Effects of geometrical parameters on performance of miniature centrifugal pump

Liu, Guiqin

2014

Liu, G. (2014). Effects of geometrical parameters on performance of miniature centrifugal pump. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/61904>

<https://doi.org/10.32657/10356/61904>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

**EFFECTS OF GEOMETRICAL PARAMETERS
ON PERFORMANCE OF MINIATURE
CENTRIFUGAL PUMP**

LIU GUIQIN

**SCHOOL OF MECHANICAL AND AEROSPACE
ENGINEERING**

2014

**EFFECTS OF GEOMETRICAL PARAMETERS
ON PERFORMANCE OF MINIATURE
CENTRIFUGAL PUMP**

LIU GUIQIN

LIU GUIQIN

School of Mechanical and Aerospace Engineering

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

2014

Acknowledgements

In the first place, I would like to express my deepest sense of gratitude to my supervisor, Professor Chan Weng Kong, who was abundantly helpful and offered invaluable assistance, advice, and guidance throughout the work in every step of my research. His sharp sight, patient explanation and continuous encouragement have enlightened me to a more profound and extensive academic future.

I would like to thank CNC technician Mr. Kong Seng Ann, Fluid Mechanics Lab technician Mr. Yap Pow Kim and Thermal Fluid Research Lab 1 technician Mr. Yuen Kee Hock for helping me on milling impellers and volutes for my experimental research. Besides, I appreciate all staffs and students in Thermal and Fluid Research Lab for their suggestions and helps.

Last but not least, I take this opportunity to express my appreciation to my beloved families. My parents and husband deserve special mention for their inseparable support and endless love through the duration of my studies.

Abstract

In this project, the effects of blade geometrical parameters on the performance of a miniature centrifugal pump are examined. Geometrical parameters such as blade angle, inner radius, blade thickness, blade number, volute outlet width, together with the flow coefficient are investigated. The performances of the pumps designed are analyzed theoretically, numerically and experimentally. Energy models including Euler head, slip loss, leakage loss, friction loss, blockage loss, diffusion loss and shock loss are considered in the theoretical analysis. 3-D numerical studies using a commercial software FLUENT showed that the results are consistent with theoretical results. Experimental studies on the pump characteristics and velocities in the impeller passage were performed. The numerical and theoretical results are consistent with experimental results.

Numerical and experimental results showed that the traditionally designed impeller cannot operate at the design point because the pressure generated is negative. The impeller channel is short and the fluid does not have enough time to receive energy from rotating impeller. Modified models registered positive pressure generated.

Theoretical results show that wall friction loss and tip leakage loss are negligible, while the slip loss is most significant. The blockage loss, diffusion loss and disk friction loss increase with flow rate, and decrease with the radius ratio. The leakage loss is significant when the radius ratio r_1/r_2 is larger than 0.4, and the shock loss should be considered when the blade angle β is smaller than 45° . The leakage flow rate is 15% of the total flow rate when the radius ratio is 0.4 and is 52% when the radius ratio is 0.9. The shock loss is 9% of the theoretical head when the blade angle is 45° and is 70% when the blade angle is 30° .

The effect of flow coefficient is studied. This parameter is important in the design procedure, and the recommended value for macro-size pump is not suitable for miniature pumps. When the flow coefficient is varied for a given outer radius and operating speed, the maximum pressure generated is obtained at the optimum flow coefficient. Simulation results show the optimized flow coefficient is in the range of 1.47 to 2 for the 20 mm diameter model.

The effects of blade angle, inner radius, blade number and thickness are investigated. With increasing blade angle, and decreasing blade thickness, the pressure generated is increasing. Optimized values of inner radius and blade number exist. According to the theoretical results, the optimum radius is 5.4 mm and 2.7 mm for 20 mm and 10 mm diameter models respectively. The optimum blade number is 13 for 20 mm diameter models while 11 for 10 mm diameter models.

Two volute design methods, namely, Constant Mean Velocity (CMV) method and Constant Angular Momentum (CAM) method are compared. For the same impeller operating at the same rotational speed, the difference between them is negligible, because the energy losses are mainly due to the volute length. For the same volute height and outlet width, the difference in the volute length is only 1% for the two methods.

The results obtained in the present study provide us guidelines on the design and performance study of miniature centrifugal pump.

