow being nearly perpendicular to the cutting plane as shown in Figure 5.14. This is also due to the circulating flow induced by the vortex located at the middle of the volute close to the blade channel exit. The positive flow can only be found at a small region near the UIS.

At the suction side of blade channel 2 as shown in Figure 5.15 (g), the flow patterns within the blade channel are similar to those in Figure 5.15 (f). The velocity magnitude is reduced near the exit of the blade channel and the flow at the outer brim of the blade channel is in the negative radial direction with small magnitude. This can be observed earlier in Figure 5.14 that there is a back flow from the trailing edge of the suction side. The flow at most of the region in the volute is in the positive radial direction. A vortex is

formed near the volute floor. Comparing with the vortex located at the similar region at the suction side of blade channel 1 shown earlier in Figure 5.15 (d), it can be found that the vortex has increased in both size and strength with the increase in volute cross sectional area.

Figures 5.15 (h), (i) and (j) demonstrate the sectional views of the flow near the pressure side, middle plane and suction side of blade channel 3 respectively. As shown in these figures, the flow patterns at the outlet of the blade channel and in the volute are similar in the three cutting planes. Affected by the vortex induced near the volute floor, the circulating flow is in the positive axial direction along the outer brim of the blade channel, it then turns into the positive radial direction at the region close to the UIS as shown in Figure 5.15 (h). From Figures 5.15 (h) to (j), it can be seen that the vortex in the volute has increased in size with the increase in the volute cross sectional area and the vortex center near the volute floor has slightly shifted to the center.

Figure 5.15 (k) shows the sectional view of the flow near the pressure side of blade channel 4. Due to the increase of static pressure in the volute, stronger negative flow is observed at the outlet of the blade channel. It can be seen that negative flow occurs at the LIS and then makes a 180° turn to become positive flow near the UIS. In the volute, the center of vortex is located at the middle and close to the blade channel outlet. The flow pattern in the blade channel outlet is correlated to the clockwise circulating flow around the vortex and thus causing the drastic variation of the flow at the channel outlet.

At the middle plane of blade channel 4 as shown in Figure 5.15 (l), the size and strength of the vortex in the volute has further increased. The circulating flow around the vortex has almost occupied the whole space of the volute passage. Meanwhile, negative flow

occupies nearly the whole outer brim of the blade channel and positive flow can only be observed at a small region near the UIS.

At the sectional plane “m” of blade channel 4 as shown in Figure 5.15 (m), the vortex in the volute is moved toward the ceiling and begins to dissipate. The secondary flow at the lower left part of the volute is in the negative radial direction with a higher velocity magnitude, while at the upper right hand region of the volute, the secondary flow is in the positive radial direction but small in magnitude. The flow at the outer brim of the blade channel is negative but has much lower velocity magnitudes than those at the middle and pressure side of blade channel 4. It is interesting to note that the flow in the blade channel of cutting planes ‘k’, ‘l’ and ‘m’ are all in the positive radial direction, especially for the cutting plane close to the suction side. This is expected, since the cutting plane is not following the profile of the blade and is in fact quite a distance away from the suction side for cutting plane ‘m’, the flow is in the reverse radial direction along the suction side as can be observed earlier in Figure 5.14.

From Figures 5.15 (b) to (m), it can be observed that the 8BB pump model shows a different flow pattern from those of the 16FB and 16SB models. This is due to the length of the backward bending impeller blade channel being much longer than those of the other two models and the cutting planes are very much different too. This has not only provided another view of the flow but has also enhanced the understanding of the flow field of the 8BB pump model. More importantly, this has reflected the flow characteristics of the backward bending blade design, that the flow direction from the blade channel is inclined backward to the rotating direction of the impeller. The flow in the blade channel of the cutting plane of the 8BB model is basically smooth and positive

towards the volute, even at blade channel 4, which is the nearest to the splitter plate tip. Furthermore, it can be seen in all the cutting planes that the flow are not being bounced back when they are approaching the volute sidewall due to the backward flow direction from the blade channels.

These are very different from the other two pump models. However, it is observed that the flow at the outer brim of the blade channel in the cutting plane is the most disturbed among the three pump models. This has reflected the flow characteristics of the backward bending blade design, that the flow direction from the blade channel is inclined at a larger angle (sometimes is even opposite) to the tangential and forward flow in the volute passage, the interaction between them has created the most disturbed flow as can be observed from Figures 5.15 (b) to (m). With the increase in the volute spacing, vortex at the volute floor gradually occupies the whole volute space. The vortex center was always located in the volute but not in the blade channel, the induced circulating flow has greatly affected the flow at the outer brim of the blade channel. Although these advantages of the 8BB pump model may be good in industrial application because less energy is wasted in the vortical flow in the blade channel. However, it may not be good enough to be applied in a blood pump because there are still some other important requirements that are not well met by the 8BB pump model, such as sufficient washout on the internal surface of the blade channel and the ability to produce high pressure head.

5.4 Comparison of the Shear Stress of the Pump Models

For the blood pump, the high rotating speed of the impeller would produce remarkable high shear stresses on the blood and subsequently cause damage to the blood cells. Therefore, the assessment of the pump inner shear stress distribution is indispensable in

the design process of the blood pumps. The inner shear stress distributions shown in this section are instantaneous and under the pump operating conditions. Note that the calculation method of the inner shear stress has been outlined earlier in Chapter 3.

The contour maps in Figures 5.16 (a) and (b) demonstrate the distributions of the scalar shear stresses at the center plane of the 16FB pump model. Note that Figures 5.16 (a) (also Figures 5.18 (a) and 5.20 (a)) and (b) (also Figures 5.18 (b) and 5.20 (b)) are instantaneous scalar shear stress distributions with the location of the impeller in Figure 5.16 (a) (also Figures 5.18 (a) and 5.20 (a)) is 11.25° (1.125×10^{-3} second) ahead of that in Figure 5.16 (b) (also Figures 5.18 (b) and 5.20 (b)). It can be seen that the tips of the splitter plate and cutwater are at the trailing edges of the impeller blades in Figure 5.16 (a) (also Figures 5.18 (a) and 5.20 (a)) while the tips are at the middle of the impeller blades in Figure 5.16 (b) (also Figures 5.18 (b) and 5.20 (b)).

As shown in Figures 5.16 (a) and (b), for most of the regions in the pump, the shear stress is lower than 60 Pa. The high shear stresses occur at the inlet and outlet regions of the blade channel, the leading and trailing edges of the impeller blade and around the tips of the cutwater and splitter plate. It can be seen that in most of the high shear stress regions, the shear stresses are not higher than 330 Pa. The scalar shear stresses higher than 330 Pa is confined to the small regions around the tips of the cutwater and splitter plate and the leading edge of the impeller blade. These locations of high shear stress regions are anticipated since the blood at these regions experiences higher velocity gradient due to the rotation of impeller.

Figure 5.16 Scalar shear stress distributions in the 16FB pump model with different impeller locations, note that impeller position in (a) is 11.25° ahead of (b)

Figure 5.17 Mass weighted distribution of scaled shear stress within the 16FB pump model. The impeller position in (a) is 11.25° ahead of (b) (as in Figure 4.16)

Comparing the shear stress distributions in Figures 5.16 (a) and (b), it can be seen that with the rotation of the impeller, the high shear stress regions at the inlets and outlets of the blade channels are moving along with the impeller. Furthermore, the magnitude of the shear stresses around the tips of the splitter plate and cutwater vary with the different locations of the impeller blades.

In order to have a more thorough comparison of the shear stress distributions due to the different impeller positions, the mass weighted distributions of scaled shear stress on the whole flow field within the pump are given in Figures 5.17 (a) and (b). The time phases of the mass weighted shear stress distribution demonstrated in Figures 5.17 (a) and (b) are the same as those in Figures 5.16 (a) and (b) respectively. The mass weighted distribution was obtained by integrating over the entire flow field in the pump and sorting out the percentage of fluid mass according to the shear stress exerted on them and the higher mass percentage of the blood distributed on the higher shear stress reflects that the pump would cause a higher shear stress to the flow. As shown in Figures 5.17 (a) and (b), the top values of the scalar shear stress are different at the different blade channel orientations. The top shear stress value, which is about 1,900 Pa as shown in Figure 5.17 (a), occurs when the blades sweep over the tips of the splitter plate and cutwater. When the splitter plate and cutwater tips are located at the middle of the impeller blades, the maximum value of the scalar shear stress is reduced to 1,600 Pa, as shown in Figure 5.17 (b). In addition, the mass percentage of shear stress which is greater than 1,000 Pa in Figure 5.17 (a) is higher than that in Figure 5.17 (b). These indicate that, at the moment when the impeller blades are sweeping over the tips of the splitter plate and cutwater, the blood between the moving impeller blades and the tips of the splitter plate and cutwater would experience stronger shear stress and subsequently result in a greater damage to the blood cells.

Figures 5.18 (a) and (b) demonstrate the scalar shear stress distribution at the center plane of the 16SB pump model. It can be seen that for most of the areas in the 16SB pump model, the scalar shear stress is below 90 Pa. High shear stresses occurs at inlets and outlets of the blade channels and around the tips of the splitter plate and cutwater. Furthermore, due to the vortical flow in the blade channels of the 16SB model, the scalar shear stresses in the blade channels are generally higher than those of the 16FB pump and fall within the range from 60 to 120 Pa as shown in Figure 5.18 (a) and (b).

The mass weighted shear stress distributions shown in Figures 5.19 (a) and (b) indicate that the mass percentage on the shear stress is not apparently different at different impeller orientations. Note that the time intervals of mass weighted shear stress distributions demonstrated in Figures 5.19 (a) and (b) are the same to those in Figures 5.18 (a) and (b) respectively. The peak shear stress value when the splitter plate and cutwater tips are at the trailing edges of the impeller blades is about 950 Pa while the peak shear stress is about 850 Pa when the splitter plate and cutwater tips are located at the middle of impeller blades. For both orientations of the impeller blade, the percentages of the mass with shear stress over 700 Pa are much lower than those of the 16FB pump model.

Figures 5.20 (a) and (b) show the scalar shear stress distribution at the center plane of the 8BB pump model with two different orientations of the impeller blade relative to the volute. In Figure 5.20 (a), the tips of the splitter plate and cutwater are located at the trailing edges of the impeller blades while in Figure 5.20 (b), the tips of the splitter plate

Figure 5.18 Scalar shear stress distributions in the 16SB pump model with different impeller locations, note that impeller position in (a) is 11.25° ahead of (b)

Figure 5.19 Mass weighted distribution of scaled shear stress within the 16SB pump model. The impeller position in (a) is 11.25° ahead of (b) (as in Figure 4.18)

and cutwater are located between the impeller blades. As shown in the figures, the scalar shear stress is lower than 60 Pa in most of the regions. Higher shear stresses in the pump occur at the inlets and outlets of the blade channels and around the tips of the splitter plate and cutwater. In these areas, the shear stresses are generally between 150 and 300 Pa. The areas with shear stresses higher than 420 Pa can scarcely be observed in Figures 5.20 (a) and (b). Furthermore, the different geometries of the blade channels of the 8BB pump model have some effects on the shear stress distributions. As shown in the figures, the shear stresses are generally lower at the inlets of blade channels 2, 4, 6 and 8 than those of the blade channels 1, 3, 5 and 7.

The mass weighted distributions of the scalar shear stresses in the 8BB pump model with different impeller orientations are shown in Figure 5.21 (a) and (b). Note that the time intervals of the statistical results shown in Figure 5.21 (a) and (b) are the same as those in Figure 5.20 (a) and (b) respectively. It can be seen that, when the splitter plate and cutwater tips are at the trailing edge of the impeller blades, the peak scalar shear stress is about 850 Pa, while when the tips are at the middle of the impeller blades, the peak shear stress is about 930 Pa. Furthermore, as shown in Figure 5.21 (a), when the splitter plate and cutwater tips are at the trailing edges of the impeller blades, the mass weighted shear stress is basically lower than 700 Pa with only a very small portion of the blood experiencing shear stresses ranging from 700 Pa to 850 Pa. This indicates that, for the 8BB pump model, more severe shear stresses would be induced when the splitter plate and cutwater tips are located at the middle of the impeller blades. The results of the 8BB impeller are opposite to those found in the 16FB and 16SB models, which would produce more severe shear stress when the trailing edges of the impeller blades are sweeping by the splitter plate and cutwater tips as discussed earlier.

Figure 5.20 Scalar shear stress distributions in the 8BB pump model with different impeller locations, note that impeller position in (a) is 22.5° ahead of (b)

Figure 5.21 Mass weighted distribution of scaled shear stress within the 8BB pump model. The impeller position in (a) is 22.5° ahead of (b) (as in Figure 4.20)

This difference can be attributed to the flow pattern in the different blade designed channels. It has been discussed earlier that the outflow from the blade channel to the volute is not uniform at different angular positions. Higher shear stresses would be induced when there are stronger flows from the rotating blade channels impinging on the stationary tips of the splitter plate and cutwater. In the 16FB and 16SB blade channels, there are strong positive flows going out along the blade pressure side. Therefore, when the trailing edges of impeller blades are sweeping by the tips of the splitter plate and cutwater, the strong positive flow would strike the tips and subsequently induce higher shear stresses to the flow. However, in the 8BB blade channels, there is no dominating positive flow along the pressure side. Due to fluid inertia, more out-flowing blood with stronger positive momentum is not restricted to the pressure side of the backward bending blade but goes smoothly between the blades. Therefore, higher shear stresses would occur when the splitter plate and cutwater tips are located between the blades.

Figure 5.22 shows the comparison of the mass weighted distributions of the scaled shear stresses among the 16FB, 16SB and 8BB pump models. Note that the comparison is carried out under the same rotating speed of the impellers and the same pump flow rate, with the tips of the splitter plate and cutwater located at the trailing edge of the impeller blades. Although the mass weighted shear stress distributions within the pump do not provide a direct correlation to the hemolysis estimation of the pump models, it will be very useful in evaluating the different pump designs. While the peak scaled shear stresses in the 16FB, 16SB and 8BB pump models are about 1900 Pa, 950 Pa and 930 Pa respectively, the mass weighted shear stress distributions lower than 20 Pa are about 51%, 67% and 76% respectively. For the shear stresses ranging from 40 Pa to the maximum, the mass weighted distribution of the 16FB pump model is the highest (24.5%) among the

three models, while the 16SB and 8BB pump models are the medium (20.4%) and the lowest (13.2%) respectively. From the results, it can be seen that the 8BB pump model produces the smallest fluid mass percentage at high shear stress among the three pump models, while the 16FB pump model produces the largest fluid mass percentage at high shear stress.

