Development of an axial flow ventricular assist device using computational fluid dynamics and particle image velocimetry measurement

Su, Boyang

2011

Su, B. Y. (2011). Development of an axial flow ventricular assist device using computational fluid dynamics and particle image velocimetry measurement. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/45500>

<https://doi.org/10.32657/10356/45500>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

**DEVELOPMENT OF AN AXIAL FLOW
VENTRICULAR ASSIST DEVICE USING
COMPUTATIONAL FLUID DYNAMICS AND
PARTICLE IMAGE VELOCIMETRY
MEASUREMENT**

DEVELOPMENT OF AN AXIAL FLOW VENTRICULAR
ASSIST DEVICE USING COMPUTATIONAL FLUID
DYNAMICS AND PARTICLE IMAGE VELOCIMETRY
MEASUREMENT

SU BOYANG

**SCHOOL OF MECHANICAL AND AEROSPACE
ENGINEERING**

2011

SU BOYANG 2011

**DEVELOPMENT OF AN AXIAL FLOW
VENTRICULAR ASSIST DEVICE USING
COMPUTATIONAL FLUID DYNAMICS AND
PARTICLE IMAGE VELOCIMETRY
MEASUREMENT**

SU BOYANG

SU BOYANG

School of Mechanical and Aerospace Engineering

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

2011

Abstract

Due to the limited heart donors available, there are many people with end-stage heart disease who die every year. Ventricular Assist Devices (VADs) are thus, developed to maximize the survival and minimize the morbidity of these patients. Axial flow VADs have gained tremendous interests due to its smaller size and favorable anatomical fit. In the development of an axial flow VAD, besides the hydraulic performance, hemolysis and thrombosis are important parameters closely related to the internal flow conditions to be considered. Therefore, the present study focuses on the hemodynamics of the axial flow VAD using Computational Fluid Dynamics (CFD) which allowed the VAD model to be further optimized based on the simulation results. These simulation results were validated later using Particle Image Velocimetry (PIV) measurements.

The original design of the axial flow VAD had a three-blade straightener, a four-blade impeller, and a three-blade diffuser. The impeller was shrouded by a cylinder, where magnet was embedded. Its numerical model was firstly built with proper operating conditions mimicking the physiological requirements of a human body. The simulation results showed that the flow field inside was not desired.

The impeller was the critical component and hence a group of impellers with different wrap angle and blade height were tested for optimization. Finally, the impeller with 270° wrap angle and 2 mm blade height was selected, because it had the highest pressure head, best hydraulic efficiency and lowest average scalar shear stress. Impellers with three different widths (0.1, 0.2 and 0.3 mm) of clearance gap between the impeller shroud and the pump casing were investigated. The model with 0.2 mm

clearance gap was selected, because it had better hydraulic performance and lower blood damage index.

It was of great interest to simulate and compare the axial flow VAD models with and without impeller shroud after the impeller blade design was optimized, since there is no such comparison done so far. It was shown that the VAD model without impeller shroud had better hydraulic performance with the same level of blood damage index compared to its counterpart. Generally, in both models, the flow fields in the straightener and impeller were smooth, but flow separation is found in the diffuser region, especially near the diffuser hub. As a result, the diffuser blades were detached from the diffuser hub that was integrated and rotated with the impeller hub. Finally, the diffuser with 0.2 mm clearance gap between the stationary blades and the rotating hub was recommended, because it had acceptable hydraulic and hematologic performances.

In order to validate the CFD results, a 1:1 acrylic prototype of the axial flow VAD with impeller shroud was fabricated for measurement. For the ease of comparison, the same planar surfaces were chosen in both numerical and experimental results. The PIV measurements showed that the numerical and experimental results matched well hence giving a reasonable validation of the numerical simulation.

Keywords:

Computational Fluid Dynamics; Axial Flow VAD; Particle Image Velocimetry

Acknowledgement

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

I would also express my sincere thanks to Dr. Ji Wenfa and Dr. Song Guoliang for the enlightening comments and discussions. My thanks were also given to Mr. Yap Pow Khim of Fluid Mechanics Laboratory and Mr. Yuan Kee Hock of Thermal and Fluids Research Laboratory.

My gratitude must also be given to my friends, who have given me help and encouragement during the progress of the work. Special thanks must go to my parents and my wife. Their morale and consistent supports have always inspired me to improve myself.

Last but not least, I would like to thank Nanyang Technological University for the financial support and good research environment.

Table of Contents

Cover Page	i
Title Page	ii
Abstract	iii
Acknowledgement	v
Table of Contents	vi
Nomenclature	ix
List of Figures	x
List of Tables	xvi
Chapter 1: Introduction.....	1
1.1 Heart Disease and Clinical Significance of Artificial Heart Pumps	1
1.2 Ventricular Assist Device and Challenges Faced	3
1.3 Axial Flow VAD in the Current Study	4
1.4 Objectives and Scopes	6
1.5 Outlines of the Report	8
Chapter 2: Literature Review	10
2.1 Blood Properties	10
2.1.1 Hemolysis	12
2.1.2 Hemolysis Tests	16
2.2 Thrombus	17
2.3 Pulsatile or Non-pulsatile Circulation	19
2.4 Maglev	22
2.5 Ventricular Assist Devices	25
2.5.1 Centrifugal Flow VADs	26
2.5.2 Axial Flow VADs	28
2.6 Computational Fluid Dynamics and Experimental Measurements	33

2.6.1 Computational Fluid Dynamics	33
2.6.2 Experimental Measurements	42
2.7 Summary	49
Chapter 3: Method and Theory	51
3.1 Physical Models	51
3.2 Basic VAD Design Theory	53
3.3 Basic Design Objectives	54
3.4 Turbulence Model	56
3.5 Mesh Generation	61
3.6 CFD Configuration	68
3.7 Boundary Conditions	70
3.8 Numerical Validation	72
Chapter 4: Experimental Methodology.....	74
4.1 Experimental Axial Flow VAD Model	74
4.1.1 Test Rig	74
4.1.2 Blood Analog	78
4.1.3 Seeding Particle	80
4.2 PIV System	81
4.2.1 Illumination System	82
4.2.2 CCD Camera	83
4.2.3 FlowMap System Hub	85
4.2.4 FlowManager Software	86
4.3 Experimental Procedure	88
Chapter 5: Original Axial Flow VAD.....	91
5.1 Flow Fields and Static Pressure Distributions	91
5.2 Hemolysis	102
5.3 Summary	111
Chapter 6: Optimization of Impeller.....	113
6.1 Original Impeller	115

6.2 Different Impeller Designs	120
6.3 Elimination of Short Blades	128
6.4 Influences of Clearance Gap	131
6.5 Model with Clearance Gap between Impeller Tip and Pump Casing	138
6.6 Summary	146
Chapter 7: New Axial Flow VAD and Its Optimizations	150
7.1 New Model One (NM1)	151
7.1.1 Static Pressure and Velocity Vector Distributions	156
7.1.2 Hemolysis	165
7.2 New Model Two (NM2)	167
7.2.1 Static Pressure and Velocity Vector Distributions	170
7.2.2 Blood Damage Index	173
7.3 Model with Clearance Gap between Diffuser Hub and Blades	173
7.4 Summary	181
Chapter 8: Experimental Results and Comparisons with Numerical Results.....	184
8.1 Pump Characteristic Curves Comparison between CFD and Experimental Results	184
8.2 Flow Field in the Straightener	185
8.3 Flow Field in the Impeller	189
8.4 Flow Field in the Diffuser	195
8.5 Summary	201
Chapter 9: Conclusions and Future Works	202
9.1 Conclusions	202
9.2 Future Works	206
References	209
Appendix	224

Nomenclature

δ_{ij}	Kronecker Delta, which equals to 1 for $i = j$ and 0 for $i \neq j$
ε	Turbulent kinetic energy dissipation rate
η	Hydraulic efficiency (%)
μ	Dynamic viscosity ($Pa \cdot s$)
μ_t	Turbulent viscosity ($Pa \cdot s$)
ν	Kinematic viscosity (m^2 / s)
ρ	Density (kg / m^3)
σ_{ij}	Molecular shear stress (Pa)
τ	Scalar shear stress (Pa)
τ_{ij}	Shear stress (Pa)
ω	Turbulent kinetic energy specific dissipation
I	Turbulent intensity (%)
k	Turbulent kinetic energy
M	Mechanical torque ($N \cdot m$)
$\dot{m}$	Mass flow rate (m^3 / s)
r	Radius with respect to the center axis (m)
Re	Reynolds number
Re_{DH}	Reynolds number of the flow with the hydraulic diameter
t	Stress exposure time (s)
u	Velocity vector (m / s)
y^+	Dimensionless wall distance $y^+ = \frac{u_* y}{\nu}$

List of Figures

Figure 1.1:	Schematic view of an implanted LVAD	3
Figure 1.2:	Assembly of an enclosed-impeller axial flow VAD. 1. Pump casing; 2. Axial magnetic bearing; 3. Axial gap sensor; 4. Self-bearing motor and self-sensing coils; 5. Permanent magnets; 6. Stator; 7. End cap; 8. Diffuser; 9. Impeller; 10. Radial gap sensor; 11. Rotor; 12. Straightener.	5
Figure 2.1:	Relationship between viscosity and rate of shear for human erythrocytes suspended in their own plasma at 25°C for various volume concentrations Hn of erythrocytes [28]	11
Figure 2.2:	RBC deformation and droplet deformation in shear flow: (top row) configurations of RBCs in an increasing shear flow; (bottom row) spherical droplet deforming into a general ellipsoid [35]	13
Figure 2.3:	Experimental set-up for hemolysis tests [48]	17
Figure 2.4:	Illustrations of both (a) HeartMate I and (b) II [67]	22
Figure 2.5:	Conceptual representations of maglev systems. A: external motor-driven system; B: direct-drive system; C: self-bearing or bearingless system [76]	24
Figure 2.6:	Implantation of PediPump (a) intravascularly and (b) extravascularly [82]	26
Figure 2.7:	(a) Photograph of HeartQuest, (b) Exploded view of the pump and (c) Five DOF of rotor movement [84, 85]	27
Figure 2.8:	The components of the MSCP [86]	28
Figure 2.9:	MicroMed-DeBakey VAD for adult cardiac failure patients [10]	29
Figure 2.10:	(a) Implementation of the Impella acute pump system into the left ventricle. (b) Photograph of the left ventricular micro-axial VAD. (c) Geometry of the impeller [27]	30
Figure 2.11:	A cutaway view of Jarvik 2000 VAD [88]	31
Figure 2.12:	Photograph of the inner components of Magnevad [78]	32
Figure 2.13:	Configuration of the axial Valvo pump [13]	33
Figure 2.14:	Flow chart of a CFD-based design [94]	35
Figure 2.15:	Design history and iterations of LEV-VAD [95]	36

Figure 2.16: Surface topology of the impeller with an optimized nose of Micro-Axial pump [27]	37
Figure 2.17: Schematic illustration of the LDA set-up [106]	44
Figure 2.18: Schematic illustration of the PIV system components [108]	45
Figure 2.19: Velocity distributions in the passage of impeller 16-forward-swept blade [109]	47
Figure 2.20: Comparison of mean axial velocity profile: numerical results (solid line); PIV result (dash line) [27]	49
Figure 3.1: Illustration of the inner components of the original axial flow VAD	52
Figure 3.2: Physical configurations of the inner components of the new model	52
Figure 3.3: Velocity triangles at inlet and outlet	54
Figure 3.4: Subdivisions of the near wall region	59
Figure 3.5: Typical shapes of grid cells in 3D	61
Figure 3.6: Surface meshes on the original axial flow VAD	64
Figure 3.7: (a), (b) and (c) Intersections of two interfaces at three different conditions; (d) information exchanging through interfaces and (e) interfaces adopted in this study.	67
Figure 3.8: Overview of the segregated solution method	69
Figure 3.9: Grid scheme test	73
Figure 4.1: Schematic view of the test rig for PIV measurement	75
Figure 4.2: (a) Exploded view of the axial VAD within the test rig, and (b) Drawing of bearing holder	77
Figure 4.3: Architecture of the FlowMap PIV instrumentation	81
Figure 4.4: Double cavity Nd:YAG PIV laser	83
Figure 4.5: Image pair obtained from CCD camera	84
Figure 4.6: Processing stages in the correlator unit	86
Figure 4.7: Demonstration of the measurement positions on the cross section.	89
Figure 5.1: Static pressure distributions along the surfaces of the blades and hubs of three inner components of the original axial flow VAD model.	92

Figure 5.2:	Velocity vector distributions on the circular planes in the straightener region at (a) $r=4\text{mm}$, (b) 5 mm and (c) 6.5 mm	94
Figure 5.3:	Relative velocity vector distributions on the circular planes in the impeller region at (a) $r=4\text{ mm}$, (b) 5 mm and (c) 6.5 mm	96
Figure 5.4:	Velocity vector distributions on the circular planes in the diffuser region at (a) $r=4\text{ mm}$, (b) 5 mm and (c) 6.5 mm	99
Figure 5.5:	Schematic view of blood flow through each region	100
Figure 5.6:	Axial and radial velocity vector distribution on the longitudinal section of the clearance gap (Region is highlighted by the red rectangular in the schematic view)	101
Figure 5.7:	Scalar shear stress distributions on the surfaces of the blades and hub of three inner components of the original axial flow VAD model	102
Figure 5.8:	Scalar shear stress distributions on the cross sectional planes perpendicular to axial axis of the (a) straightener, (b) impeller, (c) diffuser as indicated in (d) the original axial flow VAD model	105
Figure 5.9:	Mass-weighted scalar shear stress distributions in (a) straightener, (b) impeller, (c) gap and (d) diffuser of the original axial flow VAD	107
Figure 5.10:	Distribution of turbulent stress to viscous stress ratio on a cross section in the impeller region.	108
Figure 5.11:	Blood damage index distributions for the original axial flow VAD	109
Figure 5.12:	Tracks of typical particles: (a) with a smooth flow and (b) backflow at the interfaces for time history (i) the Z position, (ii) the scalar shear stress and (iii) the accumulated blood damage index	111
Figure 6.1:	(a) Physical and (b) numerical model of the original impeller model.	114
Figure 6.2:	Static pressure distributions on circular planes at (a) $r=4\text{ mm}$, (b) 5 mm , and (c) 6.5 mm respectively	116
Figure 6.3:	Relative velocity vector distributions on circular planes at (a) $r=4\text{ mm}$, (b) 5 mm and (c) 6.5 mm	119
Figure 6.4:	Scalar shear stress distributions in the impeller domain of the original impeller model	120
Figure 6.5:	Average static, dynamic pressure distributions along Z axis in P2, P5 and P8. SP: Static Pressure, and DP: Dynamic Pressure.	125

Figure 6.6:	Relative velocity vector and streamline distributions on the circular plane at $r=6.5$ mm in (a) P2, (b) P5 and (c) P8	127
Figure 6.7:	Scalar shear stress distributions in the impeller domain of P2, P5 and P8	128
Figure 6.8:	(a) Physical configuration of P8B2 and (b) Relative velocity vector and streamline distributions on the circular plane at $r=6.5$ mm in the impeller region of P8B2	129
Figure 6.9:	Average static and dynamic pressure distributions in P8B2	130
Figure 6.10:	Average static and dynamic pressure distributions in the three models	134
Figure 6.11:	Tangential and radial velocity fields in the middle circumferential sections of the clearance in (a) G2, (b) G3 and (c) Zhu et al.'s axial VAD [20]	135
Figure 6.12:	Scalar shear stress distribution induced by the clearance gap in G2 and G3	137
Figure 6.13:	Schematic view of (a) G2 and (b) NS.	138
Figure 6.14:	Average static, dynamic and total pressure distributions along Z axis in G2 and NS	140
Figure 6.15:	Radial and axial velocity vector distribution on the longitudinal cross section of the impeller in (a) NS and (b) G2. (c) Demonstration of the cross section position.	144
Figure 6.16:	Relative tangential and radial velocity vector distributions in (a) NS and (b) G2 on the cross section at $Z=10$ mm	145
Figure 6.17:	Scalar shear stress contours on the longitudinal cross section in the impeller regions of (a) NS and (b) G2	146
Figure 7.1:	(a) Characteristic curves of NM1 under various rpms. (b) Hydraulic efficiency versus flow rate under various rpms. (c) Leakage flow rate versus rotating speeds of NM1 under various inlet flow rates.	153
Figure 7.2:	Static pressure distributions along the blades and hubs of NM1	154
Figure 7.3:	Transient static pressure distributions in NM1 at four equally spaced angular position	155
Figure 7.4:	Average static pressure (SP) and dynamic pressure (DP) distributions along Z axis in NM1	157
Figure 7.5:	Velocity vector distributions in the straightener region on circular planes at the radii of (a) 6 mm, (b) 6.5 mm and (c) 7 mm.	158

Figure 7.6:	Relative velocity vector distributions in the impeller region on circular planes at the radii of (a) 6 mm, (b) 6.5 mm and (c) 7 mm.	160
Figure 7.7:	Velocity vector distributions in the diffuser region on circular planes at the radii of (a) 6 mm, (b) 6.5 mm and (c) 7 mm.	162
Figure 7.8:	Axial and radial velocity vector distributions on the longitudinal cross section in the impeller region of NM1. (a): Overall view, (b) Gap and (c) Main Flow.	165
Figure 7.9:	(a) Scalar shear stress distributions on the inner components of NM1, and (b) Mass-weighted scalar shear stress distribution in each region of NM1 as well as the entire domain. ST: straightener; IMP: impeller; GAP: clearance gap between impeller shroud and pump casing; DF: diffuser; WHOLE: Entire model.	167
Figure 7.10:	Schematic view of (a) NM1 and (b) NM2.	168
Figure 7.11:	(a) Characteristic curves of NM2 under various rpms. (b) Hydraulic efficiency versus flow rate under various rpms.	169
Figure 7.12:	Average static and dynamic pressure distributions along Z axis in NM2	171
Figure 7.13:	Static pressure and radial and axial velocity vector distribution on the longitude cross section along the Z axis of the impeller in NM2	172
Figure 7.14:	Physical configurations of the diffusers in (a) NM2 and (b) RDHs.	174
Figure 7.15:	Average static pressure distributions along Z axis in RDHs	175
Figure 7.16:	Velocity vector distributions and streamlines on the circular plane at $r=6$ mm in the diffuser of RDHs	178
Figure 7.17:	Scalar shear stress contours on the cross section 5 mm downstream from the diffuser inlet in (a) RDH1, (b) RDH2, (c) RDH3 and (d) NM2	180
Figure 7.18:	Schematic view of NM3	181
Figure 8.1:	Experimental pump characteristic curves in comparison with the CFD predictions using SST $k-\omega$ turbulence model.	185
Figure 8.2:	Demonstration of the positions of the cross sectional planes in the straightener region	186
Figure 8.3:	Axial and radial velocity vector distribution on the cross sectional plane A in the straightener region (a) PIV measurement and (b) Numerical simulation	187

Figure 8.4:	Velocity vector distribution on the cross sectional plane B in the straightener: (a) PIV measurement and (b) Numerical simulation	189
Figure 8.5:	Positions of the cross sectional planes at two impeller angular positions of (a) $\theta=0^\circ$, i.e. Planes C at the center and Plane D with 6.5 mm offset from the center line, and (b) $\theta=70^\circ$, i.e. Plane E for trailing edge.	190
Figure 8.6:	Axial and radial velocity vector distribution on the cross sectional Plane C of the impeller: (a) PIV measurement and (b) Numerical simulation	192
Figure 8.7:	Relative velocity vector distribution around the impeller leading edge on the cross sectional Plane D: (a) PIV measurement and (b) Numerical simulation	193
Figure 8.8:	Relative velocity vector distribution around the impeller trailing edge on the cross sectional Plane E: (a) PIV measurement and (b) Numerical simulation	195
Figure 8.9:	Demonstration of the regions of the cross sectional Planes F and G in the diffuser	196
Figure 8.10:	Axial and radial velocity vector distribution on the cross sectional Plane F in the diffuser: (a) PIV measurement; (b) Numerical simulation	197
Figure 8.11:	Velocity vector distributions around the diffuser leading edge on the cross sectional Plane G in the diffuser: (a) PIV measurement and (b) Numerical simulation	199
Figure 8.12:	Comparison of the velocity profiles obtained from PIV (solid line) and CFD (dotted line) results in (a) straightener, (b) impeller and (c) diffuser	200

List of Tables

Table 2.1:	Summary of threshold shear stresses and exposure time on hemolysis [34]	14
Table 2.2:	Parameters of the selected axial flow VADs	33
Table 2.3:	(a) VAD MIH ratio classification [14], and (b) Measured versus predicted NIH for CentriMag VAD [102]	42
Table 2.4:	Various blood analogs used in LDA or PIV experiments	46
Table 3.1:	NIH for VADs and their clinical outcomes	56
Table 3.2:	The application and associated requirements for blood pumps with X and (X) indicating mandatory and preferable features respectively	56
Table 4.1:	Refractive index matching fluids used in various cardiovascular measurement experiments [113]	79
Table 4.2:	Specification of CCD camera	85
Table 6.1:	Impellers with different configurations	121
Table 6.2:	Efficiency, power consumption and static pressure heads generated across the impeller and the pump in different models.	123
Table 6.3:	Blood damage index and average scalar shear stress in each regions of each model. SSS: Scalar Shear Stress	137
Table 6.4:	Pressure, efficiency, leakage flow rate (based on the percentage with respect to the pump output, 5 L/min), power and blood damage index in the models simulated in this chapter under the same operating conditions of 5 L/min flow rate and 10,000 rpm rotating speed.	147
Table 7.1:	Mass-weighted average scalar shear stress and blood damage index induced in each region of NM2 and NM1. SSS: Scalar shear stress	173
Table 7.2:	Pressure, efficiency and power consumptions in RDH3	176
Table 7.3:	Pressure, efficiency, power and blood damage index of the models simulated in this chapter under the same operating conditions of 5 L/min flow rate and 9,500 rpm rotating speed.	182

Chapter 1:

Introduction

1.1 Heart Disease and Clinical Significance of Artificial Heart Pumps

Heart is one of the most important organs in the human body. If the heart ever ceases to supply blood, the body begins to malfunction and after a very short period of time the person dies. Heart is composed of muscle that pumps blood, beating approximately 72 times per minute with cardiac output of 5 L/min at 120 mm Hg systolic pressure and 80 mm Hg diastolic pressure. According to World Health Organization (WHO), an estimated 17.5 million people die of Cardiovascular Diseases (CVDs) in 2005, representing 30% of all global deaths [1]. It includes coronary heart disease (heart attacks), cerebrovascular disease, hypertension, peripheral artery disease, rheumatic heart disease, congenital heart disease and heart failure. A substantial number of these deaths can be attributed to tobacco smoking, which increases the risk of dying from coronary heart disease and cerebrovascular disease by 2-3 folds. Physical inactivity and unhealthy diet are other main risk factors which increase individual risks to CVDs. Among these deaths, heart attacks and strokes contribute 7.6 million and 5.7 million deaths respectively in 2005. Furthermore, according to the estimation of WHO, CVDs will cause 20 million deaths by 2015, mainly due to heart attacks and strokes.

The heart failure, which is also called congestive heart failure usually, occurs when the heart cannot pump sufficient blood to the human body. It can result from coronary

CHAPTER 1: INTRODUCTION

artery disease, myocardial infarction, high blood pressure, heart valve disease, congenital heart defects and so on. It can be treated by medication and conventional cardiac surgeries such as angioplasty, valve replacement and bypass surgery. However, severe heart failures usually cannot be treated by medication or less invasive surgical procedures, and transplantation becomes an accepted form of surgical treatment. With approximately 30,000 patients listed worldwide awaiting heart transplants every year, only 3,500 transplantations are performed annually [2]. The increasing incidences of heart failures combined with the limited available hearts require that the optimization of alternative therapies should be urgently pursued in order to maximize the survival and minimize the morbidity of the patients.

Because of the problem of the deficient heart donors and matches between donors and patients, Mechanical Circulatory Support Systems (MCSSs) are being adopted increasingly for patients with severe heart failure. MCSSs avoid the risks associated with immunosuppressant and rejection and could be produced in the quantities required to treat all the patients who might otherwise die before receiving a donor heart. The use of these devices has improved the functional status and life quality of patient with end-stage heart disease, and at the same time reduces their mortality [3-5]. Initially, these devices were developed to bridge patients with end-stage heart disease to heart transplantation, but these devices could also be used as a bridge to cardiac recovery or even as a long-term alternative to heart transplantation due to the advance in technique over the last decade [6]. Although MCSS can be divided into different types according to the different aspects of feature, it is generally divided into two types: Total Artificial Heart (TAH) and Ventricular Assist Device (VAD).

CHAPTER 1: INTRODUCTION

1.2 Ventricular Assist Device and Challenges Faced

The VAD supplements the output of the native left ventricle and is not intended to completely replace the left ventricular function. The VAD operates in parallel with beating ventricle to provide adequate blood flow. In numerous implant cases, the inflow cannula of the VAD is surgically inserted into the apex of the left ventricle so that blood flows from the left ventricle to the pump as shown in Figure 1.1. Due to a rotating impeller in the VAD, energy is imparted to the blood, and the blood exits the device through the outflow cannula. The outflow cannula is then connected back into the systemic circulation via aorta. Most experimental VADs fall into one of two categories: axial or centrifugal. Centrifugal flow VADs, though much less expensive and simpler in design, are larger in size than axial flow VADs. Axial flow VADs have gained tremendous interest due to their small size, and favorable anatomical fit given by their tubular configuration, [7-9].

Figure 1.1: Schematic view of an implanted LVAD

CHAPTER 1: INTRODUCTION

The development of VADs poses a challenge because the working fluid is blood, especially for axial VADs with high speed rotating impeller. Besides pump performance, a number of specific design requirements such as blood compatibility, durability, control feasibility, reliability and portability for patient mobility must be considered [10]. Blood damage (hemolysis) and blood aggregation (thrombosis) are two main features considered during the hematologic design, which aims to minimize the blood damage in the pump [11]. Due to the complex structure of VAD and unique properties of blood, it is essential to optimize the local flow dynamics inside a VAD by avoiding recirculation regions and minimizing the residence time of blood in order to reduce the hemolysis and thrombus formation to an acceptable level. With the development of the magnetic suspension, VADs have no moving parts in contact. Hemolysis and thrombosis are reduced because of the absence of a shaft, contact bearings and seals. Therefore, the expected life span of an axial VAD is prolonged.

1.3 Axial Flow VAD in the Current Study

Figure 1.2 shows the initial design of the axial flow VAD, which is developed by Lim and Zhang [12] of Nanyang Technological University. This mainly consists of a straightener, a rotor and a diffuser, which are all housed in a pump casing. The impeller is tightly fitted inside a cylinder where the permanent magnets are also installed, so the rotor is a combination of the impeller and cylinder. The self-bearing motor and the self-sensing coils are fixed in the stator, and the stator is installed at the corresponding position in the casing. The gap is the inter space between the rotor and the casing. When the axial flow VAD is operating, there will be a forward force generated by the impeller according to Newton's third law and the axial magnetic bearing coil is used to counteract the forward force with the monitor of the axial gap sensors to maintain an acceptable axial gap. Moreover, during the operation, in order

CHAPTER 1: INTRODUCTION

to avoid the damage by the contact between the rotor and the casing, the radial gap sensors are adopted. Because the axial flow VAD with the impeller magnetically suspended has no mechanical contact between the bearing and the flowing blood, the blood damage induced by the wear and heat is minimized and its life span is enhanced.

Figure 1.2: Assembly of an enclosed-impeller axial flow VAD. 1. Pump casing; 2. Axial magnetic bearing; 3. Axial gap sensor; 4. Self-bearing motor and self-sensing coils; 5. Permanent magnets; 6. Stator; 7. End cap; 8. Diffuser; 9. Impeller; 10. Radial gap sensor; 11. Rotor; 12. Straightener.

For the design of an axial flow VAD, induced hemolysis is a main concern besides its hydraulic performance. Although hemolysis is related to many factors, only shear stress and exposure time are considered during the engineering design [13-15]. It has been found in both axial and centrifugal VADs that high shear stress occurred near the tip region where strong flow interaction between the high-speed rotating impeller and the stationary pump casing [16, 17]. The current axial flow VAD has a shrouded

CHAPTER 1: INTRODUCTION

impeller, which is designed and expected to solve this problem, and instead the flow interaction between the stationary pump casing and the fast rotating cylindrical impeller shroud occurs. Consequentially, the clearance gap between these two pump components is of interest to investigate, and it could influence the hemolysis level, the pump efficiency and the magnetic forces. Generally, the larger the clearance gap, the smaller the shear stress under a certain rotating speed and higher wash-out effect in the gap, but it reduces the efficiency and the magnetic forces. Moreover, the comparison between the normal impeller and the shrouded impeller should be conducted, so the advantages and disadvantages of each design could be evaluated. The geometry of the axial flow VAD significantly influences the flow field, so the streamlined flow path is ideal to avoid vortices or stagnation flow regions that would induce deposition of platelets. Also, a low shear stress level should be maintained to minimize the blood damage and restrain the platelet activation.

1.4 Objectives and Scopes

Since the first implantation of the mechanical circulation support device by DeBakey in 1963 [18], VADs are increasingly used for patient with severe heart failure. HeartMate II, Jarvik 2000 FlowMaker and Berlin Heart INCOR are the main axial flow VADs clinically implanted. Berlin Heart INCOR is almost the only clinically available axial flow VAD with the magnetic suspension technology. Its main advantage is that the impeller is suspended by either magnetic levitation or hydraulic bearing between the impeller and housing so as to reduce the hemolysis and thrombosis and to increase the durability [19]. The suspension of the impeller usually needs two radial bearings and one thrust bearing, but it is difficult to combine the radial and thrust bearings [20]. In order to make the axial flow blood pump compact, the thrust bearing is eliminated in the present design. The impeller is enclosed by a

CHAPTER 1: INTRODUCTION

cylinder, in which the permanent magnet is embedded, and the impeller shroud is magnetically suspended by two hybrid bearings [21]. As a result, this design has stronger electromagnetic coupling between the rotor and the stator with higher efficiency in the aspect of magnetic suspension and control. Moreover, it has larger moment of inertia so as to increase the rotating stability of the impeller.

Understanding the inner flow field of the blood pump is crucial, because it is closely related to the hemolysis and thrombosis besides hydraulic performances. Computational Fluid Dynamics (CFD) is a widely used tool for the design and development of biomechanical devices, because it can conveniently and efficiently predict the model with detailed flow field. Furthermore, it can investigate the effects of the design change at early stage so as to minimize the unexpected knock-on effect at later stage. The main objective of the thesis is to develop an axial flow VAD based on the original design. With the aid of CFD, the inner flow field of the original design, the improvements due to the optimization, and the influences of the impeller shroud can be obtained.

The CFD results have to be experimentally validated, using measurement technique such as Laser Doppler Anemometer (LDA) or Particle Image Velocimetry (PIV). The main drawback of LDA is that the time required to measure an entire flow field is much longer. PIV is capable to capture the whole flow field information in a relative short time. Therefore, the measurement of the flow is conducted on the 1:1 pump model of the prototype using PIV. The PIV measurement is commonly used for centrifugal blood pumps [22-26]. However, few PIV measurements on axial flow VAD are available in the literature due to the small size and high-rotating speed [27]. The measurement results are used to compare with the corresponding CFD results for validation purpose in the thesis.

CHAPTER 1: INTRODUCTION

The scopes of the project are as follows:

1. To build the numerical model of the original axial flow VAD so as to conduct its numerical simulation.
2. To investigate various parameters of the flow in the axial flow VAD such as the static pressure distributions, velocity distributions and blood damage index.
3. To improve the axial flow VAD based on the simulation result. Impellers with various parameters are simulated in order to find an optimized model with an acceptable hydraulic performance and low blood damage index.
4. To numerically evaluate the improved models. The static pressure and velocity vector distributions are illustrated, and the blood damage indices are also determined, in order to show the improvements over the original model.
5. To investigate the influences of the clearance gap between (a) the impeller shroud and the pump casing, and (b) the diffuser blades and the diffuser hub.
6. To compare the flow parameters such as pressure head generated, flow fields, scalar shear stress, blood damage index, etc. for impellers with and without shroud.
7. To measure the flow fields in the straightener, impeller and diffuser regions of the axial flow VAD using PIV. The experimental results are further used to validate the numerical results.

1.5 Outlines of the Report

This report consists of nine chapters. Chapter 2 presents the previous works on the research and improvements of the other VADs. The VADs will be introduced with a broader background of the various types of artificial heart assist devices. Chapter 3 presents the numerical analysis of the pump models. Both physical and numerical models used for the simulation will be described. The details of the mesh generation

CHAPTER 1: INTRODUCTION

and the numerical methods are also included in Chapter 3. Chapter 4 introduces the PIV measurement methods, the experimental pump model and the arrangements of the test rig for measurements.

Chapter 5 illustrates the numerical results of the original VAD. This chapter will show and discuss the distributions of the static pressures, velocity vectors, streamlines and etc. The estimation on the blood damage will also be included. Chapter 6 focuses on the impeller optimization through comparison of impellers with different parameters such as wrap angle and blade height. Furthermore, the effects of the impeller shroud and the clearance gap between the impeller shroud and the pump casing are also estimated. Chapter 7 shows the detailed inner flow of the optimized axial flow VAD design and its modifications. In addition, the VAD model without impeller shroud and the VAD models with clearance gap between the diffuser blades and the hub, are also illustrated. Chapter 8 introduces the PIV experimental results. Furthermore, the comparisons between the experimental results and simulation results are carried out. Chapter 9 concludes the study and proposes the future work.