Table of Contents

Cover Page	i
Title Page	ii
Acknowledgements	iii
Abstract	iv
Table of Contents.....	vi
Nomenclature	xi
List of Figures	xvi
List of Tables.....	xxii
1. Introduction.....	1
1.1 Background	1
1.2 Objectives	2
1.3 Scope of Work	3
2. Literature Review	5
2.1 General Introduction of Pumps	5
2.2 Centrifugal Pump Geometry	6
2.2.1 Impeller Geometry	7
2.2.2 Critical Dimensionless Parameters	8
2.2.3 Volute Profile.....	11
2.2.4 Critical Parameters in the Design of Volute	14
2.3 Centrifugal Pump Design.....	14
2.3.1 Impeller Design Methods.....	15
2.3.2 Volute Design Methods	16

2.4	Research on Miniature Pumps	18
2.4.1	Design and Performance	18
2.4.2	Pump Miniaturization Challenges.....	22
2.5	Pump Scaling Effect	23
2.6	Energy Models in Centrifugal Pumps.....	25
2.6.1	Euler Head	25
2.6.2	Slip Loss.....	26
2.6.3	Leakage Loss	27
2.6.4	Friction Loss	30
2.6.5	Blockage Loss	32
2.6.6	Diffusion Loss.....	32
2.6.7	Shock Loss	33
2.7	Effect of Blade Geometry on Pump Performance	33
2.8	Computational Fluid Dynamics	36
2.9	Particle Image Velocimetry Method.....	38
2.10	Summary	40
3.	Design of Miniature Centrifugal Pump	42
3.1	Impeller Design Procedure	42
3.2	Volute Design Procedure	45
3.2.1	Volute Profile with Constant Mean Velocity (CMV) Method	46
3.2.2	Volute Profile with Constant Angular Momentum (CAM) Method	47
3.3	Design Example of Miniature Pump.....	48
3.3.1	Specifications	49
3.3.2	Original Design of Impeller	49
3.3.3	Modified Models of Impeller.....	52

3.3.4	Original Design of Volute.....	55
3.3.5	Modified Models of Volute	57
3.4	Summary	58
4.	Computational Fluid Dynamics Modeling	59
4.1	Introduction of the Software FLUENT	59
4.2	Turbulent Models	60
4.3	Governing Equations	62
4.3.1	$k-\omega$ Model	63
4.3.2	Reynolds Stress Model	64
4.4	Physical Model and Boundary Conditions	65
4.5	Grid Generation	68
4.6	Convergence	70
4.7	Grid Independency	71
4.8	Simulation Procedure	73
5.	Experimental Setup	74
5.1	Testing Models.....	74
5.2	2-D PIV Measurement System	75
5.2.1	Tracer Particles	76
5.2.2	Illumination System	78
5.2.3	CCD Camera	80
5.2.4	Measurement Principle	82
5.2.5	Experimental Procedure.....	83
5.3	Pump Performance Measurement System	85
6.	Results and Discussions.....	89
6.1	Analysis of Uncertainty	91

6.2	Pump Performance Predicted by Theoretical Model	92
6.3	Pump Performance Predicted by Numerical Studies	102
6.3.1	Turbulent Models.....	102
6.3.2	Pump Performance Study	105
6.3.3	Pump Similarity	108
6.4	Pump Performance (Experimental Studies).....	111
6.4.1	Effect of Blade Geometry on Pump Performance	111
6.4.2	Pump Similarity	113
6.5	Comparison of the Theoretical, Numerical and Experimental Results.....	116
6.6	Effect of Blade Geometry on Pump Performance	119
6.6.1	Effect of the Blade Angle on Pump Performance.....	119
6.6.2	Effect of the Blade Inner Radius on Pump Performance	122
6.6.3	Effect of the Blade Thickness on Pump Performance	123
6.6.4	Effect of the Blade Number on Pump Performance	125
6.7	Effect of the Flow Coefficient on Pump Performance.....	128
6.8	Comparison of Original Designed Impeller with Modified Impeller	130
6.8.1	The Original-design Model 10.....	130
6.8.2	The Modified Model 5	131
6.9	Effect of the Volute Geometry on Pump Performance	132
6.10	Flow Field Investigation	135
6.10.1	Flow Field of 20 mm Diameter Model with the Originally Designed Impeller	135
6.10.2	Flow Field of 20 mm Diameter Model with the Modified Impeller...	137
6.10.3	Flow Field of the Radial Impeller Model	138
6.11	Summary	141

7. Conclusions and Future Works.....	144
7.1 Conclusions.....	144
7.1.1 Impeller Design.....	144
7.1.2 Volute Design	145
7.1.3 Energy Models	146
7.1.4 Numerical Models.....	146
7.1.5 Effect of the Pump Geometry	146
7.1.6 Performance Characterization.....	148
7.1.7 Flow Field	148
7.1.8 Pump Similarity	148
7.2 Future Works	148
References.....	150
Appendix.....	161