This finding should be expected because high shear stress in the pump mainly occurs at the outlet of the impeller due to the collision of blood between the outflow from the rotating impeller and the tangential flow along the volute especially at the tips of the cutwater and splitter plate.

Figure 5.22 Comparison of mass weighted distributions of the scaled shear stress of different pump models

The blade trailing edge of the 8BB impeller produces a lower absolute flow velocity at the impeller outlet during rotation than the 90° blade trailing edge angle of the 16FB and 16SB impellers. The lower velocity of the impeller outflow will cause milder collisions and thus induce a lower shear stress on the blood. The 16FB and 16SB impellers have the same blade trailing edge angle, however, as the results shown earlier, there are higher disturbances of the flow in 16SB blade channels than those in the 16FB blade channels. Therefore, the positive radial momentum of the outflow from the 16SB blade channel will be lower than that of the smooth positive flow along the pressure side of the 16FB blade channel. The outflow of the 16SB blade channel would thus have softer collisions as compared with that of the 16FB. This is why the 16SB pump model has caused a lower shear stress to the blood than the 16FB pump model.

5.5 A Brief Summary

Under the same impeller rotating speed of 2000 rpm and pump flow rate of 5 l/min, the 16SB pump model produces the highest pressure head among the three impellers, while the 8BB pump model produces the lowest. Figure 5.23 shows a relationship between the pressure head and pump flow rate of the three pump models based on the simulation results. It can be seen in Figure 5.23 that with the increase in pump flow rate, the pressure head of the 16SB pump model decreases more rapidly than that of the 16FB pump model and the pressure head generated by the 16SB pump model is lower than that of the 16FB model at the flow rates higher than 5.65 l/min. While under the same flow rates, the 8BB pump model produces a much lower pressure head than the 16FB and 16SB pump models.

The double volute has dominant effects on the flow in the pump and induces axis symmetrical flow in the impeller blade channel of all the three pump models. Meanwhile,

Chapter 5 Simulations of Flow in between the Impeller Blades for the Three Pump Models

the impeller profiles also have remarkable effects on the flow pattern in the blade channel. Both 16FB and 16SB models produce vortical flow in the blade channel, especially the 16SB model, while the flow in the blade channel of 8BB model is quite smooth and no apparent reverse flow was observed. The flow is quite disturbed at the blade channel outlet of all the three pump models. Both the positive and negative flows occur there due to the interaction between the channel outlet flow and the flow in the volute. In the pump volute, various secondary flow patterns occur in all the three pump models and are closely correlated to the outlet flow patterns from the impeller. Under the same impeller rotating speed and pump flow rate, the 16FB model produces the highest shear stress in the flow among the three pump models and the 8BB model produces the lowest shear stress. A comparison of the flow characteristics are listed in Table 5.1.

Figure 5.23 Head capacity curves of the 16FB, 16SB and 8BB pump models

From the comparisons in the table, it can be found that there is not a single impeller among the three pump models that has advantages over the others in all aspects. The 16FB and 16SB models can generate higher pressure heads than the 8BB model and are more effective in achieving the required flow rate and pressure head. However, the 16FB will also cause the highest shear stress to the flow among the three impellers while the 16SB pump model has low power efficiency due to the large hydraulic losses based on the highly disturbed flow in the blade channel. The 8BB model induces the lowest peak shear stress to the flow among the three pump models. However, the pressure head generated by the 8BB model is much lower than the other two pump models.

As a compromise, the 16SB model could be suggested as the most appropriate selection among the three models because it can generate relatively high pressure head with relatively mild shear stress although the flow in the blade channel of the 16SB model is the most disturbed. The disturbed flow on the other hand is good in mixing the flows in the blade channel and produces soft collisions between the channel outflow and the tangential flow along the volute passage and reduces the impact on the volute, especially at the beginning of the cutwater and splitter plate when the spacing between the volute and impeller is small. These would subsequently reduce the shear stress generated in the flow. In addition, the reduced shock can probably decrease the counterforce exerted on the impeller and hence reduce the difficulty in controlling the magnetically suspended impeller to rotate at the center position at high speeds. Furthermore, the vortex induced in the blade channel will produce constant washout on the inner surface of the blade channel and thus reduce the chances of thrombus formation.

	16FB	16SB	8BB
Pressure head generated at pump flow rate of 5 l/min and impeller rotating speed of 2000 rpm (mmHg)	109.4	113.5	98.5
Symmetrical flow pattern in the blade channel	yes	yes	yes
Effect of volute pressure on the outflow from blade channel to the volute	yes	yes	yes
Reverse flow and vortex found in the blade channel	medium	high	no
Flow disturbance at the blade channel outlet	high	high	high
Vortex found in the volute	yes	yes	yes
Peak shear stress induced in the flow (Pa)	1900	950	930
Mass percentage of blood with shear stress higher than 250 Pa	0.717 %	0.502 %	0.250 %

Table 5.1 The comparison of flow characteristics of the 16FB, 16SB and 8BB pump models

CHAPTER 6

NUMERICAL RESULTS OF FLOW IN THE GAP

The effect of the gap flow on the pump performance is of great importance. It has been reported by Akamatsu et al. (1995) that although the configuration of the magnetically suspended impeller has phased out the use of shaft and seal which have eliminated the major contamination problems, the risk of thrombus formation at the impeller shrouds and the inner surfaces of pump casing cannot be totally prevented. A larger gap will produce effective washout of the blood in the region and hence reduce the risk of thrombus formation, however, it may result in lower pump efficiency due to the increase in leakage flow. In the model, the gap has a width of 0.2 mm, which has been shown [Yamada et al., 1997] to be the optimum width for the pump. However, there is still a lack of flow field study in the gap. The detailed flow field in the gap will be presented and discussed in this chapter based on the numerical simulation results.

There are two gaps in the Kyoto-NTN blood pump, which are the spaces between the parallel surfaces of the pump casing and the upper and lower impeller shrouds. A schematic view of the gap configuration is shown in Figure 6.1. Note that Figure 6.1 has been presented earlier as Figure 3.2 in Chapter 3 and is shown here for ease of reference. The flow in the gap is connected to the main flow by radial clearance between the impeller and pump volute. As marked by the solid black arrows in Figure 6.1, a portion of blood enters the gap from the exit of the impeller and then flows through the gap due to the pressure difference between the eye and the pump volute and subsequently re-enter the impeller blade channels through the eye.

6.1 Numerical Results of Flow in the Gap

Figure 6.1 A schematic view of pump configuration

The static pressure in the volute has a dominant effect on the flow in the gap of the pump. Figure 6.2 shows the volute static pressure distributions at the exit of impellers of the 16FB, 8BB and 16SB pump models. In the figure, the X-axis represents the angular location and the Y-axis indicates the static pressure at the impeller exit. Note that the angular location in Figure 6.2 starts from the tip of the cutwater and increases in anticlockwise direction as usual convention. It can be observed that at the sections of $10^\circ < \theta < 170^\circ$ and $190^\circ < \theta < 350^\circ$, the pressures are increasing in general. There are two drastic drops following the peak values of static pressure of all the three pump models at $\theta = 0^\circ$ and 180° , where the tips of the cutwater and splitter plate are located respectively. Furthermore, the pressure drop at $\theta = 180^\circ$ is steeper than that at $\theta = 0^\circ$. This indicates that the static pressure distribution at the impeller exit is seriously affected by the cutwater and splitter plate. The differences between the impeller blade profiles have also caused some variations in the static pressure distribution at the exit of the impeller. As shown in Figure 6.2, the 16SB pump impeller has the largest fluctuation of static pressure at the impeller exit and the pressure

difference between the angular locations at $\theta=0^\circ$ and 180° is larger than those of the 8BB and 16FB pump models. While the 8BB pump demonstrates the mildest fluctuation of static pressure at the exit of the impeller and the pressure difference between the angular locations at $\theta=0^\circ$ and 180° is also the smallest among the three pump models.

Figure 6.2 Static pressure distributions at the exit of the 16FB, 8BB and 16SB impellers

Figures 6.3 (a), (b) and (c) show the gap pressure distributions at different radii of the 16FB, 8BB and 16SB models respectively. The normalized pressure is presented as $\psi = \Delta p / (r_{imp}^2 \omega^2 \rho)$, where Δp , r_{imp} , ω and ρ are the pressure difference, outer radius of the impeller, impeller rotating speed and density of the fluid respectively. It can be observed that the pressure is decreasing orderly with the decrease in radial location for all the three pump models. This is important since the blood is flowing from the high pressure to low pressure and thus in the correct direction from the volute through the gap to the eye. Furthermore, there is no cross point detected in the pressure distributions in

Figures 6.3 (a), (b) and (c). This has indicated that the pressure decreased generally in the negative radial direction and no circulating flow would be induced in the gap.

Figure 6.3 The static pressure distributions in the gaps of the (a) 16FB (b) 8BB and (c) 16SB pump models

For all the three pump models, the pressure distribution in the gap has been affected by the pressure at the impeller exit. It can be observed that the pressure distribution at the outer gap brim ($R/R_{imp} = 0.96$) follows closely to the volute pressure distribution at the impeller exit as shown earlier in Figure 6.2. It can also be seen in Figures 6.3 (a), (b) and (c) that the pressure distributions in the three models are approximately symmetrical, i.e., the pressure distributions from 0° to 180° are similar to those from 180° to 360° .

There are some differences in gap pressure distributions among the three pump models. The 16SB model has the steepest pressure drop at the angular locations of 0° and 180° and the pressure distribution along the angular location is not as smooth as those of the 16FB and 8BB models. It can be seen that the static pressures at dimensionless radius

$R/R_{\text{imp}} = 0.96$ of the 16FB and 8BB models are similar to each other except that the static pressure variation in the 16FB model is a little larger than that in the 8BB model. In addition, as shown in Figure 6.3 (a), (b) and (c), at the angular segments of 5° to 55° , and symmetrically, 185° to 235° , the pressure distribution contours are more compact in the 16FB and 16SB pump models than those in the 8BB model. This indicates that the pressure gradients along the radial direction in the gap of the 16FB and 16SB models are lower than those of the 8BB model at these angular locations. These differences in the pressure gradients shall be reflected in the gap radial velocity distributions and will be presented in the later section.

The flow in the gap is mainly 2-dimensional which consists of the radial and tangential velocity components. Figure 6.4 shows the tangential velocity distributions at the radius of 20 mm in the gap of the 16FB model. The tangential velocity distributions in the gap of the 8BB and 16SB pump models are very similar to those of the 16FB model and are thus not shown here. It can be observed that the tangential velocities are distributed linearly in the gap. Due to the no-slip condition, the tangential velocity is zero at the pump casing wall and equals to the local tangential velocity of the rotating impeller shroud wall.

Figure 6.4 Tangential velocity distributions at radius of 20 mm but at two different angular locations in the gap of the 16FB pump model

The radial velocity distributions at four different radial locations in the gap of the 16FB pump model are shown in Figures 6.5 (a) and (b). Note that the radial velocities demonstrated in Figure 6.5 (a) and (b) are located at the angular location of $\theta=0^\circ$, the radial velocities at other radial locations have a similar profile to those at $\theta=0^\circ$ but with different magnitudes. Further note that the dimensions of the R (radius) and Z (axial) axes in the figures are presented in meters. The upper line in the figures represents the surface of the impeller shroud and the lower line indicates the surface of the pump casing. The gap radial velocity distributions of the 8BB and 16SB pump models are similar to that of the 16FB model and are not shown here.

As shown in Figures 6.5 (a) and (b), the gap radial velocity profiles are parabolic in shape and heading towards the eye. The velocity profiles are, however, slightly skew with the maximum velocity located nearer to the pump casing surface at the outer radial region and slowly become symmetrical parabolic shape at the inner radius. At the surface of the pump casing and impeller shroud, the radial velocity is reduced to zero due to the no-slip boundary condition on the solid surface. Figure 6.5 (c) shows an analytical solution of the radial flow in the parallel facing gap by Chan et al. (2000), A close agreement between the present numerical results and the analytical solutions can be observed. Since there is a smaller annular area for the fluid in the smaller radial region of the gap, the average radial velocity in the inner radial region is larger than that in the outer radial region in order to satisfy the flow continuity. Therefore, the blood flushing effect in the gap should be stronger at the inner radius than that at the outer radius. The flushing effect is termed as washout mechanism which is required in the design nature of this pump so as to prevent the blood from being trapped and being kept circulating in the

narrow gap for a long period of time, and thus reduce the risk of hemolysis and subsequently thrombus formation.

Figure 6.5 Radial velocity profiles at (a) outer radii (b) inner radii of the gap at angular position of 0° of the 16FB pump model and (c) Analytical result of the radial flow in the parallel facing gap [Chan et al., 2000]

The radial velocity contributes to the mass flow rate in the gap. The mass flow rate through the two gaps of the 16FB, 8BB and 16SB pump models are listed in Table 6.1. The percentage shown in Table 6.1 is the ratio of the mass flow rate in the pump gaps to the mass flow rate at the pump outlet. The mass flow rate in the pump gaps, which is known as leakage flow rate, was obtained by taking the difference in flow rate between the circumference of the impeller and the exit of the pump. Note that the gap mass flow rates of the three pump models in Table 6.1 are under the same impeller rotating speed (2000 rpm) and pump flow rate (5 liter/min). As shown in the table, the percentage of the gap mass flow rate is the highest for the 8BB pump model and is the lowest for the 16FB model. It can be seen that the percentages of the gap mass flow rate are ranging from 25% to 28%. These percentages of the leakage flow are well within the range of 20-40% found by Yamada et al. (1997) and in agreement with the 25% obtained by Chua et al. (2003) through the numerical integration of the measured radial velocity profiles in the gap of a 5:1 enlarged 16FB pump model.