Chapter 2:

Literature Review

2.1 Blood Properties

The complex physical and chemical natures of blood propose a tremendous challenge to the VAD design. Blood consists of a suspension of cells, primarily erythrocytes (red blood cells), leukocytes (white blood cells) and platelets in a Newtonian medium (plasma) with the viscosity of 1 mPa. Red blood cells (RBCs) are composed of hemoglobin slurry in cytoplasm without a nucleus. Hemoglobin readily binds oxygen in the lung and facilitates gas exchange to ensure cellular metabolic function. White blood cells (WBCs) are essential for the immune system response to foreign substances or organisms in the body. Leukocytes and macrophages are accustomed to destroy and remove these foreign substances. Platelets are typically activated by many factors to initiate the coagulation cascade for clot formation and to control bleeding [10].

The volume percentage of RBCs, the hematocrit, is typically 40-45% for a human adult, while that of platelets constitutes 1%. Therefore, blood is essentially a suspension of erythrocytes in plasma and shows anomalous viscous properties. At low shear rates, the apparent viscosity increases markedly when the hematocrit is higher than 14%. Brooks et al. [28] examined the effect of hematocrit on the flow properties of human blood at shear rates up to 700 s^{-1} . An illustration of their findings is shown in Figure 2.1. Up to hematocrit level of approximately 14% (H_n equals to 12.6%, and

CHAPTER 2: LITERATURE REVIEW

note that H_n is equivalent to 0.96 of hematocrit), blood is almost Newtonian. As the hematocrit increases, the viscosity increases and it reaches an asymptotic value above approximately 100 s^{-1} for all volumes of hematocrit as shown in the figure. At shear rate below 100 s^{-1} the viscosity increases markedly with the increase in hematocrit. As a consequence, the viscosity of blood is associated with the shear rate and the percentage of hematocrit. Joles et al. [29] and Eckmann et al. [30] found that the non-Newtonian behavior of blood only occurs in a very small portion of the stress range in VAD: 0-0.7 Pa. Additionally, Song et al. [31] found that different constant values of blood viscosity (3.5 mPa·s and 6 mPa·s) had no virtual influence on the velocity fields. For the numerical simulation of VADs, blood is usually assumed to be Newtonian with constant viscosity and density [27, 32, 33].

Figure 2.1: Relationship between viscosity and rate of shear for human erythrocytes suspended in their own plasma at 25°C for various volume concentrations H_n of erythrocytes [28]

CHAPTER 2: LITERATURE REVIEW

2.1.1 Hemolysis

Hemolysis (blood damage) is the breakdown or destruction of RBCs, and causes the contained protein hemoglobin (Hb) to be released into the surrounding medium, which may activate platelets and subsequently the coagulation cascade. Continuous lysing or damage of RBC reduces its ability to effectively transport oxygen. If as little as 0.5% of RBCs are lysed, the serum will have a visually obvious pinkish color due to hemoglobin. An excess of plasma-free hemoglobin is toxic and may cause renal failure [14].

RBC measures $6.8 \mu\text{m}$ in diameter and $2.2 \mu\text{m}$ in thickness with a distinctive biconcave shape. These RBCs have an average volume of $91.5 \mu\text{m}^3$ and a surface area of $141.6 \mu\text{m}^2$ with an average lifespan of 120 days. A RBC at rest is a biconcave disc with a viscoelastic membrane, and its membrane has a relaxation time of 200 ms depending on the age of the cell. Owing to its biconcave shape, a RBC has 40% excess surface area compared to a sphere of the same volume and it can survive under an estimated strain of 6% before rupturing. Coins-tack shaped structure called rouleaux breaks as shear stress increases, and RBC becomes dispersed. Leverett et al. [34] found that above 150 Pa, extensive hemolysis occurs due to shear stress. Figure 2.2 shows these configurations of RBC in a shear flow [35]. The leftmost state corresponds to no shear and the rightmost state shows a hemolyzing RBC with pores on the surface.

Hemolysis occurs in vivo due to various medical conditions such as antigen-antibody reactions, hemolytic anemias, toxins and poisons. Mechanical RBC rupture occurs due to artificial heart valves and treatments such as hemodialysis and use of the heart-lung bypass machine, and mechanical hemolysis is the most common mode of

CHAPTER 2: LITERATURE REVIEW

hemolysis especially for the application of artificial blood-contacting devices [36]. Minimal blood damage is a fundamental requirement of artificial biomechanical devices. Hemolysis is a sensitive indicator of blood trauma, and in-vitro assessment provides valuable feedback for optimization of VAD design features such as surface finish, blade shape and running clearance during prototype development. It can be concluded that the prediction of the mechanical hemolysis is a crucial issue for the design and application of VADs.

Figure 2.2: RBC deformation and droplet deformation in shear flow: (top row) configurations of RBCs in an increasing shear flow; (bottom row) spherical droplet deforming into a general ellipsoid [35]

According to a theory first proposed by Rand [37], erythrocytes release their contents when exposed to mechanical stress that exceeds a threshold over time. The hemolysis of blood increases with not only constantly applied shear but also the exposure time to such shear [38]. Leverett et al. [34] summarized the state of RBCs under different

CHAPTER 2: LITERATURE REVIEW

flow conditions as listed in Table 2.1, which shows the inverse relationship between the exposure time and the shear stress threshold. Sutura and Mehrjardi [39] used a concentric viscometer with a rotating inner cylinder generating steady and homogeneous turbulent Couette flows. They found significant cell alteration when the shear stress is over 250 Pa for 4 minutes and the concentration of hemoglobin in the suspending medium was quite small until the stress exceeds 250 Pa. In 1984, Sallam and Hwang [40] determined hemolytic threshold of 400 Pa at 1 ms, and in 1987, Blackshear and Blackshear [36] obtained hemolytic thresholds of 1000 Pa when exposed for less than 1s. In 1988, Hasenkam et al. [41] proposed that Reynolds shear stress of 400 Pa was the threshold for lethal damage to erythrocytes for an exposure time of about 1 ms. In 1999, Grigioni et al. [42] suggested that the Reynolds shear stress for blood cell damage threshold value of 400 Pa for 1 ms was probably underestimated and should be raised to between 400 Pa and 600 Pa. In 2004, Throckmorton et al. [43] recommended blood damage threshold of approximately 500 Pa for exposure time of 100 ms.

Table 2.1: Summary of threshold shear stresses and exposure time on hemolysis [34]

Type of exposure	Order of exposure time (s)	Threshold level (Pa)
Turbulent jet	10^{-5}	4000
Oscillating wire	10^{-4}	560
Oscillating bubble	10^{-3}	450
Capillary flow	10^{-2}	500
Capillary flow	10^{-2}	450-700
Concentric cylinder	10^2	150
Concentric cylinder maximum stress 60 Pa	$10^2 - 10^3$	Relatively little hemolysis per unit time
Concentric cylinder maximum stress 20 Pa	10^3	Relatively little hemolysis per unit time
Concentric cylinder maximum stress 60 Pa	10^3	Relatively little hemolysis per unit time

CHAPTER 2: LITERATURE REVIEW

It can be seen that there are different and controversial criteria for the cell rupture in the literature. Although the mechanism of hemolysis is not fully understood, it is generally believed that the threshold for cell lysis could be affected by the following factors.

1. Measurement method. Grigioni et al. [42], who used three-dimensional Laser Doppler Anemometer (LDA), mentioned that the orientation of the measurement system affects the shear stress.
2. Nature of either laminar or turbulent flow [35, 37, 44, 45]. Wurzinger et al. [46] used viscometer to measure hemolysis at laminar shear stress up to 255 Pa and exposure times of 700 ms, whereas difference results were obtained by Suter and Mehrjardi [39] using turbulent shear stress.
3. Material used in experiment. Paul et al. [45] found that the application of the carbon ring seals has strong effect on the measured hemolysis level, as compared with the result obtained by Wurzinger et al. [46] under comparable conditions.

In the field of artificial organs, foreign material interaction and flow induced mechanical damage are the main factors [47]. Generally, the hemolysis induced by the foreign material interaction is ignored in the CFD estimation [14, 20, 42, 48, 49]. In addition to the complexity of the interaction and the difficulty of the simulation of this process, the coating of the biocompatible material can minimize this issue [50, 51]. Therefore, in the field of artificial organs, only mechanical hemolysis is concerned.

CHAPTER 2: LITERATURE REVIEW

2.1.2 Hemolysis Tests

Hemolysis test has been taken as a routine and standard method in the evaluation of hemolysis features in vitro study of rotary VADs. Figure 2.3 shows a schematic view of a hemolysis test arrangement conducted by Arvand et al. [48]. The pump is connected to a reservoir bag using silicone tubes with pressure indicator and flow meter mounted to monitor pressure heads and flow rates respectively. Fresh bovine blood, goat blood or human blood is used, and the reservoir bag is immersed in a water bath with temperature kept at around 37°C. Pressure across the pump is adjusted using resistance throttle applied over a 10 cm length tube to minimize potential for hemolysis due to local narrowing of the tube. After the circuit is filled with the blood suspension, all visible air bubbles should be vented from the blood sampling port and no air interface remained in the reservoir. During the hemolysis test, blood aliquots are sampled from the circulating blood at a regular time interval, and the plasma concentration of hemoglobin is measured thereafter for each sample.

To evaluate blood damage from medical devices such as VADs with continuous blood flow, ASTM F1841-97 standard [52] proposes three relations for the laboratory measurement of the damage to erythrocytes: the Normalized Index of Hemolysis (NIH), the Normalized milligram Index of Hemolysis (mgNIH) and the Modified Index of Hemolysis (MIH). The NIH and mgNIH measure the increase in plasma-free hemoglobin in grams and milligrams per 100 L of pumped blood respectively. The MIH measures the increase in plasma-free hemoglobin, normalized by the total quantity of hemoglobin in the volume of pumped by the device in a closed circuit. The MIH is the engineer's measurement of choice because it is unit-free [14].

$$NIH(g/100L) = \Delta_{freeHb} \times V \times \frac{100 - Ht}{100} \times \frac{10^2}{Q \times T} \quad 2.1$$

CHAPTER 2: LITERATURE REVIEW

$$MIH = \frac{\Delta freeHb}{Hb} \times V \times \frac{100 - Ht}{100} \times \frac{10^6}{Q \times T} \quad 2.2$$

where $\Delta freeHb$ (L/min) is the increase in plasma-free hemoglobin concentration (g/L) over the time interval, V is the circuit volume (L), Q is the flow rate (L/min), Ht is the hematocrit (%) and T is the sampling time interval (min).

Figure 2.3: Experimental set-up for hemolysis tests [48]

2.2 Thrombus

A thrombus or blood clot is the final product of the blood coagulation step, and it is also very important to predict thrombus formation for designing medical devices with blood flow. Compared with hemolysis, the mechanism of thrombus formation is much more complicated and there are many complicated processes to induce thrombus in blood flow [45]. Once damage of blood vessel occurs, platelets and von Willebrand Factor (vWF) adhere to the wall. After platelets react with collagen, fibrinogens react with them under the proper shear rate and clots are generated by platelet, vWF and

CHAPTER 2: LITERATURE REVIEW

fibrinogen. These aggregations are called thrombus in vivo. In artificial organs, especially rotary VADs, these aggregations can be found at wall and other parts.

In animal experiments, Yamane et al. [53] observed that the shear rates of 1300-1700 s^{-1} and 300 s^{-1} might be corresponding to the white and red thrombus formation thresholds respectively, and they proposed the following mechanisms. In the case of white thrombus or platelet aggregation, platelets are thought to be activated by high shear rate flow and to deposit on surfaces with low shear rate. In the case of red thrombus or fibrin gel capturing RBCs, it can be argued that low shear rate simply induces coagulation of RBCs. From the experiences of Nose [54] in the development of various types of cardiac prostheses, he had following comments. During the first two weeks of postoperative period, the white thrombus likely forms inside the implanted prosthesis. After two weeks, all the blood contacting surfaces are pacified by proper protein so that white thrombus is more likely not to form inside for a long period of time. The usage of the anticoagulation coating such as 2-methacryloyloxyethyl phosphorylcholine (MPC) polymer can reduce the platelet adhesion, complement activation and protein absorption so that it is effective against thrombus formation [55].

The shape, structure and size of thrombus depend directly on local blood flow conditions [56]. Higher shear flows may result in endothelial damage triggering pathological platelet clumping to the vessel wall. Low shear stress levels can produce platelet aggregates with low levels of granule released from platelets or RBC response [57]. For a regional stagnant flow, interactions between platelets and RBCs, platelets and platelets or even RBCs and RBCs contribute to thrombus formation. Additionally, the integrity of the fibrin clot is affected by the local shear flow [58]. Hashimoto et al. [59] found that thrombus formed easily when blood was subjected to low shear rate of

CHAPTER 2: LITERATURE REVIEW

less than 500 s^{-1} . A low shear flow results from recirculation and vortices, which trap the cells leading to an increase in the local wall shear stress [60]. Moreover, abrupt cross-sectional widening of flow regions may produce flow separation and potentially secondary flow, creating circumstances favorable for thrombus formation.

2.3 Pulsatile or Non-pulsatile Circulation

The human heart is a pulsatile pump, so the ventricle ejects its contents resulting in both a pressure wave form and a volumetric flow wave form after filling with blood. A long-standing question is whether arterial pulsation is essential in maintaining adequate blood flow and aerobic metabolism in key organs. A considerable amount of research has been done on the change in hemodynamic with respect to pulsatile versus non-pulsatile circulation in human and animals. The benefit of pulsatile perfusion on peripheral organs is probably mediated by its effect on systemic vascular resistance and on the microcirculation. The pulsatility improves splanchnic perfusion and plays a fundamental role in the movement of lymph into and out of the intestine, the prevention of edema, and the maintenance of capillary patency through the prevention of sludging. The brain microcirculation and cerebrospinal fluid movement are reported to be improved with pulsatility. Pulsation improves aerobic tissue metabolism. The influence of non-pulsatile flow on the renin-angiotensin system and catecholamine release remains controversial [61].

Baba et al. [62] implanted an undulation artificial heart pump in a healthy adult female goat in order to estimate the influence of flow pattern change from pulsatile to non-pulsatile on the microcirculation. They observed a decrease in velocity of erythrocytes in capillaries and deterioration in capillary density when the flow pattern was changed from pulsatile to non-pulsatile. There is a possibility that non-pulsatile

CHAPTER 2: LITERATURE REVIEW

perfusion causes microvascular dysfunction, which could become more severe with prolonged non-pulsatile VAD use or permanent non-pulsatile total arterial heart implantation. Sezai et al. [63] investigated the microcirculation of major organs during pulsatile and non-pulsatile circulation after acute myocardial infarction model in the pig. Their results indicated that the pulsatile circulation was superior in the end organ microcirculations and effective in maintaining the circulation.

The necessity of maintaining pulsatile circulation was challenged in 1953 by Wesolowski et al. [64] who successfully maintained the circulation and normal organ function using non-pulsatile blood flow for a brief period. Potapov et al. [61] used the continuous flow implantable DeBakey VAD device to detect the flow in peripheral vessels and measured transactional Doppler signals in patients after implantation. Patients with severe heart failure can be kept alive with a continuous flow VAD. The use of DeBakey VAD resulted in normal organ function and provides unloading of the left ventricle, which can lead to partially restored myocardial function. Despite the non-pulsatile flow produced by DeBakey VAD, the pressure changed due to contractions of unloaded left ventricle and the partially recovered right ventricle provided a nearby physiological pulsatile flow. The first clinical experience with the DeBakey VAD is positive and leads to its continuous use. Recent statistical data indicates that DeBakey VAD has supported over 200 patients at fourteen different heart centers and in seven countries around the world [10]. The mean flow rate for these trials ranged from 3.6-5.0 L/min and cerebral blood flow and organ function were maintained with few signs of hemolysis and thrombosis [65]. The pre- and post-transplant outcomes between pulsatile and non-pulsatile flow VADs were compared by Garatti et al. [66]. They concluded that the continuous flow pumps were similarly

CHAPTER 2: LITERATURE REVIEW

effective on bridging to transplantation rates, mortalities, and post-transplant incidences of treated rejections when compared to pulsatile pumps.

Figure 2.4 (a) shows the implantable pulsatile VAD, HeartMate I, that comprises an actuator coupled to a pusher-plate VAD, a flexible compliance volume, a controller/battery pack, and a transcutaneous energy transmission system coil [67].

Figure 2.4 (b) shows the second generation non-pulsatile rotary axial VAD, HeartMate II, with only one moving part and textured blood contacting surface.

Compared with pulsatile VADs, non-pulsatile VADs have some superiority in the areas of hemolysis, the size and the mechanical simplicity, since they do not require valves or compliance chamber [68, 69]. The current clinically available pulsatile VADs are too large to implant in some patients such as women and children [11, 54].

The size reduction in non-pulsatile pump allows for its implantation in a smaller space and minimizes any detrimental effects on adjacent organs and tissues. The pulsatile devices have the inherent problems of positive displacement that induces the turbulent flow and associated shear forces [70]. The repetitive motion of the flexible diaphragm causes significant stress to the material used, which can eventually lead to degeneration of the blood contacting surface, resulting in thrombosis, calcification, decreased compliance and ultimately device failure. Although non-pulsatile VADs could also induce high hemolysis level, their inherent advantage makes them more likely to avoid severe hemolysis. Sutura et al. [71] showed that pulsed exposure resulted in more thrombocyte aggregation than continuous exposure for the same exposure time and the same maximum stress. Furthermore, the pulsatile flow that requires the necessary mechanical deformations of some components has a lower efficiency and higher energy consumption, and it increases the requirement for the power supply system. Although there is no clear conclusion, non-pulsatile circulation

CHAPTER 2: LITERATURE REVIEW

has been widely employed in the clinical setting of cardiopulmonary bypass or circulatory support [62].

Figure 2.4: Illustrations of both (a) HeartMate I and (b) II [67]

2.4 Maglev

Application and development of maglev have improved the performance of VADs dramatically. The maglev allows suspension of a rotating impeller without any mechanical contact, and thus eliminates wear and heat generation that will take place at the contact surface. It counteracts radial loads by utilizing a magnetic field to support rotor, rather than a mechanical force as in fluid film or roller bearings, and has order of magnitude smaller power consumption than oil film bearings. For primitive models without maglev, contact surfaces between a shaft and bearings can cause hemolysis and thrombosis through shearing and heat [72]. Thrombus can also be experienced as a result of stagnation flows at these contact surfaces and at the seals. Most importantly, eliminating the use of shaft and seals has greatly reduced the contacting problem due to a lack of lubricant oil. The technology of maglev was first applied in rotating machinery by the University of Virginia research group in 1950 [73]. Application for rotary VADs was reported by Akamatsu et al. [74] and Allaire et

CHAPTER 2: LITERATURE REVIEW

al. [75]. The application of the magnetically suspended VAD shows its reliability and it prolongs the expected lifespan of VADs.

Figure 2.5 shows the concept of magnetic suspension and it can be categorized into three types, which are (A) external motor-driven system, (B) direct-drive system, and (C) the self-bearing system [76]. In the external motor-driven system, a motor is used to induce a magnetic coupling force to the impeller, while a separate levitation system controls the impeller suspension. The shortcomings of this system may be the requirement of mechanical bearings to support the rotation of the motor. The Terumo pump is an example of an external motor-driven axial levitation system. A chronic animal study in sheep for duration of 862 (ex vivo) and 482 (in vivo) days demonstrated the reliability and stability of the maglev system [77]. In the direct-drive system, the impeller becomes the motor-rotor, while a separate levitation system is built into the system to realize magnetic suspension. The Jarvik 2000 is an example of a direct-drive system. In the third type, both the drive and levitation coils share the same stator core to make it like a self-bearing system. Each levitation can be implemented in the thrust or radial direction or a combination of both directions. Magnevad is an example of self-bearing system with the motor power consumption of 5 W and the magnetic suspension power consumption of 0.33 W [78].

CHAPTER 2: LITERATURE REVIEW

Figure 2.5: Conceptual representations of maglev systems. A: external motor-driven system; B: direct-drive system; C: self-bearing or bearingless system [76]

CHAPTER 2: LITERATURE REVIEW

2.5 Ventricular Assist Devices

As mentioned earlier, the MCSS can be categorized into TAH and VAD, and the major difference is the requirement of cardiac replacement. The VAD has been considered as a useful therapeutic modality for bridge-to-transplantation, bridge-to-recover, or even long-term cardiac support [79-81]. Since the continuous circulation has its superiority, many types of VADs have been developed at institutions around the world at various stages of design and development as a trend of future long-term implantable VADs. According to the type of assist, they can be categorized as left VAD, right VAD, and bi-VAD, and the possible strategies are shown in Figure 2.6. For either left VAD or right VAD, it can be placed across the semilunar valve with the inlet located inside the ventricle so that blood can be ejected to provide corresponding ventricle support as shown in Figure 2.6 (a). Alternatively, the inlet cannulation of left VAD can be placed at the apex with the outlet connected back to aorta, and the inlet of right VAD is located at the superior vena cava with blood ejected to the left pulmonary arteries through outlet as shown in Figure 2.6 (b). In addition, the bi-VAD support is the combination of left and right VADs. Generally, the continuous flow VADs are based on the spinning of a rotor, so they are also called rotary pumps. There are two main types: centrifugal and axial pumps. The axial flow pump usually has a smaller size compared with the centrifugal flow pump, but its rotating speed is much higher [82]. Centrifugal flow pumps are suitable to generate higher pressure and lower flow rates, whereas axial flow pumps are usually used to generate lower pressure and higher flow rates [83].

CHAPTER 2: LITERATURE REVIEW

Figure 2.6: Implantation of PediPump (a) intravascularly and (b) extravascularly [82]

2.5.1 Centrifugal Flow VADs

WorldHeart (Oakland, CA, U.S.A.) has developed an advanced centrifugal flow VAD, HeartQuest, with full magnetic suspension of the rotor as shown in Figure 2.7 [84, 85]. The pump housing is made of titanium that covers all the magnetic material required for the suspension system and motor. The pump is 35 mm in height and 75 mm in diameter, and it weighs 440 g with 22 mL priming volume. Total power consumption from the battery for the entire system is nominally 12 W, and the noise and vibration are very low such that they are detectable only with stethoscope. Flow visualization was used to observe the flow, and it showed that the flow was well attached with no areas of recirculation. Male calves weighting 90-120 kg were used for the animal trials. Hematology results were in the normal level, and the device was thrombus-free in a 116-day experiment without anticoagulation after 43 days.

CHAPTER 2: LITERATURE REVIEW

Figure 2.7: (a) Photograph of HeartQuest, (b) Exploded view of the pump and (c) Five DOF of rotor movement [84, 85]

An implantable left VAD featuring a magnetically suspended centrifugal pump (MSCP) from Terumo, NTN, and Setsunan University has been developed as shown in Figure 2.8 [86]. The MSCP is a shaftless rotor pump providing contact-free rotation of the impeller, and it can be used for a long-term. The titanium prototype was evaluated in vitro using bovine blood. The pump generated up to a flow rate of 10 L/min at the maximum pressure head of 150 mm Hg, and the normalized hemolytic index was lower than 0.003 when the flow rate was maintained at 5 L/min at pressure head of 100 mm Hg. The pump was implanted in six sheep at the Oxford Heart Centre, and no anticoagulation was utilized. Renal and hepatic functions were kept normal, and no thrombus formation was found on the blood contacting surfaces of the pump.

CHAPTER 2: LITERATURE REVIEW

Figure 2.8: The components of the MSCP [86]

2.5.2 Axial Flow VADs

The MicroMed-DeBakey VAD has been designed in cooperation with NASA/Johnson Space Center, Baylor College of Medicine and MicroMed Technology Incorporated. The DeBakey VAD is driven by a brushless DC motor with the stator embedded around the middle of the flow tube as shown in Figure 2.9. It is 68 mm in length and 24 mm in width with a weight of 94 g [3]. Inside the flow tube, there are three main components with different functions, which are the straightener, the inducer-impeller, and the diffuser along the flow direction. The stationary flow straightener acts as the front bearing support of the downstream inducer-impeller that is the only rotating component to impart energy to the blood. The impeller has rod-shaped permanent magnets embedded within six blades and is activated magnetically by the motor stator, positioned outside the flow tube. The diffuser that contains the rear bearing is used to redirect the high tangential velocity blood axially resulting in a pressure-build of the blood [5, 31]. It could provide a flow rate of 5-6

CHAPTER 2: LITERATURE REVIEW

L/min against 100 mm Hg with the inducer-impeller rotating at around 10,000 rpm, and requires less than 10 W of input power. Animal tests were operated in calves to estimate the overall performance as well as hemodynamic effects before the clinical use [87]. Their study showed that the hemolysis induced by DeBakey VAD did not occur.

Figure 2.9: MicroMed-DeBakey VAD for adult cardiac failure patients [10]

Impella CardioSystems (Aachen, Germany) has developed a micro-axial VAD with minimal invasive treatment for the purpose of a temporary support of the left ventricle in acute situations [27]. It has an overall outer diameter of 4 mm and is placed on a catheter via the arterial femoral into the left ventricle as shown in Figure 2.10. The pump is designed to provide a maximum blood flow of 2.4 L/min with pressure head of 50 mm Hg when the integrated motor drives the impeller at the speed of 50,000 rpm. Unlike most of the axial VADs, it is inserted into the left ventricular through the femoral artery, into the ascending aorta, across the valve and into the left ventricle. This device has been clinically used, and some of its advantages include low anticoagulation required, low complication rate, increase of coronary and end-organ perfusion and so on.

CHAPTER 2: LITERATURE REVIEW

Figure 2.10: (a) Implementation of the Impella acute pump system into the left ventricle. (b) Photograph of the left ventricular micro-axial VAD. (c) Geometry of the impeller [27]

The Jarvik 2000 Heart was initially developed by Jarvik Heart Inc. and the Texas Institute in 1988, and it can be placed within the left ventricle without the inlet cannula with 25 mL in volume [88]. It has a single impeller located at the center of the titanium housing as shown in Figure 2.11, with the normal power consumption between 3 and 7 W. The impeller with 2 titanium hydrodynamic blades, which contains magnet, is suspended by 2 ceramic bearings, and a brushless motor contained within the housing creates the electromagnetic force necessary to rotate the impeller. In addition, the blood flow is redirected through the outflow graft by stator blades located near the pump outlet. The impeller rotates between 8,000 to 12,000 rpm and generates the flow rate up to 6 L/min under the physiological conditions. At the Texas Heart Institute from 1991-1999, Jarvik 2000 was implanted in approximately 37 calves to assess the performance, and these calves were supported for 70 days [88].

CHAPTER 2: LITERATURE REVIEW

The average hemolysis level was around 0.00082 ± 0.000054 g/L. More than 200 patients have received the Jarvik 2000 as either a bridge to transplant (79%) or permanent implant (21%). Around 70% of the patients treated as bridge to transplant have either undergone transplantation or are supported by the Jarvik 2000.

Figure 2.11: A cutaway view of Jarvik 2000 VAD [88]

Gold Medical Technologies Inc. develops the miniature axial flow LVAD, Magnevad, since 2001. The pump can provide 6 L/min with the pressure head of 100 mm Hg when it rotates at 12,500 rpm, and the motor power consumption is typically 5 W [78]. Although it is a continuous pump, it is designed to produce a pulsatile flow by adjusting the rotating speed. Diastolic pressure of 80 mm Hg and systolic pressure of 120 mm Hg could be obtained at 13,000 rpm and 11,000 rpm at 5.2 L/min respectively. Figure 2.12 shows the inner components of Magnevad. The rotor has a small auxiliary blade adjacent to the inlet of each of the two main blades that are 180° apart, and it is used to force the incoming flow to adhere to the main blade without flow separation over a wide range of flow rate. The diffuser has eight helical blades in order to maintain a short axial length. Magnevad II is developed as an improved version, and it is completely passive compared with previous model, which uses a magnetically suspended rotor requiring active axial control [89]. The NIH of Magnevad was within 0.001-0.002 g/100L during the in vitro testing, and it is much

CHAPTER 2: LITERATURE REVIEW

smaller compared with the body's normal replenishment rate of 0.005 g/100L. It was implanted into a sheep for one hour with 1/3-1/2 of the estimated cardiac output maintained. The experimental result showed that the animal's kidney and heart remained grossly free of thrombus and infarcts for the implanted duration.

Figure 2.12: Photograph of the inner components of Magnevad [78]

Valvo pump implanted at the heart valve position is shown in Figure 2.13 [9, 13]. The axial VAD consists of a direct current brushless motor, an impeller, and a guide vane. The motor and impeller assembly are fixed to a stainless steel casing via the guide vane. The casing is a cylinder of 38 mm in diameter with both inlet and outlet of 15mm in diameter. The NIH was 0.056, and the flow rate of 5 L/min was obtained at a pressure difference of 100mm Hg at 8,150 rpm with the old impeller machined by a conventional milling machine. The pump performance is improved by the usage of the impeller manufactured by an NC milling machine. After that, the NIH of this axial VAD was 0.030 ± 0.003 and a flow rate of 5 L/min was obtained at a pressure difference of 100 mm Hg at 7,000 rpm. Therefore, Mitarmura et al. [9] concluded that the newly developed intracardiac axial flow pump can produce sufficient outflow with an allowable level of hemolysis.

CHAPTER 2: LITERATURE REVIEW

Figure 2.13: Configuration of the axial Valvo pump [13]

Table 2.2 summarizes these axial flow VADs for the ease of comparison. It can be found that the rotating speeds of these pumps vary around 10 krpm and the flow rates are around 5 L/min, except Impella, which has an extremely smaller size. NIH varies significantly among different pumps, though they all claimed it was acceptable. It can also be found that the power consumption is basically less than 10 W, and it varies for different designs.

Table 2.2: Parameters of the selected axial flow VADs

	Rotating Speed (krpm)	Flow Rate (L/min)	Pressure Head (mm Hg)	NIH (g/L)	Power (W)
MicroMed-DeBakey	10	5~6	100	0.0002	<10
Impella	50	2.4	50	-	-
Jarvik	8~12	<6	-	0.00082±0.000054	3~7
Magnevad	12.5	6	100	0.00001~0.00002	5
Valvo	7	5	100	0.0003±0.00003	-

2.6 Computational Fluid Dynamics and Experimental Measurements

2.6.1 Computational Fluid Dynamics

CFD technique as a design tool has been widely adopted to simulate and analyze fluid flows within and around objects of interest, and it significantly influences the development of biomechanical devices [90-93]. CFD has the ability to evaluate design

CHAPTER 2: LITERATURE REVIEW

quickly at 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 blood flow to reduce the risk of unexpected knock-on effects that would otherwise become apparent only at a later stage. CFD technique has been used in the development of any blood contacting medical device including heart valves, VADs, stents, grafts, blood access, purification devices, etc. The challenge facing a VAD design is to optimally combine hematologic design with hydraulic design. In particular, designers using traditional methods often do not realize that local high shear stress levels and/or flow stasis have the potential to invalidate a design due to adverse hematologic consequences. The most promising avenue for achieving this task involves the use of CFD [94]. When a final design has been reached, CFD analysis can be used to confirm that design goals have been achieved. Detailed CFD results of flow field can be used often to support and explain experimental results, potentially strengthening regulatory submissions and providing a scientific base for clinical application. In addition, CFD is able to provide the flow field at the region that measurement techniques are not able to be used. Figure 2.14 illustrates the basic architecture and flowchart of a CFD-based design process [94]. The overall goal of the design process is to systematically evolve from some inferior initial designs to an optimized design. The main modules of the design process include: (1) CFD analysis, (2) design evaluation, (3) design modification, and (4) geometry modification.

CHAPTER 2: LITERATURE REVIEW

Figure 2.14: Flow chart of a CFD-based design [94]

Song et al. [95] firstly completed a baseline analysis by means of classic pump design formulas and empirical diagrams to obtain initial pump geometry. Based upon that, a CFD model was generated and the initial design was executed using TASCflow (ANSYS Corp. Inc.) as the CFD solver. LEV-VAD model has approximately a total of 352,500 grids, and both steady-state and transient-state were simulated. Although transient-state requires more computational source, it is essential to ensure a reliable and effective design. The $k-\varepsilon$ model was selected coupled with a log-law wall function to characterize near-wall flow conditions. Since the simulation result was closely related to the grid quality, the skew angles were between 15° and 165° and the y^+ was maintained around 2. The detailed information of flow field exposed the deficiencies of the initial design and provided the information of fluid forces and moments for optimizing the pump design. A number of design iterations were performed to reach the prototype stage. It was found that a proper mechanical design does not necessarily create an ideal VAD. Figure 2.15 shows the design and development of an axial VAD, which displays the main phase (UAP-1 to UAP-4) before the final version of LEV-VAD [95]. After the CFD design was completed, a plastic model was made for the

CHAPTER 2: LITERATURE REVIEW

PIV test, and the steady numerical results agreed well with the measurements over the entire range of operational conditions with the discrepancy less than 10%.

Figure 2.15: Design history and iterations of LEV-VAD [95]

Triep et al. [27] estimated and compared three different impeller designs according to hydraulic efficiency, blood traumatization and thrombus formation by CFD. Since the grid quantity and quality are significantly related to the numerical region, O-grid and H-grid were used to mesh around the blade and the gap between the impeller blade tip and the pump casing with 441 blocks constructed, and 1.1 million grids were generated in the impeller. The surface topology of the impeller with an optimized nose is shown in Figure 2.16. The commercial CFD software, Star-CD (CD-ADAPCO), was selected to analyze the designs by means of a finite volume approach. Due to the absence of rotor-stator interaction, only steady-state in the rotating reference frame was carried out using low-Reynolds $k-\varepsilon$ turbulence model. The simulation results showed that the achieved optimization reduced the rotating speed to

CHAPTER 2: LITERATURE REVIEW

45,000 rpm from 50,000 rpm when generating the same flow rate, and it decreased blood damage index by 50%. Furthermore, the numerical results agreed well with those obtained from PIV.