Nomenclature

A_0	cross-sectional area	m^2
A_{impeller}	impeller outlet area	m^2
A_T	cross section area at throat	m^2
A_{throat}	volute throat area	m^2
b_2	blade height	m
b_3	volute height	m
C_f	friction coefficient	
C_m	mean velocity	m.s^{-1}
C_u	tangential velocity in the volute channel	m.s^{-1}
d	volute outlet width	m
D	impeller diameter	m
D_h	hydraulic diameter	m
d_{image}	diameter of particle size of the captured image	m
d_s	specific diameter	
d_p	particle diameter	m
D_ω	divergence term of ω equation	$\text{J.kg}^{-1}.\text{s}^{-1}$
E_{blk}	blockage loss	m
E_{df}	disk friction loss	m
E_{wf}	impeller wall friction loss	m
E_i	external body force	N
E_{slip}	slip loss	m
E_{th}	theoretical Euler head	m
F	body force	N

$f_{\#}$	recording aperture	m
g	gravitational acceleration constant	m.s^{-2}
G_k	turbulent generation term of k equation	$\text{J.kg}^{-1}.\text{s}^{-1}$
G_{ω}	turbulent generation term of ω equation	$\text{J.kg}^{-1}.\text{s}^{-1}$
ΔH	head generated	m
k	turbulent kinetic energy	J.kg^{-1}
K	empirical factor developed by Anderson	m.s^{-2}
K_b	loss coefficient of blockage loss	
K_3	empirical design factor in CMV design method	
l	impeller length	m
L	tolerance offset	
N	rotational speed	rpm
n_s	specific rotational speed	
N_s	pump's specific speed	$\text{rpm}(\text{gpm})^{0.5}(\text{ft})^{-0.75}$
ΔP	pressure generated	Pa
P_{df}	power of disk friction loss	W
Q	flow rate	$\text{m}^3.\text{s}^{-1}$
Q_{lk}	leakage flow rate	$\text{m}^3.\text{s}^{-1}$
Q_{tip}	tip leakage flow rate	$\text{m}^3.\text{s}^{-1}$
Q_{θ}	flow rate of volute at angle	$\text{m}^3.\text{s}^{-1}$
r_1	impeller inner radius	m
r_2	impeller outer radius	m
r_3	volute basic circle radius	m
r_4	volute outlet profile radius	m

Re	Reynolds number	
r_e	impeller eye radius	m
r_s	impeller shaft radius	m
S	scalar factor to calculate particle diameter	
t	blade thickness	m
Δt	laser pulse duration	s
U_1	tangential speed at the impeller inlet	m.s ⁻¹
U_2	tangential speed at the impeller outlet	m.s ⁻¹
V_1	absolute speed at the impeller inlet	m.s ⁻¹
V_2	absolute speed at the impeller outlet	m.s ⁻¹
V_{m1}	radial component of V_1	m.s ⁻¹
V_{m2}	tangential component of V_2	m.s ⁻¹
$V_{\theta 1}$	tangential component of V_1	m.s ⁻¹
$V_{\theta 2}$	radial component of V_2	m.s ⁻¹
U_p	Pressure uncertainty	Pa
U_Q	Flow rate uncertainty	l.min ⁻¹
U_S	Uncertainty	
$\bar{V}$	velocity captured by PIV system	m.s ⁻¹
$V_{impeller}$	velocity after impeller blockage	m.s ⁻¹
V_{slip}	slip velocity	m.s ⁻¹
V_{tip}	tip velocity	m.s ⁻¹
W_1	relative velocity at the impeller inlet	m.s ⁻¹
W_2	relative velocity at the impeller outlet	m.s ⁻¹
W_{m1}	radial component of W_1	m.s ⁻¹

ΔX	particle displacement	m
Y_k	diffusion term of k equation	$\text{J.kg}^{-1}.\text{s}^{-1}$
Y_ω	diffusion term of ω equation	$\text{J.kg}^{-1}.\text{s}^{-1}$
Z	blade number	
β_1	the blade inlet angle	°
β_2	the blade outlet angle	°
β_{f1}	the flow inlet angle in blade channels	°
β_{f2}	the flow outlet angle in blade channels	°
δ	radial gap	m
ε	blockage factor	
θ	volute angle relative to the throat	°
λ	laser wavelength	nm
μ	fluid dynamic viscosity	Pa.s
μ_t	fluid turbulent viscosity	Pa.s
ν	fluid kinematic viscosity	$\text{m}^2.\text{s}^{-1}$
ρ	density	kg.m^{-3}
σ	solidity	
σ_s	slip factor	
τ	shear stress	Pa
ϕ	flow coefficient	
ϕ_i	impeller outlet flow coefficient	
ψ	head coefficient	
Ω_s	specific rotational speed	
ω	rotational speed	rad/s