	Gap Flow Rate (kg/s)	Percentage (%)
16FB	0.0219	25.1
8BB	0.0240	27.4
16SB	0.0228	26.1

Table 6.1 Gap mass flow rates and percentages relative to the pump flow rate of the 16FB, 8BB and 16SB pump models

Figures 6.6 (a), (b) and (c) show the absolute velocity distributions in three different cutting planes located at $\xi / d = 0.25, 0.50$ and 0.75 in the gap respectively of the 16FB pump model, where ξ is the distance measured from the inner surface of pump casing to

the cutting plane and d is the width of the gap. The vector plots at all the gap cutting planes of the 8BB and 16SB models are similar to those of the 16FB model and are not shown here. Note that the planes at $\xi / d = 0.25$ and 0.75 are the nearest and the furthest cutting planes to the pump casing surface respectively. It can be observed that the velocity distributions vary with the location of the cutting plane in the gap. At $\xi / d = 0.25$, the fluid has the lowest velocity magnitude among the three planes due to the boundary layer effects of the stationary pump casing. At $\xi / d = 0.75$, the flow has the highest velocity magnitude and is mainly in the tangential direction since the rotating impeller shroud has induced a higher tangential velocity of the fluid at the plane. At the regions of inner radii, the velocities tend to skew more towards the eye than those at outer radii. This is because the radial velocity magnitude is generally higher at the inner radii region than those at the outer radii region in the gap as demonstrated earlier in Figure 6.5 (a) and (b). From the vector plots, it can be observed that the flow in the gap is smooth and no vortex is identified in all the three cutting planes.

Figure 6.6 Velocity distributions in the gap of the 16FB pump model at view planes of **(a)** $\xi/d = 0.25$; **(b)** $\xi/d = 0.5$ and **(c)** $\xi/d = 0.75$

Figures 6.7 (a), (b) and (c) show the gap radial velocity distributions at different radii along the center plane of the 16FB, 8BB and 16SB model gaps respectively. Note that the velocity shown in the Y-axis of the figure is an absolute value and the velocity is in fact negative as it represents the flow in the reverse radial direction towards the eye. It can be seen in Figure 6.7 that the radial velocity magnitudes in the gap are decreasing from the inner radius to the outer radius. This is not unexpected since there is a smaller annular area for the smaller radial circumference of the gap, higher radial velocity magnitude is thus required to satisfy the flow continuity. For all the three pump models, the radial velocity magnitudes increase in the angular segments of about $20^\circ \leq \theta \leq 160^\circ$ and $200^\circ \leq \theta \leq 340^\circ$. The peaks of the radial velocity occur at some angular distance (about 4° to 30° from the outer to inner radii respectively) before 0° and 180° , where the tips of the cutwater and the splitter plate are respectively located. The peaks of the radial velocity at the outer radii occur closer to the tips of the splitter plate and the cutwater than those at the inner radii in the gap. The high radial velocity magnitude will cause a stronger washout at these regions in the gap. The lowest radial velocity magnitude occurs at some angular distance (about 10° to 40° from the outer to inner radii) after 0° and 180° . These regions have thus the lowest washout effect in the gap. It is expected to see that the radial velocity distribution in the gap is corresponding to the static pressure distributions discussed earlier in Figures 6.3 (a), (b) and (c), since higher pressure differences between the inner and outer radii of the gap will produce higher radial pressure gradients and thus resulting in higher radial velocity magnitudes at these angular locations.

It can be seen from Figures 6.7 (a), (b) and (c) that the peak gap radial velocity at the inner radii of the 8BB model is slightly higher than those of the 16FB and 16SB models.

This higher peak radial velocity could be due to the higher radial pressure gradient in the gap of the 8BB pump model as shown earlier in Figure 6.3 (b).

Figure 6.7 The radial velocity distributions at different radii at center plane ($\xi / d = 0.5$) of the gaps of (a)16FB (b)8BB and (c)16SB pump models

Figures 6.8 (a) and (b) show the velocity vectors and the radial velocity distributions at the gap center plane obtained by Chua et al. (2002) respectively. The measurements were carried out on a 5:1 enlarged pump model with the 16FB impeller. As shown in Figure 6.8 (a), there is an angular segment of flow ranging from $\theta=100^\circ$ to $\theta=240^\circ$ directed strongly towards the eye while the flows in other regions are mainly in the tangential direction [Chua et al., 2002]. The radial velocity distribution of Chua et al. (2002) as shown in Figure 6.8 (b) demonstrated that there was a unique peak of the radial velocity magnitude in the gap, which occurred at the angular position of 180° , where the tip of the splitter plate is located. The measured radial velocity distributions of Chua et al. (2002) are different from the present simulation results of the 16FB model, shown earlier in Figures 6.6 (b) and 6.7 (a), in two ways. Firstly, the measured gap radial velocity

magnitude has a larger increment at the inner radii than those at the outer radii by Chua et al. (2002), while the present numerical simulation has a smaller increase at the inner radii than those at the outer radii. Secondly, the measured gap radial velocity distribution has only one peak at the tip of the splitter plate [Chua et al., 2002], while there are two peak gap radial velocities located before both the tips of the cutwater and splitter plate in the present simulation results. Possible reasons for the differences might be due to the measurements being carried out on a 5:1 enlarged pump model, in which the flow characteristics may not be exactly the same as the prototype due to the scaling effect. More discussions on the scaling effects due to the incomplete implementation of dimensional analysis will be presented in Section 7.3 of Chapter 7. Furthermore, the medium used in the experiment was air, which was much lighter but had higher kinematic viscosity than blood. These would cause a different velocity distribution in the gap when compared with blood analog used in simulation. In addition, the use of hot-wire

(a)

Figure 6.8 Measurements of (a) velocity vector distributions, and (b) the radial velocity distributions at different radii, at center plane ($\xi / d = 0.5$) of the gap [Chua et al. (2002)]

probe for measurement has caused some modifications on the pump casing, which can be freely rotated 360° in order to mount the hot-wire probe and measure the gap velocity [Chua et al., 2002]. Although this may not modify the geometry of the pump, the flow condition may be changed due to the measurement method.

Figures 6.9 (a), (b) and (c) (also Figures 6.10 (a), (b) and (c)) show the gap radial velocity distributions at different radii of the planes located at $\xi / d = 0.25$ (also $\xi / d = 0.75$) for 16FB, 8BB and 16SB models respectively. Note that the plane of $\xi / d = 0.25$ (also $\xi / d = 0.75$) is located near the pump casing (also the impeller shroud). It can be seen that, for all the three pump models, the radial velocity distributions at the planes of $\xi / d = 0.25$ and $\xi / d = 0.75$ have similar trends as those occurring at the middle gap plane, i.e., the radial velocity magnitude increases in the angular sections of $20^\circ \leq \theta \leq$

160° and $200^\circ \leq \theta \leq 340^\circ$, while at the angular locations around 0° and 180° , the radial velocity magnitude decreases rapidly.

However, it can be observed that the radial velocity magnitudes at the planes of $\xi/d = 0.25$ and $\xi/d = 0.75$ are generally lower than those at the middle plane of the gap. It should be worth noting that at a small angular region from 190° to 210° , minus velocity magnitude, which indicates that the flow is in the positive radial direction, occurs at the outermost radius (i.e. $r/r_{\text{imp}}=0.96$) at the plane of $\xi/d = 0.75$ as shown in Figure 6.10. The low but positive radial velocity at the plane of $\xi/d = 0.75$ should be due to the higher tangential velocity of the flow at the impeller shroud than that at the plane of $\xi/d = 0.25$ as shown earlier in Figures 6.6 (a) and (c). The fluid with a higher tangential velocity would have a higher centrifugal force which would then overcome the pressure force pushing the fluid in the negative radial direction and have resulted in the flow in the positive radial direction, although this happens in a very small region. Under the effects of both the pressure gradient and the centrifugal force of the fluid, the radial velocity magnitude is generally lower at plane $\xi/d = 0.75$ than that at plane $\xi/d = 0.25$.

(a)

(b)

(c)

Figure 6.9 The radial velocity distributions at different radii at the plane of $\xi / d = 0.25$ of the gaps of (a)16FB (b)8BB and (c)16SB pump models

Figure 6.10 The radial velocity distributions at different radii at the plane of $\xi / d = 0.75$ of the gaps of (a)16FB (b)8BB and (c)16SB pump models

Based on the velocity distributions in the pump gap, the wall shear stresses (WSS, defined as $\mu \frac{\partial u}{\partial y} \Big|_{y \rightarrow 0}$, where y is the normal distance starting from the wall boundary) on the surface of the impeller shroud and pump casing are computed. It has been known that the thrombus generation in the pump gap is not only related to the washout effect of the gap flow but also related to the magnitude of shear stress on the wall. The preferable WSS value on the gap inner surface should be high enough to prevent flow stagnation at the wall and at the same time it should be low enough to prevent severe lysis of blood cells.

Figures 6.11 (a), (b) and (c) show the WSS contours on the impeller shroud surface of the gaps of the 16FB, 8BB and 16SB pump models respectively. From the figures, it can be seen that the WSS distributions on the shroud surface at the gap of the three pump models share a similar pattern. At most of the regions, the WSS is about 50 to 80 Pa. Higher WSS, which is about 90 to 110Pa, occurs at the outer radii of the surface. The higher WSS is due to the higher tangential velocity difference between the blood just entering the gap and the high rotating speed of the impeller shroud. Besides, at some angular distance after the tips of the cutwater and splitter plate, the WSS is generally lower than those at the other locations due to the less mass flow rate of blood there as demonstrated earlier in Figures 6.10 (a), (b) and (c).

Figure 6.11 WSS on the impeller shroud surfaces of the gaps of the (a)16FB (b)8BB and (c)16SB pump models

Comparing the WSS contours on the impeller shroud surface at the gap of the 16FB, 8BB and 16SB pumps, it can be found that the regions with WSS higher than 80 Pa in the 16FB pump model are larger in area than those in the 8BB and 16SB pump models while the WSS distributions on the impeller shroud surface of the 8BB and 16SB models are similar. Furthermore, as shown in the Figures 6.11 (a), (b) and (c), at almost all the impeller shroud surfaces for the three pump models, the WSS is higher than 20 Pa, at which the aggregation in human blood is disrupted [Mazumdar, 1992].

Figures 6.12 (a), (b) and (c) demonstrate the WSS contours on the pump casing surface of the gap of 16FB, 8BB and 16SB pump models respectively. Note that the pump casing surface of the gap is stationary during pump operation. It can be seen that the WSS distributions on the pump casing surface are different from those on the impeller shroud surface but share similar characteristics among the three pump models. As shown in Figures 6.12 (a), (b) and (c), the WSS on the pump casing surface of the gap is generally higher than that on the impeller shroud surface. The WSS is about 100 Pa for most of the areas of the pump casing surface. Higher WSS occurs near the tips of the splitter plate and cutwater and covers more area before than after the tips as can be observed in Figure 6.12. This is due to the higher radial velocity magnitude at some angular distance before $\theta=0^\circ$ and 180° at $\xi/d = 0.25$ plane as shown earlier in Figures 6.9 (a), (b) and (c). The highest value of the WSS on the pump casing surface is about 170 Pa, which is well below the threshold of 400 Pa proposed by Pual et al. (2003) and is not supposed to deliver massive blood cell damages.

Comparing the gap WSS on the pump casing surface of the 16FB, 8BB and 16SB pump models, it can be seen that the area with WSS higher than 100 Pa of the 16FB and 8BB models is slightly larger than that of the 16SB model. It can also be observed in Figures 6.12 (a), (b) and (c) that the lowest value of the WSS on the pump casing surface at the gap of all the three pump models is higher than 20 Pa, which is the threshold for the aggregate human blood to be disrupted [Mazumdar, 1992].

Figure 6.12 WSS on the inner pump housing surfaces of the gaps of the (a)16FB (b)8BB and (c)16SB pump models

6.2 A Brief Summary

The different impeller profiles do not cause significant difference to the gap flow. Under the pump operating condition, the 8BB has the highest leakage flow rate through the gap among the three pump models while the 16FB has the lowest. However, the leakage flow rates do not differ much and are within the range of 25% to 28% of the pump mass flow rate. The gap velocity distribution is mainly affected by the static pressure distribution. The radial velocity profile in the gap is parabolic in shape. The lowest radial flows occur at some angular distance after the tips of the splitter plate and cutwater and thus the washout effect would be the lowest at these regions. However, the cutwater and splitter plate are the initiation of the washout mechanism since they trigger a snowball effect by creating the whole sector of the flow towards the eye. The WSS on the gap surface is correlated with the radial velocity distribution. The top WSS is found to be about 170 Pa, which is below the threshold to cause massive blood cell damage [Pual et al., 2003], while the lowest WSS is higher than 20 Pa, which is the threshold for disrupting the aggregation of blood cell on the gap surface [Mazumdar, 1992].

CHAPTER 7

MEASUREMENTS OF FLOW IN BETWEEN THE IMPELLER BLADES FOR THE THREE PUMP MODELS

In this chapter, the measurement results of the flow in the blade channels with the same size (i.e. 1:1) as the prototype of the 16FB, 16SB and 8BB pump models are presented and discussed. The experimental arrangements and the blood analog used in the pump model have been reported earlier in Chapter 4. In the measurements, the impeller rotating speed and the pump flow rate are 2000 rpm and 5 l/min respectively, which are the same flow conditions as those in the simulation. The velocities are measured at six radial locations, namely 14mm, 16mm, 18mm, 20mm, 22mm and 24mm of the center plane of the blade channel. Note that the measuring plane is parallel to the upper and lower impeller shrouds and is located at the middle of the blade height. For ease and consistency in the presentation, the blade channels of the impellers are numbered (as those in the numerical simulation) in an anticlockwise direction and starting from the angular location of $\theta = 0^\circ$, where the tip of the cutwater is located.

It is well known that the computational fluid dynamics offers much more versatility and better resolution than experimental measurements, however, computational results must be validated by careful comparison with the experimental data. This is the reason why the flows in the pump model are measured and compared with the numerical simulation results in this chapter. In addition, the measurements and numerical simulations will also be compared with the measurements on a 5:1 enlarged pump model obtained by Ong (2004).

7.1 Flow in the 16FB Impeller Passages

Figures 7.1 (a) and (b) (also Figures 7.3 (a) & (b) and Figures 7.5 (a) & (b)) show the measured velocity vectors at the center plane of the blade channels and the numerical simulation results in the blade channels of the 16FB (also 16SB and 8BB) pump model respectively. Note that the simulation results have been shown earlier in Figure 5.4 (also Figures 5.9 and 5.14) for the 16FB (also 16SB and 8BB) model and are shown here for ease of reference and comparison. The vectors shown in Figure 7.1 (a) (also Figures 7.3 (a) and 7.5 (a)) are relative to the rotating speed of the impeller. Note that since the measurement volume of the laser beam has a length of about 2 mm in the fluid medium, the velocity measured has in fact covered the fluid flow within a range of about 1 mm above and below the reference center plane. In other words, the flow velocity measured at the center plane is an average value of the flow in a 2 mm region. The same conditions are also applied to the LDA measurements of 16SB and 8BB impellers. Note that the height of the impeller blade is 3.5 mm.