Figure 2.16: Surface topology of the impeller with an optimized nose of Micro-Axial pump [27]

2.6.1.1 Estimated Hemolysis

In order to reduce the costs of design and development of artificial organs, various numerical approaches have been conducted to quantify the shear-induced hemolysis. Giersiepen et al. [96] developed a simple but effective correlation between shear stress, exposure time and the extent of damage to erythrocytes by the power law under the condition of uniform shear stress. The effectiveness of a power-law formulation as a mathematical model of blood trauma is considered to be suitable only in the presence of the predominance of simple laminar flow. A number of cases show that the power law function overestimates the blood damage [48, 97-99], and Mitoh et al. [13] suggested possible reasons. Firstly, the equation proposed by Giersiepen et al. [96] was obtained from a one-dimensional shear stress field using a Couette

CHAPTER 2: LITERATURE REVIEW

viscometer. In the experiments, RBCs were exposed to three-dimensional shear stresses. Secondly, this equation was obtained for shear stress up to 255 Pa and exposure time of 3-700 ms. In the axial pump, there are regions where the shear stress exceeds 255 Pa. Thirdly, it is not known whether the damaged membranes of RBCs are restored as time passes. The RBCs are repeatedly exposed to high shear stress in experiments but exposed to high shear stress only once in CFD analysis. However, the power-law equation provides an acceptable prediction for comparison purposes [13, 14, 49, 99].

$$dHb / Hb = 3.62 \times 10^{-7} \times t^{0.785} \times \tau^{2.416} \quad 2.3$$

where dHb/Hb is the ratio of plasma-free hemoglobin to total hemoglobin, τ is the characteristic scalar shear stress (Pa) and t is the stress exposure time (s).

There are generally two approaches to apply it. The most widely used model is based on accumulated blood damage along a particle track or a streamline. A number of particles are released from the inlet of the numerical domain, and the accumulated blood damages along these tracks are averaged at outlet as the estimated hemolysis. The detailed equations and procedure are described in the following paragraphs.

Trace data of particles along a streamline can be collected, and the shear stress induced in each particle can be calculated. It is assumed that the damage index (D) associated with each particle has a value of zero initially when the particle enters into a VAD, and increases monotonically due to accumulation of blood damage along the particle trace. When D reaches a value of 1, this particle breaks. The equations are described as follows [13, 49]:

$$d_{p,i} = 3.62 \times 10^{-7} \times \tau_i^{2.416} \Delta t_i^{0.785} \quad 2.4$$

CHAPTER 2: LITERATURE REVIEW

where Δt_i is the time interval between t_i and t_{i+1} .

The damage accumulation from time zero to t_i is given by the following

$$D_{p,i} = D_{p,i-1} + (1 - D_{p,i-1})d_{p,i} \quad 2.5$$

Hemolysis index (E) is defined as the mean damage:

$$E = \frac{1}{N} \sum D_p \quad 2.6$$

where N is the total number of particles.

The scalar shear stress developed by Bludszuweit [100] is from comparative stress theory of fluids that is analogous to the Mises yield criterion for solid materials. The scalar shear stress value includes the six components of the stress tensor and represents the level of shear experienced by the blood. The scalar shear stress estimation provides information about regions of high stress levels (possible hemolysis) and extremely low levels (like stasis risking thrombosis). In the current study, this approach was adopted to account for the three-dimensional shear field and calculated the scalar shear stress according to Bludszuweit's stress formula:

$$\tau = \left[\frac{1}{3}(\tau_{ii}^2 + \tau_{jj}^2 + \tau_{kk}^2) - \frac{1}{3}(\tau_{ii}\tau_{jj} + \tau_{jj}\tau_{kk} + \tau_{kk}\tau_{ii}) + (\tau_{ij}^2 + \tau_{jk}^2 + \tau_{ki}^2) \right]^{1/2} \quad 2.7$$

$$\tau_{ij} = \sigma_{ij} + \rho \overline{u'_i u'_j} \quad 2.8$$

$$\sigma_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad 2.9$$

$$\rho \overline{u'_i u'_j} = -\frac{2}{3} \rho k \delta_{ij} + \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad 2.10$$

CHAPTER 2: LITERATURE REVIEW

where δ_{ij} is the Kronecker Delta which equals to 1 for $i=j$ and 0 for $i \neq j$, k signifies the turbulent energy term, and μ and μ_t are the molecular and turbulent viscosities respectively.

The drawback of particle track is that erroneous predictions and computational problems may arise when the particle velocity is near to zero, especially for complex geometries, which are more sensitive to the distribution of particles or streamline. In order to avoid these problems, Garon and Farinas [14] and Farias et al. [101] propose an approach to predict hemolysis without requiring particle track, streamline or residence time. Additionally, it is not true that the hemolysis at the outlet is the sum of the local hemolysis percentages due to the non-linear dependency on time, though the total residence time is the sum of the exposure times in each numerical cell. Therefore, they proposed the linear damage D_I , and details are described as follow:

$$D_I = D^{1/0.785} = (3.62 \times 10^{-7})^{1/0.785} \tau^{2.416/0.785} t \quad 2.11$$

Since the power-law is based on the Couette viscometer, it can be assumed that the shear stress is constant throughout the material volume. Hence, the time derivative of the linear damage function is constant and takes the following form:

$$\frac{d}{dt} D_I = (3.62 \times 10^{-7})^{1/0.785} \tau^{2.416/0.785} \quad 2.12$$

Although the measured shear stress in that experiment is lower than 255 Pa, it is assumed that this equation can be applied outside the range. The left hand side of Eq. 2.12 is transformed to Eulerian kinematics from the Lagrangian kinematics:

$$\frac{d}{dt} D_I = \left(\frac{\partial}{\partial t} + \bar{u} \cdot \nabla \right) D_I = \sigma \quad 2.13$$

CHAPTER 2: LITERATURE REVIEW

$$\sigma = \left(3.62 \times 10^{-7}\right)^{1/0.785} \tau^{2.416/0.785} \quad 2.14$$

where σ represents the rate of hemolysis production per unit time.

In order to simplify the relationship, the solution is assumed to be time-independent.

$$\int_V (\bar{u} \cdot \nabla) D_I dV = \int_V \sigma dV \quad 2.15$$

Assume that the damage fraction is zero at the inlet section. After the integration by part, Equation 2.15 leads to:

$$\int_S \bar{u} \cdot \bar{n} D_I dS = \int_V \sigma dV \quad 2.16$$

where S is the outlet cross-sectional area, V is the volume of blood, and $\bar{n}$ is outward normal to the outlet cross section:

$$D_I = \frac{1}{Q} \int_V \sigma dV \quad 2.17$$

where Q is the flow rate. This is the average linear damage generated by the device.

According to the relationship in Equation 2.11, the damage D is given by

$$D = D_I^{0.785} = \left(\frac{1}{Q} \int_V \sigma dV \right)^{0.785} \quad 2.18$$

Comparisons between experiment and numerical simulation were performed in hemodialysis cannula, axial VAD and CentriMag centrifugal VAD [14, 101, 102]. CFX-TASCflow (ANSYS) was used to solve the numerical model with adequate number of hexahedral meshes constructed, and the flows in these three models were laminar. By taking one model as the baseline, the numerical computations fairly matched the experimental data for both hemodialysis cannula and VAD as shown in

CHAPTER 2: LITERATURE REVIEW

Table 2.3 (a). For the centrifugal VAD, it agreed well with the hemolysis test at the operating condition as shown in Table 2.3 (b), since the maximum shear stress was maintained below 100 Pa in centrifugal VAD within the validated shear stress range.

Table 2.3: (a) VAD MIH ratio classification [14], and (b) Measured versus predicted NIH for CentriMag VAD [102]

(a)

Prototypes	MIH ratios	
	Experiments	Numerical (3D)
V	1	1
VI	0.69	0.67
VII	n/a	0.62

(b)

Experimentally measured NIH (g/100 L)							Numerical predicted NIH
Pump 1	Pump 2	Pump 3	Pump 4	Pump 5	Pump 6	Average	NIH (g/100 L)
0.0027	0.0023	0.0034	0.0025	0.0039	0.0026	0.0029	0.0033

2.6.2 Experimental Measurements

There are many techniques available to visualize flow field such as Flow Visualization (FV), Laser Doppler Anemometer (LDA), Particle Image Velocimetry (PIV), Particle Tracking Velocimetry (PTV), and Ultrasound Doppler Anemometer (UDA), but only LDA and PIV are introduced in this literature review. In the biomechanical area, both LDA and PIV are widely adopted as non-invasive techniques to measure velocity vectors, and the measured flow fields cover from laminar to turbulent such as coronary artery bypass graft, heart valve and artificial heart in a transparent prototype using sophisticated, state-of-the-art laser technology. Also, they are the recommended methods to validate the numerical results in various areas.

CHAPTER 2: LITERATURE REVIEW

The more traditional point-based LDA technique is considered to be the ‘gold standard’ due to its high accuracy and resolution, and the statistical information it provided does not place a heavy demand on computer storage [103-105]. It is capable of measuring the mean and fluctuating components of fluid velocity so that turbulence parameters can be collected such as turbulent kinetic energy, turbulent production rates and Reynolds stresses. Since it depends on the specific immutable frequency of laser radiation for direct measurements of velocity, it does not require calibration. Principally, a single laser beam is split into two equal intensity beams, which are then focused at the measuring volume. The scattered light is collected when particles are moving through this volume, and the frequency shift between the incident and scattered light is proportional to the component of particle velocity. The particle velocity at this volume measured by a couple of laser beams lies in the plane of the two laser beams and perpendicular to their bisector. In order to determine the velocity direction, one of the laser beams has a frequency slightly different from that of the other. All three components of velocity can be measured by crossing four or six laser beams of different wavelengths or polarization in the same measuring volume and separating out the scattered light using filters. Typically, 100,000 data per measurement location are recommended. With the use of traverse system in moving the measuring laser, it is capable of measuring an area of interest in the flow field. Figure 2.17 shows a schematic view of a LDA system used in Moghtaderi's experiment [106].

CHAPTER 2: LITERATURE REVIEW

Figure 2.17: Schematic illustration of the LDA set-up [106]

The time required to analyze an entire flow field can be a primary drawback of point-based LDA. The PIV technique has the ability to capture the whole flow field information in a relatively short time. The PIV measurements use seeding particles, and successive image pairs within a laser-illuminated plane are captured using a high-speed digital camera [25, 107, 108]. Using a pulsed laser, each image pair is obtained during a relatively short time interval compared with the transit time of the particle through the measurement region. Since the time interval is known, the local velocity in any subregion of these acquired images can be calculated by comparing the particle displacement between a pair of images. This approach completely maps the instantaneous velocity field within the planar illuminated region. Figure 2.18 displays a schematic view of a typical PIV detection system [108]. Browne et al. [105] experimentally investigated the steady flow downstream of the St. Jude Bileaflet heart valve by both LDA and PIV. During the experiment, the advantages and disadvantages of each technique were clearly found. Although LDA is capable of

CHAPTER 2: LITERATURE REVIEW

measuring a higher density of points, it is impractical due to the time required. At least 3000 samples were collected by LDA at each location, whereas 16 samples were obtained by PIV mainly due to the large storage demand and the limited data transfer rate. Generally, the frequency that PIV system can achieve is in the order of 10-15 Hz, and it is insufficient to capture the turbulent flow structures that have typical frequencies in the order of 10 Hz, 100 Hz and higher. In contrast, the sampling frequency of LDA can achieve up to 20 MHz. The experimental results show that PIV technique is sufficient to capture the mean velocity but not for higher moment turbulence quantities.

Figure 2.18: Schematic illustration of the PIV system components [108]

Refraction will occur when light travels through the interface between different transparent materials with different refractive index, so the refractive index matching is necessary to provide an excellent optical access without lens effects due to the curved model wall. During the experimental studies on the flow field within a VAD, a blood analog is required, which should have not only the kinematic viscosity approximating to that of blood but also the refraction index of the transparent material

CHAPTER 2: LITERATURE REVIEW

used for the prototype. Although the blood is non-Newtonian, the blood analog is Newtonian in most VAD experiments due to the high shear rate of the flow. Blood behaves as Newtonian fluid for shear rate higher than 100 s^{-1} , and it is assumed to be Newtonian in most publications in this field [20, 32, 109]. Table 2.4 illustrates some blood analogs used in LDA or PIV measurement with the corresponding kinetic viscosity, density and refractive index at the indicated temperature.

Table 2.4: Various blood analogs used in LDA or PIV experiments

Blood analog	T (°C)	μ (Pa·s)	ρ (kg/m ³)	RI	Reference
Glycerin (20%); Saturated sodium iodide (79%); Water (1%)	25	6.6	1,750	1.485	Baldwin et al. [110]
Dibutyl phthalate	50	5.0	---	1.485	Day et al. [25]
Glycerin (35%); Water (65%)	21	3.6	1,060	---	Triep et al. [27]

Chua et al. [109] studied the velocity and shear stress distributions in the impeller passage and the volute of a 5:1 centrifugal ventricular assist device with three impellers that was sandwiched by two shrouds. The two-component LDA (TSI Inc.) and 0.5 W argon laser operating in a backward-scattered mode were used to measure the flow velocity, and the 5:1 enlarged model was made of Perspex for the optical access. Note that the vector obtained in this study was an ensemble average of 2,000 samples. The 16 forward-swept-blade impeller has better performance than the 16-straight-blade and 8 back-swept-blade impellers in terms of smooth flow pattern and less high shear stress regions in the passage. Furthermore, it was found that the flow field in blade channels of the impeller was axis symmetrical as shown in Figure 2.19, because the double volute design was applied so as to enhance the balance of the magnetically suspended impeller.

CHAPTER 2: LITERATURE REVIEW

Figure 2.19: Velocity distributions in the passage of impeller 16-forward-swept blade [109]

Day and McDaniel [22, 23] investigated the flow in a centrifugal blood under steady and transient conditions by PIV technique. Due to the optical access of the prototype, the impeller was mounted on a shaft instead of being suspended by magnetic bearing. A Kodak digital CCD camera was used for the image acquisition, and it was synchronized with a dual-cavity pulsed Nd:Yag laser by software and hardware from TSI Inc. A pulse generator was used to create a controlled delay between the shaft encoder and the PIV system to measure the impeller at desired position. Besides the mean flow patterns, the viscous and turbulent stresses were also measured at several flow conditions. For the steady flow, no recirculation or stagnant flow was found at the designed flow rate, and shear stress levels were below the threshold that causes

CHAPTER 2: LITERATURE REVIEW

hemolysis and thrombus. For the transient flow, the flow rate varied from 0 to 11 L/min during each cycle, and the transient flow fields were compared to those at constant flow rates in order to analyze the transient effects. It was found that the quasi-steady assumption was a fair treatment of the transient flow.

Triep et al. [27] optimized micro-axial VAD by CFD and used PIV technique to validate the numerical results. The experiment was done on a 1:1 acrylic pump model, which was integrated into a hydraulic mock loop. A double-pulsed neodymiumdoped yttrium aluminum garnet laser (Photonics, Edinburgh Scotland) with a power of 30 mJ/pulse and a wavelength of 532 nm was used, and focused to a 0.2 mm light sheet. A telecentric lens was used in combination with a charge-couple device sensor to record the particle images in the light sheet. A light barred was used to synchronize the laser pulse to a desired angular phase position of the impeller, and the pulse separation was 10 μ s. The final results were the averages of 50 PIV recordings at the same phase position. One of the comparisons of velocity profiles between CFD and PIV at several axial locations is shown in Figure 2.20, and it demonstrates a good qualitative and quantitative agreement, especially in the core region. A discrepancy between the CFD and PIV results occurs in the regions with very high-velocity gradient.

CHAPTER 2: LITERATURE REVIEW

Figure 2.20: Comparison of mean axial velocity profile: numerical results (solid line); PIV result (dash line) [27]

2.7 Summary

Every year many people with severe heart failure worldwide are awaiting heart transplants, but the heart donors are inadequate. Therefore, artificial heart pumps such as VAD are developed to maximize the survival and improve the quality of patient's life. Although the human heart is a pulsatile pump, continuous VADs have been widely employed because of its superiority such as smaller size and less hemolysis. Generally, there are two types of VADs: axial and centrifugal, and axial flow VAD gains tremendous interest due to its small size and favorable anatomical fit. During the design and development of the VAD, the hematologic design is considered besides the conventional hydraulic design, since the working fluid is human blood. Although the exact mechanisms of both hemolysis and thrombus formation are not clearly understood, the researchers are trying to explain them using mechanical factors. Hemolysis is related to the shear stress and the exposure time by the power law, and thrombus formation is related to the stagnation flow, recirculation, and flow

CHAPTER 2: LITERATURE REVIEW

separation. Therefore, these phenomena should be minimized or even eliminated if possible, besides the more complicated physiological and biological factors. Hemolysis test is a standard process, which could measure the blood damage induced by the biomechanical device before the animal trial, and several parameters are widely used to indicate the hemolysis level such as NIH. Since the rotating impeller can be magnetically suspended, there is no moving part in physical contact, and it reduces the heat generation and the stagnation flow. Therefore, the lifespan of artificial heart is prolonged and both hemolysis and thrombus are reduced.

CFD technique has been used increasingly to simulate and evaluate biomechanical devices such as VADs, and it is a useful design tool before the expensive prototype fabrication and testing. Moreover, various numerical models have been invented to predict blood damage, and they show a good match with the hemolysis test. After the CFD simulation, the experimental validation is necessary, and there are many techniques available such as FV, LDA and PIV. Each technique has its advantages and limitations. FV can only be used for the centrifugal pump according to its working principle. LDA is the most robust technique to measure 2D or 3D flow fields. It can provide good measurements on mean velocity and fluctuating components, but it requires a long time to complete the entire flow field. PIV technique is sufficient to capture the mean velocities at the entire flow field by pairs of images except the turbulence parameters. Only few LDA and PIV measurements have been performed on the VAD, especially for the axial VAD probably due to the small size and the high rotating speed. During the experimental measurement, refractive index matching is necessary to provide a good optical access and to reduce lens effect.

Chapter 3:

Method and Theory

3.1 Physical Models

Figure 3.1 shows the original VAD model, which consists of a three-blade straightener, a four-blade impeller, and a three-blade diffuser inside the pump casing. The straightener is 12 mm in length with the hub diameter of 4 mm near leading edge and 6 mm hub diameter near trailing edge. The impeller is 20 mm in length with 6 mm constant hub diameter. The impeller is shrouded by a cylinder where the magnets are embedded so that the impeller and the cylinder are suspended and driven by magnetic forces. The cylinder has an outer diameter of 18.4 mm, and the inner diameter of the casing is 19 mm at that section. As a result, a 0.3 mm clearance gap is present between the pump casing and the cylinder, and this is the specific design of the present axial VAD. The diffuser is 16.2 mm in length with the hub diameter of 6 mm. For the blades of the straightener, impeller, and diffuser, the outer diameters are 15 mm. Under the desired operating condition, when the impeller is rotating at high speed, due to the suction effect, blood flows through the straightener so that it is regulated and accelerated along axial direction. When blood enters the impeller region, it flows along the blade profile without separation, and the kinetic energy is transferred to blood in order to produce adequate axial and tangential velocities. In addition, the static pressure is built up along the flow channel. The diffuser is placed downstream to convert the tangential velocity to static pressure by straightening the flow direction.

CHAPTER 3: Method and Theory

Figure 3.1: Illustration of the inner components of the original axial flow VAD

During the development of the axial flow VAD using CFD technique, the geometries of the inner components (straightener, impeller and diffuser), have been improved in order to induce a lower hemolysis as shown in Figure 3.2. In the new model, the whole length of straightener is 20 mm with the blade length of 8 mm. Its outer diameter remains as 15 mm, while the hub diameter increased to 11 mm. The impeller is still 20 mm long with the bigger hub diameter of 11 mm, and the blade number has reduced from four to two with the blade wrap angle increased from 90° to 270° . The diffuser is 20 mm in length, and the blade length is 10 mm with 11 mm hub diameter. The detailed drawings of these components are shown in Appendix. It can be found that geometries of the inner components in the new model are more streamlined without sharp change.

Figure 3.2: Physical configurations of the inner components of the new model

CHAPTER 3: Method and Theory

3.2 Basic VAD Design Theory

The axial flow VAD is a micro-turbo-machinery. The impeller, which is actually a rotor of the brushless motor, imparts the electrical energy to the flow of blood. Velocity triangle can be used to calculate the basic performance of the impeller as demonstrated in Figure 3.3. W denotes the relative velocity with respect to the impeller; V denotes the absolute velocity; U denotes the blade speed, with the subscripts 1 and 2 indicating the inlet and outlet respectively. Note that some texts and vectors are printed in white for the clear presentation in the figure. The absolute velocity is obtained by adding the relative velocity, W , to the blade velocity, U .

$$V = W + U$$

Newton's second law is applied to basic design, and it states that the torque on the impeller is equal to the rate of change of the fluid angular momentum:

$$T = \dot{m}(V_{u1}r_1 - V_{u2}r_2)$$

where r is the mean radius of the hub and blade tip, and $\dot{m}$ is the mass flow rate.

The pump power, P , is the product of the torque, T , and the rotating speed, ω .

$$P = \dot{m}(V_{u1}U_1 - V_{u2}U_2)$$

Note that $\omega r_1 = U_1$ and $\omega r_2 = U_2$

The Euler's head equation is shown as:

$$H = \frac{1}{g}(U_2V_{u2} - U_1V_{u1})$$

The hydraulic efficiency, η , is calculated as:

CHAPTER 3: Method and Theory

$$\eta = \frac{\dot{m}}{\rho} \left(\frac{P_o - P_i}{M\omega} \right)$$

where $\dot{m}$ is the mass flow rate (kg/s), ρ is the blood density (kg/m³), P_o is the total pressure at the outlet (Pa), P_i is the total pressure at the inlet (Pa), M is the applied mechanical torque (N·m), and ω is the rotating speed (rad/s).

Figure 3.3: Velocity triangles at inlet and outlet

3.3 Basic Design Objectives

The axial flow VAD is basically a turbo-machinery device. The primary requirement is the hydraulic performance, i.e. the VAD should be able to generate a desired flow rate at certain pressure head, instead of power consumption. The typical operating condition is 5 L/min at 13.33 kPa (100 mm Hg) for left VAD [111, 112]. Although lower power consumption (or higher hydraulic efficiency) is desired in this and many other developments of rotary blood pump, it is only considered when the hydraulic performance is satisfied. In addition, the power consumption varies based on the

CHAPTER 3: Method and Theory

many parameters such as type (centrifugal or axial), mechanism (magnetic suspension or shaft), size, operating speed and so on. Although the power consumption is around 10 W generally, there is no clear criterion. The next consideration is the hematologic performance (i.e. blood damage), which includes both thrombosis and hemolysis. Although hemolysis is related to many factors, only exposure time and shear stress are concerned in the CFD study of a VAD. Giersiepen et al. [96] proposed an equation to estimate the mechanical hemolysis, which is the blood damage index in numerical simulation. In the in-vitro test using fresh human or animals' blood, normalized index of hemolysis (NIH) is commonly used. Table 3.1 lists the NIH for VADs and their corresponding outcomes [113]. Nose [54] suggested that the acceptable NIH should be less than 0.004 g/100L at 13.3 kPa (100 mm Hg). Since in-vitro test was not conducted in this study, the comparison was not performed. However, blood damage index is usually assumed to be linearly related to NIH [5, 13, 114]. According to comparison between blood damage index and hemolysis conducted by Yano et al. [49], the corresponding values are 0.131 and 0.033 g/100L. If the blood damage index is linearly proportional to the hemolysis, i.e. blood damage=linear coefficient*hemolysis, the coefficient is around 3.97. If the coefficient is also valid in the present model, the blood damage index is 0.016 corresponding to the hemolysis of 0.004 g/100L. Unlike hemolysis, there is no widely accepted parameter to indicate the content of thrombosis directly in CFD simulation. Instead, thrombosis is minimized by reducing or eliminating the stagnation flow region in the CFD study phase before the in-vitro or in-vivo test. Behbahani et al. [111] listed the application and associated requirements of blood pumps with different suggested period of usage in Table 3.2, which indicates that the first requirement is the hydraulic performance followed by hematologic performance (thrombosis and hemolysis).

CHAPTER 3: Method and Theory

Table 3.1: NIH for VADs and their clinical outcomes

NIH (g/100L)	Clinical Outcomes
>0.06	Increased level of plasma-free hemoglobin
>0.04	No increase of plasma-free hemoglobin, but requires blood transfusion
<0.04	Physiological acceptable
<0.02	Clinically acceptable
<0.01	Design objective of the blood pump

Table 3.2: The application and associated requirements for blood pumps with X and (X) indicating mandatory and preferable features respectively

Blood pump model	Typical application	Durability requirement	Hydraulic performance	Anti-thrombogenicity	Low sublethal cell damage	Small size	Without extra-corporeal connection
Short-term: support during clinical high-risk operation	support during extracorporeal cardiopulmonary bypass	2 days	X				
Short-term: support during transient cardiopulmonary failure	ECMO, PCPS, postcardiotomy cardiac failure	2 weeks	X	X			
Middle-term: bridge to therapy	treatment of dilated cardiomyopathy	6 months	X	X	X	(X)	
Long-term: bridge to recovery	support until recovery of the heart	2 years	X	X	X	X	(X)
Long-term: bridge to transplant	support until transplant	> 2 years	X	X	X	X	(X)
Long-term: destination therapy	permanent support	> 5 years	X	X	X	X	X

3.4 Turbulence Model

The working fluid within the artificial heart is blood, which is widely assumed to be Newtonian in the axial VAD simulations [27, 31, 32]. Song et al. [31] investigated different constant values of blood viscosity: 0.0035 Pa·s and 0.006 Pa·s, and virtually no difference was found in the velocity fields. In this study, the viscosity of blood was set at 0.0035 Pa·s with a density of 1,050 kg/m³ [33, 85]. Reynolds number based on the impeller tip diameter was about 3.5×10^4 at the desired rotating speed of 10,000

CHAPTER 3: Method and Theory

rpm, and was beyond the critical Re of 2000-3000 for internal flows [115]. Since the flow within the axial VAD was turbulent, a proper turbulence model was necessary to simulate the internal flow field. The Reynolds-average Navier-Stokes (RANS) based modeling approach greatly reduces the required computational effort and resources, and it is widely adopted for practical engineering applications. The solution variables in the instantaneous Navier-Stokes equations are decomposed into the mean and fluctuating components. For example, the velocity is decomposed into mean velocity and fluctuating velocity ($U_i = \overline{U_i} + U'_i$). For the incompressible Newtonian fluid, Reynolds-averaged form of the conservation of mass equation can be reduced to:

$$\frac{\partial}{\partial x_i} (\overline{U_i}) = 0 \quad 3.1$$

where U represents the velocity vector, x is the spatial component, i corresponds to Einstein summation convention and the bar denotes a mean value. Applying the Reynolds-averaging process for incompressible fluids, the conservation equation describing the mean momentum is derived:

$$\frac{\partial}{\partial t} (\overline{U_i}) + \frac{\partial}{\partial x_j} (\overline{U_i U_j}) = -\frac{1}{\rho} \frac{\partial \overline{P}}{\partial x_i} - \frac{1}{\rho} \frac{\partial}{\partial x_j} (\overline{\tau_{ij}} + \rho \overline{u'_i u'_j}) \quad 3.2$$

where ρ is the fluid density, P represents the pressure and τ corresponds to the shear stress. The Reynolds stress tensor, $\rho \overline{u'_i u'_j}$, is considered as one of the most difficult quantities to be determined in turbulence flow modeling.

During the selection of the turbulence model, there are some factors to be considered such as the physics of the flow, the computation time, the available computational sources, and the acceptable accuracy. The Reynolds stress model is the most elaborate

CHAPTER 3: Method and Theory

turbulence model, and it closes the Reynolds-averaged Navier-Stokes equations by solving transport equation for the Reynolds stress together with an equation for the dissipation rate. However, it requires seven additional transport equations in three-dimension, which means it will consume much more computing sources. It can be found in the literature that most numerical simulations of rotary VADs have been conducted by either k - ε or k - ω turbulence model [7, 20, 27, 32, 43, 47, 93, 116-124], which are elaborated in the next few paragraphs.

The standard k - ε model is robust, economic and accurate for a wide range of turbulent flows, but it is valid only for fully turbulent flows, when the effect of molecular viscosity is negligible. In general, turbulence models based on the ε -equation under-predicts the onset and amount of separation, which gives an overly optimistic performance. Therefore, this prediction is not on the conservative side from engineering stand-point. Since the wall significantly influences the turbulent flow through the no-slip condition that is satisfied at the wall, the large gradients of solution variables, the momentum, and other scalar transports occur at the near-wall region. Therefore, the near-wall modeling could significantly affect the accuracy of the wall-bounded turbulent flow.

There are three layers at the near-wall region as shown in Figure 3.4. In the inner layer, called viscous sublayer, the viscosity plays a dominant role in the momentum and mass transfer. In the outer layer called fully-turbulent layer, turbulence plays a major role. In the interim region, the effects of molecular viscosity and turbulence are equally important. For the k - ε model, there are several options such as standard wall function and enhanced wall treatment in FLUENT (Version 6.3.26). The standard wall function is not well suited for flow with strong body force such as flow near a rotating disk or impeller. The enhanced wall treatment can provide a more robust

CHAPTER 3: Method and Theory

treatment, which is a near-wall modeling method that combines a two-layer model with enhanced wall functions. If the near-wall mesh is fine enough to be able to resolve the laminar sublayer (typically $y^+ \approx 1$, and can go up to $y^+ \leq 5$), then the enhanced wall treatment will be identical to the traditional two-layer zonal model. It needs less computational requirement than the two-layer model and avoids the excessive error for intermediate meshes that are not only too fine for the near-wall cell centroid to lie in the fully turbulent region but also too coarse to resolve the sublayer. FLUENT achieves this by blending linear (laminar) and logarithmic (turbulent) laws-of-the-wall. This approach allows the fully turbulent law to be easily modified and extended to take into account other effects such as pressure gradients and variable properties. Thus, an enhanced wall treatment should be 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.

Figure 3.4: Subdivisions of the near wall region

CHAPTER 3: Method and Theory

For the k - ω turbulence model, Standard k - ω model based on Wilcox k - ω model and Shear Stress Transport (SST) k - ω model are available in FLUENT. The Standard k - ω model incorporates modifications for low-Reynolds-number effects, compressibility, and shear flow spreading, and it has closer agreement with measurements for wall-bounded and free shear flows. The SST k - ω model is similar to the standard k - ω , but includes several refinements so that it behaves like k - ω in near wall region and behaves like standard k - ε in the free stream. In other words, it can be used as a low-Reynolds number model without any extra damping function in the inner parts of the boundary layer, and switches to k - ε behaviors in the free-stream avoiding the common k - ω problem. Therefore, it is more accurate and reliable for a wider class of flows such as adverse pressure gradient flow and separating flow. However, SST k - ω turbulence model produces higher turbulence levels in regions with large normal strain such as stagnation and strong acceleration regions. Song et al. [26] found that the results from SST k - ω turbulence model matched well with those from PIV measurements so that it was selected as the default solver in this study. For the k - ω model, the wall boundary conditions are treated in the same way as k - ε when enhanced wall treatment is used, which means that the appropriate low-Reynolds-number boundary conditions will be applied for the fine meshes.

The blood temperature rises due to the heat generated in motor and frictional loss when blood flows through the axial VAD, but it is usually ignored during the numerical simulation. Song et al. [31] investigated this phenomena through the solid-fluid coupling, and it was found that majority of the generated heat was removed by the flowing blood with the remaining heat transferred to the tissue. Since blood and tissue have high specific heat, the maximum temperature increases were less than 0.1°C and 0.2°C to blood and tissue, respectively, due to the high specific heat of

CHAPTER 3: Method and Theory

blood and tissue. As a result, it was impossible to damage blood and tissue, and the density and viscosity of blood do not vary. Therefore, the thermal effect is not considered in this study and many other investigations [31].

3.5 Mesh Generation

In order to solve the fluid flow, the first step in the modeling procedure is to generate grids for the flow geometry. Mesh generation is the process of splitting the whole computational domain into a set of discrete cells and the cells are usually tetrahedrons, hexahedrons, prisms or pyramids in three-dimension as shown in Figure 3.5. Edges of these cells form lines of the computational mesh, and the points located at the corners or in the center of a cell are grid nodes, where the flow parameters are obtained. The most basic form of mesh classification is based upon the connectivity of the mesh: structured or unstructured. An unstructured mesh is characterized by irregular connectivity, and it is possible to generate a mesh inside a domain of arbitrary and complex geometry. In contrast, a structured mesh is characterized by regular connectivity, and the computational domain is divided into blocks according to some user-specified topology in order to generate the structured mesh. With the same quantity of grids, the structured grids generally give more accurate results than the unstructured grids, especially when the grid lines are aligned with the flow, though building the structured grids is usually more time consuming.

Figure 3.5: Typical shapes of grid cells in 3D

CHAPTER 3: Method and Theory

The mesh quality plays a significant role in the accuracy and stability of the numerical computation, the attributions associated with mesh quality such as smoothness, skewness, and aspect ratio should be considered during the mesh generation. The smoothness is a measure of changes in grid volume between adjacent of an equilateral grid. The skewness is defined as the difference between the grid shape and the shape of an equilateral grid of the equivalent volume, and the high skewed grid can decrease the accuracy and stability of the solution resulting in convergence difficulty. The aspect ratio is used to indicate the stretching of a cell, which is the ratio of the maximum distance between the cell centroid and face centroids to the minimum distance between the nodes of the cell. For the grids within the viscous boundary layer near the wall, they usually have a high aspect ratio than other regions. The computational generated mesh should be fine enough to resolve the physical phenomenon happening inside the domain. Therefore, in the region where solution is expected to be complicated, especially the region near the sharp feature or the abrupt variation contour, 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 variation) or at regions of interest where greater accuracy is required. Since the enhanced wall treatment was adopted in the simulation, it required fine mesh at near-wall region, and y^+ should be maintained around 1 with the maximum acceptable value of 5. Furthermore, it is recommended that at least 10 cells should be generated within the viscosity affected near-wall region ($Re_y < 200$) in order to resolve the mean velocity and turbulent quantities in that region. However, y^+ and Re_y are not straight forward values, and are calculated after the simulation.