ω_k rotational vector

ω_t turbulent kinetic energy dissipation rate J.kg^{-1}

List of Figures

Fig. 2-1 Three types of blade shape [29]	7
Fig. 2-2 Impeller geometry of centrifugal pump in the meridional direction [20]	8
Fig. 2-3 Impeller geometry of centrifugal pump in the axial perpendicular direction [30]	8
Fig. 2-4 Closed, semi-open and open impellers [30]	8
Fig. 2-5 Cordier diagram based on n_s and d_s [32]	10
Fig. 2-6 Top view of volute geometry [20]	11
Fig. 2-7 Side view of volute geometry [20]	12
Fig. 2-8 Cross section shapes of volute casings [30]	12
Fig. 2-9 Three types of tongue profiles [36]	13
Fig. 2-10 Comparison of the pressure distribution around the periphery of impeller for CAM and CMV volute [20]	18
Fig. 2-11 Impeller of the miniature pump designed by Lei [69]	20
Fig. 2-12 Pump developed by Antaki and Diao [70]	21
Fig. 2-13 Pump performance for fluid with different viscosity [125]	34
Fig. 2-14 Flow chart of CFD simulation process [137]	37
Fig. 2-15 Basic requirements for a PIV system [142]	40
Fig. 3-1 Selection of outlet flow coefficient and head coefficient of pumps [20]	43
Fig. 3-2 K_3 as function of the specific speed	47
Fig. 3-3 Empirical data curve of Anderson [24]	48

Fig. 3-4 Inlet and outlet velocity triangles of the originally designed impeller.....	51
Fig. 3-5 Profile of the originally designed impeller (actual size)	51
Fig. 3-6 Inlet and outlet velocity triangles of impeller model 5	55
Fig. 3-7 Profile of the impeller model 5(actual size)	55
Fig. 3-8 Volute profile of the Constant Mean Velocity method (actual size).....	56
Fig. 3-9 Volute profile of the Constant Angular Momentum method (actual size).....	56
Fig. 4-1 Simulation model of the pump	65
Fig. 4-2 Velocity profile in the inlet	66
Fig. 4-3 Velocity vector in the inlet	66
Fig. 4-4 Example of simulation residual.....	70
Fig. 4-5 Grid independence check	73
Fig. 5-1 Testing models of 20 mm diameter.....	74
Fig. 5-2 Testing model of 10 mm diameter	75
Fig. 5-3 Setup of the radial blade model.....	75
Fig. 5-4 PIV measurement system	76
Fig. 5-5 Control panel for PIV system.....	79
Fig. 5-6 Double-Q-switching mechanism [156]	80
Fig. 5-7 Image pairs captured by the CCD camera.....	81
Fig. 5-8 Flowsense 2M camera.....	82
Fig. 5-9 Elements and processes in a 2-D particle image velocimeter system [158] ...	82
Fig. 5-10 Prototype of the front and back view of the testing rig.....	86

Fig. 5-11 Actual testing rig	88
Fig. 6-1 Energy losses versus blade angle for 20 mm diameter impellers at 112.5 rad/s	94
Fig. 6-2 Energy losses versus blade angle for 10 mm diameter impellers at 450 rad/s	95
Fig. 6-3 Energy losses versus radius ratio for 20 mm diameter radial impeller at 112.5 rad/s.....	96
Fig. 6-4 Energy losses versus radius ratio for 20 mm diameter impeller with blade angle 56° at 112.5 rad/s.....	97
Fig. 6-5 Energy losses versus radius ratio for 10 mm diameter radial impeller at 450 rad/s.....	98
Fig. 6-6 Energy losses versus radius ratio for 10 mm diameter impeller with blade angle 56° at 450 rad/s.....	98
Fig. 6-7 Energy losses versus flow rate for 20 mm diameter radial impeller at 112.5 rad/s.....	99
Fig. 6-8 Energy losses versus flow rate for 20 mm diameter impeller with blade angle 56° at 112.5 rad/s	100
Fig. 6-9 Energy losses versus flow rate for 10 mm diameter radial impeller at 450 rad/s	100
Fig. 6-10 Energy losses versus flow rate for 10 mm diameter impeller with blade angle 56° at 450 rad/s	101
Fig. 6-11 Different turbulent models for 20 mm diameter models with radial blades	103
Fig. 6-12 Relative velocity distribution of radial impeller with Reynolds Stress Model	104