As those observed in the simulation results, which are presented earlier in Chapter 5, the flows in the blade channels are also found to be approximately symmetrical in the measurement results. It can be seen in Figure 7.1 (a) that the flows in quadrant I (and II) are symmetrical to those in quadrant III (and IV), that is blade channels 1 and 9 share a similar flow pattern and it is the same for blade channel pairs of 2 and 10, 3 and 11, ..., 8 and 16. The symmetrical flow pattern in the blade channel obtained by the LDA measurement has once again shown the influence of the double volute design, which has the tips of the cutwater and splitter plate starting at the angular positions of $\theta=0^\circ$ and 180° respectively.

Figure 7.1 (a) The measured velocity distributions at the middle plane and (b) the numerical simulated 3-D velocity distribution of the 16FB impeller blade channels

However, it can be seen from the measurement results in Figure 7.1 (a) that the symmetry of the flow is far from perfect. The velocity magnitudes in the blade channels at the lower half of impeller ($180^\circ < \theta < 360^\circ$) are smaller than those in the respective blade channels at the upper half of impeller ($0^\circ < \theta < 180^\circ$). Besides the measurement uncertainty, it is found that there is a defect in the pump model which has caused that the flow conditions not exactly the same as those of the numerical model. Due to limitations in fabrication, it is detected that when the two half of the pump casings are being assembled together, a tiny leaking strip is formed at the splitter plate, i.e., the two half of the splitter plates are not contacting each other firmly but have a tiny empty space between them. The tiny empty strip is too small to be detected by the naked eye in the assembled pump and unfortunately, it was found when all the measurements were done. Note that the width of the empty strip is varying from 1.1mm at $\theta=180^\circ$ to 0.5mm at $\theta=360^\circ$. Due to time constraints, it was too late to make a new pump model and repeat the measurements. With this tiny strip, the pressure at the inner volute was thus increased due to the additional pressure from the outer volute. This has explained why the velocity magnitude is generally smaller in the blade channels at the lower half of the impeller. The flow patterns in the corresponding blade channel pairs (1 and 9, 2 and 10, etc) are nevertheless qualitatively symmetrical to each other. The comparison between the results of the measurement and the simulation is therefore concentrated on the upper half of the impeller in all the three pump models hereafter.

It can be seen that the measured flow fields in blade channel 1 are quite similar to those of the simulation. The fluids flow out along the pressure side and there is backward flow near the trailing edge due to a clockwise vortex, which spans the outer brim of the blade channel and results in the outward flow near the trailing edge of the suction side. Inside

the blade channel, another vortex can be observed near the middle of the suction side and has resulted in negative flow along the suction side. All these flow patterns are very similar to those found in the simulation, although there are only a limited number of vectors obtained and presented in Figure 7.1 (a) due to the spatial resolution of the LDA measurements.

The measured flow patterns in blade channel 2 are also similar to those in the simulation. The only difference between them is that the vortex generated in the blade channel near the suction side is difficult to be identified in the measurement results, with only backflow along the suction side can be seen vaguely. This should be due to that the measurement is taking an average value at the center plane (with a length of 1mm on the upper and lower sides of the plane), the positive and negative vectors will cancel out each other in the averaging process and results in a small velocity magnitude there.

It can be observed that the measured flows along the pressure sides of blade channels 3, 4 and 5 become stronger in Figure 7.1 (a) and there is no vortex found in these blade channels. This is consistent with the increasing stronger through flows observed in the same blade channels of the simulations as shown in Figure 7.1 (b). It can also be found in Figure 7.1 (b) that the vortex observed earlier in blade channels 1 and 2 has vanished and the reverse flow along the suction side has disappeared.

From the measured flows in blade channels 6, 7 and 8, reverse flow along the suction side has increased due to these blade channels being close to the highest pressure region at the splitter plate. A vortex can also be clearly identified in blade channels 7 and 8 near the

suction side. These are the flow patterns found in the simulation in blade channels 6, 7 and 8 as demonstrated in Figure 7.1 (b).

Through the comparisons, it can be found that the experiment and numerical simulation have very similar flow patterns in the 16FB impeller. The agreement has reasonably validated the numerical simulation results.

Figure 7.2 shows the measured velocity distributions obtained by Ong (2004) in the blade channels of a 5-times enlarged 16SB pump model. Note that the impeller blades have the same orientation with respect to the pump volute as those of the present 1:1 pump model. There are some similarities between the flows in the blade channels of the 5:1 and present 1:1 pump models. As those found in the 1:1 pump model, the flow patterns in the blade channels of the 5:1 pump model are basically symmetrical, i.e. the flow patterns in the blade channels of quadrant I are similar to those of quadrant III and the flow patterns in the blade channels of quadrant II are similar to those of quadrant IV.

Besides the similarity, the main difference between the flow patterns of the 5:1 and 1:1 pump models is that, in the 5:1 pump model as shown in Figure 7.2, the flow is blocked at the outer rim of the blade channels 1, 2 and 3 (also 9, 10 and 11) due to the small space between the impeller and the volute. With the increase of the volute space, the flow to the volute increases from blade channel 4 (also 12) onwards and reaches the maximum at blade channels 7 and 8 (also 15 and 16). However, in the 1:1 pump model, there is obvious reverse flow from the volute to the blade channel along the suction side of blade channels 6, 7 and 8 (also 14, 15 and 16) due to the highest static pressure before the splitter plate tip (also the cutwater tip) and the flow increased from blade channel 1 (also

9) to blade channel 5 (also 13) and then decreased again from blade channel 6 to 8 (also from blade channel 14 to 16). It is interesting to note that the influence of the double volute design on the 5:1 and 1:1 pump models are similar since they both produce symmetrical flow. However, the way that the flows are symmetrical is different in the two models. Besides the limitations due to manufacturing process, the 1:1 model pump should have produced the flow closer to the actual flow in the prototype. The differences in flow pattern measured between the 1:1 and 5:1 pump models should be due to the scaling effect of enlargement since the 5:1 pump was built based on dimensional analysis, which has some limitations. Its effect on the measurements of the enlarged pump model will be discussed further in Section 7.3.

Figure 7.2 Velocity distribution in the blade channels of the 16FB impeller on a 5:1 pump model [Ong, 2004]

7.2 Flow in the 16SB Impeller Passages

The measured velocity vectors in the blade channels of the 16SB pump model are presented in Figure 7.3 (a). It can be seen in the figure that the measured flows are more disturbed as compared with those in the 16FB model as shown earlier in Figure 7.1 (a). The flows in the 16SB model are, however, more symmetrical than those in the 16FB model. In short, the flows in blade channels 1, 2, 3 and 4 (also 5, 6, 7 and 8) in quadrant I (also III) are symmetrical to those in blade channels 9, 10, 11 and 12 (also 13, 14, 15 and 16) in quadrant II (also IV) respectively. These have once again demonstrated that although the blades are different in shape for the 16FB and 16SB models, the double volute design has significant and dominant effects on the main flow characteristics. Due to the manufacturing defect in the splitter plate as discussed earlier in Section 7.1, the symmetry in flow pattern for the 16SB model are only qualitative and comparisons between the measured results and numerical results is focused on the upper half ($0^\circ < \theta < 180^\circ$) of the impeller.

Comparing the measured flow patterns in blade channel 1 with those of the numerical simulation, it can be seen that the positive flows are strong along the pressure side, although the positive out flows measured are smaller in magnitude as compared to those in the simulation at the outer rim of blade channel 1. A similar strong vortex can be found at the center of blade channel 1 but slightly close to the entrance and the suction side in both the measured and numerical results. It can be observed from blade channel 2 to 5, the vortex which is found in blade channel 1 is slowly losing its strength from blade channel 2 to 3 and totally vanishes in blade channel 5 in both the measured and simulation results. At the same time, the positive flow along the pressure side has increased in velocity magnitude from the entrance to the exit. Note that in all these blade

Figure 7.3 (a) The measured velocity distributions at the middle plane and (b) the numerical simulated 3-D velocity distribution of the 16SB impeller blade channels

channels, the velocity vectors are decreasing in magnitude from the pressure to suction side.

From blade channel 6 to 8, it can be observed that a vortex is formed initially near the suction side of blade channel 6 and growing in strength from blade channel 7 to 8 and it has shifted closer to the pressure side in blade channel 8. This should be due to the high pressure at the splitter plate which has caused the reverse flow at the suction side of the channel and the straight blade design which has also strengthened the effect of the high pressure and forced the vortex center to shift closer to the pressure side. The positive flow along the pressure side is maintained from blade channel 6 to 8, although it can be seen that the flow velocity magnitude is decreasing slightly with the advancing of the blade channel. The reduction in velocity magnitude is especially obvious at the blade channel entrance, particularly in blade channel 8, where the strong effect of vortex can be seen. Comparing to the 16FB model, it can be found that vortices in the blade channels of the 16SB impeller are more prevalent. The number of blade channels of the 16SB impeller that contained vortex is more than that of the 16FB impeller. In addition, the vortex in the 16SB impeller blade channel is larger in size than that in the 16FB blade channel. The larger and stronger vortex would extend to the inlet of the blade channel and thus cause reverse flow even at the blade channel inlet. All these can be found in both the measured and numerical results, although it is less obvious in the measured flow pattern due to the limited spatial resolution in the 1:1 model.

Through the comparison, it can be seen that the experimental results share quite similar flow patterns in the blade channels with those of the numerical simulation. The agreements have reasonably validated the numerical simulation of the flow in the 16SB pump model.

Figure 7.4 shows the measurement of velocity distribution in the blade channel of 16SB impeller on a 5:1 pump model by Ong (2004). Similarity of the flows in the 16SB blade channel can be drawn from the 1:1 and 5:1 pump models; the flows in 5:1 pump model are also symmetrical, that is, flows in blade channels 1, 2, 3 and 4 (also blade channels 5, 6, 7 and 8) of quadrant I (also II) are similar to those in blade channels 9, 10, 11 and 12 (also blade channels 13, 14, 15 and 16) of quadrant III (also IV). This has shown the same significant effect of the double volute design on the flow pattern of both the 1:1 and 5:1 pump models. Besides this similarity, it can be seen that the detailed flow patterns in the blade channels of the 1:1 and 5:1 models are totally different. First of all, it can be observed in the 5:1 model that the vortex which is initially induced in blade channel 1 has persistently existed till nearly blade channel 5, while in the 1:1 model, the vortex dissipates and disappears in both blade channels 4 and 5. Furthermore, it can be clearly seen in the 5:1 models that the flow are nearly in reverse direction at the outer rim of the channels from blade channel 1 to 5. It seems that there is a blockage effect due to the small opening space between the outer volute (started from the tip of the cutwater) and the impeller from blade channel 1 to 5. There is only a small portion of positive flow along the pressure side in these blade channels (1 to 5) for the 5:1 model, which is very much different from the measured results of the 1:1 model, where the positive flow is increasing in magnitude from blade channel 1 to 5 as shown earlier in Figure 7.3 (a).

The through flow characteristics is obvious in the 5:1 model from blade channel 6 to 8, due to the wider opening space between the outer volute and the impeller at these angular locations. In addition, it can be observed in blade channels 6, 7 and 8 that there is no vortex or even reverse flow in the 5:1 model except that the positive flow is skewed to the

pressure side at the entrance. It seems that the high pressure at the splitter plate has no effect on the flow in these blade channels, which is very much different from those of the present 1:1 model, where the vortex is formed in both blade channels 7 and 8. The reverse flow along the suction sides of blade channels 6, 7 and 8 for the 1:1 pump is obviously due to the increasing high pressure along the outer volute before the splitter plate. It is apparent that the size of space between the volute and the impeller has greatly affected the flow patterns in the blade channels of the 5:1 pump model while the flow patterns of the 1:1 pump model are governed by the pressure difference between the volute and the blade channel.

Figure 7.4 Velocity distribution in the blade channels of the 16SB impeller on a 5:1 pump model [Ong, 2004]

It can be concluded that although the flow patterns of the 5:1 and 1:1 pumps are axis symmetrical, the detailed flow patterns are quite different for the two pumps. These should be due to the scaling effects of the enlarged pump and will be discussed further in Section 7.4.

7.3 Flow in the 8BB Impeller Passages

As shown in Figure 7.5 (a), the measured velocity distributions in the blade channel of the 8BB impeller are much smoother than those of the 16FB and 16SB impellers. The flow in the blade channel basically follows the profile of the impeller blade. Similar to the flow in the blade channels of the 16FB and 16SB pump models, the measured flows in the 8BB model also demonstrate symmetrical flow pattern. It can be observed in Figure 7.5 (a) that the flow pattern in blade channel 1 is a mirror image of those in blade channel 5, although it is not a perfect match due to the measurement uncertainty and manufacturing defect as mentioned earlier. Similar flow patterns are shared between blade channel pairs of 2 and 6, 3 and 7 and 4 and 8. These are due to the double volute design of the pump and the same effects have shown in the numerical simulated flow distributions in the blade channels of the 8BB pump model in Figure 7.5 (b).

It can be seen that the measured velocity distributions in blade channel 1 (also 3) are very similar to those of the simulation. Blood flows smoothly and almost uniformly from the inlet to the outlet along blade channel 1 (also 3) except those at the outmost radius, the velocity magnitude is reduced near the trailing edge of the suction side. Note that odd number blade channels 1 and 3 have the same shape which is bounded by the thicker blade at the pressure side and thinner blade at the suction side.

Figure 7.5 (a) The measured velocity distributions at the middle plane and (b) the numerical simulated 3-D velocity distribution of the 8BB impeller blade channels

For blade channel 2, the flow pattern of the measurement is also quite similar to those of the simulation. Blood is flowing smoothly from the entrance through the blade channel space till the exit. There is a backflow from the volute into the blade channel near the suction side of the channel, however, the positive flow is still dominant and the backflow is restricted to half of the channel space at higher radial regions.

A similar flow pattern can be observed in blade channel 4, however, due to the high pressure at the splitter plate, the backflow has occupied a bigger region and the positive forward flows along the pressure side have smaller magnitudes than those in blade channels 2 and 3. Similar flow patterns can be observed in Figure 7.5 (b) for the numerical results and the measurements have thus validated the simulation model.

However, there are minor differences between the measured and numerical results of the 16FB, 16SB and 8BB models. This should be due to three possible reasons: Firstly, the much lower resolution and 2-dimensional scope of the experimental measurement would restrict it to reveal detail flow structures as the numerical simulation. Secondly, the simulation result provided is instantaneous of the transient flow in the pump, while LDA measures the flow based on time-averaging over a number of cycles with the reference of impeller angular location provided by a shaft encoder. Last but not the least, due to the limitation of fabrication, the geometries of the experimental pump model could not be perfect as those of the numerical model and would cause slight differences of the flow in the experimental pump model from those of the prototype.