CHAPTER 3: Method and Theory

Figure 3.6 shows the surface meshes generated on the straightener, impeller, shroud and diffuser. The meshes were generated by ICEM (ANSYS), and all the grids were structured. According to the flow characteristics within the pump, the simulation domain was divided into three regions, and they were straightener regions, impeller region, and diffuser region. Note that the clearance gap was included in the impeller region. For the straightener zone, the geometry and the flow field were relatively simple and the velocity gradient was relatively mild. Therefore, coarser grids would be allowed in this region. The flow fields within the impeller and the diffuser regions were more complicated and had great effects on the pump performance so that denser grids were generated. The clearance gap between the rotor cylinder and the axial pump casing was the specific configuration of the pump with 0.3 mm clearance, and the mesh size in the gap was critical with 12 layers of grids generated in between. In order to have a stable flow, an extended pipe was adopted at the pump inlet and outlet with the length of the inlet pipe 5 times of the pipe diameter. Since a poor quality grid will give inaccurate solutions and/or slow convergence, the skewness angles were maintained between 20° and 160° , and the maximum aspect ratio was below 120. Since either the $k-\varepsilon$ model with enhanced wall treatment or the SST $k-\omega$ model was selected, sufficient fine grids were generated at the near-wall region with the y^+ maintained around 1~2.

CHAPTER 3: Method and Theory

Figure 3.6: Surface meshes on the original axial flow VAD

Interfaces were used to connect different components of the axial VAD, and it is positive for FLUENT to handle the situation that the grid node locations are not identical at the boundaries where two domains meet, i.e. the grid on the two interfaces are non-conformal. The intersection between interface zones is updated at each new time step for the sliding mesh model. The resulting intersection produces one interior zone with fluid cells on both sides and a pair of “wall” zones with fluid cell on only one side as demonstrated in Figure 3.7 (a). If the problem is periodic, the intersection produces one interior zone and a pair of periodic zones as demonstrated in Figure 3.7 (b). These two “periodic” zones match each other and exchange information, whereas the information exchanging does not occur between a pair of “wall” zones. If the interfaces intersect entirely as demonstrated in Figure 3.7 (c), only one interior zone is produced. The flow information is exchanged through the intersection of the two interface zones rather than from the interface zone faces. For example, in Figure 3.7

CHAPTER 3: Method and Theory

(d), to compute the flux across the interface into cell IV, face D-E is ignored and faces d-b and b-e are used instead, bringing information into cell IV from cells I and III, respectively. For the numerical model of the axial VAD, neighboring regions were connected via interfaces, but the interfaces were not located at the physical interface, where different regions physically connected. Instead, the interfaces were selected at numerical interface, which were 0.2 mm away from the physical interfaces as shown in Figure 3.7 (e), in order to reduce the convergence difficulty and the numerical diffusion.

CHAPTER 3: Method and Theory

(b)

(c)

CHAPTER 3: Method and Theory

Figure 3.7: (a), (b) and (c) Intersections of two interfaces at three different conditions; (d) information exchanging through interfaces and (e) interfaces adopted in this study.

CHAPTER 3: Method and Theory

3.6 CFD Configuration

In this study, FLUENT (Version 6.3.25, ANSYS) was mainly used as the default CFD solver in the simulation. Therefore, only FLUENT is introduced in this section. FLUENT provides two numerical methods: segregated 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 coupled approach gives a performance advantage in solving high-speed compressible flows, and the segregated solver has traditionally been used for incompressible and mildly compressible flows so that segregated solver was selected in this study. In the segregated solver, each equation is solved separately, and the under-relaxation factors are included in the discretized equations, which can improve stability of iterative process and limit change in variable during iterations. In transient flow, the advancing of the solution by one time step normally requires a number of outer iterations as shown in Figure 3.8. The convergence is checked at the end of each iteration. If it is converged, the calculation will proceed to next time step. Otherwise, another outer iteration starts. The general steps in the outer iteration loop are described as follows:

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 velocity obtained in Step 2 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

CHAPTER 3: Method and Theory

then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass fluxes such that continuity is satisfied.

4. Where appropriate, equation for scalar such as turbulence, energy, species and radiation are solved using the previously updated values of the other variables.

5. A check for convergence of the equation set is made.

Figure 3.8: Overview of the segregated solution method

There are several pressure interpolation schemes available in FLUENT such as standard, second-order, and PRESTO!. Although the standard scheme is robust for most cases, second-order scheme was selected due to improved accuracy. Both first-order and second-order discretization schemes are available in FLUENT, and the

CHAPTER 3: Method and Theory

second-order scheme is recommended to obtain more accurate results, especially for complex flows. During the simulation, Second-Order Upwind Scheme was used as discretization method, which brings about a similar result to Quadratic Upwind Interpolation of Convective Kinematics (QUICK) Scheme in the study and reduces the numerical diffusion of First-Order Upwind Scheme. 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 to solve this problem. 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 converging rate with relaxation factor of 0.5-0.7 and 1 for momentum and pressure equations respectively.

Generally, it is difficult to judge convergence by examining the residual if no scaling is involved. FLUENT scales the residual with a scaling factor representative of the flow rate through the domain. This scaled residual is a more appropriate indicator for most problems, and the solver's scaled residual for the continuity equation is defined

as $\frac{R_{iterationN}^C}{R_{iteration5}^C}$, where $R_{iteration5}^C$ is the maximum residual value in the first five iterations,

and $R_{iterationN}^C$ is the residual value at current iteration. By default, the scaled residual is 10^{-3} in FLUENT, which is acceptable for most cases, and it was set to be 10^{-4} in this study to enhance the convergence [43].

3.7 Boundary Conditions

The physiological pressure output of human heart is from 80 mm Hg to 120 mm Hg, so the static pressure at the pump outlet was within this range. Velocity inlet was

CHAPTER 3: Method and Theory

applied with 4 to 6 L/min flow rate according to the physiological requirement of human body. The pressure across the VAD could be determined after the simulation.

The turbulence intensity, I , defined as the ratio of the root-mean-square of the turbulent velocity fluctuations, u' , to the mean flow velocity, u_{avg} was estimated from Eq. 3.3 derived from an empirical correlation for pipe flows:

$$I = \frac{u'}{u_{avg}} = 0.16 \text{Re}_{D_H}^{-1/8} \quad 3.3$$

where $\text{Re}_{D_H} = \frac{\rho u_{avg} D_H}{\mu}$ is the Reynolds number of the flow with the hydraulic diameter as the characteristics length. Under the desired flow rate, the turbulent intensity was around 5~6%.

There are three methods provided to solve the problems that involve both stationary and moving zones including Multiple Reference Frame (MRF) model, the mixing plane model, and the sliding mesh model. The MRF model is the simplest, and it is a steady-state approximation, which is a suitable approach when the interaction between rotor-stator is weak. In addition, it is a suitable choice for the initial calculation in the transient sliding mesh model. The sliding mesh model is the most accurate method for simulating the rotor-stator interaction, but it requires more computational resources. According to the flow nature of the axial VAD, MRF was selected as the initial calculation, and then the sliding mesh was used to simulate the flow accurately. MRF/sliding mesh was applied to the impeller region with the straightener and diffuser set to be stationary regions, and the pump casing was stationary walls. For the pump casing in the rotor region, which forms clearance gap with the outer surface of the impeller shroud, it was set to be rotating wall with zero absolute velocity. The impeller hub, blade and shroud have zero relative velocity with

CHAPTER 3: Method and Theory

respect to the reference frame. No-slip boundary condition was applied to wall so that the blood on the wall has zero relative velocity with respect to the wall.

3.8 Numerical Validation

CFD has its superiority to the in-vivo and in-vitro methods such as more versatility and higher resolution, but its accuracy must be validated. In this test, the static pressure head across the pump model was used for the comparison criteria. Additionally, the SST $k-\omega$ turbulence model was selected as the default solver and the same boundary conditions were applied to all the cases (Velocity Inlet: 0.471 m/s, Pressure Outlet: 13.3 kPa (100 mm Hg), and Rotating Speed: 10,000 rpm). The impeller region rotated with respect to the stationary straightener and diffuser as the flow time advanced when the sliding mesh model was adopted. The fully implicit scheme was used in the flow study, because it has the superiority of unconditional stability with respect to the time-step size. There was thus no stability criterion that needs to be met in determining the time-step size. However, in the transient flow the convergence rate and accuracy still have some relationships with the time-step size. For the segregated solver, the Δt must be small enough to resolve time dependent features, and the time-step size's order of magnitude can be estimated as:

$$\Delta t \approx \frac{\text{typical cell size}}{\text{characteristic flow speed}} \text{ so that accurate unsteady characteristics of the flow}$$

can be resolved. Better time step must reduce the whole computation time and express the flow field alteration clearly.

In this study, time steps of 1/100, 1/200, and 1/300 per time-period have been selected and examined. Note that the time period is defined as the time of the impeller rotating one revolution. To eliminate the starting effect of the transient flow, the computation was carried out over a relatively long time to make sure the flow in the pump was

CHAPTER 3: Method and Theory

periodic. Although the simulation results of 1/200 per time-period were almost identical to those of 1/300 time-period, 1/300 time-period was chosen, since the angular positions of the rotating impeller can be equally spaced between two adjacent straightener blades with 120° pitch angle. Figure 3.9 shows the grid scheme test for the new model with seven different grid schemes, and the calculated static pressure head of each model is adopted for comparison. It can be found that the static pressure head with 438,680 grids (A) has a larger discrepancy with the rest grid schemes. As the grid size increases, the difference of the static pressure with the densest model (G) with about 1.87 million grids becomes smaller. Besides the accuracy, the computational resource required is another consideration, since a model with larger grid number requires more physical memory and computational time. The static pressure head generated by grids C (781,680) is closed that generated by the densest grid G with the difference of 1%. Therefore, grid C was selected as a compromise between the accuracy and computational resource.

Figure 3.9: Grid scheme test

Chapter 4:

Experimental Methodology

4.1 Experimental Axial Flow VAD Model

4.1.1 Test Rig

A full-scale axial flow VAD model was fabricated in order to use PIV for measuring the flow field and subsequently validate the CFD results. In order to simulate the blood circulation, a closed flow loop was designed and fabricated as illustrated in Figure 4.1. The blood analog was firstly injected into the upper reservoir, which was an acrylic tank, and then the test chamber was fully filled through the flexible tubes, with the valve near the lower reservoir closed, while the valve located at the downstream of the flow meter was open. When changing the components or the experiment was completed, the valve near the lower reservoir tank (which was a 2.5 L beaker) was open so that the blood analog would flow into the beaker for the storage of the next experiment. When the VAD model was in operation, the flow rate was controlled by the throttle valve located at the downstream of the flow meter, and the pressure head across the pump model and the flow rate could be obtained from the pressure transducer and the flow meter respectively.

Since PIV system was used for the measurement, the test model must be transparent for the illumination of the laser sheet and the image pairs capturing. An acrylic model of the axial flow VAD, NM1, was fabricated, and it consisted of a straightener, an enclosed impeller, a diffuser, a rectangular chamber and a driving mechanism.

CHAPTER 4: Experimental Methodology

Although the impeller should be magnetically suspended, it was impossible for the PIV measurement due to the opaque magnets. Therefore, a stain steel shaft was inserted through the impeller hub and then tightly fitted. The shaft was further connected to the servomotor through a coupling as shown in Figure 4.1, which was also mounted with 3 channel optical shaft encoder. Since the signal of the shaft encoder was of TTL, a BNC cable consisting of channel I and GND signals from the shaft encoder was used as 'once per revolution' trigger input for System Processor, which sent signals to both Laser System and CCD Camera. Therefore, the camera captured the image pair of planar flow field, which was sequentially illustrated with adjustable time delay by the laser sheets from both frames. The image pair was then sent back to System Processor, and finally stored in PC through Communication Link. As a result, a pair of seeding particle images with time delay was captured at certain impeller angular position, and the velocity vector map was derived from the movement of particle during the time delay.

Figure 4.1: Schematic view of the test rig for PIV measurement

CHAPTER 4: Experimental Methodology

Figure 4.2 (a) shows the exploded view of the axial flow VAD. During the operation, the vibration amplitude of the impeller must be kept to a minimum, since the radial clearance gap between the cylinder and the pump casing was only 0.2 mm. A bearing holder was designed as shown in Figure 4.2 (b), and the outer diameter of the front end was small enough to locate and fix the hollow diffuser in position. Two bronze journal bearings were inserted into the two holes located at both ends of the bearing holder, and tightly fixed. The bearing holder was then inserted into the posterior component. The bearing holder was made of aluminum, and it was iodized into black color so as to avoid strong reflection from laser sheet. The impeller had two helical blades with 180° separated, and it was a single component made from a piece of acrylic solid rod. The impeller was placed between the straightener and the diffuser as shown in this figure, and the straightener was glued together with the pump casing in order to fix its position. Due to the specific design of the impeller, it was shrouded by a cylinder that was also made of acrylic. Both impeller and cylinder were polished, and glued together to become a single component using chloroform. As mentioned earlier, the impeller was driven by a stain steel shaft, which was inserted through the bearings and bearing holder. The pump casing was capped by two components connected to the test rig, and was inserted into a rectangular test chamber, since the surface should be flat to avoid distortion of the laser sheet as shown in Figure 4.2 (a). The inter space between the pump casing and the test chamber was filled with blood analog in order to have the same refraction index. The chamber was properly oriented in order for the laser sheet to enter the chamber perpendicularly and the camera lens was mounted parallel to the top chamber surface as shown earlier in Figure 4.1. Furthermore, the chamber was mounted on a platform, which could be adjusted to different positions so that the different flow area could be measured using PIV.

CHAPTER 4: Experimental Methodology

(a)

(b)

Figure 4.2: (a) Exploded view of the axial VAD within the test rig, and (b) Drawing of bearing holder

CHAPTER 4: Experimental Methodology

4.1.2 Blood Analog

The physical properties of human blood have been introduced in the literature review of Chapter 2, and the main concern for the PIV measurement (and many other flow measurements) is the refraction index matching, which is a more important parameter than the density and kinetic viscosity in achieving a good measuring result. Furthermore, it is more challenging for the velocity measurement in axial VADs, when the laser sheet must be able to transmit through the transparent acrylic blade, and the scattered light can be captured by the CCD camera. If refraction occurs at the interface between the blood analog and the acrylic component, a dropout in acquiring vectors may happen near that region due to strong random reflection.

In order to avoid these problems, a working fluid with a close refractive index of acrylic plastic ($RI=1.49$) was considered. Although the refractive index matching is the first priority, the desired fluid should be low cost, safe, easy to prepare and stable. During the past decades, many blood analogs have been used in various cardiovascular measuring experiments, and Hassan and Dominguez-Ontiveros [125] listed the refractive index of some commonly used fluids as shown in Table 4.1.

During the PIV measurement, the working fluid involved mixing 40% of glycerin and 60% of aqueous ammonium thiocyanate (NH_4SCN) solution by volume [126]. Its refractive index was 1.488 to match that of the acrylic model with a density of $1,155 \text{ kg/m}^3$ and a dynamic viscosity of $3.8 \text{ mPa}\cdot\text{s}$. Note that the density of blood used in simulation was $1,050 \text{ kg/m}^3$ and the dynamics viscosity was $3.5 \text{ mPa}\cdot\text{s}$. The kinetic viscosities of the working fluid and blood were 3.29 and $3.33 \text{ m}^2/\text{s}$ respectively and they were quit close.

CHAPTER 4: Experimental Methodology

Table 4.1: Refractive index matching fluids used in various cardiovascular measurement experiments [113]

Fluid	Refractive Index	CAS #	Hazard	Fire	Reactivity	Density (g/cm ³)	Boiling point/(melting point) (°C)	Flash point (°C)	Dynamic viscosity (cP)	Cost
Water	1.33					1	100		1	
Glycerin	1.47	56-81-5	1	0	1	1.262	182	160	934	\$
Zinc iodide solution	1.33–1.62	10139-47-6	3	0	1	4.74	(445)		1–10.	\$\$\$\$
Sodium iodide solution	1.33–1.5 (60%)	7681-82-5	2	0	1	3.67	(651)			\$
Potassium thiocyanate sol.	1.33–1.49	333-20-0	1	0	0	1.89	(173)		1–2.4	\$
Ammonium thiocyanate sol.	1.33–1.5	1762-95-4	1	0	1	1.3	(152)		1–2.1	\$
Sodium thiocyanate sol.	1.33–1.48	540-72-7	2	0	1	1.74	(287)		1–7.5	\$
Kerosene	1.45	8008-20-6	2	2	0	0.8	175–325	82		\$
Silicone oil mixture	1.47	63148-58-3	0	0	0	1.102		316	90–110	\$\$\$\$
Mineral oil	1.48	8020-83-5	0	0	0	0.862				\$
Turpentine	1.47	8006-64-2		3		0.87	150	35	1.49	
Solvent naphtha	1.5					0.67				
Soybean oil	1.47	8001-22-7				0.93			69	\$\$\$
Olive oil	1.47					0.92			84	\$\$\$
Castor oil	1.48	8001-79-4				0.96	313		986	\$
Tung oil	1.52	8001-20-5				0.93				
Cassia oil	1.6									
Dibutylphthalate	1.49	84-74-2	2	1	0	1.043	340	171		\$
Tetraline	1.541	119-64-2	2	2	1	0.969	204	75		\$\$\$
Benzyl alcohol	1.54	100-51-6	2	1	1	1.044	90	96	5	\$\$\$
Ethyl alcohol	1.362	64-17-5	2	3	1	0.79	78	14		\$\$\$
Methylnaphthalene	1.615	90-12-0	3	2	0	1.001	240	82		\$\$\$\$
Chloronaphthalene	1.632	90-13-1	2	1	0	1.194	111	121		\$\$\$\$
200 fluid (Dow Corning)	1.404		1	3	0	0.816	152	30	1	
550 fluid (Dow Corning)	1.4935		0	1	0	1.065	35	101	125	
556 fluid (Dow Corning)	1.46		3	1	0	0.98	65	101	20	
710 fluid (Dow Corning)	1.533		0	1	0	1.11	35	101	500	
Cargille RI matching fluids	1.3–2.3									\$\$\$\$
D-Limonene	1.473	5989-27-5	3	2	0	0.843	176	48	0.8	\$\$\$\$
Methylsalicylate	1.526	119-36-8	2	1	1	1.184	219	96	4.09	\$\$\$\$
Diethylphthalate	1.504	84-66-2	0	0	0	1.118	294	156	12	\$
p-Cymene	1.491	99-87-6	2	2	0	0.857	177	47	0.876	\$
Isopropyl alcohol	1.378	67-63-0	2	3	1	81	12	81		\$

CHAPTER 4: Experimental Methodology

4.1.3 Seeding Particle

The velocity distribution of the flow is obtained through the suspended seeding particles in the blood analog. It is thus important to choose the seeding particle. The main parameters to be considered in the selection are the particle size, relative density of particle with respect to fluid, and concentration of particle. According to the manual of FlowMap PIV, the camera images of seeding particles should have a diameter of at least 2 pixels, preferable 3 pixels or more [127], and Equation 4.1 was used to calculate the particle image size.

$$d_{image} \approx \left[\frac{d_p^2}{S^2} + \left(2.44 \left(1 + \frac{1}{S} \right) f_{\#} \lambda \right)^2 \right]^{1/2} \quad 4.1$$

where d_p is the diameter of the seeding particles; $f_{\#}$ is the recording aperture; S is the scalar factor; and λ is the wavelength of the illuminating light source.

Although the seeding material should be chosen to maintain neutrally buoyant in the fluid, it is usually a secondary consideration in many situations. In this experiment, hollow glass spheres with a mean diameter of 10 μm for the size distribution between 2 and 20 μm . The density of the seeding particles should fulfill the requirement of a minimum 5 particles per interrogation area for the cross-correlation to attain a good signal-noise ratio. In the present PIV test, the measurement area was $10.76 \times 8.26 \text{ mm}^2$, so the scale factor, S , was 1.25. The recording aperture, $f_{\#}$, was 5.6~8, and thus the camera image of seeding particle had a diameter 2~3 pixels under the wavelength of 532 nm.

CHAPTER 4: Experimental Methodology

4.2 PIV System

The PIV system used in the present study is the Dantec FlowMap PIV system (Dantec Measurement Technology, Denmark). It consists of illumination system, CCD camera, FlowMap System Hub which includes correlator units, input buffers and synchronization unit, and a PC with FlowManager software installed as demonstrated in Figure 4.3. The FlowMap processor is connected to both the illumination system and the camera in order to synchronize the pulsed light-sheet with the recording of the camera images. It is also capable to connect other device to the FlowMap processor so as to synchronize the measurements with external events using TTL trigger signals.

Figure 4.3: Architecture of the FlowMap PIV instrumentation

CHAPTER 4: Experimental Methodology

4.2.1 Illumination System

For the illumination system, it should satisfy several criteria. Firstly, the light budget should be sufficient high to ensure the intensity of scatted light from the seeding particle is above the optical noise level of the system. Secondly, the duration of the light pulse should be such that the particle does not move significantly during its exposure to the light-pulse. Thirdly, the time between successive light pulses should be such that the flow field does not vary significantly. Finally, the location and dimensions of the measurement plane should be well defined. In the axial flow VAD, the flow is three-dimensional, so the thickness of the laser sheet should satisfy Equation 4.2 in order to minimize the effects of the out-of-plane velocity component.

$$\frac{v_z \Delta t}{z_l} \leq 0.25 \quad 4.2$$

where z_l is the laser sheet thickness, v_z is the out of plane velocity component, and Δt is the time interval. When v_z was about 7.8 m/s and Δt was about 10 μ s, the z_l should be larger than 0.32 mm, and the selected thickness in the experiment was 0.5 mm.

Usually a pulsed Nd:YAG laser (Neodymium Yttium Aluminum Garnet) with double cavities is used as the light source because of its high light intensity, and its principal layout is shown in Figure 4.4. The Gemini PIV 200-15 (New Wave Research, USA), a double cavity pulsed Nd:YAG laser system with a repetition rate of 15 Hz, a pulse time of 5-10 ns and 200 mJ emitting ability, was used as the illumination source in the FlowMap system. The two pulsing beams from a double-cavity Q-switched Nd:YAG laser are combined and formed into pulsing light-sheets using optics. Since most cameras are not sensitive to the wavelength of 1064 nm, the frequency is doubled by harmonic generator. Since harmonic generator is not efficient, the residual infra-red is dumped by the harmonic separator. Q-switched pulse, which is the cavity closed for

CHAPTER 4: Experimental Methodology

the majority of the time and is only allowed to open for brief period, can effectively freeze the motion of particle, and two cavities ensue the widest possible ranges of time between pulse. Communication lines between the pulsed laser and the synchronization unit in the PIV FlowMap processor are required for user control of the data acquisition process. The synchronization unit will instruct the laser cavity when to fire their flash lamps and when to allow their Q-switches to emit laser radiation.

Figure 4.4: Double cavity Nd:YAG PIV laser

4.2.2 CCD Camera

The CCD camera used in the FlowMap PIV system is HiSense MkII camera, and it consists of a high resolution Hamamatsu C8484-05 digital CCD chip and Nikon AF Micro-Nikkor lens 60/2.8. It is a cross-correlation camera with the time separation down to 0.2 μ s, and contains a high-performance progressive scan interline CCD chip with no mechanical shutter, which contains 1280 $\times$ 1024 light sensitive cells and an equal number of storage cells. Table 4.2 lists the specifications of the CCD camera. Asynchronous reset model was used to drive the camera when it is used with the FlowMap system. In this model, the FlowMap processor was acted as the master clock, driving both camera and illumination system in order to ensure the camera and laser are synchronized. During a PIV recording sequence, the first laser pulse is timed to expose the first frame, which is transferred from the light sensitive cells to the

CHAPTER 4: Experimental Methodology

storage cells immediately after the laser pulse. The second pulse is then fired to expose the second frame. The storage cells and light sensitive cells contain the first and second frames respectively, and then these two frames are transferred sequentially to the video output for acquisition and cross-correlation calculation by the FlowMap processor. A pair of input image from CCD camera is shown in Figure 4.5 as a sample.

(a)

(b)

Figure 4.5: Image pair obtained from CCD camera

CHAPTER 4: Experimental Methodology

Table 4.2: Specification of CCD camera

Dimensions	Camera Housing 135×65×65 mm
Lens mount	C-mount
CCD type	Progressive scan interline, Hamamatsu C8484-05
Active pixels	1280×1024 pixel
Camera bit resolutions	12-bit
Pixel pitch	6.45×6.45 μm
Dynamic range	1800:1
Readout noise	10 electrons r.m.s
Pixel clock rate	14.7 MHz
Max. single-frame rate*	12.2
Pulse interval range*	0.2 μs and up
Double frame rate*	Up to 5.6 Hz

4.2.3 FlowMap System Hub

This unit is a core member of the system, and it has a modular structure including correlator unit, input buffer and a synchronization unit. The correlation unit is to determine the vector maps, following the processing stages as illustrated in Figure 4.6. The input buffer allows continuous throughput of image maps from the camera to the correlator unit. It can read image maps from a CCD camera, store image maps in a memory area and either divide stored image maps into interrogation unit and send them to correlator unit or send the entire image to the PC. The synchronize unit provides communication links between the processor and other elements in the PIV system such as illumination device and camera as well as the user's own devices.

CHAPTER 4: Experimental Methodology

Figure 4.6: Processing stages in the correlator unit

4.2.4 FlowManager Software

FlowManager software runs on the Windows Operating system as an ordinary application, and it controls the entire process of PIV measurement. The communication between PC and the FlowMap processor is via a FlowMap System Hub installed in the FlowMap processor through TCP/IP communication link. During the measurement sequence, the FlowManager software controls the necessary synchronization sequence to acquire data, and stores the recording data. The actual measurement sequence is initiated by the occurrence of a trigger event, which can be an external TTL input to the processor, or original from the PC. For the pulsed laser, there are four modes including asynchronous, free-run, triggered and window triggered, which responded to the external triggering signal from the shaft encoder, and it was selected. The image pair acquisition was synchronized to the impeller rotation using 'once-per-revolution' signal from the shaft encoder. The signal is sent to the FlowMap System Hub processor, which then fires the laser and acquires the images from the PIV camera, so images could be taken at certain angular position of

CHAPTER 4: Experimental Methodology

the impeller. Moreover, the timing signals are used to control the camera to make sure that the CCD sensor is only exposed to the light from the illumination system, since the ambient light will reduce the signal-noise ratio. In order to measure the flow at different angular position, a delay was set between the external signal and the image acquisition. Two successive images are then transferred to the FlowMap processor and stored in the database of FlowManager software.

Once the particle images are acquired, the instantaneous particle-displacement fields can be calculated using the Fast Fourier Transform (FFT) Cross-Correlation (CC) routines within the FlowManager software, in order to increase the computational speed. FFT CC requires $N^2 \log 2N$ operations, whereas direct CC requires $N^2 M^2$, where N is the length of one side of the correlation and M is the size of the correlation map. Note that M must be the same as N for FFT [128]. The image maps are divided into interrogation areas of a rectangular shape and the length of each side of the rectangular is a number of pixels expressed in multiply of 2: 16, 32, 64, or 128. During the experiments, the interrogation area of 32×32 pixels was used with 50% overlap. The pulse duration of this PIV is 10 ns, and the particle movement during this time duration is very short so that the streaking effect can be neglected. For the pulse separation, it varied 10~25 μs at different measurement regions according to the local flow velocity. Subsequently, post-processing validation methods including peak height validation and velocity range validation would be applied on the raw vector map. Furthermore, calculation was adopted to derive other information from the vector maps such as streamline and vorticity. Finally, the data are ready for exporting in numerical format.

CHAPTER 4: Experimental Methodology

4.3 Experimental Procedure

Firstly, blood analog was mixed with seeding particles, and placed in a container for a day. Because the densities of the seeding particles were not uniform, lighter particles stayed near the top surface, whereas heavier particles deposited. Therefore, the blood analog at the middle region contained the particles with closer density to the blood analog, and was selected for PIV measurement. Before the measurement, it is important to make sure that all the PIV components, including the monitoring computer, laser, CCD camera and FlowMap System Hub, and the trigger signal cable from the shaft encoder are properly connected. The test model was then assembled and installed. After that, the fluid filled the entire test rig, and the air bubbles were removed by running the impeller at low speed. Turned on the PIV system, and let it run under continuous mode. Then, adjust the camera, positioning the test chamber, powers of the laser sheet from both frames and so on until the best image pair was obtained. During the measurement, the impeller was driven by the servo-motor at 9,500 rpm. When the motor was running, ‘once-per-revolution’ signal was sent to the PIV System Hub. Under the window triggered mode, the image pairs were only stored at certain angular position as introduced earlier. In order to get the planar flow field at other angular positions of the impeller, a trigger delay (dt) is used. When the impeller (motor) rotates at that specific angle, a trigger is sent to PIV system, image pairs are captured at a new angle (ωdt) after dt . Note that ω is the rotating speed of the impeller (rad/s).

Note that the flow field in the 0.2 mm clearance gap between the casing and impeller shroud, and the two gaps between the straightener & impeller and between the impeller & diffuser would not be measured, since the space was insufficient to generate enough vectors as demonstrated by yellow color in Figure 4.7. Additionally,

CHAPTER 4: Experimental Methodology

due to the presence of the opaque shaft and the bearing holder at the center, the laser sheet could not pass through the entire cross section, and only half was measured as demonstrated in black color in this figure. Since the PIV measuring area was much smaller than the entire area of the model, the measurement was conducted part by part as demonstrated in Figure 4.7. In order to present the measurement area clearly, blue and cyan colors are used alternatively. It can be observed that the measured area is larger than the corresponding flow region indicated by black, and its coverage varies according to the local cross section area in order to save image acquiring time, storage space and computing source during post-processing. Additionally, the measurement area partially overlaps with its adjacent one. At each measurement, 100 image pairs were collected.

Figure 4.7: Demonstration of the measurement positions on the cross section.

After the measurements were accomplished, FlowManager was used for the post-processing of the image pairs, and the results were exported in numerical format for further processing. The steps are described below:

- I. Masking. Drew an outline for each measurement location in order to mask the images outside the flow field, and then applied it to image pairs.
- II. Mean pixel values. Calculate the mean pixel values at each position for images in data set for both frames, and obtained one mean image pair.

CHAPTER 4: Experimental Methodology

- III. Image arithmetic. The mean image pair was subtracted from each image in the corresponding data set, leaving only the illuminated particle images against a black background in each image pair ideally.
- IV. Adaptive correlation. Use iterations for offsetting the second window for cross-correlation analysis. Began with a larger interrogation area, and the final area was 32 pixels $\times$ 32 pixels after 2 refinement steps. The correlation areas were overlapped by 50% in both horizontal and vertical direction.
- V. Peak Validation. Validated vectors based on the correlation peak height ratio between the highest peak and the second highest peak, and vectors below the ratio of 1.2 were deleted.
- VI. Range Validation. Rejected the vectors, which were outside the local velocity range.
- VII. Moving Average Validation. Validated vectors were based on the comparison between neighboring vectors. The rejected vectors were replaced by estimated vectors using interpolation.
- VIII. Masking Vector. The same mask was applied to the obtained vector field so that the vectors under the mask were disabled.
- IX. Vector Statistics. Calculate the mean velocity vector map for export.

The obtained velocity vector map was based on local coordinate with the lower left corner recognized as its reference. After it was imported into Tecplot, and it was transferred to the corresponding location according to the coordinate of the numerical model. The same procedure was applied to every velocity vector map, and the vector maps in each region were combined together.

Chapter 5:

Original Axial Flow VAD

The physical configuration of the original axial flow VAD has been introduced in Chapter 3, and its simulation results including the static pressure and velocity vector distributions will be presented in this chapter based on the boundary conditions of 10,000 rpm rotating speed and 5 L/min flow rate. In addition, the blood damage index of the original VAD is evaluated, which is used as the parameter of the baseline model for comparison with the improved design later.

5.1 Flow Fields and Static Pressure Distributions

Figure 5.1 shows the static pressure distributions along the surfaces of the blades and hubs of three inner components at certain angular position. Note that the impeller is rotating in clockwise direction when looking in the positive Z direction from the straightener and this is applied to all models in the study. In the straightener, a high static pressure region is observed near the tips of all the blades close to its trailing edge, due to the flow from the rotating impeller. In addition, the static pressure on the straightener hub basically decreases along Z axis. In the impeller region, it can be observed that the static pressure increases in the radial direction along impeller blades where more energy is imparted into the blood at higher radial location. Moreover, the static pressures on the pressure side are higher than those on the suction side at the same Z position. It can be observed on the impeller hub that the static pressures are not uniformly distributed and low static pressure regions are found near the trailing edge, which implies irregular flow fields. Due to the curvature of the diffuser blade,

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

the static pressures on side P are higher than those on side S, indicating the interaction between swirling flow and stationary blades. It can also be found that the static pressure increases along the diffuser channel due to the flow deceleration, which is converting the kinetic energy into static pressure.

Figure 5.1: Static pressure distributions along the surfaces of the blades and hubs of three inner components of the original axial flow VAD model.

Since the flow is three-dimensional, the flow patterns at different radial location may vary. Therefore, three flow fields in the sectional circular planes are selected at $r=4$ mm (near the hub), 5 mm (middle) and 6.5 mm (near the pump casing) in the straightener, impeller and diffuser are shown in Figures 5.2, 5.3 and 5.4 respectively.