Fig. 6-13 Relative velocity distribution of radial impeller with laminar model	105
Fig. 6-14 Numerical results of pump performance of 20 mm diameter models at 112.5 rad/s (Laminar model).....	106
Fig. 6-15 Numerical results of pump performance of 20 mm diameter models at 225 rad/s (Reynolds Stress Model).....	107
Fig. 6-16 Numerical results of pump performance of 10 mm diameter models at 450 rad/s (laminar).....	108
Fig. 6-17 Numerical results of pump similarity of models with radial impeller	109
Fig. 6-18 Numerical results of pump similarity of models with blade angle 72 °.....	109
Fig. 6-19 Numerical results of pump similarity of models with blade angle 63 °.....	110
Fig. 6-20 Numerical results of pump similarity of models with blade angle 56 °.....	110
Fig. 6-21 Numerical results of pump similarity of models with blade angle 17°.....	111
Fig. 6-22 Experimental results of pump performance of 20 mm diameter models at 112.5 rad/s.....	112
Fig. 6-23 Experimental results of pump performance of 20 mm diameter models at 225 rad/s.....	112
Fig. 6-24 Experimental results of pump similarity of radial models 13 and 26	114
Fig. 6-25 Experimental results of pump similarity of impeller model 2 with blade angle 72°	114
Fig. 6-26 Experimental results of pump similarity of impeller model 12 with blade angle 63°	115

Fig. 6-27 Experimental results of pump similarity of impeller model 5 with blade angle 56°	115
Fig. 6-28 Experimental results of pump similarity of impeller model 10 with blade angle 17°	116
Fig. 6-29 Pump performance of 20 mm diameter radial model (model 13) at 112.5 rad/s.....	117
Fig. 6-30 Pump performance of 20 mm diameter radial (model 13) at 225 rad/s	118
Fig. 6-31 Pump performance of 10 mm diameter model with radial blades	118
Fig. 6-32 Effect of blade angle of 20 mm diameter models	120
Fig. 6-33 Effect of blade angle of 10 mm diameter models	121
Fig. 6-34 Effect of blade inner radius r_1 of 20 mm diameter models	122
Fig. 6-35 Effect of blade inner radius r_1 of 10 mm diameter models	123
Fig. 6-36 Effect of blade thickness of 20 mm diameter models	124
Fig. 6-37 Effect of blade thickness of 10 mm diameter models	124
Fig. 6-38 Effect of blade number of 20 mm diameter models.....	126
Fig. 6-39 Effect of blade number of 10 mm diameter models.....	126
Fig. 6-40 Pump performance of 20 mm diameter models at 112.5 rad/s with different Z	127
Fig. 6-41 Pump performance of 20 mm diameter models at 225 rad/s with different Z	128
Fig. 6-42 Effect of flow coefficient on pump performance of 20mm diameter models	129

Fig. 6-43 Pump performance of original impeller model 10 at 112.5 rad/s.....	130
Fig. 6-44 Pump performance of original impeller model 10 at 225 rad/s.....	131
Fig. 6-45 Pump performance of impeller model 5 at 112.5 rad/s	132
Fig. 6-46 Pump performance of impeller model 5 at 225 rad/s	132
Fig. 6-47 pump performance of CMV and CAM volute with different volute outlet width	133
Fig. 6-48 Pump performance of pumps with CMV and CAM volutes (outlet width 20.2 mm).....	134
Fig. 6-49 Static pressure distribution in the middle plane of model 10	136
Fig. 6-50 Relative velocity distribution in the middle plane of model 10	136
Fig. 6-51 Static pressure distribution in the middle plane of model 5	137
Fig. 6-52 Relative velocity distribution in the middle plane of model 5	138
Fig. 6-53 Velocity distribution near the volute tongue	138
Fig. 6-54 Sketch of the impeller channel zone	139
Fig. 6-55 Captured picture of impeller zone 1	140
Fig. 6-56 Relative velocity distribution of the experimental and numerical results in impeller channel 1 at 225 rad/s	140
Fig. 6-57 Relative velocity distribution of the experimental and numerical results in impeller channel 2 at 112.5 rad/s	141

List of Tables

Table 2-1 Comparison of different miniature pumps	21
Table 3-1 Guideline for volute height.....	45
Table 3-2 Guideline for volute base circle.....	46
Table 3-3 Pump requirements and selected empirical parameters.....	49
Table 3-4 Parameters of volute	49
Table 3-5 Velocity triangles of impeller inlet and outlet of the original model	50
Table 3-6 Modified impeller models with different flow coefficient	53
Table 3-7 Modified impeller models with different geometrical parameters	53
Table 3-8 10 mm diameter models	53
Table 3-9 Velocity triangles of impeller inlet and outlet of model 5.....	54
Table 3-10 Volute with different mean velocity C_m	57
Table 3-11 Volute with different mean momentum K	57
Table 5-1 Seeding materials for liquid flows [155]	77
Table 5-2 Seeding materials for gas flows [155]	77
Table 5-3 Laser specifications	78
Table 6-1 Geometrical parameters and conditions of models	89
Table 6-2 Design requirements of models	91
Table 6-3 Energy equations	93
Table 6-4 Flow conditions and geometrical parameters	120

1. Introduction

1.1 Background

Advances in microelectronics and large scale integration technologies have resulted in high powered and compact devices that generate very high heat fluxes [1]. For current electronic chips, the heat flux generated ranges from tens to hundreds W/cm^2 , and for semiconductor lasers, this value is almost $2000 \text{ W}/\text{cm}^2$ [2]. Furthermore, temperatures at the chip surface must be kept at relatively low levels ($\sim 100 \text{ C}$) for safety purposes. Therefore, reliability of these devices is critically dependent on the heat dissipation and cooling system.