The velocity distribution in the blade channels of 8BB impeller on the 5:1 pump model obtained by Ong (2004) is shown in Figure 7.6. Note that the orientation of the pressure

side of blade channel 1 is about 5° before the cutwater for the 5:1 model which is slightly different from the present 1:1 model where the pressure side blade is located at the cutwater. There is a similar flow characteristic between the 1:1 and 5:1 pump models. As those observed in the 1:1 pump model, the flow patterns in the blade channels of the 5:1 pump model are approximately symmetrical, i.e. the flow patterns in the blade channel of quadrant I (also quadrant II) are similar to those in the blade channel of the quadrant III (also IV).

Figure 7.6 Velocity distribution in the blade channels of the 8BB impeller on a 5:1 pump model [Ong, 2004]

However, there are obvious differences between the flow patterns in the blade channels of the 1:1 and 5:1 pump models. As the cases of the 16FB and 16SB models, the flow

pattern of the 5:1 pump is seriously affected by the space in between the outer volute (also inner volute) and the impeller. It can be observed in Figure 7.6 that the flow in blade channel 1 is quite disturbed where backward flow can be found at the region close to the suction side. The forward flow at the trailing edge of the pressure side is basically due to the location of the trailing edge, which is 5° before the beginning of cutwater. This has resulted in the rushing out of the flow because of the large space between the inner volute and the impeller at this location. Due to the gradually increasing space between the volute and the impeller, the positive forward flow is increasing as shown in blade channels 2 and 3 of the 5:1 pump model. It can be seen that the flow patterns are similar to each other for the 5:1 and 1:1 models in blade channel 2 although the forward flow velocity is slightly smaller in the 5:1 model. There are flow separations at the suction side of the blade channel 3 for the 5:1 model as shown in Figure 7.6, and in general the flow is not as smooth and uniform as compared with those in the 1:1 model as shown in Figure 7.5 (a).

The obvious difference is in blade channel 4, where through flow characteristics can be observed in the 5:1 model while there is backflow in the 1:1 model near the suction side as discussed earlier. The differences should be due to the scaling effect of the 5:1 pump model and will be discussed in the next section.

7.4 Limitation of Dimensional Analysis

From the comparison of the flow in the blade channels of the 1:1 and 5:1 pump models for the 16FB, 16SB and 8BB impeller designs, it can be concluded that the five-time enlarged pump model indeed shares some similar flow characteristics with the 1:1 pump model. However, remarkable flow pattern differences between the 1:1 and 5:1 models exist in all the 16FB, 16SB and 8BB impeller designs. This indicated that the five-time

Chapter 7 Measurements of Flow in between the Impeller Blades for the Three Pump Models

enlarged pump model may not be able to produce true inner flow patterns of the prototype although it was built according to the flow similarity law. This is an important finding, since it may explain why we should not use the enlarged model but the exact size model in order to have a better reflection of the actual true flow.

The imperfect flow in the 5:1 enlarged pump model could be due to incomplete similarity. Under complete flow similarity between the enlarged pump model and the prototype, all the equations from (7.1) to (7.5) should be satisfied.

$$\phi(\pi_H, \pi_Q, \pi_{Re}, \pi_P, \dots) = 0 \quad (7.1)$$

where

$$\pi_H = \frac{H_p}{\rho_p D_p^2 N_p^2} = \frac{H_m}{\rho_m D_m^2 N_m^2} \quad ; \quad \text{Head coefficient} \quad (7.2)$$

$$\pi_Q = \frac{Q_p}{N_p D_p^3} = \frac{Q_m}{N_m D_m^3} \quad ; \quad \text{Flow coefficient} \quad (7.3)$$

$$\pi_{Re} = \frac{\rho_p N_p D_p^2}{\mu_p} = \frac{\rho_m N_m D_m^2}{\mu_m} \quad ; \quad \text{Reynolds number} \quad (7.4)$$

$$\pi_P = \frac{P_p}{\rho_p N_p^3 D_p^5} = \frac{P_m}{\rho_m N_m^3 D_m^5} \quad ; \quad \text{Power coefficient} \quad (7.5)$$

H Pressure head (Pa);

Q Volume flow rate (l/min);

N Rotational speed of the impeller (1/s);

P Impeller driving power (W);

D Impeller diameter (m);

ρ Fluid density (kg/m^3);

μ Fluid dynamic viscosity ($\text{Pa}\cdot\text{s}$);

Subscript p denotes the prototype;

Subscript m denotes the enlarged pump model.

The dimensionless parameters, π_H , π_Q , π_{Re} and π_P , which are obtained from the flow condition of the prototype, are the same magnitudes specified for the pump models in order to maintain flow similarity. However, this usually cannot be fully satisfied in practice. The independent variables, D , N , ρ , μ and H will affect the pump operation, while P and Q are dependent on these independent variables and can only be obtained through experiments. Note that if Q replaces H as independent variable then H will become a dependent variable and vice versa. Because impeller diameter D has been determined in fabricating the 5:1 enlarged pump model, there are four independent variables that are controllable in the enlarged pump model. Therefore, it can be seen that the four independent variables, N , ρ , μ and H , must be carefully selected in order to satisfy all the four equations from (7.2) to (7.5).

However, it is difficult to obtain the solution of N , ρ , μ and H such that the four dimensionless coefficients can be satisfied simultaneously since the dependent variables P and Q can only be achieved through experiments and would be affected by some other inter related factors such as imperfect geometry of the experimental pump model as compared to the prototype, vibration of the pump model, different heat generation between the pump model and prototype and so on. Practically, instead of satisfying the full set of dimensionless coefficients simultaneously, only major dimensionless

coefficients are satisfied. For the 5:1 enlarged blood pump model as shown in Table 7.1 [Ong, 2004], the density of the fluid was prepared and similar to human blood and the rotating speed of the impeller was set at 200 rpm [Ong, 2004]. Same Reynolds number as the prototype was achieved by adjusting the viscosity of the working fluids in order to satisfy equation 7.4 for dynamic similarity. The pressure head of the 5:1 model is adjusted to 30 Pa during experiment, which is close to the standard value of 30.427 Pa in order to satisfy the similarity law. However, the flow rate attained by the 5:1 pump model during experiments was 82 l/min, which is much higher than the standard 62.5 l/min in order to satisfy the similarity law. As shown in Table 7.2, there are some discrepancies between the flow coefficients of the 5:1 pump model and the prototype. This indicates that the enlarged pump model would not be able to achieve a complete similarity. That is the reason why the flow patterns in the enlarged pump model cannot completely and truly reflect the flow of the prototype. Nevertheless, the larger flow coefficient of the 5:1 model could probably explain why the outflow from the impeller blade channels of the 5:1 model is not affected by static pressure as those happening in the 1:1 pump model but is restricted by the capacity of the volute passage. Under almost the same head coefficient as the prototype, the 5:1 model pump with more than 31% flow coefficient could have caused higher flow velocities in the blade channels. These high velocity fluids with high momentum would impact harder on the volute. When the space between the impeller and the volute is small, for example, the impeller passages in the 1st and 3rd quadrants, the flow will be bounced back to the impeller and cause backflow at the outer rim of the impeller as can be observed in the 5:1 pump model results obtained by Ong (2004). Through flow characteristics are observed at the 2nd and 4th quadrants in the enlarged pump when the space between the impeller and volute is large [Ong, 2004]. These are very different from the 1:1 pump that the flow characteristics are being affected strongly

by the pressure distribution along the volute passage. Since the 1:1 pump model have a good balance between all the coefficients especially the head and flow coefficients, the flow obtained should have a better reflection of the true flow in the prototype than those of the 5:1 enlarged pump model.

	Prototype (blood)	Model (blood analog)
Density ρ (kg/m ³)	1055	1070
Kinematic Viscosity ν (m ² /s)	0.33x10 ⁻⁵	0.85x10 ⁻⁵
Head H (mmHg)	120	30
Power P (W)	1.33	4.176
Power (motor) (W)	4.447	126
Torque T (Nm)	0.0214	0.3126
Diameter D (mm)	50	250
Speed N (rpm)	2000	200
Discharge Q (l/min)	5	82

Table 7.1 Operating parameters of the prototype pump and the 5:1 enlarged model [Ong, 2004]

	Prototype (blood)	Model (blood analog)	Percentage Difference
Reynolds Number	1.52x10 ⁶	1.47x10 ⁶	-3.3%
Head Coefficient	1.51x10 ⁻³	1.49x10 ⁻³	-1.3%
Flow Coefficient	20.00	26.24	31.2%
Power Coefficient	5.04x10 ⁻⁷	4.99x10 ⁻⁷	-1.0%

Table 7.2 Dimensionless coefficients of the prototype pump and the 5:1 enlarged model

Furthermore, in order to achieve complete flow similarity of the enlarged pump, the dimensionless surface roughness coefficient, $\frac{\delta_r}{D}$, where δ_r is absolute roughness of pump inner surface, should also be considered [Douglas et al., 1985], since the surface roughness would have some effects on the flow in the pump. However, this coefficient was not included in the enlarged pump model of Ong (2004). This could be another source of incomplete similarity between the enlarged model and prototype.

It can be seen from the measured results that incomplete similarity could cause some remarkable difference between the flow patterns of the enlarged pump model and the prototype. Therefore, more elaborate arrangements would be required to further enhance the flow similarity between the enlarged pump and the prototype. Otherwise, considering the difficulties in achieving the complete similarity of the enlarged pump, investigation should be directly carried out using the prototype. However, a more conclusive remark can only be made through a more extensive experimental investigation.

7.5 Particle Flow Angle at the Inlet and Outlet of Impeller

It has been shown earlier that the blade profile designs have significant effects on the fluid flow from the eye of pump through the impeller channels to the volute. Shock loss at the impeller blade is an important element that causes differences between the actual and theoretical mass flow rates against pressure head generated and is also correlated to the blood hemolysis level in the pump. In an ideal situation at the design point, flow relative to the impeller is assumed to enter and leave tangentially to the blade profile. This ideal condition is called shockless condition. It is evident that resistance to flow is a minimum

if the fluid enters the blade channel at an angle (β) which is same as the blade leading edge angle β_1 as demonstrated in Figure 7.7 (a). When the fluid approaches the blade at an angle which is different from β_1 , higher shock losses can always be expected. Figure 7.7(b) shows the situation when the flow is not approaching the impeller blade at the

Figure 7.7 The velocity triangles at the enter of impeller with the flow enter the impeller at different angle

designed angle (β_1) but is striking directly on the blade. Consequently, energy losses would be increased due to the turbulence generated by the impact of the fluid against the blade. At the impeller outlet, the loss is mostly caused by a high shear rate due to a low average volute velocity and high impeller outflow velocity [Stepanoff, 1948; Massey, 1979; Fox and McDonald, 1992].

The flow angles of the fluid particles at the inlet and outlet of the blade channel are thus further investigated and presented in this section. The flow angle is defined as the angle

between the relative velocity and the tangent at the circumference as shown in Figure 7.8. The tangent to the circumference (at a particular radial location) is taken as the reference line and the positive angle is defined in the anticlockwise direction from the tangent and vice versa.

Figure 7.8 The definition of the fluid particle flow angle, β

The inlet (and outlet) flow angle distributions of the fluid particles in the blade channel of the 16FB, 16SB and 8BB impellers are shown in Figures 7.9 (a) (also 7.9 (b)), 7.10 (a) (also 7.10 (b)) and 7.11 (a) (also 7.11 (b)) respectively. In these figures, the boundary of the blade channel is represented by the vertical partition line and the number of the blade channel is also indicated in the corresponding partition. Note that the X-axis of the figures shows the angular position of the blade channel starting from the tip of the cutwater (0°) and increases in anticlockwise direction. The Y-axis of the figures represents the fluid

flow angle. A long bold horizontal line is used to mark the blade leading edge angle (or trailing edge angle) and a long horizontal dashed line is used to highlight the zero degree angle, which represents the tangent to the impeller. Note that the fluid flow angle above the horizontal dashed line in the figures represents flow in the positive radial direction, i.e., from the eye to the blade channel inlet or from the blade channel outlet to the volute. The flow angle below the horizontal dashed represents flow in negative radial direction, i.e., from the blade channel inlet to the eye or from the volute to the blade channel outlet. Due to the skewness of the blade channel, the inlet angle of the 16FB and 8BB impellers starts from channel number 16 and 7, as shown in Figures 7.9 (a) and 7.11 (a) respectively. The inlet angle of the 16SB impeller and the outlet angle of the 16FB, 16SB and 8BB impellers are all presented starting from channel 1 as shown in Figures 7.10 (a), 7.9 (b), 7.10 (b) and 7.11 (b) respectively.

7.5.1 Flow Angle of Impeller 16FB

The fluid flow angles at the inlet of 16FB impeller, as shown in Figure 7.9 (a), are all in the positive radial direction. At most of the blade channels, the inlet flow angles are within the range of 25° to 60° . These angles are considered close to the blade leading edge angle, β_1 , which is 25° . Inlet flow angles of over 80° only occur at blade channels 16, 8, 10 and 15. However, there are only a few fluid inlet angles more than 80° at these blade channels. As shown in Figure 7.9 (a), the fluid flow inlet angles are generally larger at the left partition boundary, which is the pressure side of the blade channel, while at the right boundary, the suction side of the blade channel, the inlet flow angles are generally close to the blade leading edge angle. This indicates that shock losses are larger at the pressure side than those at the suction side of the 16FB blade channel inlet.

At the outlet of impeller 16FB as shown in Figure 7.9 (b), the flow angles are positive at most of the channels. Negative flow angle can only be observed at a few locations in blade channels 1, 8, 9, 15 and 16. This verified again that negative flow at the impeller outlet was more prone to occur near the tips of the cutwater and splitter plate due to the high pressure. At the pressure and suction sides of the blade channel as shown in Figure 7.9 (b), the differences between the outlet flow angles and the blade trailing edge angle (90°) are basically less than 20° . This reveals that the outflow of the 16FB impeller was quite smooth near the blade trailing edge. As shown in Figure 7.9 (b), the outlet flow angles have great variation at the middle of the blade channel. The largest and smallest outlet flow angles are about 160° and 40° respectively. However, because the vectors with the largest and smallest outlet flow angles are not close to the blade tips and small in magnitude, the fluid particles with these outlet angles would not collide severely with the impeller blades.