On the sectional circular plane at $r=4$ mm in the straightener as shown in Figure 5.2 (a), a vortex is induced at the middle of the channel due to the influence of the strong vortex located at the upstream of the channel close to the blade on the $r=6.5$ mm sectional circular plane as found in Figure 5.2 (c). In addition, it can be observed in Figure 5.2 that the area affected by the vortex reduces from high radius to low radius. Due to the rotating impeller, the swirling flow impinges heavily on the straightener blade near the tip region of the trailing edge with a backflow induced as shown in

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

Figure 5.2 (c), and it is correlated to the high static pressure near the tip region of the straightener blade trailing edge as shown earlier in Figure 5.1. It can be deduced from Figure 5.2 that the irregular flow field in the straightener channel is related to the improper design of the impeller blade profile at the leading edge section.

(a) $r=4$ mm

(b) $r=5$ mm

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

(c) $r=6.5$ mm

Figure 5.2: Velocity vector distributions on the circular planes in the straightener region at (a) $r=4$ mm, (b) 5 mm and (c) 6.5 mm

On the sectional circular plane at $r=4$ mm of the impeller as shown in Figure 5.3 (a), the blood flows smoothly along the channel between pressure (bottom) and suction (top) sides about three quarter length of the impeller channel, probably due to the mild pressure difference at these regions. Whereas the dominant flow at the last quarter of the impeller channel is in backward direction. In addition, the vortices induced near the trailing edge should be related to the zero trailing edge angle of the impeller blade, which has resulted in high static pressure difference between pressure and suction sides. As the circular plane increased to $r=5$ mm in Figure 5.3 (b), which is around the middle between the impeller shroud and hub, the flow pattern near the impeller entrance is still smooth, and the backflow near the exit region is reduced. On the circular plane close to the impeller tip at $r=6.5$ mm as shown in Figure 5.3 (c), the flow pattern is quite irregular. Based on the observations on different sectional circular planes, it can be found that the blood flows into the impeller channel at low

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

radial location, and it interacts with the backflow from the impeller exit inducing vortices. As a result, a portion of blood flows backward along the impeller shroud into the straightener channel, which is consistent with the observation as shown earlier in Figure 5.2 (c), and the blood at the other region flows out of the impeller channel close to the impeller shroud. The flow field in the impeller channel is mainly due to the poor impeller blade design, which has a small wrap angle of 90° and cannot gradually accelerate the blood without flow separation. Moreover, it seems that higher hub to tip ratio could reduce the difference of flow pattern at various circular planes. In other words, the impeller hub diameter should be increased, while keeping the current impeller tip diameter constant, it is thus preferred to have smaller blade height.

(a) $r=4$ mm

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

(b) $r=5$ mm

(c) $r=6.5$ mm

Figure 5.3: Relative velocity vector distributions on the circular planes in the impeller region at (a) $r=4$ mm, (b) 5 mm and (c) 6.5 mm

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

In Figure 5.4 (a), it can be observed that the flow in the diffuser channel at $r=4$ mm are all in the negative Z direction back into the impeller. In addition, the tangential component of the velocity is large in magnitude and is opposite to the rotating direction of the impeller. The backflow at the leading edge is consistent with the backflow region close to the impeller hub at its exit as shown earlier in Figure 5.3 (a). It can also be found that a vortex is present near the trailing edge in the clockwise direction due to the static pressure difference between side P and side S. At the downstream of the diffuser blade, the tangential velocity component is much higher than the axial velocity components. On the circular plane at $r=5$ mm in Figure 5.4 (b), the tangential velocity component at the diffuser entrance is in the same direction of the impeller, and the blood initially flows along both sides of the diffuser blade near the leading edge. Due to the vortex located at the trailing edge, the flow in the channel near side S is in reverse direction and a small region near side P is in forward direction. Another vortex is thus induced, near the diffuser leading edge in the counterclockwise direction between the forward and backward flows. At the downstream of the diffuser blade, the flow direction is slightly tilted to positive Z direction than that at the circular plane of $r=4$ mm. In Figure 5.4 (c), the dominant flow direction is forward, since the flow at higher radius has larger velocity magnitude so that it is more resistible to the adverse pressure gradient. Due to the shape of the diffuser blade and the incoming flow direction, the blood leaving the impeller tends to flow along side P. For the flow close to side S, the blood cannot overcome the adverse pressure gradient, and detaches from side S near the trailing edge. It can be observed from Figure 5.4 (a) to (c) that flow consists of a high tangential velocity component at the downstream of the diffuser blade, and it indicates

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

that the diffuser design should be improved in order to be more efficient in converting the tangential velocity component into the static pressure.

(a) $r=4$ mm

(b) $r=5$ mm

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

Figure 5.4: Velocity vector distributions on the circular planes in the diffuser region at (a) $r=4$ mm, (b) 5 mm and (c) 6.5 mm

Figure 5.5 shows a schematic view of blood flow through each region. Note that this figure only intends to demonstrate the flow distribution in each region of the original VAD so that the blades are not included. It can be found that the leakage flow is present in the clearance gap between the impeller shroud and the pump casing due to the adverse pressure gradient along Z axis. In addition, the simulation results show that the leakage flow rate is 0.45 L/min, which is 9.0% of the total flow rate (5 L/min). Although the clearance gap between the impeller shroud and the pump casing is not a common design in axial VAD, it is widely adopted in the centrifugal VAD. Researchers tried to eliminate the leakage flow by various means, and the only feasible solution is to design the blades on the outer surface of the shroud. However, it increases the difficulty of manufacturing the pump components and results in higher blood damage. Therefore, it is not applied in most design including the present VAD.

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

Smaller gap can reduce the vibration of rotor and increase the suspension force. In contrast, a higher leakage flow rate can provide a good washout in the trade off of higher power consumption. Through the investigation on the gap region of the original pump, it is helpful in optimizing the later designs.

Figure 5.5: Schematic view of blood flow through each region

The dominant flow direction of blood within the gap is tangential with the maximum velocity on the rotating cylinder and zero velocity on the stationary pump casing, and it is of interest to investigate the flow at the gap longitudinal cross section along Z axis. Figure 5.6 shows the axial and radial velocity vectors and static pressure contours on the cross section. The flow field within the gap is complicated, and the gap region is thus divided into five sections according to the flow pattern. It can be observed in section A that the blood close to the rotating cylinder is driven by the centrifugal force via the viscous effect, and the force is high enough to overcome the adverse pressure gradient so that blood is flowing from low radial location to high radial location. It is obvious that further away from the rotating cylinder in the Z direction of the gap the centrifugal force is diminishing and the pressure force becomes dominant resulting in backflow along the region close to the pump casing (inner side). On the opposite side along the section C of the gap, it can be observed

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

that the blood flows from the high to low radial locations near the pump casing (inner side), and this should be due to the high pressure difference between them. However, the centrifugal force becomes dominant at the region close to the rotating cylinder resulting in a flow from low to high radial locations. In section B, the axial velocity is basically parabolic in shape along the negative Z direction due to the static pressure difference and no-slip boundary condition on both the rotor cylinder and pump casing (inner side), and the flow in this region is quite smooth. The velocity profiles at sections AB and BC should be due to the transient processes between sections A & B and between sections B & C respectively. Although the flow field within the gap is complicated in 2D plane, the potential of thrombus formation is not severe when the high tangential velocity is taken into consideration.

Figure 5.6: Axial and radial velocity vector distribution on the longitudinal section of the clearance gap (Region is highlighted by the red rectangular in the schematic view)

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

5.2 Hemolysis

Figure 5.7 illustrates the scalar shear stress distributions on the surfaces of the blades and hubs of three inner components of the original axial flow VAD model. For the straightener, the high scalar shear stress located near the trailing edge is due to the backflow from the impeller as shown earlier in Figure 5.2 (c). As the blood flows into the impeller region, it is driven by the high-speed rotating impeller, so the high scalar shear stresses at the impeller leading edge are almost unavoidable. The blood is tangentially accelerated gradually along the impeller blades, and it should have a lower relative velocity with respect to the impeller near the trailing edge than that near the leading edge. Therefore, the relatively high scalar shear stresses at the impeller trailing edge could be greatly reduced if the backflow from the diffuser is removed. In contrast with the impeller, the blood at the diffuser leading edge has a high tangential velocity component, so the high scalar shear stresses induced at these locations are also inevitable.

Figure 5.7: Scalar shear stress distributions on the surfaces of the blades and hub of three inner components of the original axial flow VAD model

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

In order to show the scalar shear stress distribution inside the flow domain, Figures 5.8 (a), (b) and (c) show three cross sectional planes (A, B and C), which are selected in the straightener, impeller and diffuser respectively, with the corresponding Z positions at -5 mm, 10 mm and 15 mm as demonstrated in Figure 5.8 (d). It can be found in Figure 5.8 (a) that the scalar shear stress near side I is higher than that near side II due to the vortex induced by the impeller leading edge as shown earlier in Figure 5.2. Despite the velocity gradients, the flow nature also affects the scalar shear stress significantly. Due to the boundary layer effect, the scalar shear stress at the near wall region is mainly affected by laminar flow and is lower in Figure 5.8 (b). As the flow further away from the wall, the effect of turbulence becomes stronger than that of laminar flow. As a result, the scalar shear stress at the core region is high in Figure 5.8 (b). In the clearance gap between the impeller shroud and the pump casing, the flow is affected by the boundary layer, and the scalar shear stress is almost uniform. As the blood flows into the diffuser region, the interaction between the blood and the diffuser blade is much stronger than those in the straightener and impeller as shown in Figure 5.8 (c).

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

(a) Straightener

(b) Impeller

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

Figure 5.8: Scalar shear stress distributions on the cross sectional planes perpendicular to axial axis of the (a) straightener, (b) impeller, (c) diffuser as indicated in (d) the original axial flow VAD model

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

Figure 5.9 compares the mass-weighted scalar shear stress distributions within each region of the axial flow VAD. Although some induced scalar shear stresses can reach as high as 1,500 Pa, the corresponding percentage is too low to be seen in the figure. It can be found in the straightener that more than 90% of the scalar shear stresses are lower than 200 Pa and most of them are within 50 Pa. In addition, the percentage decreases continuously as the corresponding scalar shear stress increases, and the mass-weighted average scalar shear stress in the straightener region is 41.3 Pa. Based on the fast estimation method, the blood damage index induced in the straightener is around 1.45%. The scalar shear stress level in the impeller region is much higher than those in other regions. The mass-weighted average scalar shear stress in the impeller is 190.4 Pa with the blood damage index of 2.89%. The mass-weighted scalar shear stress distributions in the diffuser region indicate that the 3.02% blood damage index is higher than 1.45% in the straightener with the average stress of 86.5 Pa. For the scalar shear stresses in the gap region, the mass percentage initially increases in the region between 0 and 200 Pa with average scalar shear stress of 125.1 Pa, however, it dramatically decreases thereafter. Moreover, all of them are almost less than 400 Pa and the blood damage index generated by the clearance gap alone is 0.77% based on the leakage flow rate. The overall blood damage index of the original axial flow VAD is 5.92%. Note that this is not a simple summation of all the regions, since the estimation model is non-linear. In addition, the mass-weighted average scalar shear stress in the entire model is 86.3 Pa. Therefore, its blood damage index is severe, and it is correlated to the irregular flow field as discussed earlier.

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

(a) Straightener

(b) Impeller

(c) Gap

(d) Diffuser

Figure 5.9: Mass-weighted scalar shear stress distributions in (a) straightener, (b) impeller, (c) gap and (d) diffuser of the original axial flow VAD

Since the flow within the original model is turbulent, the scalar shear stress is based on both laminar and turbulent shear stress tensors. In order to estimate their individual contributions, Figure 5.10 shows the ratio of the turbulent stress to viscous stress on a cross section in the impeller region, and its position is on Plane B as shown earlier in Figure 5.8 (d). It can be observed that the viscous stress induces mainly at the near wall region. At the core of the flow, the turbulent stresses are much higher than viscous stresses so that they are the main factor in causing blood damage. Therefore, it is essential to improve the flow field inside the pump so that it can operate at lower

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

rotating speed with lower turbulence intensity. Alternatively, the hub to tip ratio can be increased so that the near wall region is increased with respect to the core region. For example, the flow in the clearance gap has a low turbulent to viscous stress ratio, because the width of the clearance gap is small.

Figure 5.10: Distribution of turbulent stress to viscous stress ratio on a cross section in the impeller region.

Although the fast estimation method can give the overall blood damage induced by the axial flow VAD, it is impossible to indicate the exact location where high or low shear stresses occur. Therefore, another method based on particle track is utilized to estimate the hemolysis, and 737 particle tracks are collected with the distributions of the corresponding blood damage index shown in Figure 5.11. The average of the blood damage indices of the 737 particle tracks are calculated, which is 14.31%. Note that the blood damage index indicates the possibility of hemolysis with the maximum value of 1. It can be deduced that the blood damage index of the original flow VAD is severe.

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

Figure 5.11: Blood damage index distributions for the original axial flow VAD

Two typical particle tracks are selected in order to indicate the detailed information on both scalar shear stress and blood damage index as shown in Figure 5.12. Note that the first horizontal line at $Z=0$ mm in part (i) of the figure indicates the interface between the straightener and impeller and the second one at $Z=20$ mm indicates the interface between the impeller and diffuser. Based on the relationship between the Z position and the residence time, Figures 5.12 (a) and (b) show (i) the Z position, (ii) the scalar shear stress induced and (iii) the accumulated blood damage against the residence time of typical particle streamlines without and with backflow near the interface between the impeller and diffuser. It can be observed in the figure that the blood at the upstream of the straightener (<-20 mm) experiences a negligible scalar shear stress and blood damage index. As the blood flows forward, the flow field is irregular with vortices as indicated earlier in Figure 5.2, the scalar shear stress fluctuates with the magnitude less than 400 Pa. Near the interface between the straightener and the impeller, the scalar shear stress increases significantly due to the high-speed rotating impeller. In the impeller channel, the blood is accelerated in the

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

tangential direction, and the velocity difference between the blood and impeller decreases so that the scalar shear stresses become smaller along the impeller. When the blood flows into the diffuser region, the swirling blood interacts with the stationary diffuser, so the scalar shear stresses near the interfaces are high resulting in high blood damage index. When backflow occurs near the interface between the impeller and the diffuser as shown in Figure 5.12 (b), blood particle experiences high scalar shear stresses twice. As the blood flows out of the diffuser region (>40 mm), the scalar shear stress is lower than 50 Pa, and the blood damage index almost remains constant. Therefore, it can be deduced that the blood damage is significantly related to the flow field, and the vortices should be eliminated, especially near the interfaces, if possible.

(a)

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

(b)

Figure 5.12: Tracks of typical particles: (a) with a smooth flow and (b) backflow at the interfaces for time history (i) the Z position, (ii) the scalar shear stress and (iii) the accumulated blood damage index

5.3 Summary

The aim of this investigation is to illustrate the flow fields within the original model, and to identify the critical regions with high scalar shear stress and high blood damage index. Generally, most bloods flow forwards with vortices around the interfaces between the impeller and both the straightener and diffuser. The suction effect of the rotating impeller causes a static pressure drop across the straightener, which generates a forward flow, and the static pressure rapidly increases in the impeller region due to the energy transfer. The forward flow in the impeller is close to the hub initially and then turning towards the shroud due to the vortex close to the hub at the trailing edge, which affects the static pressure generated across the impeller. The irregular flow at

CHAPTER 5: ORIGINAL AXIAL FLOW VAD

the impeller leading edge should be mainly due to the steep blade profile variation, which could not guide the flow as desired. In the diffuser region, vortices are present in the diffuser channel due to the swirling flow, adverse static pressure gradient and improper diffuser blade profile.

A leakage flow is present in the clearance gap between the impeller shroud and pump casing, mainly due to the adverse static pressure distribution. The dominant flow pattern in the clearance gap is tangential, since it is affected by the high speed rotating impeller shroud. The axial and radial velocity vector distribution on the longitudinal cross section of the clearance gap indicates that the flow field is not only affected by the static pressure gradient but also the centrifugal force. The turbulent to viscous stress ratio indicates that the flow in the clearance gap is mainly affected by the viscous stress.

It is found that the blood damage induced by the VAD is severe and it is mainly damaged by the turbulent stress. As a result, it is critical to reduce the turbulence intensity level by smoothing the flow field so as to reduce the blood damage. Some particles are sampled, and it is found that high blood damage happens at the interfaces so that it is important to design the impeller and diffuser properly in order to ensure smooth flowing route. Based on the observations above, the original model should be optimized so as to smoothen the flow field and to reduce blood damage index.

Chapter 6:

Optimization of Impeller

The simulation results show that the flow field inside the original VAD model is not up to expectation. The flow interactions between the rotating impeller and both the stationary straightener and diffuser are strong. These have induced vortices and flow separation, mainly due to the improper design of the impeller. Since the impeller is the most important component, it is of interest to investigate the flow field induced by the impeller alone without the interactions between the blades of both straightener and diffuser, i.e. the straightener and diffuser are assumed to have no blade in the simulations presented in this chapter as shown in Figure 6.1 (a). Based on the previous observation, the leakage flow rate through the clearance gap between the impeller shroud and the pump casing is quite low as compared to that through the impeller channel, so the clearance gap has limited influence on the flow field in the impeller channel. The clearance gap is thus excluded initially during the optimization process of the impeller for the ease of mesh generation and simulation. Note that the flow field in the clearance gap is presented in the later section of this chapter.

Since the ‘rotor-stator’ interaction is avoided during the impeller optimization, Multiple Reference Frame (MRF) model is sufficient, and requires less computational resources than the sliding mesh model. Moreover, in order to take the advantage of periodic flow field, only a portion of the model has been simulated according to the physical configuration. For example, although there are four blades in the original impeller model as shown in Figure 6.1 (a), only a quarter of the original impeller

CHAPTER 6: OPTIMIZATION OF IMPELLER

model is numerically built as demonstrated in Figure 6.1 (b). Stationary straightener and diffuser regions are indicated in dark gray color, and rotating impeller region is indicated in light gray color in the figure. Since the flow pattern at the desired flow rate is a major concern, the same velocity inlet was applied to each model in order to maintain 5 L/min flow rate. Additionally, the outlet pressure is fixed at 10,000 Pa and the rotating speed is 10,000 rpm.

Figure 6.1: (a) Physical and (b) numerical model of the original impeller model.

CHAPTER 6: OPTIMIZATION OF IMPELLER

6.1 Original Impeller

The static pressure distributions at $r=4$ mm, 5 mm and 6.5 mm circular planes (from the hub to the tip) are shown in Figures 6.2 (a), (b) and (c) respectively. Firstly, it can be found that the static pressures near the tip in Figure 6.2 (c) are much higher than those near the hub in Figure 6.2 (a), and the static pressure difference in radial direction is even higher than that in the axial direction from the impeller entrance to exit. Secondly, the static pressures increase steeply at higher radial location than those at lower radial location around both pressure and suction sides. Thirdly, the highest static pressure occurs in the last half of the impeller near pressure side on each radial cross section presented in Figure 6.2. The simulation results show that static pressure generated across the whole pump is only 4.5 kPa, which is much lower than the desired value, and that across the impeller is 8.6 kPa. Note that the power consumption of the original impeller model is 7.5 W with 10.4% efficiency.

(a) $r=4$ mm

CHAPTER 6: OPTIMIZATION OF IMPELLER

(b) $r=5$ mm

(c) $r=6.5$ mm

Figure 6.2: Static pressure distributions on circular planes at (a) $r=4$ mm, (b) 5 mm, and (c) 6.5 mm respectively

The same circular cross sectional planes at $r=4$ mm, 5 mm and 6.5 mm are used to demonstrate the relative velocity vector and streamline distributions in the original

CHAPTER 6: OPTIMIZATION OF IMPELLER

impeller region in Figures 6.3 (a), (b) and (c) respectively. It can be observed in Figure 6.3 (a) that near the impeller leading edge, the blood flows smoothly along both pressure and suction sides of the impeller blade without any vortex or flow separation. However, near the end of the impeller blade, the blood is skew towards the blade at the pressure side, and then flows around the trailing edge, creating a backflow region along the suction side for about 4mm in length. The flow at $r=5$ mm circular plane as shown in Figure 6.3 (b) is similar to that at $r=4$ mm circular plane as shown in Figure 6.3 (a), but with less order. The skewing of the flow toward the blade pressure side occurs much earlier at about half length of the impeller, creating a thin layer of backflow along the blade (pressure side). At the second half of the impeller, it can be seen that the flow impinges on the pressure side of the blade and then turns around the blade trailing edge with a backflow region occupying the impeller channel close to the suction side.

The flow at $r=6.5$ mm circular plane is the most complicated as shown in Figure 6.3 (c). The blood flows in negative Z direction from the pressure side to suction side around the leading edge, due to the high static pressure difference between the two sides of the impeller blade as demonstrated earlier in Figure 6.2 (c). The flow along the suction side then interacts with the backflow in the impeller channel close to suction side with a flow separation found on the blade suction side near the leading edge. The bloods at the second half of the blade channel are complicated with vortices found in the channel near the suction side. In summary, near the impeller blade trailing edge, flow separation is observed on all the three circular planes close to the suction side as shown in Figure 6.3, and it is consistent with the low static pressure region as shown earlier in Figure 6.2. The high static pressure difference between pressure and suction sides at the impeller trailing edge induces blood flow from

CHAPTER 6: OPTIMIZATION OF IMPELLER

pressure side to suction side, which is due to the zero trailing edge angle design. Generally, it can be observed that the flow pattern on different radial location varies significantly. The blood flows forward into the impeller region near the hub, whereas it flows out of the impeller close to the tip with backflow found near the hub.

(a) $r=4$ mm

(b) $r=5$ mm

CHAPTER 6: OPTIMIZATION OF IMPELLER

Figure 6.3: Relative velocity vector distributions on circular planes at (a) $r=4$ mm, (b) 5 mm and (c) 6.5 mm

Figure 6.4 shows the scalar shear stress distributions in the impeller domain of the original impeller model. Although the scalar shear stresses induced can attain the level higher than 1000 Pa, they have constituted only an extremely low mass percentage, which is too small to be distinguished in the figure. It can be found that mass percentage of scalar shear stress decreases continuously with the increase of the stress, and more than 80% of blood experience a shear stress less than 200 Pa. The mass-weighted average scalar shear stress is 132.7 Pa, and the blood damage index induced by the impeller is around 4.6% based on the fast estimation method, which has been introduced in Chapter 3.

CHAPTER 6: OPTIMIZATION OF IMPELLER

Figure 6.4: Scalar shear stress distributions in the impeller domain of the original impeller model

6.2 Different Impeller Designs

In order to improve the impeller design, various hub diameter and wrap angles are tested, and their configurations are illustrated in Table 6.1. Note that these impellers have the same outer diameter and length as the original impeller. As observed earlier in Figure 6.3, the irregular flow field near the impeller trailing edge is due to the zero trailing edge angle, which induces high static pressure difference between the pressure and suction sides. Moreover, a non-zero trailing edge angle is commonly used in other axial flow VADs [13, 27, 95], so a 30° trailing edge angle is adopted for impellers in Table 6.1. Although 15° and 45° trailing edge angles have been tested, it is found that the irregular flow near the 15° trailing edge is still persistent (not shown here), and the static pressure head generated across the impeller with 45° trailing edge angle is the lowest without further improving the flow field. While the impeller with 180° wrap angle has four blades with equal length and identical profile, the impeller with 270° or 360° wrap angle has two long blades and two short blades, and the short blade profile starts at one quarter of the length (i.e. 5 mm) from the impeller entrance as shown in

CHAPTER 6: OPTIMIZATION OF IMPELLER

Table 6.1. This design is to provide wider space for impeller channels at the leading edge and the function of the shorter blades can be activated at later section. The effect of the blade height (H) is also considered in the design as shown in Table 6.1, with three H values of 2, 2.5 and 4 mm being studied.

Table 6.1: Impellers with different configurations

H (mm)	Wrap Angle (°)		
	180	270	360
4.5	(a) P1	(b) P2	(c) P3
	(d) P4	(e) P5	(f) P6
	(g) P7	(h) P8	(i) P9

Table 6.2 shows the efficiency (η), power consumption (P), the static pressure head across impeller (H(I)) and that across the pump (H). It can be found in Table 6.2 that the power consumption (P) decreases with the blade height for every wrap angle. In the models with 180° and 270° wrap angles, the efficiency (η) varies slightly when the blade height decreases from 4.5 mm to 2.5 mm, but increases more when the blade

CHAPTER 6: OPTIMIZATION OF IMPELLER

height further decrease to 2 mm. While the efficiency (η) of 360° wrap angle is decreasing from 4.5 mm to 2 mm.

Firstly, the comparison is conducted in the ascending order of wrap angle under the same blade height, i.e. $P2 \rightarrow P3$, $P4 \rightarrow P5 \rightarrow P6$, and $P7 \rightarrow P8 \rightarrow P9$. It can be observed in Table 6.1 that the variation of gradient is in descending order from 180° to 360° wrap angle, i.e. the variation of gradient for 180° wrap angle is the largest among the three wrap angles. The large variation of gradient can induce big mismatch between the flow direction and the blade profile near the leading edge, and this results in high energy lost. As a result, the power consumptions of the models with 180° wrap angle are higher than other models with 270° or 360° wrap angles. However, the large wrap angle will result in longer flow channel in impeller, thereby increasing the frictional loss. Due to the tradeoff between the mismatch at the leading edge and the frictional loss, the efficiency increases when the wrap angle increases from 180° to 270° , and then decreases when the wrap angle further increases to 360° .

Secondly, the comparison is conducted in the descending order of blade height under the same wrap angle, i.e. $P1 \rightarrow P4 \rightarrow P7$, $P2 \rightarrow P5 \rightarrow P8$, and $P3 \rightarrow P6 \rightarrow P9$. As the blade height reduces, the contacting area between the flow and the impeller is also decreased, thereby the power consumption decreasing. The decrease of the blade height has restrained the flow, and reduced the mismatch between the flow direction and the blade profile resulting in a smoother flow field. Therefore, it is found that the static pressure head across the impeller alone ($H(I)$) increases from $P1 \rightarrow P4 \rightarrow P7$ and $P2 \rightarrow P5 \rightarrow P8$. However, when the blade height decreases, the average velocity increases due to the smaller cross sectional area. As a result, the frictional losses of the flow passing through the straightener and diffuser hub have increased as indicated by the pressure head difference between the overall pressure head (H) and the

CHAPTER 6: OPTIMIZATION OF IMPELLER

pressure head across impeller alone ($H(I)$). The increase of the static pressure head generated across the impeller alone ($H(I)$) has a tradeoff of the increased frictional loss. Comparing P1 with P4 and P2 with P5, it can be observed that the efficiency does not vary significantly, since both pressure head (H) and power consumption drops. Comparing P4 with P7 and P5 with P8, the power consumption decreases marginally, but the pressure head (H) generated increases due to the improved flow field in the impeller region. As a result, the efficiency increases. Among the models with wrap angle of 360° (i.e. P3, P6 and P9), the leading edge angle matches the flow direction, and it is not affected by the decrease of the blade height. Therefore, the frictional loss is the dominate factor among these models, and the generated pressure head drops (both H and $H(I)$) with the decrease of the blade height.

As shown in Table 6.2, the impeller with 270° wrap angle blade has a higher static pressure head generated than those with 180° or 360° wrap angle with the same hub diameter, and the blade height is significantly related to the static pressure generated. Therefore, the impellers (i.e., Models P2, P5 and P8) with 270° wrap angle blades are selected for the testing on the effects of the different blade height.

Table 6.2: Efficiency, power consumption and static pressure heads generated across the impeller and the pump in different models.

	180°				270°				360°			
Blade Height	η	P	$H(I)$	H	η	P	$H(I)$	H	η	P	$H(I)$	H
mm	%	W	kPa	kPa	%	W	kPa	kPa	%	W	kPa	kPa
4.5	P1				P2				P3			
	16.2	5.58	8.5	6.3	19.3	5.0	9.5	7.4	18.8	4.2	7.2	5.6
2.5	P4				P5				P6			
	16	3.93	9.1	4.6	20.4	3.6	10.3	6.1	9.5	3.3	4.2	1.3
2	P7				P8				P9			
	18.7	3.66	10.5	5.5	25.0	3.5	12.5	8.0	6.6	3.1	3.1	0.5

CHAPTER 6: OPTIMIZATION OF IMPELLER

Figure 6.5 shows static and dynamic pressure distributions along Z axis in P2, P5 and P8 models. It can be observed in the figure that the average static pressure profile of P2 model is not very smooth as compared with those of P5 and P8 models, due to the irregular flow field in the impeller channels. In addition, the simulation results show that the power required by P2 is 5 W with the efficiency of 19.27%. Compared with the original impeller model (7.5 W and 10.4%), P2 model has lower power consumption and higher efficiency due to the enlarged wrap angle. As the blade height is further decreased, the static pressure distributions in the impeller region generated by P5 and P8 models have less irregular variation. It can also be observed that the highest static pressure head is generated by P8 model, which has the lowest blade height ($H=2\text{mm}$). Comparing the dynamic pressure distributions in the straightener region, it is found that the impellers with $H=2$ and 2.5 mm (P5 and P8 models respectively) induce a small and gentle increase. In the impeller region, it is observed that the blade height is inversely proportional to the slope of the dynamic pressure increment and its peak, i.e. a lower blade height induces a higher dynamic pressure and shifts the peak further downstream near the impeller trailing edge. Furthermore, it is found that the blade height influences the power consumption and the efficiency. The simulation results show that the power consumptions of P5 and P8 models are 3.63 W and 3.48 W respectively, and the corresponding efficiencies are 20.35 % and 25.00%.

CHAPTER 6: OPTIMIZATION OF IMPELLER

Figure 6.5: Average static, dynamic pressure distributions along Z axis in P2, P5 and P8. SP: Static Pressure, and DP: Dynamic Pressure.

In order to investigate the influence of the blade height on the flow fields in P2, P5 and P8 models, a circular plane at $r=6.5$ mm is selected in each model, and the corresponding relative velocity vector and streamline distributions are illustrated in Figure 6.6. Note that only half of the models are simulated in order to take the advantage of axis-symmetrical shape of the design profile. It can be observed in Figure 6.6 (a) that the flow field in the impeller region of P2 is irregular with vortices, and it is consistent with the static and dynamic pressure distributions as shown earlier in Figure 6.5. When the blade height decreased from 4.5 mm to 2.5 mm, the flow pattern on $r=6.5$ mm circular plane is smooth without vortex and flow separation as illustrated in Figure 6.6 (b). The flow pattern in P8 as shown in Figure 6.6 (c) is not

CHAPTER 6: OPTIMIZATION OF IMPELLER

only similar to that in P5 after the blade height is further reduced to 2 mm, but is also further improved at the region near the exit with the flow out of the impeller directly.

(a) P2

(b) P5

CHAPTER 6: OPTIMIZATION OF IMPELLER

(c) P8

Figure 6.6: Relative velocity vector and streamline distributions on the circular plane at $r=6.5$ mm in (a) P2, (b) P5 and (c) P8

Figure 6.7 shows the scalar shear stress distributions in the impeller domains of P2, P5 and P8 models. It can be observed that the percentage of the scalar shear stress induced decreases continuously with the increased scalar shear stress for all models. Based on the simulation, the mass-weighted average scalar shear stresses in the impeller domains of P2, P5 and P8 are 95.8 Pa, 93.7 Pa and 72.9 Pa respectively, and it is consistent with the observation in Figure 6.7. Compared with the average scalar shear stress of 132.7 Pa in the original impeller, P2 is significantly improved due to the large wrap angle. Furthermore, based on the fast estimation method, the blood damage indices induced by the impellers of P2, P5 and P8 are 0.8%, 0.44% and 0.17% respectively. Therefore, it can be deduced that the blade height of 2 mm has induced the lowest scalar shear stress level in the impeller region among the three H studied.

CHAPTER 6: OPTIMIZATION OF IMPELLER

Figure 6.7: Scalar shear stress distributions in the impeller domain of P2, P5 and P8

6.3 Elimination of Short Blades

The original impeller has four identical blades, in which two blades are short and start at $Z=5$ mm, when the wrap angle increases in order to prevent the blockage of flow at the leading edge. Since the blade height has been reduced from 4.5 mm to 2 mm, the flow channel area is much reduced and the function of the short blades in guiding the flow may be unnecessary. As a result, a new impeller model, P8B2, is built, which has the same physical configurations of P8 except the elimination of the two short blades as shown in Figure 6.8 (a). Figure 6.8 (b) shows the relative velocity vector and streamline distributions in the impeller region on the circular plane ($r=6.5$ mm), and it can be found that the flow is smooth as indicated by the streamlines without vortex or flow separation. Therefore, the impeller with only two 2 mm height blades is also

CHAPTER 6: OPTIMIZATION OF IMPELLER

capable of maintaining a smooth flow field. Furthermore, it simplifies the manufacturing process and thus reduces the cost.

(a)

(b)

Figure 6.8: (a) Physical configuration of P8B2 and (b) Relative velocity vector and streamline distributions on the circular plane at $r=6.5$ mm in the impeller region of P8B2

Figure 6.9 shows the average static and dynamic pressure distributions in P8B2 model. For the ease of comparison, the distributions in P8 model are also included. The

CHAPTER 6: OPTIMIZATION OF IMPELLER

distributions of the average static and dynamic pressures in both P8B2 and P8 models are almost identical from the entrance till $Z=5$ mm, where the static pressure distribution in P8B2 model becomes higher than that in P8 model thereafter. This is expected since there is bigger channel area of P8B2 than that of P2 due to the elimination of two short blades. Note that P8 models has two short blades starting from $Z=5$ mm. The dynamic pressure distribution of P8B2 model is smoother than that of P8 model in the impeller, though the peak dynamic pressure near the impeller trailing edge in P8 model is also higher than that of P8B2 model. At the further downstream, the static and dynamic pressure distributions in both models are quite similar, and the static pressure generated in P8B2 model is higher than that in P8 model. The simulation results show that the power consumption of P8B2 model is 3.53 W with 27.14% efficiency, which are close to those of P8 (3.48 W with 25.00%).