The main heat transfer mechanism in cooling system is convection, and convective air cooling is still the most common method for removing heat from the heat generating elements. This is because air is readily available and air moving devices are relatively simple [3]. However, forced air cooling is currently approaching the maximum limit of heat removal rates [4]. The limits of air cooling capabilities force us to find alternatives, which are more effective.

One of the most effective ways to remove heat is through convective liquid cooling. Water is a suitable working fluid because of its high specific heat and low cost. For cooling effect, a miniature centrifugal pump serves as an attractive alternative if its size is sufficiently small and the flow rate is adequate.

Besides cooling devices, miniature pumps can be found in the pharmaceutical and medical industries [5, 6], liquid dispensing systems [7, 8], fuel delivery systems for miniature fuel cells [9, 10]. In pharmaceutical applications, miniature infusion pumps have been developed to inject insulin into patients [11], but they need to be improved further to achieve better performance. Miniature pumps are also in great demand as

cardiac assist device [12] because existing cardiac pumps are still relatively big in size. The diameter of existing cardiac pumps is in the range of 24 mm to 40 mm.

Many researchers have put in a lot of effort in developing miniature pumps. Currently there are many types of miniature pumps, including piezoelectrical pump [13, 14], electrostatic pump [15, 16], ultrasonic pump [17] and diaphragm pump [18]. However, a complete understanding of the design of dynamic miniature pumps is still lacking though the principle of conventional macro dynamic pump is well established.

The design of miniature size rotary pump is different from macro-size pump, because the viscous force is significant at small scales, and this results in a larger impedance [19]. The relatively larger viscous force will reduce the velocity, affect pressure distribution significantly, and hence reduce pumping efficiency. In addition, the operating Reynolds number is much lower and the flow pattern will be different. Hence, the design of miniature pumps is more complex than that of macro-size pumps.

Even though the design method for traditional macro pump is well established for many years, the theory on the design of miniature pumps has not been well established. More efforts are needed in the design procedure. This serves as the motivation of this project where the aim is to investigate the effects of the blade geometry on the performance of a miniature centrifugal pump.

1.2 Objectives

The demand of miniature size pump motivates us to investigate the effects of geometrical parameters on pump performance. In the preliminary design stage, the conventional design method will be employed to establish the preliminary design. Modifications will be made by varying the geometrical parameters to study the effects of blade angles, inner radius, blade thickness, blade number and volute outlet size.

Simple theoretical models will be developed. These will be compared with numerical studies and experimental measurements. The pump pressure generated and flow field are the two main considerations to assess the pump performance.

The objectives of this project are as follows:

1. To investigate the effects of blade geometry on a miniature centrifugal pump.
2. Geometrical parameters are modified to obtain different pump models and different flow coefficients will also be investigated.
3. To conduct theoretical and numerical studies of the various pump models.
4. To conduct experimental studies on selected pumps.
5. To compare theoretical, numerical and experimental results.
6. To improve the initiated design based on theoretical, numerical and experimental studies.

1.3 Scope of Work

The scope of this project includes a thorough literature review of existing pumps, pump design methods, research on miniature pumps, energy loss models, effects of geometrical parameters on pump performance, Computational Fluid Dynamics and Particle Image Velocimetry methods used in the centrifugal pump. These are presented in chapter 2.

Chapter 3 presents the design of miniature pumps based on traditional design procedure of the macro pump. Effects of geometrical parameters will be examined through modifications made on both impeller and volute to study their influence on the pump performance.

Chapter 4 describes the basic theory of numerical modeling and CFD simulation. The mesh generation, grid independence, governing equations, boundary conditions, turbulent models and convergence are also presented in this chapter.

Chapter 5 describes the pump models, experimental setup, pump performance measurement method, and Particle Image Velocimetry technology.

Chapter 6 presents the theoretical, numerical and experimental results. The pump performance for different blade angle, inner radius, blade number, blade thickness and volute outlet size are investigated. In order to optimize the design procedure of miniature centrifugal pumps, effect of flow coefficient is also investigated. Comparisons between the traditional designed impeller and the modified impellers, and between the Constant Mean Velocity volute and Constant Angular Momentum volute are performed.

Finally, Chapter 7 presents the conclusions and future works that will be conducted.

2. Literature Review

This chapter will give an overview of the history and the evolution of centrifugal pumps, current pump design methods and research on miniature pumps. Theoretical analysis of losses in pumps, the study of the effect of geometrical parameters on pump performance, Computational Fluid Dynamics and Particle Image Velocimetry methods will also be presented. Finally, challenges that many researchers have encountered during the miniaturization process will also be reviewed.