7.5.2 Flow Angle of Impeller 16SB

It can be seen that the inlet flow angles of the 16SB impeller as shown in Figure 7.10 (a) are varying from 25° to 80° and have an average of about 45° . The inlet flow angles are consistently smaller than the blade leading edge angle, which is 90° , except for a flow angle at blade channel 3 has risen up to about 150° , which should be due to the measurement uncertainty. The difference between the inlet flow angle and the blade leading edge angle would induce shock at the entrance of the blade channel. It can be seen in Figure 7.10 (a) that negative flows occur at the inlet of most of the blade channels except channels 3, 4, 10, 11 and 12. All the negative flows are located at the suction side of the blade channel. As discussed in Section 7.2, these negative flows are caused by the massive vortical flow in the straight blade channel. At the pressure side of the blade

channel, the inlet flow angles are all positive which indicate that the flows near the pressure side of the blade channel inlet are stable and flow towards the volute.

The outlet flow angles of the 16SB channel have shown great diversity in Figure 7.10 (b). The flow angles in blade channels 2, 3 and 4 have less variation with a range from 65° to 135° , just around the blade trailing edge angle (90°). It can be observed in these three channels that the outlet flow angles are increasing linearly from the pressure side to the suction side. In blade channels 1, 12 and 13, the outlet flow angles are small at both the pressure and suction sides, which are about 65° , while the maximum outlet flow angles, which are nearly 180° , occur at the middle of the channel. This indicates that in blade channels 1, 12 and 13, the flow is in the positive radial direction at the pressure and suction sides and shifts to nearly tangential direction at the middle of the channel. In blade channels 5, 6, 7, 8, 9, 10, 11, 14, 15 and 16, the outlet flow angles fluctuate sharply from -180° to 180° . The vast outlet flow angle variation could be attributed to the disturbed flow in the 16SB channel. As shown in Figure 7.10 (b), at the left and right boundaries of the blade channels, the differences between the outlet flow angles and blade trailing edge angle are ranging from 20° to 40° , which are larger than those of the 16FB model. This indicates a higher probability of collision of fluid with the blade trailing tip of the 16SB model.

7.5.3 Flow Angle of Impeller 8BB

The inlet flow angles of the 8BB impeller, as shown in Figure 7.11 (a), are basically distributed around the blade leading edge angle, which is 20° , with small variation ranging from 13° to 45° . These inlet flow angles would render a smooth inlet flow to the 8BB impeller. As discussed earlier, due to the different geometries of the 8BB blade

channels, the flow patterns in blade channels 1, 3, 5 and 7 are different from those in channels 2, 4, 6 and 8. As shown in Figure 7.11 (a), at the inlet of blade channels 8, 2, 4 and 6, flow angles basically decrease from the pressure side to the suction side except for some minor variations as can be observed at channels 8 and 4. Meanwhile, the distribution of the inlet flow angle demonstrates an approximate “W” shape from the pressure side to the suction side of blade channels 7, 1, 3 and 5. As shown in Figure 7.11 (a), the inlet flow angles at the left hand side of the partition are in general higher than those at the right hand side, where the inlet flow angles are about the same as the blade leading edge angle. This indicates that the inlet flow shocks of the 8BB impeller are more serious at the pressure side (left hand side) than those at the suction side (right hand side).

At the outlet of the 8BB impeller, as shown in Figure 7.11 (b), the flow angles are quite uniform from the pressure to suction sides but have more variations near the suction side. It can be found that the outlet flow angles ranged from 0° to 45° in most of the blade channel outlet space and are quite close to the blade trailing edge angle, which is 25° . At the suction side of the blade channel, the outlet flow angles differs apparently from the blade trailing edge angle except channel 1 and 7 as shown in Figure 7.11 (b). However, the vectors with these fluctuating outlet flow angles basically have low magnitudes as shown earlier in Figure 7.5 (b), the impact of the fluid at these locations on the blade tip are supposed to be mild.

7.5.4 Comparison of Flow Angle of the Different Impellers

It can be seen from the above discussion that fluctuations of flow angle at the inlet of the impeller is generally milder than those at the impeller outlet. The average inlet flow angle is about 40° to 50° in the 16FB and 16SB impellers and is about 20° in the 8BB impeller.

This is because there are 16 blades in the 16FB and 16SB impellers while the 8BB impeller has 8 blades. The 16 blades will create more blockage of the inlet flow to the impeller because they occupy a larger cross sectional area at the impeller inlet than those of the 8-blade impeller. Therefore, under the same flow rate through the pump and rotating speed of the impeller, the average inlet flow angle in the 8BB impeller would be smaller than those of the 16FB and 16SB impellers and thus closer to the blade leading edge angle.

The difference between the average inlet flow angle and the blade leading edge angle is about 40° of the 16SB impeller, which is the largest among the three impeller profiles studied. While the 16FB impeller has a difference of about 25° , the difference is thus the smallest for the 8BB impeller, in which the average inlet flow angle is only slightly larger than the blade leading edge angle. Therefore, the shock losses at the inlet of the 8BB impeller would be the smallest among the three impeller profiles. Further reduction of the shock losses at the inlet of the 16FB impeller could be achieved by increasing the blade leading edge angle by about 25° .

The outlet flow angles of the 16SB impeller are distributed about a complete 360° ranging from -180° to 180° in most of the blade channels. The 8BB impeller has a uniform outlet flow angle distribution at the pressure side and the uniformity has been maintained until the location close to the suction side. The 16FB has a wavy outlet flow angle distribution, but the variation of the angle is confined in an interval ranging from 40° to 160° in most of the blade channels. Therefore, both the 8BB and 16FB impellers have smoother outlet flow angle distributions than those of the 16SB impeller. Negative outlet flow angle can be seen at all the three impeller profiles. In the 16SB impeller, the negative flow angle

occurs in 10 of the 16 blade channels, while the 16FB and 8BB impellers have 5 and 3 blade channels that contain the negative outlet flow angle respectively. This indicated that the flow at the outlet of the 16SB blade channels has the most severe disturbance among the three impeller profiles. The difference between outlet flow angle and the blade trailing edge angle of the 16FB impeller is less than 20° at both the pressure and suction sides of the blade channel, which is lower than those of the 8BB and 16SB impellers. Thus the collision of the outlet flow with the rear tips of the 16FB impeller blades should be the smallest among the three impeller profiles.

7.6 A Brief Summary

In this chapter, the measurement results of the flow in between the impeller blades were presented and compared with those of the numerical simulation for the 1:1 pump model. Close agreements between the numerical simulation and the experiment have been found in the comparison. The agreements have reasonably validated the numerical simulation results of the pump. On the other hand, remarkable differences between the flow in the 5:1 enlarged pump model and 1:1 pump model were observed. These different flow characteristics might be due to the incomplete similarity between the enlarged pump model and prototype. Based on the experimental results, flow angle at the inlet and outlet of the impeller was investigated. The 16SB impeller was found to have the most severe shock loss at the inlet and the highest disturbance at the outlet while 16FB impeller was considered to have the mildest collision between the outlet fluids and the blade rear tip.

Figure 7.9 (a) Inlet particle flow angle distributions of the 16FB impeller

Figure 7.9 (b) Outlet particle flow angle distributions of the 16FB impeller

Figure 7.10 (a) Inlet particle flow angle distributions of the 16SB impeller

Figure 7.10 (b) Outlet particle flow angle distributions of the 16SB impeller

Figure 7.11 (a) Inlet particle flow angle distributions of the 8BB impeller

Figure 7.11 (b) Outlet particle flow angle distributions of the 8BB impeller

CHAPTER 8

CONCLUSIONS AND FUTURE WORKS

8.1 Conclusions

In this study, flows in the Kyoto-NTN centrifugal blood pump with different impeller blade profiles were investigated. Three different impellers with 16 forward-bending blades, 16 straight blades and 8 backward-bending blades respectively were studied in order to compare the effects of blade profile on the flow in the pump. Commercial CFD software package, FLUENT, was applied in the study and measurements of the flow in between the impeller blades were carried out using laser Doppler anemometer. Both the numerical and experimental pump models were built in 1:1 scale according to the prototype and the same operating condition of the pump was applied except that the shaft was still employed in driving the experimental pump model instead of the magnetically suspended impeller. Through the comparison between the numerical and measured results, it was found that the experimental results exhibited similar flow characteristics as those found in the numerical simulation, although the numerical simulation results provided much more detailed information of the flow patterns in the pump models than those of the experiment. The agreements reasonably validated the numerical simulation of the inner flow of the pump. It is worth to emphasize that this is a different but important measurement since we can scarcely find similar measurements in literature.

PRESSURE

Under the pump flow rate of 5 l/min and impeller rotating speed of 2000 rpm, the 8BB model generates a pressure head of 98.5 mmHg, which is the lowest among the three pump models, while the 16FB model generates a pressure of 109.4 mmHg and the 16SB model produces the highest pressure head of 113.5 mmHg. However, the inner static pressure distributions of the three pump models have similar patterns. The static pressure in the blade channel increases gradually in the positive radial direction and the static pressure in the volute passage increases tangentially and is generally higher than that in the blade channel. The highest static pressure in the pump occurs near the tips of the splitter plate and cutwater in the volute due to the flow stagnation there.

EFFECTS OF THE DOUBLE VOLUTE

The double volute has dominant effects on the flow in blood pump. Symmetrical flow among the blade channels occurred in all the three pump models. The symmetrical flow is expected and induced by the symmetrical pressure distribution in the double volute passages. The symmetrical flow in the blade channel will result in good balance of the radial force exerted by the flow and pressure on the impeller and subsequently ease the complicated control of the high-speed rotating magnetically suspended impeller.

EFFECTS OF THE IMPELLER BLADE PROFILE

The impeller blade profile has remarkably affected the flow pattern in the blade channel. Both the 16FB and 16SB impeller would cause circulating flow in the blade channels and the circulating flow is more prone to occur in the blade channels that are

near the tips of the splitter plate and cutwater due to the peak pressure in these regions. Furthermore, the 16SB model has more prevalent and stronger circulating flow in the blade channels than the 16FB model and the stronger circulating flow of the 16SB pump model would even extend to the channel inlet and cause negative flow there. Meanwhile, the flow in the blade channel of 8BB pump model is smoother than those of the 16FB and 16SB models and no circulating flow occurs in the blade channel of 8BB pump model.

Under the same impeller rotating speed of 2000 rpm and flow rate of 5 l/min, the 16FB, 16SB and 8BB pump models produce the peak scalar shear stresses of 1900 Pa, 950 Pa and 930 Pa respectively. The comparison of the mass weighted shear stress distributions among the three pump models indicated that the 8BB impeller produces the lowest shear stress to the fluid, while the 16FB impeller induces the highest shear stress. However, for all the three pump models, the scalar shear stresses are lower than 90 Pa in most regions of the flow field. High shear stress regions with more than 250 Pa only occupy very small area (less than 1%) of the flow fields. These high shear stress areas basically occur at the inlet and outlet of the blade channel, the beginning of the two volute passages and especially the regions around the tips of the splitter plate and cutwater.

FLOW IN THE GAP

The different impeller profiles do not produce remarkably different gap flow. The mass flow rates through the gap of the three models ranges from 25% to 28% of the pump flow rate. The flow in the pump gap mainly consists of radial and tangential velocity components. The tangential velocities are distributed linearly in the gap and

the radial velocity profiles in the gap is parabolic in shape and head towards the eye. The radial velocity magnitudes are mainly affected by and varying with the pressure at different angular positions in the volute. Two peak gap radial velocity regions are located at some angular distance before the tips of the splitter plate and cutwater. The wall shear stresses in the gaps of the models are found to be much lower than the threshold of inducing massive blood hemolysis but high enough to prevent the blood cells from being aggregated on the inner surface of the gap.

ASSESSMENT OF THE THREE PUMP MODELS

Through the comparisons of the three impellers in this study, it can be found that there are advantages and disadvantages for all the three impellers. The 16FB and 16SB pump models produce higher pressure heads than the 8BB model and therefore could be more favorable in attaining the physiologically required flow of human body. However, 16FB would induce the highest shear stress to the flow, while the 16SB model would have low power efficiency due to the massive vortical flows in the blade channels. On the other hand, the 8BB model induces the lowest shear stress to the blood, but it has also produced the lowest pressure and flow rate. The 16SB impeller, which induces less shear stress than the 16FB impeller with relative high pressure head, could be the best choice among the three pumps investigated. Furthermore, besides the ease of fabrication, the vortex induced in the blade channel would produce constant washout on the inner surface of the blade channel and therefore reduce the chances of the thrombus formation.

COMPARISON BETWEEN THE 5:1 AND 1:1 PUMP MODELS

There are significant differences between the flow patterns in the blade channel of the 5:1 and 1:1 pump models, although both the 5:1 and 1:1 models have axis symmetrical flow in the blade channel under the effect of the double volute design. The differences should be due to the 5:1 pump model not attaining a complete similarity to the prototype. Therefore, it is emphasized that a complete dynamic similarity should be strictly observed and implemented when measurements have to be performed on the enlarged pump model. Otherwise, considering the difficulty in achieving the complete similarity between the enlarged pump model and prototype, studies should be carried out using a pump model with the same dimensions of the prototype.

8.2 Future Works

Detailed flow characteristics of the Kyoto-NTN centrifugal blood pump have been found by the numerical simulation and measurement in the present investigation. The results would be useful in improving the pump design. However, there is still room for improvement and the following future works are recommended.

The impeller blade profile has significant effects on the flow of the pump. It has been found that there is no single impeller blade profile in this study having the best performance in all aspects over the others. Furthermore, the tips of the splitter plate and cutwater in the volute are two major contributors to the high shear stress in the pump. Therefore, further pump design improvements should be focused on the geometries of the impeller blade and the volute. In addition, future design should also

consider minimizing the pump overall dimensions to benefit the young and also small built Asian patients as compared to Europeans and North Americans.

The current measuring volume of LDA should be further reduced in order to perform measurements with a higher resolution. This is especially essential when the dimension of the LDA measuring volume is comparable to the flow field. With improved techniques, the velocity at different planes in the blade channels can be measured and thus, much more detailed information about the flow can be obtained.

Blood analog was used as the working fluid in this study based on the understanding that blood can be treated as Newtonian fluid when flowing in a sufficiently large channel. In future works, the current finding of the flow within the pump can be further verified by using blood as the working fluid. Ultrasound Doppler anemometer (UDA) is suggested to overcome the constraints in performing measurements due to the photic properties of blood. Furthermore, blood hemolysis test can be carried out using human or bovine blood. The measurement results can be used to validate the numerical prediction of blood hemolysis level and will improve the pump design and subsequently the pump performance.