Figure 6.9: Average static and dynamic pressure distributions in P8B2

CHAPTER 6: OPTIMIZATION OF IMPELLER

The scalar shear stress distribution in the impeller region of P8B2 model is similar to that of P8 model as shown earlier in Figure 6.7, the mass-weighted average scalar shear stress in the impeller domain of P8B2 model is 71.1 Pa with the corresponding blood damage index of 0.18%. These parameters of P8B2 model are also close to those (72.9 Pa and 0.17%) of P8 model. Finally, P8B2 model is selected instead of P8 model, because it reduces both the difficulty and cost in the manufacturing.

6.4 Influences of Clearance Gap

Since the magnet is embedded inside the impellers shroud instead of the impeller hub, there is a clearance gap between the outer surface rotor cylinder and the inner surface of pump casing. It is the specific design feature of this axial VAD, and it is also adopted in some other axial VADs [17, 20]. However, its effects on the axial VAD performance have not been studied. The clearance gap is widely present in centrifugal VAD, and its investigations [93, 120, 129-131] are helpful in understanding the similar feature in an axial VAD. Although the clearance gap results in a lower efficiency due to a leakage flow in general, a small gap width with a low flow rate through the gap reduces the washout effect so that it increases the risks of thrombus formation. Therefore, it is necessary to have optimal width of clearance gap. Besides the thrombus formation, hemolysis is another factor to be considered. During the optimization of the impeller, the clearance gap is not included in order to study the performance of the impeller alone, although it is not physically possible. In this section, the clearance gaps with 0.1, 0.2 and 0.3 mm widths will be investigated. For the ease of comparison, P8B2 is used in this section, and renamed as G0, since it does not have a clearance width. Note that P8B2 is not physically possible, and it is only used for comparison. The models with 0.1, 0.2 and 0.3 mm clearance gap are named

CHAPTER 6: OPTIMIZATION OF IMPELLER

G1, G2 and G3 respectively, where G and number denote the gap and its width respectively.

The simulation results show that the flow rates through the 0.1, 0.2 and 0.3 mm gap are 0.06, 0.38 and 0.74 L/min respectively. With respect to the 5 L/min flow rate, the percentages of the leakage flows are 1.26%, 7.43% and 14.85% respectively, so the leakage flow rate is reduced almost by half after the width of the gap is reduced from 0.3 mm to 0.2 mm. Furthermore, the percentages of the leakage flow in G3 are higher than 9.0% of the original VAD, which also has 0.3 mm clearance gap, due to the increased static pressure difference across the impeller.

The power consumptions of G1, G2 and G3 are 8.19, 5.85 and 5.76 W respectively with the corresponding efficiencies of 11.5%, 15.34% and 11.73%. Compared with G0, the three models with clearance gap have consumed higher power, since a substantial portion of torque is used to overcome the friction between the blood and the impeller shroud outer surface. However, G0 is impractical as mentioned earlier, and the G1 model with only 0.1 mm clearance gap between the pump casing and the impeller shroud has too low leakage flow rate in the gap with a high risk of hemolysis. The much higher power consumption in G1 is due to the high torque required on the impeller shroud in the 0.1 mm narrow gap. Since the leakage rate is so low, G1 has quite similar flow field and static pressure distribution to G0 in the main flow region as well as the scalar shear stress. As a result, G1 is not further discussed in this section, and it is not recommended in the design due to high power consumption and low leakage flow rate (poor wash-out effect).

Figure 6.10 shows average static pressure distributions in the three models. Note that for the average static and dynamic pressure distributions in the impeller region, only

CHAPTER 6: OPTIMIZATION OF IMPELLER

the impeller channel region between the impeller hub and the impeller shroud inner surface is considered and the clearance gap region between the outer surface of the impeller shroud and the pump casing is excluded. Note that the same condition is also applied to the later improved models unless otherwise stated. For the ease of comparison, the static pressure at $Z=-40$ mm of the three models is adjusted to zero and is set as the reference point. In other words, the static pressure distribution shown in these figures are the relative static pressure with respect to the static pressure at $Z=-40$ mm. It can be found that in the straightener upstream (from $Z=-40$ to -20 mm) and straightener regions (from $Z=-20$ to 0 mm), the average static pressure distributions of these three models are almost identical. In the diffuser and its downstream regions (from $Z=20$ to 60 mm), the shapes of the static pressure distributions are also similar, although the static pressure generated by G2 is lower than G0 but is much higher than G3. It can be found that the clearance gap affects the static pressures distribution in the impeller region, where the slope of G3 near the leading edge is the most gentle followed by G2. Therefore, a larger clearance gap results in a lower static pressure generation. As blood flows forward, the shapes of the three curves are similar except at the impeller exit, where the static pressure drop is slightly higher with a wider clearance gap.

It can be found that the dynamic pressure distribution in each model is almost the same from the pump inlet to the impeller entrance as shown in Figure 6.10. The radial velocities in all models are very low, so the dynamic pressure mainly reflects the magnitude of axial and tangential velocities. Moreover, the average axial velocities in these models are the same due to the same cross sectional area and flow rate. Therefore, the sudden shooting up of the dynamic pressure at the impeller entrance ($Z=0$ mm) in G2 and G3 are due to the swirling leakage flow from the clearance gap.

CHAPTER 6: OPTIMIZATION OF IMPELLER

Near the impeller exit ($Z=20$ mm), the dynamic pressure in G3 is the lowest, because part of the main flow enters the clearance gap. In contrast, the dynamic pressure of G0 is the highest, which does not have leakage flow.

Figure 6.10: Average static and dynamic pressure distributions in the three models

Figures 6.11 (a) and (b) show the tangential and radial velocity fields in the cross-sections of gap clearance at $Z=10$ mm of G2 and G3 respectively. Due to the no-slip boundary condition, blood on the impeller shroud (outer surface) and that on the pump casing have the highest and lowest tangential velocity respectively, and the radial velocities in the cross section are almost zero in both models. It can be observed that the velocity distribution in the 0.2 mm clearance gap is almost linear, so it is a circular laminar Couette flow. When the width of the gap increases to 0.3 mm, the velocity profile is changed, and is similar to the result in the axial VAD of Zhu et al.

CHAPTER 6: OPTIMIZATION OF IMPELLER

[20] as shown in Figure 6.11 (c). For the axial and radial velocity field in the middle longitudinal section of the impeller, the flow patterns in both models are similar to that of the original VAD as shown earlier in Figure 5.6, so they are not shown.

(a)

(b)

(c)

Figure 6.11: Tangential and radial velocity fields in the middle circumferential sections of the clearance in (a) G2, (b) G3 and (c) Zhu et al.'s axial VAD [20]

CHAPTER 6: OPTIMIZATION OF IMPELLER

Using the fast estimation method, the blood damage index ratios of G2 and G3 are 1.17 and 1.32 with respect to G0, so G2 is superior to G3 in term of the hemolysis. Furthermore, the average scalar shear stresses in the entire model of G0, G2 and G3 are 27.0 Pa, 28.2 Pa and 29.8 Pa respectively, which is also consistent with the blood damage indices estimated in the respective model.

Table 6.3 compares the blood damage index and the mass-weighted average scalar shear stress induced at each region for the three models studied in order to understand the effects of the clearance gap width. Note that according to the non-linear characteristics of the estimation method, the blood damage for the entire model is not a simple summation of all regions. It can be found that the clearance gap increases the blood damage level in the straightener, probably because the leakage flow with high tangential velocity interacts with the stationary straightener exit region. The blood damage is more severe when the width of clearance gap increases to 0.3 mm, since the leakage flow of G3 is almost doubled of G2 as mentioned earlier. In the impeller region, the blood damage index of model decreases with the decrease of clearance gap. For the diffuser, the blood damage increases with the decrease of the clearance gap, since a portion of blood from the impeller exit diverts into the clearance gap, which has reduced the impingement of blood on the stationary diffuser. In the clearance gap region, the average scalar shear stress of G2 is lower than that of G3, because the scalar shear stress of G2 is only contributed by laminar viscosity and that of G3 is contributed by both laminar and turbulent viscosities. The blood damage index is not only related to the scalar shear stress but also inversely related to the flow rate. Since the leakage flow rate in G3 is almost twice of that in G2, the blood damage index of G2 is 10% higher than that of G3 in the clearance gap.

CHAPTER 6: OPTIMIZATION OF IMPELLER

Table 6.3: Blood damage index and average scalar shear stress in each regions of each model. SSS: Scalar Shear Stress

	Straightener		Impeller		Gap		Diffuser		Overall	
	Index (%)	SSS (Pa)	Index (%)	SSS (Pa)	Index (%)	SSS (Pa)	Index (%)	SSS (Pa)	Index (%)	SSS (Pa)
G0	0.0029	2.3	0.18	71.1	-	-	0.056	40.4	0.18	27.0
G2	0.0082	3.1	0.15	68.8	0.54	117	0.048	38.8	0.21	28.2
G3	0.0301	3.8	0.16	77.9	0.48	125.1	0.043	36.9	0.24	29.8

Figure 6.12 shows the scalar shear stress distributions in the clearance gaps of G2 and G3, and it can be found that the percentage initially increases with the scalar shear stress, but significantly reduces after 200 Pa. In addition, the percentage is negligible when the scalar shear stress is higher than 350 Pa. It can also be found that the percentage of the scalar shear stress (>150 Pa) distribution in G2 is lower than that in G3, and the corresponding average scalar shear stresses are 28.2 Pa and 29.8 Pa as mentioned earlier. The simulation results show that the flow in the 0.2 mm clearance gap is almost laminar, and the larger clearance gap has activated the turbulent flow. Finally, G2 is selected for further investigation, since it has better hydraulic and hematologic performances than G3.

Figure 6.12: Scalar shear stress distribution induced by the clearance gap in G2 and G3

CHAPTER 6: OPTIMIZATION OF IMPELLER

6.5 Model with Clearance Gap between Impeller Tip and Pump Casing

Casing

Beside the current design, where the magnet is embedded inside the impeller shroud as demonstrated in Figure 6.13 (a), in most axial flow VADs the magnetic material is embedded inside either impeller blades or impeller hub as demonstrated in Figure 6.13 (b). Therefore, the later design does not have an impeller shroud, which is named as No Shroud (NS) model hereafter, and a clearance gap is maintained between the impeller tip and the pump casing. The two designs, NS and G2 models, are compared in this section. Note that the 0.2 mm clearance gap is maintained between the tip and the pump casing in NS, for the equivalent comparison with G2.

Figure 6.13: Schematic view of (a) G2 and (b) NS.

Figure 6.14 shows the average static and dynamic pressure distributions along Z axis in NS. For the ease of comparison, those of G2 are also included. The static pressures

CHAPTER 6: OPTIMIZATION OF IMPELLER

generated by NS and G2 models are 10.2 kPa and 8.02 kPa respectively. It can be observed that both static and dynamic pressure distributions in the straightener and diffuser regions are almost identical in the two models, and the main differences are in the impeller region. The static pressure of NS increases more steeply, and the static pressure across the impeller region is also higher than that of G2. In contrast, the dynamic pressure of NS is lower than that of G2 in the impeller region, especially near the impeller trailing edge, and this is because a stationary pump casing rather than the rotating impeller shroud encloses the NS impeller. In the diffuser region, the dynamic pressure of NS is slightly lower than that of G2, which means the flow in the diffuser region of NS has lower tangential velocity when the average axial velocities are the same and the radial velocities are quite low in both models. The simulation results show that NS requires 4.58 W with 22.66% efficiency, whereas, the power consumption of G2 is 5.85 W, and the efficiency is 15.34%. It is found that the powers consumed by the impeller shroud and the impeller of G2 are 2.44 W and 3.41 W respectively. Thus, the impeller shroud increases the power consumption but decreases the efficiency, which is consistent with the earlier observation.

CHAPTER 6: OPTIMIZATION OF IMPELLER

Figure 6.14: Average static, dynamic and total pressure distributions along Z axis in G2 and NS

Figures 6.15 (a) and (b) show axial and radial velocity vector distributions along the longitudinal cross sections in the impeller channels of NS and G2 respectively. For the ease of comparison, the longitudinal cross section in each model is at the same cross sectional plane as demonstrated in Figure 6.15 (c). As blood flows forward until the starting of the impeller blade, the flow near the hub has higher axial velocity than that near the tip in NS, due to the higher adverse static pressure gradient at higher radial location. In NS, an obvious backflow can be observed at the clearance gap between the first impeller tip and the pump casing as shown in enlarged Section A of Figure 6.15 (a), and it interacts with the forward flow resulting in a tip leakage vortex at the upstream close to the pump casing. Since G2 does not have a gap between the impeller tip and the pump casing, there is no backflow as expected as shown in enlarged Section A of Figure 6.15 (b). In the region between the first and second blade cross section the backflow is found close to the pump casing in NS as shown in

CHAPTER 6: OPTIMIZATION OF IMPELLER

enlarged Section B of Figure 6.15 (a). Although small in magnitude as shown in enlarged Section B of Figure 6.15 (b), the dominant flow direction of blood close to the shroud in G2 is basically in forward direction. At the second clearance gap region as shown in enlarged Section C of Figure 6.15 (a), the strength of the flow is much weaker as compared with that of the first one (Section A) in NS. In the region between the second blade cross section and the impeller exit, the flow near the pump casing has higher axial velocity as shown in enlarged Section C of Figure 6.15 (a). However, the flow field is more like a channel flow in between the impeller hub and the shroud in G2 as shown in enlarged Section C of Figure 6.15 (b). There are backflows, however, in both models near the hub at the exit. The backflow is stronger in the NS model than that of the G2 model.

Figures 6.16 (a) and (b) show the relative tangential and radial velocity vector distributions with respect to the rotating impeller of NS and G2 respectively on the cross section at $Z=10$ mm as indicated in Figure 6.15 (c). In the figure, the impeller rotates in the counterclockwise direction, and the blood flows in the clockwise direction as expected, since the vectors are presented in relative velocity with respect to the impeller. It can be found in the figure that the radial velocities in both models are quite low, and the flow fields in both models are smooth without obvious vortex or flow separation. In NS, the blood close to both the impeller hub and the pump casing inner surface has higher tangential velocity than those in the center region of the channel as demonstrated in Figure 6.16 (a). However, the blood near the impeller shroud has low tangential and axial velocities in G2 as shown in Figure 6.16 (b) and Figure 6.15 (b) respectively. As a result, blood is impossible to be deposited on the pump casing inner surface or the impeller hub despite its axial velocity in NS, but it is highly possible to be deposited on the impeller shroud inner surface in G2.

CHAPTER 6: OPTIMIZATION OF IMPELLER

CHAPTER 6: OPTIMIZATION OF IMPELLER

(b) G2

CHAPTER 6: OPTIMIZATION OF IMPELLER

(c)

Figure 6.15: Radial and axial velocity vector distribution on the longitudinal cross section of the impeller in (a) NS and (b) G2. (c) Demonstration of the cross section position.

(a) NS

CHAPTER 6: OPTIMIZATION OF IMPELLER

Figure 6.16: Relative tangential and radial velocity vector distributions in (a) NS and (b) G2 on the cross section at $Z=10$ mm

Figures 6.17 (a) and (b) show the scalar shear stress contours on the longitudinal section in the impeller regions of NS and G2 respectively, and the position of the cross section is shown earlier in Figure 6.15 (c). Comparing Figures 6.17 (a) and (b), it can be found that the impeller with shroud in G2 has a lower scalar shear stress as a whole due to the bigger area of low scalar shear stress indicated by blue color. The scalar shear stresses close to the pump casing of NS are higher than those close to the impeller shroud of G2, and high scalar shear stresses are present near the first clearance gap region in NS, which is consistent with the tip vortex region as shown earlier in Figure 6.15 (a). The blood damage index of NS is 0.24%, and is slightly higher than 0.21% induced by G2. In addition, the mass-weighted scalar shear stress

CHAPTER 6: OPTIMIZATION OF IMPELLER

in the impeller region of NS is 78.3 Pa, which is also higher than 76.3 Pa in the impeller and gap regions of G2.

Figure 6.17: Scalar shear stress contours on the longitudinal cross section in the impeller regions of (a) NS and (b) G2

6.6 Summary

Due to the strong flow interaction between the rotating impeller and stationary straightener and diffuser, the performance of the original impeller is difficult to determine. Therefore, the original impeller itself is simulated initially without the straightener and diffuser blades, and the same strategy is also used for the further improved models throughout the chapter. It is found that the flow field within the original impeller design is still not desired due to vortices induced. In addition, the flow disturbance at the upstream of the impeller is severe with strong swirling blood. Due to the zero trailing edge angle design of the original impeller blade, flow separation can be observed. The trailing edge angle is then modified and set at 30° in

CHAPTER 6: OPTIMIZATION OF IMPELLER

the improved models. Based on the fast estimation method, the blood damage index induced by the original impeller model is too high to be accepted as shown in Table 6.4.

Table 6.4: Pressure, efficiency, leakage flow rate (based on the percentage with respect to the pump output, 5 L/min), power and blood damage index in the models simulated in this chapter under the same operating conditions of 5 L/min flow rate and 10,000 rpm rotating speed.

Models	Pressure (kPa)	Efficiency (%)	Leakage Flow Rate (%)	Power (W)	Blood Damage Index (%)
Original	4.5	10.40	-	7.50	4.6
P2	7.4	19.27	-	5.00	0.8
P5	6.1	20.35	-	3.63	0.44
P8	8.0	25.00	-	3.48	0.17
P8B2 (G0)	8.7	27.14	-	3.53	0.18
G1	8.6	11.50	1.26	8.19	0.31
G2	8.0	15.34	7.43	5.85	0.21
G3	5.6	11.73	14.85	5.76	0.24
NS	10.2	22.66	-	4.58	0.24

In order to improve the flow field in the impeller, a number of impellers (from P1 to P9) with two parameters (wrap angle and blade height) are simulated. The simulation results show that the static pressure head generated by the impellers with 270° wrap angle is higher than those generated by both 180° and 360° wrap angles under the same blade height. As the blade height reduces, the flow field in the impeller is improved, and the static pressure head generated increases. Furthermore, the blood damage index is also reduced with smaller blade height. Through the comparisons among different impellers, P8 with 2 mm blade height is selected, since it has generated high static pressure head, smooth flow field and low blood damage index.

CHAPTER 6: OPTIMIZATION OF IMPELLER

Initially, there are two short impeller blades in the design so as to prevent blood blockage near the impeller leading edge, but it is found later that they are not necessary to maintain smooth flow field when the blade height is 2 mm. Finally, the impeller with only two long blades, P8B2 model is selected as the optimized impeller. It has a simpler physical configuration than the original impeller, and this has reduced the cost in manufacturing.

For the ease of simulation, the clearance gap between the rotor cylinder and the pump casing is excluded. After the optimized impeller is selected, the effects of the clearance gap with different width are estimated. It is found that the clearance gap reduces the static pressure generated across the pump, especially the impeller, and it significantly reduces the hydraulic efficiency, because more torque is required to rotate the impeller shroud. The power consumption of G2 and G3 are 5.80 and 5.76 W respectively with the corresponding efficiencies of 15.34% and 11.73%, G2 is thus performing better than G3 from the hydraulic aspect as shown in Table 6.4. With 0.2 mm clearance gap, the leakage flow rate of G2 model is 7.43%, and increases to 14.85% when the gap width increases to 0.3 mm in G3 model. Using the fast estimation method, the blood damage index of G2 model is 20% lower than that of G3 model, so G2 model is superior to G3 model in term of the hemolysis. Therefore, G2 model is selected for further improvement.

Besides the current impeller design, the magnet can also be embedded in the impeller hub or blade as in other axial flow blood pumps found in the literature, where the impeller is not shrouded [78, 95, 132]. In order to estimate the differences between these two types of impellers, NS model is simulated and compared with G2. The simulation results show that NS model requires 4.58 W with 22.66% efficiency, whereas the power consumption of G2 is 5.85 W with 15.34% efficiency as shown in

CHAPTER 6: OPTIMIZATION OF IMPELLER

Table 6.4. Additionally, the static pressure head generated by NS model is 10.2 kPa, which is higher than 8.0 kPa generated by G2. The flow fields show that the blood is possible to be deposited on the corner between the impeller shroud inner surface and impeller blade of G2. The impeller shroud induces lower blood damage index and average scalar shear stress in the impeller region. However the hydraulic performance of NS is better, so it can rotate at lower speed to generate the same pressure head. Consequently, the blood damage index will also be reduced with the lower rotating speed [27]. Therefore, it can be deduced that the impeller shroud is not beneficial in terms of hydraulic and hematologic performances. The NS model increases, nevertheless the manufacturing difficulty, since the technology to embed the magnet in the impeller blades demands higher skill than the impeller shroud.

In the current stage, only G2 design is viable, due to the limitation of the current magnetic suspension design. Moreover, the primary purpose of this study is to optimize the axial flow VAD with the impeller shrouded. Therefore, both G2 and NS designs are selected for the further investigation and comparison. However, with the advance of technology, NS design should be adopted in the future due to its superiorities on lower power consumption and higher efficiency.

Chapter 7:

New Axial Flow VAD and Its Optimizations

In the previous chapter, the impeller design has been optimized, and it is found that a large portion of kinetic energy produced by the impeller can be further converted to higher static pressure through placing a diffuser at its downstream to satisfy the pump hydraulic performance. Moreover, the blood damage index distribution in the original blood pump indicates that the diffuser is another main pump component in causing high blood damage besides the impeller. The design of the diffuser is even more challenging than that of the impeller, since its incoming flow is not only spinning with high speed but also in great variations in velocity direction. In the current study the diffuser is designed to have three blades for the ease of manufacturing and PIV measurements. Ideally, zero trailing edge angle of a diffuser blade is preferred, since it is better that the blood is flowing in only axial direction so that its tangential velocity is completely converted to static pressure. It is, however, highly possible that a vortex is present near the trailing edge due to the high static pressure difference between the two sides of the diffuser blade. Since the diameter of the impeller hub has been increased to 11 mm as discussed earlier, a new diffuser is, thus, designed with the same hub diameter and three blades with 45° trailing edge angle.

The first new axial flow VAD consists of a three-blade straightener, a two-blade impeller, and a three-blade diffuser, and it is named as New Model One (NM1)

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

hereafter. As discussed in the previous chapter, the impeller configuration without impeller shroud is superior to that with impeller shroud in terms of hydraulic performances. Therefore, the second new axial flow VAD without impeller shroud (NM2) is also numerically simulated. Based on the simulation results of both NM1 and NM2, the optimization process is then carried out for the improving of the diffuser design.

7.1 New Model One (NM1)

Figures 7.1 (a) and (b) show the static pressure head versus flow rate and hydraulic efficiency curves of NM1 with the rotating speed varying from 9,000 rpm to 11,000 rpm. For each rotating speed, the static pressure across the entire model is obtained by varying the flow rates from 3 to 7 L/min using CFD, and each point in the figure indicates a simulation for a given flow rate and rotating speed. According to the physiological requirements of human body, the operating condition under 9,500 rpm rotating speed and 5 L/min flow rate is selected, and the corresponding static pressure generated is 13.3 kPa (100 mm Hg). Note that the simulation results in this chapter are based on these operating conditions. As shown in Figure 7.1 (b), the hydraulic efficiency at the chosen operating condition is 22.44%, and the power consumption is 5.44 W. Figure 7.1 (c) shows the leakage flow rates through the clearance gap under different operating conditions. It can be found that the leakage flow rate increases with the rotating speed, since higher rotating speed generates higher static pressure across the impeller. The leakage flow rate under the present operating condition is 0.33 L/min, which is 6.7% of the pump output.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(a)

(b)

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Figure 7.1: (a) Characteristic curves of NM1 under various rpms. (b) Hydraulic efficiency versus flow rate under various rpms. (c) Leakage flow rate versus rotating speeds of NM1 under various inlet flow rates.

Figure 7.2 shows the static pressure distributions along the blades and hubs of NM1. It can be found that the static pressure decreases slightly in the straightener channel, and its variation is mild as compared to other regions. The static pressures increase steeply near the leading edges of both the impeller and diffuser, because high velocity acceleration and deceleration occur respectively along the flow channels. In addition, the static pressures increase in radial direction along the impeller and diffuser blades, which indicate that strong interaction between blood and blade occurs at higher radial location, and similar phenomenon can be observed in the original model. Furthermore, compared with the original pump model, it can be found that the static pressure distributions are more uniform in NM1, especially those on the hub, which indicates the improvements of NM1. Since the angular position of the impeller is changing with

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

time, four static pressure distributions (i.e. a 30° change in each interval) on the circular plane at $r=6.5$ mm are demonstrated in Figure 7.3 when a impeller leading edge moves from one straightener blade to the next one (120°). It can be observed that the static pressure distribution in the straightener region is affected by the rotating impeller, whereas the static pressure distribution in the diffuser region remains almost unchanged. The simulation results show that the average static pressure at the pump inlet changes periodically every 60° , and the variation with respect to the mean static pressure head across the new model is less than 1%. Therefore, the changing angular position of the impeller should not significantly affect the average static pressure distribution. Additionally, the streamline distributions vary slightly with the rotating impeller. Therefore, the simulation result at any angular position could represent reasonably those at other angular positions. Note that the simulation results presented hereafter are all based on the angular position of the impeller as shown in Figure 7.2.

Figure 7.2: Static pressure distributions along the blades and hubs of NM1

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Figure 7.3: Transient static pressure distributions in NM1 at four equally spaced angular position

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

7.1.1 Static Pressure and Velocity Vector Distributions

The average static and dynamic pressure distributions along Z axis excluding the gap region are shown in Figure 7.4. Note that the interface between the straightener and impeller is at $Z=0$ mm and that between the impeller and diffuser is at $Z=20$ mm in the figure. It can be observed that the static pressure along the straightener decreases gently in small magnitude, which demonstrates the smooth forward flow in this region. Near the impeller leading edge, the rotating impeller creates a low static pressure due to the suction effect, and the static pressure increases significantly at its leading edge region, which is consistent with the observation shown earlier in Figure 7.2. In the impeller blade channel, it further increases to a peak near the trailing edge before dropping slightly when approaching the diffuser. Another static pressure rise can be observed in the diffuser region due to the conversion of flow kinetic energy to the static pressure. The static pressure then gradually decreases, because of the frictional loss. For the dynamic pressure, it can be found that it increases slightly in the straightener region due to the reduced cross sectional area and the rotating impeller is the dominant energy source in accelerating the flow. After blood is flowing out of the impeller region, it decreases steeply due to the frictional loss and its conversion to static pressure along the diffuser.

In order to demonstrate the detailed flow patterns in NM1, the velocity vector and streamline distributions in the straightener, impeller and diffuser are illustrated in Figures 7.5, 7.6 and 7.7 respectively. Three circular planes at radii of (a) 6 mm, (b) 6.5 mm and (c) 7 mm, which are close to the hub, at the middle blade height, and near the impeller shroud respectively, are chosen in order to show the flow patterns on different radial planes. Note that the velocity vector distributions presented are based on the angular position as shown earlier in Figure 7.2. It can be found in Figure 7.5

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Figure 7.4: Average static pressure (SP) and dynamic pressure (DP) distributions along Z axis in NM1

that the flow field in the straightener region is smooth without obvious irregular flow. It can be seen in Figure 7.6 that blood flows smoothly along the streamline design of the blade as expected, though the impeller has only two blades.

(a) $r=6$ mm

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(b) $r=6.5$ mm

(c) $r=7$ mm

Figure 7.5: Velocity vector distributions in the straightener region on circular planes at the radii of (a) 6 mm, (b) 6.5 mm and (c) 7 mm.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(a) $r=6$ mm

(b) $r=6.5$ mm

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(c) $r=7$ mm

Figure 7.6: Relative velocity vector distributions in the impeller region on circular planes at the radii of (a) 6 mm, (b) 6.5 mm and (c) 7 mm.

Figure 7.7 shows that the flow field in the diffuser region is not desired with vortex and flow separation, and the flow fields on different radial planes in the diffuser channel are significantly different. It can be observed in Figure 7.7 (a) that the flow at the diffuser entrance is in tangential direction and a counterclockwise vortex is located at side S of the diffuser blade near the trailing edge. This is because blood at lower radial location has a smaller velocity magnitude, which cannot overcome the high adverse static pressure gradient along side S. As the radius increases to 6.5 mm as shown in Figure 7.7 (b), the velocity has not only increased in magnitude but also shifted more in forward axial direction, and the flow separation can still be observed on the blade side S near the trailing edge. For blood close to the pump casing, the flow field is much smoother than those at lower radial locations, and flow separation has almost disappeared in Figure 7.7 (c). Although blood leaving the diffuser trailing edge still has tangential velocity at various radial locations, its kinetic energy is low as shown earlier in Figure 7.4.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(a) $r=6$ mm

(b) $r=6.5$ mm

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(c) $r=7$ mm

Figure 7.7: Velocity vector distributions in the diffuser region on circular planes at the radii of (a) 6 mm, (b) 6.5 mm and (c) 7 mm.

Figure 7.8 shows the axial and radial velocity vector distributions and static pressure contours on the longitudinal cross section in the impeller region of NM1. The position of the sectional plane has already been demonstrated earlier in Figure 6.15 (c). It can be observed in Figure 7.8 (a) that the main flow in between the impeller hub and inner shroud surface is in the forward direction, and the rotating impeller builds up the static pressure along the flow direction. Between the pump casing and the impeller shroud outer surface, the leakage flow is present due to the design nature and static pressure difference. The flow patterns vary at different regions, and are then enlarged in order to show them clearly. Although adverse static pressure gradient is present along the radial direction in Section A of Figure 7.8 (b), the blood has enough kinetic energy to overcome this and flows from low to high radii, due to the centrifugal force induced by the rotating shroud via the viscous effect. However, the flow near the

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

pump casing at the right hand side is in the reverse direction, because the centrifugal force is diminishing and the pressure force becomes dominant along the Z axis direction. In Section AB, blood flows smoothly around the radius and changes from radial to axial direction. It can be found in Section B that the velocity profile is parabolic, and the maximum velocity is almost at the center of the gap. Again in Section BC, the flow is changing from axial to radial direction around the radial curve. It can be observed in Sections C that the static pressure gradient drives the main flow from high to low radial locations near the pump casing region (left). However via the viscous effect, the rotating impeller drives the blood from low to high radial locations (right). After blood flows out of the clearance gap, it merges with the main flow. Although, the flow field within the gap is complicated in 2D plane, the potential of thrombus formation on the shroud outer surface should not be severe when the high tangential velocity induced by the impeller shroud is taken into consideration.

In the main flow region as shown in Section D, there is a large static pressure difference between the left and right sides of the impeller blade, i.e. the color of the static pressure contour changes from blue to yellow. This is expected, because the left and right sides of the impeller blade (white in color) are the suction and pressure sides respectively. It can be observed in Section D that the blood with higher axial velocity is close to the impeller hub, whereas the blood near the impeller shroud has a low axial velocity, though the flow is basically in forward direction. As shown in Section E, the blood basically flows in forward direction except that the blood near the impeller hub at the suction side (right side) of the blade (white color) flows reversely due to the interaction of the diffuser downstream.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(a) Overall View

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Figure 7.8: Axial and radial velocity vector distributions on the longitudinal cross section in the impeller region of NM1. (a): Overall view, (b) Gap and (c) Main Flow.

7.1.2 Hemolysis

Figure 7.9 (a) shows the scalar shear stress contours on the surfaces of the pump components, and the high shear stresses locate at the leading edges of both the impeller and the diffuser due to the flow acceleration and deceleration respectively. It

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

can be found that the scalar shear stress contours in the straightener region is the lowest. The mass-weighted scalar shear stress distributions in each component and the whole pump are shown in Figure 7.9 (b). It can be found that around 99% of blood inside the entire model experiences less than 250 Pa of low shear stress, and the mass-weighted average scalar shear stress is 22.7 Pa. In addition, the distribution in each region varies significantly. In the straightener region, almost all scalar shear stresses induced are below 50 Pa, which implies extremely low blood damage, and the mass-weighted average scalar shear stress is 2.0 Pa. Compared with the straightener, the diffuser region has almost the same flow volume, but the average scalar shear stress in the diffuser region is 26.8 Pa. The scalar shear stress distributions in the impeller and the clearance gap of NM1 are similar to those discussed in the previous chapter, and the corresponding average scalar shear stresses are 116.7 Pa and 66.2 Pa. The average value in both the impeller and clearance gap is 73.7 Pa. Based on the fast estimation method, the blood damage index of NM1 is 0.39%. Although there is a variation at different impeller angular position, it is less than 1% so that the scalar shear stress distributions at different impeller angular positions should have similar distribution. Besides, the blood damage index induced at each component is also calculated. The impeller (including the clearance gap) and the diffuser produce 0.17% and 0.27% blood damages respectively, and that induced by straightener is almost negligible. Compared with the original model, the blood damage index and the average scalar shear stress in each domain of NM1 are significantly reduced.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Figure 7.9: (a) Scalar shear stress distributions on the inner components of NM1, and (b) Mass-weighted scalar shear stress distribution in each region of NM1 as well as the entire domain. ST: straightener; IMP: impeller; GAP: clearance gap between impeller shroud and pump casing; DF: diffuser; WHOLE: Entire model.

7.2 New Model Two (NM2)

As mentioned in Chapter 6, the impeller shroud results in lower efficiency with larger blood contacting surface and longer residence time. The new design with the magnets, which are embedded inside the hub as other axial flow blood pumps found in the literature [83, 133], is shown in Figure 7.10 (b), and is named as NM2 hereafter.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(a) NM1

(b) NM2

Figure 7.10: Schematic view of (a) NM1 and (b) NM2.