2.1 General Introduction of Pumps

Pumps used for water delivering systems have existed for a long time, such as Persian wheel, waterwheels since 200 B.C. [20], as well as the screw rod invented by Archimedes [21]. For most ancient water pumps, water was delivered by a displacement container, and emptied at the destination periodically.

The first modern rotary pump was developed in the 16th century [22]. As early as in 1588, four bladed vane pumps were recorded and other rotary pumps appeared subsequently. In 1689, Papin [23] invented a four-bladed impeller centrifugal pump and the first semi-open double-suction centrifugal pump with radial blades was introduced in 1818 [24]. From 1851 to 1875, the development of multi-stage centrifugal pumps with guide vanes has made high lift pumps possible. Since then, rotary pumps are becoming more popular.

Over the centuries, pumps have extended their sizes, types and applications, from macro-size hydraulic pumps to micro-scale diaphragm pumps [21, 25], from displacement pumps to centrifugal pumps, from pumping water to pumping other fluids such as air, gas, refrigerant, oil, petrol, bio-fluid, drugs, medicine, blood and some other fluids.

Pumps can be categorized based on how energy is added to the fluid, dynamic pumps and displacement pumps respectively. For dynamic pumps, energy is continuously added to the fluid with the assistance of a rotating vane or impeller. Dynamic pumps can be further classified into radial flow, axial flow and mixed flow pumps according to the outlet flow direction relative to the inlet flow direction. Radial flow pumps are used for higher pressure and lower flow rate, while axial flow pumps are primarily used for delivering a higher flow rate at a lower pressure. For displacement pumps, flow is created by trapping a fixed amount of fluid and then forcing the trapped volume of fluid to move with each revolution of the pumping element, which could be gears, lobe, rotary pistons, vanes and screw. Displacement pumps are basically classified into reciprocating and rotary types, which is based on the movement of the pressure-producing element.

In addition to the above macro-size pumps, there exists a new type of pumps, namely, micropumps, which are designed based on different principles. Since the development of the first micro pump in 1980s [26], its applications have become more extensive, such as in the areas of microelectronics cooling, space exploration, chemical and biological analysis. Currently, the most common micropumps are the dynamic micropumps and displacement micropumps respectively [27]. Recently, novel pumps based on electro wetting, electrochemical and phase transfer technologies have attracted increasing attention [28].

2.2 Centrifugal Pump Geometry

Centrifugal pumps are widely used in many areas, including fluid supply, heat removal, air conditioning, refrigeration, chemical movement, sewage movement, flood control, drug delivery and marine services.

The typical centrifugal pump has an inlet tube, a rotating impeller, and a volute. Fluid is guided by the inlet tube, accelerated by the impeller, and then discharged through the volute. In this process, mechanical energy is transferred into kinetic energy via the rotating impeller, which is then converted into pressure energy in the volute.

2.2.1 Impeller Geometry

There are three types of blade orientation, namely, backward-curved, radial and forward-curved [29] as shown in Fig. 2-1. The backward-curved impeller generates the lowest pressure head compared to that of the other two types, but is commonly used as its characteristic is most stable. Radial blades are relatively easy to be manufactured and introduce no complex bending stresses.

Fig. 2-1 Three types of blade shape [29]

The impeller geometrical features and parameters in the meridional direction and axial perpendicular direction are shown in Fig. 2-2 and Fig. 2-3. The impeller eye radius r_e , impeller outer radius r_2 , blade height b_2 , blade thickness t and the hub-shroud profile are the main geometrical parameters of the impeller. In general, r_e is approximately the same as the impeller inner radius r_1 . The hub and shroud profiles are usually defined based on the designer's experience, with consideration of the flow field in the blade passage. Based on the existence of hub and shroud, impellers can be classified into open impeller, semi-open impeller and closed impeller as shown in Fig. 2-4.

Fig. 2-2 Impeller geometry of centrifugal pump in the meridional direction [20]

Fig. 2-3 Impeller geometry of centrifugal pump in the axial perpendicular direction [30]

Fig. 2-4 Closed, semi-open and open impellers [30]

2.2.2 Critical Dimensionless Parameters

A large number of variables are usually involved in the design procedure, and dimensionless variables are often used to reduce the number of variables. Based on the dimensional variables, such as density ρ , viscosity μ , rotational speed ω and impeller diameter D , various dimensionless variables can be formed, including Reynolds

number Re , specific speed n_s , specific diameter d_s , head coefficient ψ and flow coefficient ϕ [31]. In addition, dimensionless variables can be used to predict the pump performance by conducting tests on a scale model running at the same operating conditions [31].