REFERENCES

Abouawdi N.L. and Frazier O.H., 1992, *The HeartMate: A left ventricular assist device as a bridge to cardiac transplantation*. Transplantation Proceedings; 24(5): 2002–2003.

Akamatsu T. and Tsukiya T., 1998, *Development of a centrifugal blood pump with magnetically suspended impeller and the related fluid mechanical problems*. Sadhana - Academy Proceedings in Engineering Sciences; 23 (5): 597-603.

Akamatsu T., Nakazeki T. and Itoh H., 1992, *Centrifugal blood pump with a magnetically suspended impeller*. Artif Organs; 16(3): 305-308.

Akamatsu T. and Nakazeki T., 1994, *Recent development of a centrifugal blood pump with a magnetically suspended impeller*. Proceedings of the third JSME-KSME Fluids Engineering Conference, Sendai, Japan; 305-308.

Akamatsu T. and Tsukiya T., 1997, *Development of the centrifugal blood pump with magnetically suspended impeller and the related fluid mechanical problems*. Proceedings of the Seventh Asian Congress of Fluid Mechanics; 7-10.

Akamatsu T., Tsukiya T., Nishimura K., Park C.H. and Nakazeki T., 1995, *Recent studies of the centrifugal blood with a magnetically suspended impeller*. Artif Organs; 19(7): 631-634.

Akutsu T. and Kolff W.J., 1958, *Permanent substitutes for valves and hearts*. Trans ASAIO; 4: 230-235.

Allen G.S., Murray K.D. and Olsen D.B., 1997, *The importance of pulsatile and non-pulsatile flow in the design of blood pumps*. Artif Organs; 21(8): 922-928.

Apel J., Paul R., Klaus S., Siess T. and Reul H., 2001, *Assessment of hemolysis related quantities in a microaxial blood pump by computational fluid dynamics*. Artif Organs; 25(5): 341-347.

Araki K., Taenaka Y., Masuzawa T., Inoue K., Nakatani T., Kinoshita M., Akagi H., Baba Y., Matsuo Y., Sakaki M., Anai H. and Takano H., 1993, *A new visualization study of centrifugal blood pumps developed for long term usage*. Artif Organs; 17(5): 307-311.

Arvand A., Hormes M. and Reul H., 2005, *A validated computational fluid dynamics model to estimate hemolysis in a rotary blood pump*. Artif Organs; 29(7): 531-540.

References

- Asztalos B., Yamane T. and Nishida M., 1999, *Flow visualization analysis for evaluation of shear and recirculation in a new closed-type, monopivot centrifugal blood pump*. *Artif Organs*; 23(10): 939-946.
- Badner N., Murkin J. and Lok P., 1992, *Differences in pH management and pulsatile/nonpulsatile perfusion during cardiopulmonary bypass to not influence renal function*. *Anesth Analg*; 75(5): 696-674.
- Barenblatt G.I., 1996, *Scaling, self-similarity, and intermediate asymptotics*. Cambridge University Press; 39-63.
- Bauermeister U., Affeld K. and Berger E., 1982, *The 'Tea spoon pump': a new centrifugal blood pump*. *ASAIO Abstracts*; 11:1.
- Bludszuweit C., 1994, *A theoretical approach of the prediction of hemolysis in centrifugal blood pumps*. PhD Thesis, University of Strathclyde, Glasgow.
- Bludszuweit C., 1995, *Model for a general mechanical blood damage prediction*. *Artif Organs*; 19: 583-589.
- Brown C.H., Lemuth R.F., Hellums J.D., Leverett L.B. and Alfrey C.P., 1975(a), *Response of human platelets to shear stress*. *Trans Am Soc Artif Intern Organs*; 21: 35-38.
- Brown C.H., Leverett L.B., Lewis C.W., Alfrey C.P. and Hellums J.D., 1975(b), *Morphological biochemical and functional changes in human platelets subjected to shear stress*. *J Lab Clin Med*; 86: 462-471.
- Burgreen G.W., Antaki J.F., Wu Z.J. and Holmes A.J., 2001, *Computational fluid dynamics as a development tool for rotary blood pumps*. *Artif Organs*; 25(5): 336-340.
- Burke D.J., Burke E., Parsaie F., Poirier V., Butler K., Thomas D., Taylor L. and Maher T., 2001, *The HeartMate II: design and development of a fully sealed axial flow left ventricular assist system*. *Artif Organs*; 25(5): 380-385.
- Butler K.C., Dow J.J., Litwak P., Kormos R.L. and Borovetz H.S., 1999, *Development of the Nimbus/University of Pittsburgh innovative ventricular assist system*. *Ann Thorac Surg*; 68: 790-794.
- Butler K.C., Wampler R., Griffith B., Antaki J., Kormos R. and Borovetz H., 1991, *Development of an axial flow LVAS*. *Proceedings of the International Workshop on Rotary Blood Pumps*, Vienna, University of Vienna; 148-153.

References

Chan W.K., Akamatsu T., and Li H.D., 2000, *Analytical investigation of leakage flow in disk clearance of a magnetically suspended centrifugal impeller*. Artif Organs; 24(9): 734-742.

Chatel D., 1996, *Concept of totally biological internal coating for newly shaped artificial ventricles*. Artif Organs; 20(7): 814-817.

Chiang B.Y., Burns G.L., Pantalos G.M., Dew P.A., Kinoshita M., White R.K., Mohammed S.F. and Olsen D.B., 1991, *Microbially infected thrombus in animals with total artificial hearts*. Trans. ASAO; 37: M256-257.

Chua L.P. and Akamatsu T., 2000, *Measurement of gap pressure and wall shear stress of a blood pump model*. Medical Engineering & Physics; 22: 175-188.

Chua L.P., Ong K.S., Yu C.M.S. and Zhou T.M., 2002, *Gap velocity measurements of the blood pump model*. Artif Organs; 26(8): 682-694.

Chua L.P., Ong K.S., Yu C.M.S. and Zhou T.M., 2003, *Leakage flow measurements in a bio-Centrifugal ventricular assist device model*. Artif Organs; 27(10): 942-959.

Chua L.P., Song G.L., Lim T.M. and Zhou T.M., 2006, *Numerical analysis of the inner flow field of a biocentrifugal blood pump*. Artif Organs; 30(6): 467-477.

Colantuoni G. and Hellums J.D., 1977, *The response to human platelets at short exposure time*. Trans ASAO; 23:626-631.

Cooley D.A., Liotta D., Hallman G.L., Bloodwell R.D., Leachman R.D. and Milam J.D., 1969, *Orthotopic cardiac prosthesis for two-staged cardiac replacement*. Am J Cardiol; 24:723-730.

Copeland J.G., Arabia F.A., Banchy M.E., Sethi G.K., Foy B., Long J., Kormos R.L. and Smith R.G., 1998, *The Cardio West total artificial heart bridge to transplantation: 1993-1996 national trial*. Ann Thorac Surg; 66: 1662-1669.

Daitoh N., 1978, *The effects of nonpulsatile perfusion on the renal energy metabolism*. Arch J Chir; 47(5): 52-56.

Dapper F., Neppl H., Wozniak G., Strube I., Zickmann B., Hehrlein F. and Neuhof H., 1992, *Effects of pulsatile and nonpulsatile perfusion mode during extracorporeal circulation – a comparative clinical study*. J Thorac Cardiovasc Surg; 40(6): 345-351.

De Wachter D. and Verdonck P., 2002, *Numerical calculation of hemolysis levels in peripheral hemodialysis cannulas*. Artif Organs; 26: 576-582.

References

Dieck R.H., 1992, *Measurement uncertainty methods and applications*. Instrument Society of America; 37-112.

Douglas J.F., Gasiorek J.M. and Swaffield J.A., 1985, *Fluid mechanics*, 2nd edition, Longman Scientific & Technical.

Durst F., Melling A. and Sender J., 1990, *Photodetectors for laser-Doppler measurement*. Experiment in Fluids; 9: 13-21.

FLUENT manual 6.2, 2005, ANSYS Inc. Canonsburg, PA, US.

Fox R.W. and McDonald A.T., 1992, *Introduction to fluid mechanics*, 4th edition, John Wiley and Sons.

Giersiepen M., Wurzinger L.J., Opitz R. and Reul H., 1990, *Estimation of shear stress-related blood damage in heart valve prostheses – intro comparison of 25 aortic valves*. Int J Artif Organs; 13: 300-306.

Golding L.R., Jacobs G., Murakami T., Takatani S., Valdes F., Harasaki H. and Nosé Y., 1980, *Chronic nonpulsatile blood flow in an alive, awake animal 34-day survival*. Trans Am Soc Artif Intern Organs; 26: 251-255.

Golding L.R., Murakami G., Harasaki H., Takatani S., Jacobs G., Yada I., Tomita K., Yozu R., Valdes F., Fujimoto K., Kolke S. and Nosé Y., 1982, *Chronic nonpulsatile flow*. Trans ASAIO; 28: 81-86.

Goldstein D.J., Mehmet C. and Rose E.A., 1998, *Implantable left ventricular assist devices*. New England Journal of Medicine; 339(21): 1522-1533.

Hogness J.R. and Antwerp V., 1991, *Clinical effectiveness and need for long-term circulatory support*. The Artificial Heart: Prototypes, Policies, and Patients, Washington, D.C., National Academy Press; 65-84.

Hoshi H., Shinshi T. and Takatani S., 2006, *Third-generation blood pumps with mechanical noncontact meglev bearings*. Artif Organs; 30(5): 324-338.

Ikeda T., Yamane T., Orita T. and Tateishi T., 1996, *Quantitative visualization study of flow in a scaled-up model of a centrifugal pump*. Artif Organs; 20(2): 132-138.

Ishitoya H., Kawamura M., Linneweber J., Motomura T., Ichikawa S., Nishimura I., Glueck J., Shinohara T. and Nosé Y., 2002, *Titania gel reduces thrombin generation*. Artif Organs; 26(11): 959-963.

References

Kader B., 1993, *Temperature and Concentration Profiles in Fully Turbulent Boundary Layers*. Int J Heat Mass Transfer; 24(9): 1541-1544.

Kantrowitz A., Wasfie T., Freed P.S., Rubenfire M., Wajszczuk W. and Schork M.A., 1986, *Intraaortic balloon pumping 1967 through 1982: analysis of complications in 733 patients*. Am J Cardiol; 57:976-983.

Kaplon R.J., Oz M.C., Kwiatkowski P.A., Levin H.R., Shah A.S., Jarvik R.K. and Rose E.A., 1996, *Miniature axial flow pump for ventricular assistance in children and small adults*. J Thorac Cardiovasc Surg; 111:13-18.

Kawahito K., Adachi H. and Ino T., 2000., *Platelet activation in the Gyro C1E3 centrifugal pump: Comparison with the Terumo Capiox and the Nikkiso HPM-15*. Artif Organs; 24(11): 889-892.

Kawahito K. and Nosé Y., 1997, *Hemolysis in different centrifugal blood pumps*. Artif Organs; 21(4):323-326.

Koller T. and Hawrylenko A., 1967, *Contribution to the in vitro testing of pumps for extracorporeal circulation*. J Thorac Cardiovasc Surg; 54:22-29.

Kumbasar S.D., Semiz E., Ermis C., Yalcinkaya S., Deger N., Pamir G. and Oral D., 1998, *Effect of intraaortic balloon counter-pulsation on QT dispersion in acute anterior myocardial infarction*. Int J Cardiol; 65:169-172.

Kung R.T., Yu L.S., Ochs B.D., Parnis S.M. Macris M.P. and Frazier O.H., 1995, *Progress in the development of the ABIOMED total artificial heart*. ASAIO J; 41:M245-248.

Kunov M.J., Steinman D.A. and Ethier C.R., 1996, *Particle volumetric residence time calculations in arterial geometries*. Trans ASME J Biomecha Eng; 118: 158-164.

Launder B.E. and Spalding D.B., 1974, *The numerical computation of turbulent flows*. Comput Methods Appl Mech Eng. 3:269-289.

Loree H.M., Bourque K., Gernes D.B., Richardson J.S., Poirier V.L., Barletta N., Fleischli A., Foiera G., Gempp T.M., Schoeb R., Litwak K.N., Akimoto T., Kameneva M., Watach M.J. and Liwak P., 2001, *The HeartMate III: Design and in vivo studies of a maglev centrifugal left ventricular assist device*. Artif Organs; 25(5): 386-391.

References

MacLauhlin D.K., and Tiederman W.G., 1972, *Biasing correction for individual realization for the laser anemometry measurements in turbulent flow*. Physics of Fluids; 16(2):2082-2088.

Maher T.R., Butler K.C., Poirier V.L. and Gernes D.B., 2001, *HeartMate Left Ventricular Assist Devices: A Multigeneration of Implanted Blood Pumps*. Artif Organs; 25: 422-427.

Markatos N.C., 1986, *The mathematical modeling of turbulent flows*. Appl Math Modeling; 10:190-220.

Marlinski E., Jacobs G., Deirmengian C. and Jarvik R., 1998, *Durability testing of components for the Jarvik 2000 completely implantable axial flow left ventricular assist device*. ASAIO J; 44:M741-744.

Maruyama L., Nishida M., Yamane T., Oshima I., Adachi Y. and Masuzawa T., 2006, *Hemolysis resulting from surface roughness under shear flow conditions using a rotational shear stressor*. Artif Organs; 30(5): 365-370.

Massey B.S., 1979, *Mechanics of fluids*. New York: Van Nostrand Reinhold Co.

Massiello A., Kiraly R., Butler K., Himley S., Chen J.F. and McCarthy P.M., 1994, *The Cleveland Clinic-Nimbus total artificial heart. Design and in vitro function*. J Thorac Cardiovasc Surg; 108:412-419.

Massin E.K., 1996, *Evaluation and treatments of end-stage heart disease*. In: Support and replacement of failing heart. Eds.: Frazier O.H., Macris M.P. and Radovancevic B., Philadelphia. Lippincott-Raven Publishers: 37-58.

Masuzawa T., Tsukiya T., Endo S., Tatsumi E., Taenaka Y., Takano H., Yamane T., Nishida M., Asztalos B., Miyazoe Y., Ito K., Sawairi T. and Konishi Y., 1999, *Development of design method for a centrifugal blood pump with a fluid dynamic approach: Results in hemolysis tests*. Artif Organs; 23(8): 757-761.

Mazumdar J.N., 1992, *Biofluid mechanics*. World Scientific.

McCarthy P.M. and Sabi K., 1994, *Implantable circulatory support devices as a bridge to heart transplantation*. Sem Thorac Cardiovasc Surg; 6(3): 174-180.