Figures 7.11 (a) and (b) show respectively the static pressure head versus flow rate and hydraulic efficiency curves of NM2 with the rotating speed varying from 9,000 rpm to 10,000 rpm. For each rotating speed, the static pressure across the entire model is obtained numerically by varying the flow rates from 4 to 9 L/min through simulation. For the equivalent comparison with NM1, the operating conditions of 9,500 rpm rotating speed and 5 L/min flow rate are selected. The corresponding static pressure generated is 14.7 kPa (110 mm Hg), and the power consumption is 4.25 W. As shown in Figure 7.11 (b), the flow rate and the peak efficiency increase with the rotating speed, and the hydraulic efficiency at the chosen operating condition is 31.41%. The peak efficiency for 9,000 rpm is at 7 L/min, and those for 9,500 and 10,000 rpm are at 8 L/min. Compared with NM1, it can be found that the NM2 has generated higher static pressure head and higher hydraulic efficiency, especially at

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

higher rotating speed. In addition, the flow rate, where the peak efficiency occurs, also increases in NM2, therefore, an impeller without shroud (NM2) is superior to that with shroud (NM1).

Figure 7.11: (a) Characteristic curves of NM2 under various rpms. (b) Hydraulic efficiency versus flow rate under various rpms.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

7.2.1 Static Pressure and Velocity Vector Distributions

Figure 7.12 shows the average static and dynamic pressure distributions along Z axis in NM2, and those of the NM1 are also included. It can be observed that both static and dynamic pressure distributions in the straightener and diffuser regions are almost identical in the two models, and the main differences are in the impeller region. The static pressure of NM2 increases more steeply near the impeller leading edge, and the static pressure head generated across the impeller region is much higher. Additionally, the position of the peak static pressure in the impeller region of NM2 is located further downstream, i.e. closer to the impeller trailing edge. The dynamic pressure in NM2 is lower than that in NM1, especially near the impeller trailing edge, since the impeller of NM1 has higher momentum due to the shroud cylinder. The simulation results show that NM2 requires 4.25 W with 31.41% efficiency, whereas the power consumption of NM1 is 5.44 W and the efficiency is 22.44%. Hence, the impeller shroud increases the power consumption with low efficiency, which is consistent with the earlier observation.

The flow fields in the straightener, impeller and diffuser regions of NM2 (at $r=6$, 6.5 and 7 mm circular planes) have similar flow patterns as NM1 as shown earlier from Figure 7.5 to Figure 7.7, and are not shown here. Figure 7.13 shows the axial and radial velocity vector distributions and static contours on the longitudinal cross section in the impeller region of NM2. For the ease of comparison, some parts are enlarged to show the flow field clearly. It can be observed in Figure 7.13 that the rotating impeller builds up the static pressure along the flow direction, and the static pressure increases significantly across the first impeller blade. As shown in Section 1, a strong reverse flow occurs between the impeller tip and the pump casing due to the

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Figure 7.12: Average static and dynamic pressure distributions along Z axis in NM2

static pressure difference between the pressure (right side) and suction (left side) sides of the blade (white color), and interacts with the forward flow (upstream) resulting in a tip leakage vortex rotating in counter-clockwise direction. In the region between the first and second blade cross sections the backflow is present close to the pump casing in NM2 as shown in Section 2 of Figure 7.13. At the second clearance gap region in Section 3, the velocity vector is much smaller in magnitude as compared with that of Section 1 in NM2 as shown in Figure 7.13. Beside the backflow along the gap in between the pump casing and the impeller tip, it can also be observed in the region between the second blade cross section and the impeller exit that another backflow is present close to the hub, due to the diffuser.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Figure 7.13: Static pressure and radial and axial velocity vector distribution on the longitude cross section along the Z axis of the impeller in NM2

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

7.2.2 Blood Damage Index

Table 7.1 compares the mass-weighted average scalar shear stress and blood damage index induced in each region of NM2 and NM1 based on the fast estimation method. Note that the mass-weighted average scalar shear stress and blood damage index in the impeller of NM1 are based on both impeller and gap regions. It can be found that the average stress and blood damage index induced by the impeller of NM2 are higher than those of NM1, due to the vortex induced by backflow from the pressure to suction sides of the blade with the incoming flow into the impeller channel at the region near the pump casing. However, these two parameters in diffuser of NM2 are slightly lower than those of NM1, because the blood from the impeller of NM2 has lower tangential velocity components as shown earlier in Figure 7.12 so that the interaction between the swirling blood and the stationary diffuser blades are mild in NM2. It can be found that the overall blood damage index and the average stress in NM2 are slightly lower than those of NM1. Since the differences of the blood damage indices in both models are quite marginal, these two models have almost the same blood damage index.

Table 7.1: Mass-weighted average scalar shear stress and blood damage index induced in each region of NM2 and NM1. SSS: Scalar shear stress

	Straightener		Impeller		Diffuser		Whole	
	SSS (Pa)	Index (%)	SSS (Pa)	Index (%)	SSS (Pa)	Index (%)	SSS (Pa)	Index (%)
NM1	2.0	0.01	73.7	0.17	26.8	0.27	22.7	0.39
NM2	1.9	0.00	75.0	0.19	25.8	0.25	22.1	0.37

7.3 Model with Clearance Gap between Diffuser Hub and Blades

The simulation results show that the flow field and the blood damage in the diffuser of NM2 are not desired, the configuration of the diffuser should thus, be improved.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Various axial flow VADs have been developed during the past decade, and some designs do not include a diffuser, probably due to the simplification of the design and manufacturing [9, 20]. For the axial VADs with diffuser, there are mainly two types of design. For the first type, the diffuser blades and hub are integrated as a single component such as NM2 as demonstrated in Figure 7.14 (a). For the second type, the diffuser blades are detached from the hub with clearance gap, and the diffuser hub is the extension of the impeller hub as shown in Figure 7.14 (b). Note that the stationary diffuser hub and the rotating impeller hub are indicated by blue and red respectively in the figure. The axial flow VAD model with the second kind of diffuser is named as Rotary Diffuser Hub (RDH). In order to estimate the effects of the clearance gap width between the diffuser blades and hub, RDH with 0.1 mm (RDH1), 0.2 mm (RDH2) and 0.3 mm (RDH3) clearance gaps are simulated and compared with NM2. Note that the number denotes the width of the clearance gap.

Figure 7.14: Physical configurations of the diffusers in (a) NM2 and (b) RDHs.

Figure 7.15 shows the average static and dynamic pressure distributions along Z axis of RDHs. Note that the static pressure at Z=-40 mm is set as the reference in each model. Both static pressure and dynamic pressure distributions of RDHs with various clearance gaps are almost identical in the straightener and impeller regions except in

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

the diffuser region. It is found that a smaller clearance gap produces steeper slope of the static pressure increment at the leading edge and higher static pressure in the diffuser region. The dynamic pressures decreases steeply in all the three models, and RDH1 has the steepest slope and lowest dynamic pressure generated among the three models followed by RDH2 and then RDH3. This is because the flow in the RDH with smaller clearance gap is less affected by the rotary diffuser hub. Table 7.2 shows the static pressure generated by RDHs with the corresponding efficiency and power consumption. As the width of the gap increases, it can be found in the table that the pressure, power consumption and efficiencies are all decreased.

Figure 7.15: Average static pressure distributions along Z axis in RDHs

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Table 7.2: Pressure, efficiency and power consumptions in RDH3

	Pressure (kPa)	Efficiency (%)	Power (W)
RDH1	15.9	31.0	4.71
RDH2	15.2	30.7	4.62
RDH3	14.5	29.8	4.52

As shown earlier in Figure 7.7, the flow field on the circular plane at $r=6$ mm of the diffuser is the worst among the three radial cross section planes investigated. The circular radial plane at $r=6$ mm is thus selected in RDHs for comparison, and the velocity distributions and streamlines on the plane are shown in Figure 7.16. A vortex is present near the diffuser leading edge of RDH1 at A as indicated in Figure 7.16 (a), but the blood flows smoothly along the diffuser blade thereafter. Compared with the diffuser of NM2 as shown earlier in Figure 7.7 (a), the high tangential velocity induced by the rotary diffuser hub has shifted the vortex from the side S of diffuser trailing edge to the leading edge of the opposite side diffuser blade within the same channel, and the size of the vortex is much smaller as demonstrated in Figure 7.16 (a), so the flow pattern on the circular plane at $r=6$ mm in RDH1 is significantly improved. As the width of the clearance gap further increases to 0.2 mm and 0.3 mm, the vortex near the leading edge is vanished, and the flow patterns in RDH2 and RDH3 does not vary significantly as shown in Figures 7.16 (b) and (c) respectively. In addition, the blood also flows along the diffuser blade smoothly in both RDH2 and RDH3.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(a) RDH1

(b) RDH2

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(c) RDH 3

Figure 7.16: Velocity vector distributions and streamlines on the circular plane at $r=6$ mm in the diffuser of RDHs

Scalar shear stress contours on the cross-section at 5 mm from the diffuser inlet of RDH1, RDH2 and RDH3 are shown in Figures 7.17 (a), (b) and (c) respectively, and those in NM2 is also shown in Figure 7.17 (d) for the ease of comparison. The area of high scalar shear stress region indicated by the red color in RDHs is much reduced as compared with that of NM2, and the distributions of the scalar shear stress on the cross section become smoother as demonstrated in Figures 7.17 (a) to (c). However, another high scalar shear stress region at the blade corner can be found in Figure 7.17 as highlighted by a black square, and this should be due to the flow interaction between the rotary hub and the stationary blades. Based on the fast estimation method, the blood damage indices induced by the diffusers of RDHs are all around 0.14%, and is lower than that of NM2 (0.25%). In addition, the blood damage indices for the whole pumps of RDHs are all around 0.29%, and are also lower than that of NM2 (0.37%).

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(a) RDH1

(b) RDH2

(c) RDH3

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

(d) NM2

Figure 7.17: Scalar shear stress contours on the cross section 5 mm downstream from the diffuser inlet in (a) RDH1, (b) RDH2, (c) RDH3 and (d) NM2

The diffusers with three different clearance gap widths have been simulated. Although RDH1 has the highest static pressure head, a vortex is present near the diffuser leading edge at $r=6$ mm as shown earlier in Figure 7.16 (a). The flow fields in the diffusers of both RDH2 and RDH3 are smooth, but RDH2 has generated higher static pressure with almost the same blood damage level. Therefore, RDH2 is chosen, and its schematic view is shown in Figure 7.18. The simulation results show that the efficiency of RDH2 is 30.7% and the power consumption is 4.6W under the operating conditions of 9,500 rpm rotating speed and 5 L/min flow rate. Compared with NM2, its hydraulic performance is slightly low due to the power consumption of the rotary diffuser hub, but the flow field near the diffuser hub is much improved.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Figure 7.18: Schematic view of NM3

7.4 Summary

A new axial flow VAD, NM1, has been simulated and the results are presented and discussed in this chapter. It is found that the basic physiological requirements of 5 L/min flow rate and 13.3 kPa (100 mm Hg) pressure head are satisfied when the impeller rotates at 9,500 rpm for the designed pump model as listed in Table 7.3. The simulation results show that the static pressure generated across the entire model and the static pressure distributions with respect to different angular position of the impeller does not change significantly with the variation of the static pressure head with respect to the average value less than 1%. Therefore, the simulation results at any angular position of the impeller can represent those at other angular positions. The flow fields in the straightener and the impellers are basically smooth without obvious irregular flow. However, flow separation and vortex are present in the diffuser channel due to the high static pressure difference between two sides of the diffuser blade. Its blood damage index is dramatically reduced due to the smooth flow field and smaller blade height, as compared to those of the original VAD design.

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

Table 7.3: Pressure, efficiency, power and blood damage index of the models simulated in this chapter under the same operating conditions of 5 L/min flow rate and 9,500 rpm rotating speed.

	Shroud Impeller	Rotating Diffuser Hub	Pressure (kPa)	Efficiency (%)	Power (W)	Blood Damage Index (%)
NM1	Yes	No	13.3	22.44	5.44	0.37
NM2	No	No	14.7	31.41	4.25	0.38
RDH1	No	Yes	15.9	31.02	4.71	0.29
RDH2 (NM3)	No	Yes	15.2	30.65	4.62	0.29
RDH3	No	Yes	14.5	29.84	4.52	0.29

It can be found in the literature that the impeller is not shrouded in many axial flow blood pumps [83, 133]. Therefore, NM2 without the impeller shroud has been designed, and it has the same physical configurations as NM1 in other aspect. In addition, 0.2 mm clearance gap is maintained by the impeller tip and the pump casing for the equivalent comparison. At the operating condition under 9,500 rpm rotating speed and 5 L/min flow rate, NM2 could generate 14.7 kPa (110 mm Hg) pressure head as listed in Table 7.3. It is also found that the hydraulic efficiency is further improved, compared with that of NM1. Although NM1 and NM2 have almost the same blood damage index under operating condition as shown in Table 7.3, NM2 can rotate at lower speed to generate the same pressure head with lower blood damage index. Therefore, it is suggested to eliminate the impeller shroud in the future design of the axial flow VAD.

Based on the previous simulation results, the diffuser is a critical component, which significantly affects the static pressure difference across the entire model and the blood damage level. Mainly, there are two methods for the diffuser design based on whether the diffuser blades are detached from the diffuser hub. The simulation results show that RDH has limited influences on the impeller, since the diffuser is placed at the downstream, but it improves the flow field in the diffuser region. In addition, it

CHAPTER 7: NEW AXIAL FLOW VAD AND ITS OPTIMIZATIONS

increases the slope of the static pressure profile near the diffuser leading edge. However, as shown in Table 7.3, the RDH design increases the power consumption due to the torque required to rotate the diffuser hub, and their efficiencies are slightly lower than NM2, especially when the width is being enlarged. The fast estimation method indicates that RDH reduces the blood damage indices, due to the improved flow field in the diffuser region. Among RDH models, RDH2 is superior to the others, since it has smooth flow pattern without vortex and high static pressure head generated. The static pressure head of RDH2 is 15.2 kPa, and it is slightly higher than that of NM2 under the same operating condition as shown in Table 7.3.

Chapter 8:

Experimental Results and Comparisons with Numerical Results

Since the development of an axial flow VAD with a shrouded impeller is the main target of the present study, the measurements are carried out for NM1 model. In addition, it is easier to fabricate than the model with rotary diffuser hub as developed in Chapter 7. Although NM1 numerical model has been tested with different grid schemes as illustrated in the earlier chapter, it is still necessary to be experimentally validated. In this chapter, the measurement results of the flow in the 1:1 acrylic prototype of NM1 are presented, and the experimental arrangements have been discussed earlier in Chapter 4. In the PIV measurements, blood analog is used to mimic the kinetic viscosity of the human blood and to match the refractive index of the acrylic prototype. In order to match the operating conditions used in the numerical simulation, the impeller rotating speed and the pump flow rate were 9,500 rpm and 5 L/min respectively in the experiment.

8.1 Pump Characteristic Curves Comparison between CFD and Experimental Results

Figure 8.1 compares the pump performance curves of both CFD and experimental results at different rotating speed. Note that each point corresponds to a steady state under a specific flow rate and rotating speed. Both the experimental and CFD

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

simulation results show that the static pressure head decreases with the increasing flow rate. Although the CFD simulations overestimate the experimental performance of NM1 by about 10%, the numerical results generally agree well with the experimental results.

Figure 8.1: Experimental pump characteristic curves in comparison with the CFD predictions using SST $k-\omega$ turbulence model.

8.2 Flow Field in the Straightener

Two parallel cross sectional planes are selected in the straightener along Z axis (Plane A) and 6.5 mm offset (Plane B) as demonstrated in Figure 8.2, in order to show the flow field between the straightener hub and the pump casing as well as the flow field surrounding the straightener blade respectively. Note that Plane B, which is 6.5 mm offset from the Z axis, is in the middle between the straightener hub ($r=5.5$ mm) and the pump casing ($r=7.5$ mm), and is perpendicular to the straightener blade.

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

Figure 8.2: Demonstration of the positions of the cross sectional planes in the straightener region

Figure 8.3 (a) shows the measured velocity vector distributions in the straightener cross sectional region through Z axis as demonstrated in Plane A of Figure 8.2. Note that the axial position (Z axis) of the straightener exit is set to 0 as the reference (consistent with the convention used in earlier chapter for numerical simulation), and the center of the pump is at 0 mm in radial direction. It can be observed that the flow in the straightener region is smooth and is mainly in positive axial direction. At the region close to the pump casing ($r=7.5$ mm), the fluid has lower speed as compared to that at the center of the channel. As fluid flows forward, the smooth curvature expansion of the straightener hub has accelerated the flow due to the reduced cross sectional area (perpendicular to axial axis), and guides the flow smoothly into the space between the pump casing and the hub constant diameter region, which begins at $Z=-10$ mm with a parabolic velocity profile. It can be observed in Figure 8.3 (b) that the PIV measured flow field has matched well with that obtained from SST $k-\omega$ turbulence model. It can be deduced that the simulation can predict the flow field well in the straightener, where the flow pattern is smooth.

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

Figure 8.3: Axial and radial velocity vector distribution on the cross sectional plane A in the straightener region (a) PIV measurement and (b) Numerical simulation

Figure 8.4 (a) shows the measured velocity vector distributions on the cross sectional Plane B as demonstrated in Figure 8.2. It can be found that the flow field is smooth around the straightener blade without flow separation and vortex. However, the flow at the downstream of the straightener trailing edge is irregular, because the rotating impeller blade leading edges have induced the static pressure difference between two

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

sides of the straightener blade. In addition, due to the viscous effect, the flow near the blade has a low velocity magnitude. These findings closely agree with those observed in the numerical simulation results at the same position as shown in Figure 8.4 (b). Based on the comparisons in Figures 8.2 and 8.3, it can be deduced that the numerical simulation can well predict the flow field in the straightener, where the flow field is simple and smooth. Both experimental and numerical results show that the flow field in the straightener region is mainly driven by the suction effect of the impeller, which induced a favorable pressure gradient along Z axis, and the dominant flow direction is forward except the region very close to the rotating impeller.

(a) PIV measurement

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

(b) Numerical simulation

Figure 8.4: Velocity vector distribution on the cross sectional plane B in the straightener: (a) PIV measurement and (b) Numerical simulation

8.3 Flow Field in the Impeller

Figure 8.5 (a) shows the two parallel cross sectional planes, Planes C and D, in the impeller along the centerline of Z axis and 6.5 mm offset from the centerline in order to show the flow fields between the impeller hub and the shroud and those surrounding the impeller leading edge respectively. Note that the flow field in the straightener presented earlier has also the same angular position of the impeller. In other words, the PIV image pairs used to calculate the flow fields in the stationary straightener are captured when the rotating impeller reaches the current angular position. However it can be observed that the flow field around the impeller trailing edge cannot be observed at the current impeller angular position. Therefore, another cross sectional plane, Plane E, is selected near the trailing edge after the impeller

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

leading edge has rotated 70° from the reference position as demonstrated in Figure 8.5 (b).

Figure 8.6 (a) shows velocity vector distributions (in axial and radial directions) on the cross sectional Plane C in the impeller as demonstrated in Figure 8.5 (a). As the blood analog flows into the impeller channel, the dominant flow is in positive Z direction, and the high axial velocity is located slightly close to the impeller hub ($r=5.5$ mm), especially at the further downstream until $Z=18$ mm. As a result, the blood analog close to the impeller shroud ($r=7.5$ mm) is much smaller than that close to the hub ($r=5.5$ mm), and it is consistent with the simulation results obtained in the earlier chapter. From $Z=18$ mm onwards, it can be observed that the forward velocity at the impeller shroud has increased in magnitude, while near the impeller exit, reverse flow is detected close to the impeller hub. Because the diffuser placed at the impeller downstream induces an adverse static pressure gradient at the impeller exit,

Figure 8.5: Positions of the cross sectional planes at two impeller angular positions of (a) $\theta=0^\circ$, i.e. Planes C at the center and Plane D with 6.5 mm offset from the center line, and (b) $\theta=70^\circ$, i.e. Plane E for trailing edge.

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

the blood near the rotating impeller hub, which has lower momentum than that near the impeller shroud, cannot overcome it and flows reversely. After the interaction with the original forward flow along the hub, the reverse flow has made an U-turn at around $Z=18.5$ mm. As shown in Figure 8.6 (b) the numerically predicted velocity vectors are basically similar to those from PIV measurements, so the selected turbulence model is able to predict the dominant forward flow and reverse flow close to the impeller hub near the exit. Slight differences between the PIV measurement and the simulation results can be detected near the interfaces between the impeller shroud and the impeller blade, where the seeding particles trapped there have induced relatively strong reflection resulting in error in the PIV measurements. In addition, the blood analog close to the interfaces usually has a high velocity gradient in radial direction, which could cause error due to the difficulty in PIV measurements. Ideally seeding particles within an interrogation area, where the camera image is usually divided into 32 pixels by 32 pixels in size, should be homogenous, and have exactly the same displacement. However, the displacement of individual particle is always different from each other, especially when there is a high velocity gradient. The particle displacement will greatly depend on the particle position, and the position of the seeding particle within the interrogation area is random. In other words, the estimated velocity is strongly depended on the position of the individual seeding particle within the interrogation area when there is high velocity gradient, and thus resulted in error measurement. .

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

(a) PIV measurement

(b) Numerical simulation

Figure 8.6: Axial and radial velocity vector distribution on the cross sectional Plane C of the impeller: (a) PIV measurement and (b) Numerical simulation

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

Figure 8.7 (a) shows the relative velocity vector distribution with respect to the rotating impeller near the leading edge in Plane D (see Figure 8.5 (a)). It can be observed that the blood analog flows along the impeller blade smoothly from the leading edge, and the leading edge angle of the impeller is basically aligned with the flow direction with respect to the impeller. It can be found in Figure 8.7 (b) that the simulation result has a similar flow pattern.

(a) PIV measurement

(b) Numerical simulation

Figure 8.7: Relative velocity vector distribution around the impeller leading edge on the cross sectional Plane D: (a) PIV measurement and (b) Numerical simulation

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

It can also be observed in Figure 8.8 (a) that the blood analog flows along the impeller blade smoothly around the trailing edge in Plane E (see Figure 8.5 (b)), and the flow direction is almost align with the trailing edge angle. In both Figures 8.7 and 8.8, the dominant flow direction is mainly forward and parallel to the impeller blade, so the impeller blades basically guide the blood analog from leading edge to trailing edge smoothly as desired. The numerical results as demonstrated in Figure 8.8 (b) are found to be similar to the measurements. Through the comparison between the experimental and numerical results from Figures 8.5 to 8.7, the difference between the flow patterns predicted by CFD and measured by PIV is not obvious when the flow direction is forward.

(a) PIV measurement

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

(b) Numerical simulation

Figure 8.8: Relative velocity vector distribution around the impeller trailing edge on the cross sectional Plane E: (a) PIV measurement and (b) Numerical simulation

8.4 Flow Field in the Diffuser

Figure 8.9 shows the regions of the cross sectional Planes F (at the centerline) and G (6.5 mm offset from Plane F) in the diffuser to illustrate the flow patterns between the stationary diffuser hub & the pump casing and the region around the diffuser leading edge respectively. Unlike the impeller, where the flow field near the trailing edge can be captured after it rotates 70° , the flow field near the diffuser trailing edge cannot be measured, since the diffuser is stationary. In other words, the present setting of the PIV laser sheet cannot be adjusted to reach the diffuser trailing edge due to the physical constrain in the present experimental arrangement.

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

Figure 8.9: Demonstration of the regions of the cross sectional Planes F and G in the diffuser

Figure 8.10 (a) shows the velocity vector distributions on the cross sectional Plane F in the diffuser. Note that the axial position of $Z=20$ mm is at the entrance of the diffuser region and the flow demonstrated in Figure 8.10 (a) is a continuation of the flow from the impeller as shown earlier in Figure 8.6 (a) to the diffuser. Compared with the flow patterns in both straightener and the impeller, the flow pattern in the diffuser is the most irregular with strong reverse flow close to the diffuser hub ($r=5.5$ mm). However, the forward flow becomes more dominant from $Z=26.5$ mm after the blade cross section. The similar flow pattern is predicted by the numerical simulation as shown in Figure 8.10 (b). Minor differences can be found near the diffuser hub and pump casing, because the blood analog near these region has high velocity gradient in the radial direction. In addition, due to the seeding particle deposited there, relatively strong reflection occurs, and it increases the difficulty in calculating the velocity vector through the cross correlation of the captured image pair.

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

Figure 8.10: Axial and radial velocity vector distribution on the cross sectional Plane F in the diffuser: (a) PIV measurement; (b) Numerical simulation

Figure 8.11 (a) shows the velocity vector distributions around the diffuser leading edge on the cross sectional Plane G. It can be found that the blood analog flows smoothly along the blade without obvious flow separation. Although the flow field in

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

the diffuser region is not smooth as shown earlier in Figure 8.10, at the region between $r=6.5$ mm and 7.5 mm the flow pattern is basically smooth and forward. Note that Plane G, which is parallel to Plane F with 6.5 mm offset from the pump centerline, shows the flow field in the radial region between 6.5 mm and 7.5 mm. The flow pattern shown in this figure is smooth, and a good match of the simulation result to the PIV measurement is observed in Figure 8.11 (b). Based on the flow pattern shown in Figures 8.9 and 8.10, the flow field near the diffuser hub is complicated, whereas the flow close to the pump casing is basically smooth. However, reverse flow is found near the diffuser hub, because the blood there has lower velocity and cannot overcome the adverse pressure gradient. This is consistent with the observation in Chapter 7, and it has shown again that the diffuser is another critical region in the blood pump design.

(a) PIV measurement

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

(b) Numerical simulation

Figure 8.11: Velocity vector distributions around the diffuser leading edge on the cross sectional Plane G in the diffuser: (a) PIV measurement and (b) Numerical simulation

Figures 8.12 (a), (b) and (c) show the comparison of velocity profiles in the straightener, impeller and diffuser regions respectively, in order to quantitatively compare the CFD and PIV results. Note that the solid line and dotted line denoting PIV and CFD results respectively in these figures. Generally, the CFD results agree fairly well with the PIV results, especially in the straightener region. The L^2 relative error norm of the differences in axial velocity between CFD and PIV with respect to the PIV in the straightener, impeller and diffuser are 11%, 19%, and 27% respectively.

Note that the L^2 relative error norm is defined as $\sqrt{\frac{\sum_i^N (f_i^E - f_i^N)^2}{\sum_i^N (f_i^E)^2}}$, where f_i^E and f_i^N

represent the values at node i for PIV experiment and numerical simulation respectively.

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

(a) Straightener

(b) Impeller

(c) Diffuser

Figure 8.12: Comparison of the velocity profiles obtained from PIV (solid line) and CFD (dotted line) results in (a) straightener, (b) impeller and (c) diffuser

CHAPTER 8: EXPERIMENTAL RESULTS AND COMPARISONS WITH NUMERICAL RESULTS

8.5 Summary

In this chapter, the PIV measurement results on the cross sections in three main components of NM1 model are presented and compared with the numerical simulation in Chapter 7 under the same operating condition. The minor differences between the experimental and simulation results are usually found near the intersection near the impeller hub and the impeller blade, due to the high velocity gradient and strong reflections from the seeding particles trapped there. Basically, fair agreements between the simulations and measurements can be observed, and it has reasonably validated the simulation results. Therefore, it indicates that SST $k-\omega$ turbulence model is robust, and the CFD results in earlier chapters are reliable.

Chapter 9:

Conclusions and Future Works

9.1 Conclusions

In the thesis, an axial flow VAD was developed by numerical simulation, based on the considerations of hydraulic and hematologic performances. Different designs of both impeller and diffuser were numerically investigated for comparison. The CFD software: FLUENT (Version 6.3.26) was used in the numerical simulations. Furthermore, the experimental pump model was built in 1:1 scale according to the numerical model and the operating conditions used in the simulation were applied to the experiments. PIV was used to measure the flow field in order to validate the numerical results.

At the initial stage, the internal flow field of the original model was studied as the baseline model for further improvements. The simulation results revealed that the original design could not generate adequate pressure head at the desired rotating speed, and the inner flow field was irregular. It was found in the original model that the relatively low hub to tip ratio (or blade height) resulted in two adverse effects. Firstly, the flow pattern varied significantly at different radial surfaces in each flow region, so the difficulty in guiding blood along designed blade profile increased. Secondly, the volume dominated by turbulent shear stress was relatively large at the region away from wall, as compared with that dominated by laminar shear stress at the region near

CHAPTER 9: CONCLUSIONS AND FUTURE WORKS

wall. These two effects had negative impact on the blood damage, so the original model had high blood damage potential.

At the next stage, the effects of the hub to tip ratio and blade profile were examined on the impeller alone, in order to simplify the design process and to take advantage of periodic flow. The original impeller was estimated as reference. Its flow field was still irregular even without the interactions between the upstream straightener and downstream diffuser. In the thesis, the simulation results of nine impellers (P1 to P9) with different configuration of two parameters (wrap angle and blade height) were compared. The results showed that the impellers with 270° wrap angle (P2, P5 and P8) generated the highest static pressure head among the impellers with the same blade height. Finally, the model with 2 mm blade height, P8, was found to be superior to other models, due to the high pressure head and the low blood damage index. In order to further simplify the design for the ease of manufacturing, two blades were trimmed from P8 model without significantly altering the pressure head and blood damage index. As a result, the optimized impeller model had only two blades with 270° wrap angle and 2 mm blade height.

Since the clearance gap between the impeller shroud and the pump casing was the specific feature of the present design, its effects were studied in details. Different clearance gaps were examined and compared with each other and the optimized impeller model without gap. Firstly, when the clearance gap is increased, the pressure head generated has decreased. Secondly, the pump requires additional power, due to the frictional force induced on the impeller shroud, and the power consumption decreased as the width increased. As a result, the model with 0.2 mm clearance gap had the highest hydraulic efficiency. Furthermore, the blood damage index and scalar

CHAPTER 9: CONCLUSIONS AND FUTURE WORKS

shear stress also indicated that the model with 0.2 mm clearance gap was superior to others gap width, so it was selected in further design phase.

A completed model, NM1, was designed in order to investigate the interactions between the optimized impeller and the upstream straightener and downstream diffuser. As compared with the original design, the flow fields in the straightener and impeller were smooth except that the flow field in the diffuser region was not desired with vortices and flow separations. The angular position of the impeller with respect to the straightener and diffuser does not significantly change the static pressure distribution with the variation less than 1%, though the pressure head changes periodically. Therefore, the simulation result at any angular position could represent those at other angular positions. As compared with the original axial flow VAD, NM1 had also lower blood damage index, due to the significantly improved flow field and lower blade height.

Although the axial flow VADs with or without impeller shroud have been investigated in the literature, the comparison has not been conducted. Therefore, a new model, NM2, without the shroud but with a clearance gap between the impeller tip and the pump casing was studied. The impeller shroud design has its advantages such as improved rotating stability, due to the imbedded magnet in the shroud, which simplifies and strengthens the pump control. The CFD simulation results have revealed its advantages and disadvantages from the hemodynamic point of view. Firstly, the impeller shroud design improved the leakage flow near the impeller tip and avoided the flow interaction between the impeller tip and pump casing. Secondly, the impeller shroud altered the flow rate at the peak efficiency, i.e. the flow rate is 5-6 l/min in NM1 (with shroud) and that is 7-8 l/min in NM2 (without shroud). Thirdly, the impeller shroud did not significantly change the inner flow field and the estimated

CHAPTER 9: CONCLUSIONS AND FUTURE WORKS

blood damage index. Fourthly, the main drawback of the impeller shroud was that it induced the drag, whereas the drag on the tip of impeller (without shroud) was smaller. As a result, the impeller shroud increased the power consumption. To some extent, it offsets the improved electromagnetic coupling efficiency. The final comparison of the overall efficiency, which is the combination of hydraulic and electromagnetic efficiency, between these two designs could only be obtained with the actual prototyped pump.

There were two kinds of diffuser configurations adopted in axial flow VADs. For the first type, the diffuser blades were attached to the diffuser hub. For the second type, the diffuser blades were detached from the diffuser hub, which was integrated and rotated with the impeller hub. It was found that the second design significantly improved the flow field and the reduced blood damage index as compared with its counterpart. Among the designs with different widths of the clearance gap between the diffuser blades and the hub, the one with 0.2 mm clearance was selected as a compromise between the static pressure head and the flow field near the diffuser hub.

Since the primary purpose of this study is to optimize the flow field in an axial flow VAD with an impeller shroud, NM1 was selected in the PIV experimental measurement, and it was also the only feasible model that could be fabricated due to the time and financial constraints. Although the simulation results of NM1 were numerically tested using different grid schemes, it was still necessary to be validated experimentally. In the PIV measurement, the 1:1 prototype was made of acrylic, and blood analog was used to mimic the kinetic viscosity of the blood and the refractive index of acrylic. Two cross sectional planes, with one plane passing through the Z axis centerline and the other plane parallel to the center plane but with 6.5 mm offset from it, were selected to show the flow patterns between the hub and tip and those

CHAPTER 9: CONCLUSIONS AND FUTURE WORKS

around the blades in each component. The experimental results exhibited similar flow characteristics as those found in the numerical simulation, and these agreements reasonably validated the numerical simulation of the pump inner flow.

Overall, the main scope of the thesis is to develop an axial flow VAD. In addition, the impeller shroud was the specific design feature in the model, but its effects on the hemodynamic have not been elaborated in the literature. As a result, the advantage and disadvantages of the impeller shroud have been revealed in this thesis, through the comparison between the designs with and without impeller shroud. Moreover, this study compared different diffuser designs. It has been found that the clearance gap between the diffuser blade and hub could improve the flow field. These results could provide useful information to the VAD design and optimum studies in the future. Lastly, the experiment in this study enriched the understanding of flow mechanism and validated the CFD results in this model.

9.2 Future Works

The static pressure distributions and flow fields inside both the original blood pump and the new models have been numerically estimated, and the PIV measurements were also conducted in NM1, for the validation in the present study. Furthermore, NM1 has gone through the optimized process with the new configuration of the pump suggested. However, there are some limitations involved in this investigation. In order to be consistent with the experimental model, the blade angle distribution along the impeller and diffuser blade was made constant along the radial direction, due to the limited capability of the machining tool available in NTU. The airfoil profile is not applied to the blades to reduce the difficulty of manufacturing and to prevent the cracking of the acrylic prototype during operation. The angle between the blade and

CHAPTER 9: CONCLUSIONS AND FUTURE WORKS

the hub is 90° , and the sharp angle is for the ease of mesh generation. However, the corner is rounded with small radii in the prototype. Due to the small clearance gap between the impeller shroud and the pump casing, the flow field in the gap and the leakage flow rate were not measured. The hemolysis was only estimated numerically, and should be measured experimentally in *in-vitro* test using the blood of either human or animal. The simulations and measurements were conducted under steady flow condition based on the assumption that the VAD replaces the native left ventricle completely. When the native left ventricle is partially recovered, it will produce pulsatile pressure wave. Therefore, it would be of interest to investigate the flow inside the VAD with transient flow condition, i.e. pulsatile pressure head. As an extension to the current work, the followings are suggested:

1. Although the blood damage potential of the new model has been estimated numerically, there may be a big difference between the numerically estimated blood damage and that conducted from the hemolysis test using human or bovine blood. Therefore, the hemolysis tests are recommended in order to build up a relationship between the numerical estimation and the experimental result, which will be useful in improving the design of VAD in the future.
2. Although the flow field in the diffuser channel is improved without obvious flow separation when the clearance gap between the diffuser blade and the rotating hub is 0.2 mm. There is still room for improvement on the flow field as well as reducing the blood damage potential. The swirling flow is still present at the diffuser downstream, and should be improved in the future.
3. Through the comparisons between the models with and without impeller shroud, it has been found that the impeller without shroud is a better design. As a result, the magnet material should be embedded into the impeller blades or hubs, though it

CHAPTER 9: CONCLUSIONS AND FUTURE WORKS

increases the difficulty of the magnetic levitation design and control technique.

When the impeller shroud is removed, the impeller profile can be further improved in order to make the VAD operate at high hydraulic efficiency under the selected rotating speed.

4. Since the flow field near the diffuser hub is irregular when the diffuser blades are attached to the hub, it may result in thrombus formation. This should be experimentally tested in the future in order to establish the relation between the thrombus formation and the irregular flow field. Furthermore, the same designs with the diffuser blade detached from the hub should be also tested in order to estimate the influences on the thrombus formation.
5. Although the flow parameters such as flow fields and blood damage indices under the normal flow condition have been simulated, it shall be beneficial to evaluate those at off design conditions in order to know the performances of the VAD model under different operating conditions. Therefore, the VAD can be operated at lower requirement when the native heart can still function and partially satisfy the body requirement.

References

1. **Fact Sheet Who/317.** World Health Organization (WHO) 2007. [http://whqlibdoc.who.int/fact_sheet/2007/FS_317.pdf]
2. Mielniczuk L, Mussivand T, Davies R, Mesana TG, Hendry RG, Mastersand PJ, Keon WJ, Haddad HA. **Patient Selection for Left Ventricular Assist Devices.** *Artif Organs* 2004;28:152-7.
3. DeBaakey ME. **The Odyssey of Artificial Heart.** *Artif Organs* 2000;24:405-11.
4. Olsen DB. **The History of Continuous-Flow Blood Pumps.** *Artif Organs* 2000;24:401-4.
5. Song X, Throckmorton AL, Wood HG, Antaki JF, Olsen DB. **CFD Prediction of Blood Damage in a Centrifugal Pump.** *Artif Organs* 2003;27:938-41.
6. Matsuda H, Matsumiya G. **Current Status of Left Ventricular Assist Devices: The Role in Bridge to Heart Transplantation and Further Perspectives.** *Artif Organs* 2003;6:157-61.
7. Chan WK, Wong YW, Ong W, Koh SY, Chong V. **Numerical Investigation of the Effects of the Clearance Gap between the Inducer and Impeller of an Axial Blood Pump.** *Artif Organs* 2005;29:250-8.
8. Okada Y, Masuzawa T, Matsuda K, Ohmori KI, Yamane T, Fukahori Y, Konishiand S, Ueno S, Kim SJ. **Axial Type Self-Bearing Motor for Axial Flow Blood Pump.** *Artif Organs* 2003;27:887-91.
9. Mitamura Y, Nakamura H, Okamoto E, Yozu R, Kawada S, Kim D-W. **Development of the Valvo Pump: An Axial Flow Pump Implanted at the Heart Valve Position.** *Artif Organs* 1999;23:566-71.

REFERENCES

10. Wood HG, Throckmorton AL, Untarolu A, Song X. **The Medical Physics of Ventricular Assist Devices.** *Rep Prog Phys* 2005;68:545-76.
11. Nose Y. **A Rotary Blood Pump: Its Design and Development Strategy.** *Artif Organs* 1997;21:263-4.
12. Lim TM, Zhang D. **Development of Lorentz Force-Type Self-Bearing Motor for an Alternative Axial Flow Blood Pump Design.** *Artif Organs* 2006;30:347-53.
13. Mitoh A, Yano T, Sekine K, Mitamura Y, Okamoto E, Kim D-W, Yozu R, Kawada S. **Computational Fluid Dynamics Analysis of an Intra-Cardiac Axial Flow Pump.** *Artif Organs* 2003;27:34-40.
14. Geron A, Farinas MI. **Fast Three-Dimensional Numerical Hemolysis Approximation.** *Artif Organs* 2004;28:1016-25.
15. Alemu Y, Bluestein D. **Flow-Induced Platelet Activation and Damage Accumulation in a Mechanical Heart Valve: Numerical Studies.** *Artif Organs* 2007;31:677-88.
16. Tamagawa M, Akamatsu T, Saito K. **Prediction of Hemolysis in Turbulent Shear Orifice Flow.** *Artif Organs* 1996;20:553-9.
17. Zhang Y, Song X, Gui X, Sun H, Zhang H, Zhu X, Hu S. **A Novel Integrated Rotor of Axial Blood Flow Pump Designed with Computational Fluid Dynamics.** *Artif Organs* 2007;31:580-5.
18. Chua LP, Ong KS, Song G, Ji W. **Measurements by Laser Doppler Velocimetry in the Casing/Impeller Clearance Gap of a Biocentrifugal Ventricular Assist Device Model.** *Artif Organs* 2009;33:360-72.
19. Morshuis M, El-Banayosy A, Arusoglu L, Koerfer R, Hetzer R, Wieselthaler G, Pavie A, Nojiri C. **European Experience of Duraheart(Tm) Magnetically**

REFERENCES

- Levitated Centrifugal Left Ventricular Assist System.** *European Journal of Cardio-thoracic Surgery* 2009;35:1020-8.
20. Zhu X, Zhang M, Zhang G, Liu H. **Numerical Investigation on Hydrodynamics and Biocompatibility of a Magnetically Suspended Axial Blood Pump.** *ASAIO J* 2006;52:624-9.
21. Lim TM, Cheng S, Chua LP. **Parameter Estimation and Actuator Characteristics of Hybrid Magnetic Bearing for Axial Flow Blood Pump Applications.** *Artif Organs* 2009;33:509-31.
22. Day SW, McDaniel JC. **Piv Measurements of Flow in a Centrifugal Blood Pump: Time-Varying Flow.** *Transaction of ASME* 2005;127:254-63.
23. Day SW, McDaniel JC. **Piv Measurements of Flow in a Centrifugal Blood Pump: Steady Flow.** *Transaction of ASME* 2005;127:244-53.
24. Sankovic JM, Kadambi JR, Mehta M, Smith WA, Wernet MP. **Piv Investigation of the Flow Field in the Volute of a Rotary Blood Pump.** *ASME Transaction of ASME* 2004;126:730-34.
25. Day SW, McDaniel JC, Wood HG, Allaire PE, Song X, Miles PP, Lemireand SD. **A Prototype Heartquest Ventricular Assist Device for Particle Image Velocimetry Measurement.** *Artif Organs* 2002;26:1002-5.
26. Song X, Wood HG, Day SW, Olsen DB. **Studies of Turbulence Models in a Computational Fluid Dynamics Model of a Blood Pump.** *Artif Organs* 2003;27:935-37.
27. Triep M, Brucker C, Schroder W, Siess T. **Computational Fluid Dynamics and Digital Particle Image Velocimetry Study of the Flow through an Optimized Micro-Axial Blood Pump.** *Artif Organs* 2006;30:384-91.
28. Brooks DE, Goodwin TW, Seaman GVF. **Interactions among Erythrocytes under Shear.** *J Appl Physiol* 1970;28:172-7.

REFERENCES

29. Joles JA, Willekes-Koolschijn N, Koomans HA. **Hypoalbuminemia Cause High Blood Viscosity by Increasing Red Cell Lysophosphatidycholine.** *Kidney Int* 1997;52:761-70.
30. Eckmann DM, Bowers S, Stecker M, Cheung AT. **Hematocrit Volume Expander, Temperature and Shear Rate Effects on Blood Viscosity.** *Anesh Analg* 2000;91:539-45.
31. Song X, Wood HG, Olsen D. **Computational Fluid Dynamics (CFD) Study of the 4th Generation Prototype of a Continuous Flow Ventricular Assist Device (Vad).** *J Biomech Eng* 2004;126:180-7.
32. Kido K, Hoshi H, Watanabe N, Kataoka H, Ohuchi K, Asama J, Shinshi T, Yoshikawa M, Takatani S. **Computational Fluid Dynamics Analysis of the Pediatric Tiny Centrifugal Blood Pump (Tinypump).** *Artif Organs* 2006;30:392-9.
33. Chua LP, Su BY, Lim TM, Zhou TM. **Numerical Simulation of an Axial Blood Pump.** *Artif Organs* 2007;31:560-70.
34. Leverett LB, Hellums JD, Alfrey CP, Lynch EC. **Red Blood Cell Damage by Shear Stress.** *Biophys J* 1972;12:257-73.
35. Arora D, Behr M, Pasquali M. **Tensor-Based Measure for Estimating Blood Damage.** *Artif Organs* 2004;28:1002-15.
36. Blackshear PL, Blackshear GL. **Handbook of Bioengineering:** McGraw-Hill Inc New York 1987.
37. Rand R. **Mechanical Properties of the Red Cell Membrane: Ii Viscoelastic Breakdown of the Membrane.** *Biophys J* 1964;4:303-16.
38. Yeleswarapu KK, Antaki JF, Kameneva MV, Rajagopal KR. **A Mathematical Model for Shear-Induced Hemolysis.** *Artif Organs* 1995;19:576-82.

REFERENCES

39. Sutura SP, Mehrjardi MH. **Deformation and Fragmentation of Human Red Blood Cells in Turbulent Shear Flow.** *Biophys J* 1975;15:1-10.
40. Sallam AM, Hwang NHC. **Human Red Blood Cell Hemolysis in Turbulent Shear Flow: Contribution of Reynolds Shear Stresses.** *Biorheology* 1984;21:783-97.
41. Hasenkam JM, Westphal D, Nygaard H, Reul H, Giersiepen M, Stodkilde-Jorgensen H. **In Vitro Stress Measurements in the Vicinity of Six Mechanical Aortic Valves Using Hot-Film Anemometry in Steady Flow.** *Journal of Biomechanics* 1988;21:243-47.
42. Grigioni M, Daniele C, D'Avenio G, Barbaro V. **A Discussion on the Threshold Limit for Hemolysis Related to Reynolds Shear Stress.** *Journal of Biomechanics* 1999;32:1107-12.
43. Throckmorton AL, Untaroiu A, Allaire PE, Wood HG, Matherne GP, Lim DS, Peeler BB, Olsen DB. **Computational Analysis of an Axial Flow Pediatric Ventricular Assist Device.** *Artif Organs* 2004;28:881-91.
44. Akamatsu T, Tsukiya T, Nishimura K, Park CH, Nakazeki T. **Recent Studies of the Centrifugal Blood Pump with Magnetic Bearing.** *Artif Organs* 1995;19:631-4.
45. Paul R, Apel J, Schugner S, Shwindke P, Reul H. **Shear Stress Related Blood Damage in Laminar Couette Flow.** *Artif Organs* 2003;27:517-29.
46. Wurzing LJ, Opitz R, Blasberg P, Schmid-Schonbein H. **Platelet and Coagulation Parameters Following Millisecond Exposure to Laminar Shear Stress.** *Thromb Haemost* 1985;54:381-6.
47. Apel J, Paul R, Klaus S, Siess T, Reul H. **Assessment of Hemolysis Related Quantities in a Microaxial Blood Pump by Computational Fluid Dynamics.** *Artif Organs* 2001;25:341-47.

REFERENCES

48. Arvand A, Hormes M, Reul H. **A Validated Computational Fluid Dynamics Model to Estimate Hemolysis in a Rotary Blood Pump.** *Artif Organs* 2005;29:531-40.
49. Yano T, Sekine K, Mitoh A, Mitamura Y, Okamoto E, Kim D-W, Nishimura I, Murabayashi S, Yozu R. **An Estimation Method of Hemolysis within an Axial Flow Blood Pump by Computational Fluid Dynamics Analysis.** *Artif Organs* 2003;27:920-5.
50. Moen O, Fosse E, Dregelid E, Brockmeier V, Andersson C, Høga, sen K, Venge P, Mollnes TE, Kierulf P. **Centrifugal Pump and Heparin Coating Improves Cardiopulmonary Bypass Biocompatibility.** *The Annals of Thoracic Surgery* 1996;62:1134-40.
51. Yoshikawa M, Nakata KI, Ohtsuka G, Takano T, Glueck J, Fujisawa A, Makinouchi K, Yokokawa M, Nosé Y. **Feasibility of a Tiny Gyro Centrifugal Pump as an Implantable Ventricular Assist Device.** *Artif Organs* 1999;23:774-9.
52. ASTM. **Standard Practice for Assessment of Hemolysis in Continuous Flow Blood Pumps.** *Annual Book ASTM Standard* 1997:1288-92.
53. Yamane T, Maruyama O, Nishida M, Toyoda M, Tsutsui T, Shigeta T, Jikuyaand O, Sankai Y. **The Most Profitable Use of Flow Visualization in the Elimination of Thrombus from a Monopivot Magnetic Suspension Blood Pump.** *Artif Organs* 2004;28:390-7.
54. Nose Y. **Design and Development Strategy for the Rotary Blood Pump.** *Artif Organs* 1998;22:438-46.
55. Kihara S, Yamazaki K, Litwak KN, Litwak P, Kameneva MV, Ushiyama H, Tokuno T, Borzelleca DC, Umezu M, Tomioka J, Tagusari O, Akimoto T, Koyanagi H, Kurosawa H, Kormos RL, Griffith BP. **In Vivo Evaluation of a Mpc Polymer Coated Continuous Flow Left Ventricular Assist System.** *Artif Organs* 2003;27:188-92.

REFERENCES

56. Dinnar U. **Cardiovascular Fluid Dynamics**: Boca Raton: CRC Press, Inc 1981.
57. Moake JL, Turner NA, Stathopoulos NA, Nolaseo L, Hellums JD. **Shear-Induced Platelet-Aggregation Can Be Mediated by Vwf Released from Platelet, as Well as by Exogenous Large or Unusually Large Vwff Multimers, Requires Adenosine-Diphosphate, and Is Resistant to Aspirin.** *Blood* 1988;71:1366-74.
58. Ramstack JM, Zuckerman L, Mockros LF. **Shear-Induced Activation of Platelets.** *Journal of Biomechanics* 1979;12:113-25.
59. Hashimoto S, Maeda H, Sasada T. **Effect of Shear Rate on Clot Growth at Foreign Surfaces.** *Artif Organs* 1985;9:345-50.
60. Wurzinger LJ, Blasberg P, Schmid-Schonbein H. **Towards a Concept of Thrombosis in Accelerated Flow: Rheology, Fluid Dynamics and Biochemistry.** *Biorheology* 1985;22:437-500.
61. Potapov EV, Loebe M, Nasser BA, Sinawski H, Koster A, Kuppe H, Noon GP, DeBakey ME, Hetzer R. **Pulsatile Flow in Patients with a Novel Nonpulsatile Implantable Ventricular Assist Device.** *Circulation* 2000;102:183-7.
62. Baba A, Dobsak P, Mochizuki S, Saito I, Isoyama T, Shibata L, Takiura M, Abe Y, Chinzei T, Vasku J, Imachi K. **Evaluation of Pulsatile and Nonpulsatile Flow in Microvessels of the Bulbar Conjunctiva in the Goat with an Undulation Pump.** *Artif Organs* 2003;27:875-81.
63. Sezai A, Shiono M, Orime Y, Nakata K, Hata M, Iida M, Kashiwazaki S, Kinoshita J, Nemoto M, Koujima T, Eda MFK, Hirose H, Yoshino T, Saito A, Taniguchi Y, Ysezai. **Major Organ Function under Mechanical Support Comparative Studies of Pulsatile and Nonpulsatile Circulation.** *Artif Organs* 1999;23:280-5.
64. Wesolowski SA, Fisher JH, Welch CS. **Perfusion of the Pulmonary Circulation by Nonpulsatile Flow.** *Surgery* 1953;33:370-5.

REFERENCES

65. Bendowski R, Morley D, Abdelsayed S, Noon GP. **Clinical Update and Transition to Destination Therapy for the Micromed Debakey Vad.** *10th Congr Inte Soc of Rotary Blood Pumps.* Osaka Japan 2002.
66. Garatti A, Bruschi G, Colombo T, Russo C, Lanfranconi M, Milazzo F, Frigerio M, Vitali E. **Clinical Outcome and Bridge to Transplant Rate of Left Ventricular Assist Device Recipient Patients: Comparison between Continuous-Flow and Pulsatile-Flow Device.** *European Journal of Cardio-thoracic Surgery* 2008;34:275-80.
67. Burke DJ, Burke E, Parsaie F, Poirier V, Bulter K, Thomas D, Taylor L, Mather T. **The Heartmate II: Design and Development of a Fully Sealed Axial Flow Left Ventricular Assist System.** *Artif Organs* 2001;25:380-5.
68. Araki K, Taenaka Y, Masuzawa T, Inoue K, Nakatani T, Akagi M, Kinoshitaand H, Baba H, Matsuo Y, Sakaki M, Anai H, Takano H. **A New Visualization Study of Centrifugal Pumps Developed for Long Term Usage.** *Artif Organs* 1993;17:307-12.
69. Nose Y, Kawahito K, Nakazawa T. **Can We Develop a Non-Pulsatile Permanent Rotary Blood Pump? Yes, We Can.** *Artif Organs* 1996;20:467-74.
70. Bulter KC, Maher TR, Borovetz HS, Kormos RL, Kamenewa JF, Antakiand MV, Griffith BP, Zerbe T, Schaffer FD. **Development of an Axial Flow Blood Pump Lvas.** *ASAIO J* 1992;38:296-300.
71. Sutura SP, Nowak MD, Joist JH, Zeffren DJ, Bauman JE. **A Programmable, Computer-Controlled Cone-Plate Viscometer for the Application of Pulsatile Shear Stress to Platelet Suspensions.** *Biorheology* 1988;25:449-59.
72. Chung MKH, Zhang N, Tansley GD, Qian Y. **Experimental Determination of Dynamic Characteristics of the Ventrassist Implantable Rotary Blood Pump.** *Artif Organs* 2004;28:1089-94.

REFERENCES

73. Beams JW. **Magnetic Suspension for Small Rotors.** *Rev Sci Inst* 1950;21:182-4.
74. Akamatsu T, Nakazeki T, Itoh H. **Centrifugal Blood Pump with a Magnetically Suspended Impeller.** *Artif Organs* 1992;16:305-8.
75. Allaire PE, Kim HC, Maslen EH, Olsen DB, Bearnson GB. **Prototype Continuous Flow Ventricular Assist Device Supported on Magnetic Bearing.** *Artif Organs* 1996;20:582-90.
76. Hoshi H, Shinshi T, Takatani S. **Third-Generation Blood Pumps with Mechanical Noncontact Magnetic Bearing.** *Artif Organs* 2006;30:324-38.
77. Nojiir C, Kijima T, Maekawa J, Horiuchi K, Kido T, Sugiyama T, Mori T, Sugiura N, Asada T, Ozaki T, Suzuki M, Akamatsu T, Akutsu T. **Terumo Implantable Left Ventricular Assist System: Results of Long-Term Animal Study.** *ASAIO J* 2000;46:117-22.
78. Goldowsky M. **Magnevad-the World's Smallest Magnetic-Bearing Turbo Pump.** *Artif Organs* 2004;28:945-52.
79. Borovetz HS, Ramasamy N, Zerbe TR, Portner PM. **Evaluation of an Implantable Ventricular Assist System for Humans with Chronic Refractory Heart Failure.** *ASAIO J* 1995;41:42-8.
80. Tayama E, Olsen DB, Ohashi Y, Benkowski R, Morley D, Noon GP, Nose Y, DeBakey ME. **The DeBakey Ventricular Assist Device: Current Status in 1997.** *Artif Organs* 1999;23:1113-6.
81. Goldstein D, Oz M. **Cardiac Assist Devices:** New York: Future Publishing Company 2000.
82. Duncan BW, Dudzinski DT, Gu L, Mielke N, Noecker AM, Kopcak MW, Fukamachi K, Cingoz F, Ootaki Y, Smith WA. **The Pedipump: Development Status of a New Pediatric Ventricular Assist Device: Update II.** *ASAIO J* 2006;52:581.

REFERENCES

83. Song X, Throckmorton AL, Untaroiu A, Patel S, Allaire PE, Wood HG, Olsen DB. **Axial Flow Blood Pumps**. *ASAIO J* 2003;49:355-64.
84. Bearnson GB, Jacobs GB, Kirk J, Khanwilkar PS, Nelson KE, Long JW. **Hearquest Ventricular Assist Device Magnetically Levitated Centrifugal Blood Pump**. *Artif Organs* 2006;30:339-46.
85. Curtas A, Wood HG, Allaire PE, McDaniel JC, Day SW, Olsen DB. **CFD Modeling of Impeller Design for the Hearquest Lvad**. *ASAIO J* 2002;48:552-61.
86. Nojiri C, Kijima T, Maekawa J, Horiuchi K, Kido T, Sugiyama T, Mori T, Sugiura N, Asada T, Umenura W, Ozaki T, Suzuki M, Akamatsu T, Westaby S, Katsumata T, Saito S. **Development Status of Terumo Implantable Left Ventricular Assist System**. *Artif Organs* 2001;25:411-13.
87. Fossum T, Morley D, Benkowski R. **Chronic Survival of Calves Implanted with the Debakey Ventricular Assist Device**. *Artif Organs* 1999;23:802-6.
88. Frazier OH, Myers TJ, Gregoric ID, Khan T, Delgado R, Croitoru M, Miller K, Jarvik R, Westaby S. **Initial Clinical Experience with the Jarvik 2000 Implanted Axial-Flow Left Ventricular Assist System**. *Circulation* 2002;105:2855-60.
89. Goldowsky M, Lafaro R, Reed G. **Magnevad Status of Design Improvements Human Blood Results and Preliminary Sheep Trail**. *Artif Organs* 2005;29:855-57.
90. Anderson JB, Wood HG, Allaire PE, McDaniel JC, Olsen DB, Bearnson G. **Numerical Studies of Blood Shear and Washing in a Continuous Flow Ventricular Assist Device**. *ASAIO J* 2000;46:486-94.
91. Antaki JF, Ghattas O, Burgreen GW, He B. **Computational Flow Optimization of Rotary Blood Pump Components**. *Artif Organs* 1995;19:608-15.
92. Miyazoe Y, Sawairi T, Ito K. **Computational Fluid Dynamics Analyses to Establish Design Process of Centrifugal Blood Pumps**. *Artif Organs* 1998;22:381-5.

REFERENCES

93. Wu JC, Antaki JF, Wanger WR, Snyder TA, Paden BE, Borovetz HS. **Elimination of Adverse Leakage Flow in a Miniature Pediatric Centrifugal Blood Pump by CFD-Based Design Optimization.** *ASAIO J* 2005;51:636-43.
94. Burgreen GW, Antaki JF, Wu ZJ, Holmes AJ. **Computational Fluid Dynamics as a Development Tool for Rotary Blood Pumps.** *Artif Organs* 2001;25:336-40.
95. Song X, Untaroiu A, Wood HG, Allaire PE, Throckmorton AL, Day SW, Olsen DB. **Design and Transient Computational Fluid Dynamics Study of a Continuous Axial Flow Ventricular Assist Device.** *ASAIO J* 2004;50:215-24.
96. Giersiepen M, Wurzingler LJ, Opitz R, Reul H. **Estimation of Shear Stress-Related Blood Trauma in Heart Valve Prostheses in Vivo Comparison of 25 Aortic Valves.** *Artif Organs* 1990;12:130-6.
97. Schima H, Muller MR, Tsangaris S, Gheiseler G, Schulusche C, Losert U, Thomas H, Wolner E. **Mechanical Blood Traumatization by Tubing and Throttles in in Vitro Pump Tests: Experimental Results and Implications for Hemolysis Theory.** *Artif Organs* 1993;17:164-70.
98. Wachter DD, Verdonck P, Verhoeven R, Hombrouckx R. **Red Cell Injury Assessed in a Numerical Model of a Peripheral Dialysis Needle.** *ASAIO J* 1996;42:524-9.
99. Wachter DD, Verdonck P. **Numerical Calculation of Hemolysis Levels in Peripheral Hemodialysis Cannulas.** *Artif Organs* 2002;26:576-82.
100. Bludszuweit C. **Model for General Mechanical Blood Damage Prediction.** *Artif Organs* 1995;19:583-9.
101. Farinas MI, Garon A, Lacasse D, N'Dri D. **Asymptotically Consistent Numerical Approximation of Hemolysis.** *Transaction of ASME* 2006;128:688-96.

REFERENCES

102. Zhang J, Gellman B, Koert A, Dasse KA, Gilbert RJ, Griffith BP, Wu ZJ. **Computational and Experimental Evaluation of the Fluid Dynamics and Hemocompatibility of the Centrimag Blood Pump.** *Artif Organs* 2006;30:168-77.
103. Maymir JC, Deutsch S, Meyer RS, Geselowitz DB, Tarbell JM. **Mean Velocity and Reynolds Stress Measurement in the Regurgitant Jets of Tilting Disk Heart Valves in an Artificial Heart Environment.** *Ann Biomed Eng* 1998;26:146-56.
104. Meyer RS, Deutsch S, Maymir JC, Geselowitz DB, Tarvbell JM. **Three-Component Laser Doppler Velocimetry Measurements in the Regurgitant Flow Region of a Bjork-Shiley Monostrut Mitral Valve.** *Ann Biomed Eng* 1997;25:1081-91.
105. Browne P, Ramuzat A, Saxena R, Yoganathan AP. **Experimental Investigation of the Steady Flow Downstream of the St. Jude Bileaflet Heart Valve: A Comparison between Laser Doppler Velocimetry and Particle Image Velocimetry Techniques.** *Annals of Biomedical Engineering* 2000;28:39-47.
106. Moghtaderi B. **Application of Laser Doppler Velocimetry (Ldv) to Study the Influence of Heat Transfer on the Structure of Gravity Currents.** *Exp Thermal and Fluid Sci* 2004;28:843-52.
107. Day SW, McDaniel JC, Curtas A, Wood HG, Allaire PE, Fan N, Landrotand H, Olsen D, Long J. **Optical Measurement of Blood Velocity and Shear in Heartquest Lvad.** *ASAIO J* 2000;46:188.
108. Day SW, McDaniel JC, Wood HG, Allaire PE, Landrot N, Curtas A. **Piv Measurement of Blood Velocity in a Continuous Flow Ventricular Assist Device.** *ASAIO J* 2001;47:406-11.
109. Chua LP, Ong KS, Song G. **Study of Velocity and Shear Stress Distributions in the Impeller Passages and the Volute of a Bio-Centrifugal Ventricular Assist Device.** *Artif Organs* 2008;32:376-87.

REFERENCES

110. Baldwin JT, Tarbell JM, Deutsch S, Geselowitz DB, Rosenberg G. **Hot Film Wall Shear Probe Measurement inside a Ventricular Assist Device.** *ASME J Biomech Eng* 1988;110:326-33.
111. Behbahani M, Behr M, Hormes M, Steinseifer U, Arora D, Coronado O, Pasquali M. **A Review of Computational Fluid Dynamics Analysis of Blood Pumps.** *European Journal of Applied Mathematics* 2009;20:363-97.
112. Hoshi H, Asama J, Hijikata W, Hara C, Shinshi T, Yasuda T, Ohuchi K, Shimokohbe A, Takatani S. **Hemolytic Performance of a Maglev Disposable Rotary Blood Pump (Medtech Dispo): Effects of Maglev Gap Clearance and Surface Roughness.** *Artif Organs* 2006;30:949-54.
113. DiSesa VJ. **Advanced Therapy of Cardiac Surgery.** 2nd edition ed. Hamilton, Ontario, Canada, 2004.
114. Song X, Throckmorton AL, Wood HG, Antaki JF, Olsen DB. **Quantitative Evaluation of Blood Damage in a Centrifugal Vad by Computational Fluid Dynamics.** *J Fluids Eng* 2004;126:410-8.
115. Akamatsu T, Tsukiya T, Ozaki T. **Fluid Engineering Aspect for the Development of the Centrifugal Blood Pump with Magnetically Suspended Impeller.** Berlin: Springer-Verlag 2000.
116. Bludszuweit C. **Three-Dimensional Numerical Prediction of Stress Loading of Blood Particles in a Centrifugal Pump.** *Artif Organs* 1995;19:590-6.
117. Takiura K, Masuzawa T, Endo S. **Development of Design Methods of a Centrifugal Blood Pump with in Vitro Test, Flow Visualization and Computational Fluid Dynamics: Result Hemolysis Tests.** *Artif Organs* 1998;22:393-8.
118. Allaire PE, Wood HG, Award RS, Olsen DB. **Blood Flow in a Continuous Flow Ventricular Assist Device.** *Artif Organs* 1999;23:769-73.

REFERENCES

119. Miyazoe Y, Sawairi T, Ito K, Konishi Y, Yamane T, Asztalos M, Nishidaand B, Masuzawa T, Tsukiya T, Endo S, Taenaka Y. **Computational Fluid Dynamics Analysis to Establish the Design Process of a Centrifugal Blood Pump: Second Report.** *Artif Organs* 1999;23:762-8.
120. Anderson J, Wood HG, Allaire PE, Bearnson G, Khanwilkar P. **Computational Flow Study of the Cfvad 3 Blood Pump.** *Artif Organs* 2000;24:377-85.
121. Zhu L, Zhang X, Yao Z. **Shape Optimization of the Diffuser Blade of an Axial Blood Pump by Computational Fluid Dynamics.** *Artif Organs* 2010;34:185-92.
122. Throckmorton AL, Kishore RA. **Design of a Protective Cage for an Intravascular Axial Flow Blood Pump to Mechanically Assist the Failing Fontan.** *Artif Organs* 2009;33:611-21.
123. Taskin ME, Fraser KH, Zhang T, Gellman B, Fleischli A, Dasse KA, Griffith BP, Wu ZJ. **Computational Characterization of Flow and Hemolytic Performance of the Ultramag Blood Pump for Circulatory Support.** *Artif Organs* 2010;34:1099-113.
124. Throckmorton AL, Kapadia JY, Chopski SG, Bhavsar SS, Moskowitz WB, Gullquist SD, Gangemi JJ, Haggerty CM, Yoganathan AP. **Numerical, Hydraulic, and Hemolytic Evaluation of an Intravascular Axial Flow Blood Pump to Mechanically Support Fontan Patients.** *Annals of Biomedical Engineering* 2011;39:324-36.
125. Hassan YA, Dominguez-Ontiveros EE. **Flow Visualization in a Pebble Bed Reactor Experiment Using Piv and Refractive Index Matching Techniques.** *Nuclear Engineering and Design* 2008;238:3080-5.

REFERENCES

126. Doyle MG, Vergiaud JB, Tavoularis S, Bourgault Y. **Numerical Simulations of Blood Flow in Artificial and Natural Hearts with Fluid-Structure Interaction.** *Artif Organs* 2008;32:870-9.
127. Dantec-Dynamics. **Flowmanager Software and Introduction to Piv Instrumentation** 2002.
128. Roth GI, Katz J. **Five Techniques for Increasing the Speed and Accuracy of Piv Interrogation.** *Measurement Science and Technology* 2001;12:238-45.
129. Anderson JB, Wood HG, Allaire PE, Olsen DB. **Numerical Analysis of Blood Flow in the Clearance Regions of a Continuous Flow Artificial Heart Pump.** *Artif Organs* 2000;24:492-500.
130. Chua LP, Ong KS, Yu SCM, Zhou T. **Leakage Flow Rate and Wall Shear Stress Distribution in a Biocentrifugal Ventricular Assist Device.** *ASAIO J* 2004;50:530-6.
131. Chua LP, Song GL, Yu SCM, Zhou TM. **Computational Fluid Dynamics of Gap Flow in a Biocentrifugal Blood Pump.** *Artif Organs* 2005;29:620-28.
132. Zhang Y, Zhan Z, Gui X, Sun HS, Zhang H, Zheng Z, Zhou JY, Zhu XD, Li GR, Hu SS, Jin DH. **Design Optimization of an Axial Blood Pump with Computational Fluid Dynamics.** *ASAIO J* 2008;54:150-5.
133. Song X, Throckmorton AL, Wood HG, Allaire PE, Olsen DB. **Transient and Quasi-Steady Computational Fluid Dynamics Study of a Left Ventricular Assist Device.** *ASAIO J* 2004;50:410-17.

Appendix

A: Straightener

REFERENCES

B: Impeller

REFERENCES

C: Diffuser

CAD drawings of (A) the straightener, (B) impeller and (C) diffuser for NM1 model. The unit is mm. The black dot is the control point to define the spline of the hub contour in both straightener and diffuser.