Mesana T.G., 2004, *Rotary blood pumps for cardiac assistance: A "Must"? Artif Organs*; 28(2): 218-225.

References

Miyazoe Y., Sawairi T., Ito K., Konishi Y., Yamane T., Nishida M., Masuzawa T., Takiura K. and Taenaka Y., 1998, *Computational fluid dynamics analyses to establish design process of centrifugal blood pumps*. *Artif Organs*; 22(5): 381-385.

Nakazawa T., Benkowski R., Makinouchi K., Takami Y., Ohtsubo S., Glueck J., Kawahito K., Sueoka A., Schmallegger H., Schima H., Wolner E. and Nose Y., 1998, *Recent advances in the gyro centrifugal ventricular assist device*. *ASAIO J*; 44(1): 94-97.

Nichols W.W. and O'Rourke M.F., 1990, *McDonald's blood flow in arteries: theoretic, experimental and clinical principles*. Philadelphia: Lea & Febiger.

Nishida M., Asztalos B., Yamane T., Masuzawa T., Tsukiya T., Endo S., Taenaka Y., Miyazoe Y., Ito K. and Konishi Y., 1999, *Flow visualization study to improve hemocompatibility of a centrifugal blood pump*. *Artif Organs*; 23(8): 697-703.

Nojiri C., Kijima T., Maekawa J., Horiuchi K., Kido T., Sugiyama T., Mori T., Sugiura N., Asada T., Shimane H., Nishimura K., Ban T., Akamatsu T., Ozaki T., Ito H., Suzuki M. and Akutsu T., 1997, *More than 1 year continuous operation of a centrifugal pump with a magnetically suspended impeller*. *ASAIO J*; 43:M548-552.

Nosé Y., 1991(a), *Toward a totally implantable artificial heart: Development status at Cleveland Clinic*. *Artificial Heart 3: Heart Replacement*. Springer Verlag; 147-167.

Nosé Y., 1991(b), *Intrathoracic cardiac prosthesis: is it really not clinically acceptable*. *Artif Organs*; 15(3):161-163.

Nosé Y., 1993, *Changing trends in circulatory support technology*. *Artificial Heart 3: The Development of Biomation in the 21st Century*. Harwood Academic. 1-9.

Nosé Y., 1998, *Design and development strategy for the rotary blood pump*. *Artif Organs*; 22(6): 438-446.

Nosé Y. and Furukawa K., 2004, *Current Status of the Gyro Centrifugal Blood Pump-Development of the permanently Implantable Centrifugal Blood Pump as a Biventricular Assist Device (NEDO Project)*. *Artif Organs*; 28(10): 953-958.

Nosé Y., Ohtsubo S. and Tayama E., 1997, *Therapeutic and physiological artificial heart: Future prospects*. *Artif Organs*; 21(7): 592-596.

Ohara Y., Makinouchi K., Glueck J., Sutherland B., Shimono T., Naito K., Tasai K., Orime Y., Takatani S. and Nosé Y., 1994, *Development and evaluation of antithrombogenic centrifugal pump: the Baylor C-Gyro pump eccentric inlet port model*. *Artif Organs*; 21: 673-679.

References

Oku T., Harasaki H., Smith W. and Nosé Y., 1988, *Hemolysis, a comparative study of four nonpulsatile pumps*. Trans ASAIO; 34(3): 500-504.

Ong K.S., 2004, *Measurements in a 5:1 enlarged bio-centrifugal pump model*. Ph.D. thesis, Nanyang technological university.

Pantalos G.M., 1993, *Artificial heart: past, present and future*. Artif Organs; 17(10): 826-827.

Park C.H., Nishimura K., Akamatsu T., Tsukiya T., Matsuda K. and Ban T., 1996, *A new magnetically suspended centrifugal blood pump: in vitro and preliminary in vivo assessment*. Artif Organs; 20(2): 128-131.

Paul R., Apel J., Klaus S., Schügner F., Schwindke P. and Reul H., 2003, *Shear stress related blood damage in laminar couette flow*. Artif Organs; 27(6): 517-529.

Perri L., *New Heart Assist Device Designed as Permanent Implant For End-Stage Heart Failure*. <http://www.psu.edu/ur/heartdevices/release3.htm>

Philips W.M., 1993, *The artificial heart: History and status*. Journal of Biomedical Engineering; 115(555): 429-436.

Pierce W.S., Sapirstein J.S. and Pae W.E.J., 1996, *Total artificial heart: from bridge to transplantation to permanent use*. Ann Thorac Surg; 61: 342-346.

Pinotti M. and Paone N., 1996, *Estimating Mechanical blood trauma in centrifugal blood pump: laser Doppler anemometer measurements of the mean velocity field*. Artif Organs; 20(6): 546-552.

Ramstack J.M., Zuckerman L., Mockros L.F., Caprini J.A. and Vagher J.P., 1978, *Effect of platelet count and hematocrit on the coagulability of heparinized blood*. Res Commun Chem Pathol Pharmacol; 21(2): 327-342.

Richenbacher W.E. and Pierce W.S., 1997, *Assisted circulation and the mechanical heart*. In: Braunwald E, ed. Heart Disease: A textbook of cardiovascular medicine. Philadelphia: W.B. Saunders Company; 534-547.

Rodi W., 1984, *Turbulence models and their application in hydraulics*. 2nd edition. Netherlands; IAHR, 9-46.

Sakaki M., Taenaka Y., Tatsumi E., Nakatani T., Kinoshita M., Akagi H., Masuzawa T., Matsuo Y., Inoue K., Baba Y., Anai H., Araki K., Takano H. and Kasugai T., 1992, *Pulmonary function in a nonpulsatile pulmonary circulation*. ASAIO J; 38(3): 366-369.

Sakuma I., Fukui Y. and Dohi T., 1996, *Study of secondary flow in centrifugal blood pumps using a flow visualization method with a high-speed video camera*. *Artif Organs*; 20(6): 541-545.

Sallam A.M. and Hwang N.H.C., 1984, *Human red blood cell hemolysis in a turbulent shear flow, contribution of Reynolds shear stress*. *Biorheology*; 25: 783-797.

Schöb R., 2002, *Centrifugal pump without bearings or seals*, *World Pumps*; July: 34-37.

Scholz K.H., Figulla H.R., Schweda F., Smalling R.W., Hellige G., Kreuzer H., Aboul H.W. and Wampler R.K., 1994, *Mechanical left ventricular unloading during high risk coronary angioplasty: first use of a new percutaneous transvalvular left ventricular assist device*. *Cathet Cardiovasc Diagn*; 31:61-69.

Shumakov V.I., 1993, *The total artificial heart: New aspects in prophylaxis and therapy of infections and blood coagulation disorders*. *Artificial Hearts 3: The Development of Biomedicine in the 21st Century*. Harwood Academic; 35-45.

Siegel-Itzkovick J., 2000, *Israeli surgeons implant first permanent artificial ventricle*. *BMJ*; 321(7258): 399.

Souhami R.L. and Moxham J., 1994, *Textbook of medicine*., London: Churchill Livingstone.

Stepanoff A.J., 1948, *Centrifugal and axial flow pumps*. John Wiley and Sons.

Sugita Y., Golding L.R., Jacobs G., Harasaki H., Yozu R., Sato N., Fujimoto L.K., Morimoto T., Snow J., Olsen E., Smith W., Murabayashi S., Kambic H., Kiraly R. and Nosé Y., 1984, *Comparison of osmotic and body fluid balance in chronic nonpulsatile biventricular bypass (NPBVB) and total artificial heart (TAH) experiments*. *Transactions - American Society for Artificial Internal Organs*; 30: 148-154.

Sutera S.P., Croce P. and Mehrjardi M., 1972, *Hemolysis and subhemolytic alterations of human RBC induced by turbulent shear flow*. *Trans ASAIO*; 18: 583-589.

Sutera S.P. and Nowak M.D., 1988, *A programmable, computer-controlled, cone-plate viscometer for the application of pulsatile shear stress to platelets suspensions*. *Biorheology*; 25: 449-459.

References

Taguchi S., Yozu R., Iseki H., Soma Y. and Inoue T., 1988, *Effects of nonpulsatile and pulsatile right ventricular bypass on pulmonary circulation*. Trans ASAO; 34: 213-219.

Takami Y., Andrade A., Nakazawa T., Makinouchi K., Glueck J., Benkowski R. and Nosé Y., 1997(b), *Eccentric inlet port of the pivot bearing supported Gyro centrifugal pump*. Artif Organs; 21(4): 312-317.

Takami Y., Makinouchi K., Nakazawa T., Glueck J., Benkoski R. and Nosé Y., 1996, *Effect of surface roughness in hemolysis in a pivotbearing supported Gyro centrifugal blood pump (C1E3)*. Artif Organs; 20(11):1155-1161.

Takami Y., Yamane S., Makinouchi K., Glueck J. and Nosé Y., 1997(a), *Mechanical white blood cell damage in rotary blood pumps*. Artif Organs; 21(2):138-142.

Takami Y., Nakazawa T., Makinouchi K., Tayama E., Glueck J., Benkowski R. and Nosé Y., 1997(c), *Hemolytic effect of surface roughness of an impeller centrifugal blood pump*. Artif Organs; 21(7): 686-690.

Takami Y., Yamane S., Makinouchi K., Niimi Y., Sueoka A. and Nosé Y., 1998, *Evaluation of platelet adhesion and activation on materials for an implantable centrifugal blood pump*. Artif Organs; 22(9): 753-758.

Tamagawa M., Akamatsu T. and Saitoh K., 1996, *Predication of hemolysis in turbulence shear orifice flow*. Artif Organs; 20(6):553-559.

Tominaga R., Nakashima A., Fukumura F., Fukae K., Hisahara M., Morita S., Masuda M., Kohno H., Mayumi H., Kawachi Y., Yasui H. and Tokunaga, 1993, *Left ventricular assist circulation in patients with postoperation cardiogenic shock*. Artificial Heart 3: The Development of Biomation in the 21st Century. Harwood Academic; 155-163.

Treichler J., Rosenow S.E., Damm G., Naito K., Ohara Y., Mizuguchi K., Makinouchi K., Takatani S. and Nose Y., 1993, *A fluid dynamic analysis of a rotary blood pump for design improvement*. Artif Organs; 17(9): 797-808.

Tsukiya T., Akamatsu T. and Ozaki T., 1997, *Fluid dynamics design of Kyoto-NTN Magnetically Suspended Centrifugal Blood pump*. ASME FEDSM; 97-3425.

Umezu M., 1996, *Effect of surface roughness on mechanical hemolysis*. Artif Organs; 20(1): 20-29.

Wesolowski S., Fisher J. and Welch C., 1953, *Perfusion of the pulmonary circulation by nonpulsatile flow*. Surgery; 33: 370-375.

References

Whicher S.J. and Brash J.L., 1978, *Platelet-foreign surface interactions: Release of granule constituents from adherent platelets*. J biomed Mater Res; 12: 181-184.

Wieselthaler G.M., Schima H., Hiersmayr M., Pacher R., Laufer G., Noon G.P., DeBakey M., Wolner E., 2000, *First clinical experience with the DeBakey VAD continuous-axial-flow pump for bridge to transplantation*. Circulation; 101: 356-359.

Yamada T., Nishimura K., Park C.H., Kono S., Yuasa S., Tsukiya T., Akamatsu T., Matsusa K. and Ban T., 1997, *Chronic animal experiment with magnetically suspended centrifugal pump*. Artif Organs; 21(7): 635-640.

Yamane T., Asztalos B., Nishida M., Masuzawa T., Takiura K., Taenaka Y., Konishi Y., Miyazoe Y. and Ito K., 1998, *Flow visualization as a complementary tool to hemolysis testing in the development of centrifugal blood pump*. Artif Organs; 22(5): 375-380.

Yamane T., Kodama T., Yamamoto Y., Shinohara T. and Nosé Y., 2004, *Flow visualization in a centrifugal blood pump with an eccentric inlet port*. Artif Organs; 28(6): 564-570.

Yanta W.J. and Smith R.A., 1985, *Measurement of turbulent transport properties with a laser-Doppler velocimeter*. AIAA; 73: 169-179.

Yasuda T., Shimokasa K., Funakubo A. and Fukui Y., 2000, *Effect of wall hardness on hemolysis*. Artif Organs; 24(8): 614-617.

Yeleswarapu K.K., Antaki J.F., Kamenewa M.V. and Rajagopal K.R., 1995, *A mathematical model for shear-induced hemolysis*. Artif Organs; 19: 576-582.

Yu S.C.M., Chua L.P. and Leo H.L., 2001, *The angle-resolved velocity measurement in the impeller passages of a model biocentrifugal pump*. Proc Instn Mech Engrs; 215(C): 547-568.

Zhang J., Gellman B., Koert A., Dasse K.A., Gilbert R.J., Griffith B.P. and Wu Z.J., 2006, *Computational and experimental evaluation of the fluid dynamics and hemocompatibility of the CentriMag blood pump*. Artif Organs; 30(3): 168-177.

GLOSSARY

cardiomyopathy	A disease or disorder of the heart muscle, especially of unknown or obscure cause.
creatinine	A creatine anhydride formed by the metabolism of creatine, that is found in muscle tissue and blood and normally excreted in the urine as a metabolic waste.
cytokines	Substances, usually peptides or proteins, that act as growth factors, i.e. affect the growth division or differentiation of cells. Examples include interferon and platelet derived growth factor.
cytosol	The fluid component of cytoplasm, excluding organelles and the insoluble, usually suspended, cytoplasmic components.
decompensation	Failure of the heart to maintain adequate blood circulation, marked by labored breathing, engorged blood vessels, and edema.
hematocrit	The percentage by volume of packed red blood cells in a given sample of blood after centrifugation.
hemodynamics	A branch of physiology concerned with circulatory movements of the blood and the forces involved in circulation.
hemolysis	The destruction or dissolution of red blood cells with subsequent release of hemoglobin.
immunosuppression	Suppression of the immune response, as by drugs or radiation, in order to prevent the rejection of grafts or transplants or control autoimmune diseases.
ischemia	Conditions in which there is not enough oxygen-rich blood supply to heart muscle to meet the heart's needs
ischemic	Pertaining to, affected with, ischemia
lysis	The break-up of a cell after the rupture of its cell wall.
myocarditis	Inflammation of the myocardium
prosthesis	An artificial substitute for a missing part of the body.
sinoatrial	Pertaining to the sinus venosus and the atrium of the heart
thrombus	(pl. thrombi) A solid mass formed from the constituents of blood within the blood vessels or the heart.