The Reynolds number is the ratio of the inertia forces to the viscous forces, and is defined as equation (2.1) in turbo-machines. ω is the impeller rotational speed and D_2 is the impeller outer diameter. The transition Reynolds number between laminar flow and turbulent flow varies between 2000 and 10000 [31] for turbo-machines.

$$Re = \frac{\rho \omega D_2^2}{\mu} \quad (2.1)$$

The specific speed n_s and the specific diameter d_s are critical in the pump design procedure. The specific speed n_s , is defined as the speed of an ideal pump geometrically similar to the actual pump, which when running at this speed will raise a unit of volume of fluid, in a unit of time through a unit of head, as shown in equation (2.3). The specific diameter d_s , indicates the relative size of a machine for a given set of operating conditions as shown in equation (2.2). The specific speed n_s , determines the impeller type and geometry. Low specific speed will lead to a radial and narrow long impeller, while high specific speed refers to axial and short impeller. The specific speed diameter, d_s , determines the pump size operating at the highest efficiency. Based on experimental data, the relationship between the specific speed and the specific diameter at the best performance can be summarized by the Cordier diagram [32] as shown in Fig. 2-5. Besides experimental data, theoretical studies were also conducted to support the Cordier diagram [33]. For centrifugal pumps, the specific speed n_s should be between 0.2 to 1.

$$d_s = \frac{D(g\Delta H)^{1/4}}{\sqrt{Q}} \quad (2.2)$$

$$n_s = \frac{N\sqrt{Q}}{(g\Delta H)^{3/4}} \quad (2.3)$$

Fig. 2-5 Cordier diagram based on n_s and d_s [32]

The head coefficient ψ and flow coefficient ϕ are two key factors in specifying the pump geometry and performance at off-design operating conditions. Generally, the flow coefficient refers to the volumetric flow rate passing through the impeller, which implies the relationship between flow rate and geometrical shape. The inlet flow coefficient can be expressed as the ratio of the radial velocity to the tip speed as shown in equation (2.4). The head coefficient represents the pressure generated for a pump running at given speed, and is defined as given by equation (2.5).

$$\phi = \frac{Q}{ND^3} \quad (2.4)$$

$$\psi = \frac{g\Delta H}{N^2 r_2^2} \quad (2.5)$$

Macro-size pumps designed using the above equations have shown to operate with high efficiency. However, in the case of miniature size pumps, where the Reynolds number is relatively low, the traditional approach may not be appropriate and further studies are necessary.

2.2.3 Volute Profile

The volute is used to collect fluid leaving the impeller and transforms the kinetic energy into pressure energy [34]. The volute profile, as shown in Fig. 2-6 and Fig. 2-7, is defined by distributing its cross-sectional area A on a “base circle” of radius r_3 . The volute is a spiral-shaped passage surrounding the impeller exit, and the volute cross section increases gradually along the impeller periphery. The volute profile starts from the volute tongue and ends at the volute throat.

Fig. 2-6 Top view of volute geometry [20]

Fig. 2-7 Side view of volute geometry [20]

There are basically several types of the volute cross-sectional shape as shown in Fig. 2-8. The shapes must be selected so as to suit the pump type, taking into account casing stresses and deformations [30]. In the volute design process, a rectangular cross-section shape is adopted to ensure that the volute outlet flow velocity is not in the axial direction. In addition, the rectangular type is the simplest shape from a manufacturing point of view [35].

Fig. 2-8 Cross section shapes of volute casings [30]

Tongue configuration, including tongue profile and tongue angle is one of the most important factors influencing the performance of the volute. The flow pattern around

several tongue configurations was measured using particle-tracking velocimeter by Ayder et al. [36]. Three volute tongue designs are illustrated in Fig. 2-9. Volute (a) has a full tongue and did not allow any flow recirculation, volute (b) has a reduced tongue length thereby allowing recirculation, while volute (c) has a rounded tongue. It is concluded that by cutting back the volute tongue, the volute performance was improved, and the performance was further improved by providing a rounded tongue, like volute C.

Fig. 2-9 Three types of tongue profiles [36]

Hira [35] found that the maximum efficiency point shifts towards to a lower flow rate as the radial clearance was increased, either by reducing the tongue length or by decreasing the tongue angle. With a shorter tongue and a smaller tongue angle, the pump will have broader efficiency curves without decreasing the pump efficiency. Wong et al. [37] conducted numerical investigations on the effect of radial gap and volute tongue position. Results showed that a radial clearance of 10% and a tongue position at 30 degrees will generate the best performance. Boehning [38] experimentally investigated the hydraulic radial forces for three types of volute, separately design-singular, circular, and double volute. When the pump is running at a design point of 5 l/min and 2500 rpm, the single volute experienced the lowest radial force while the circular volute yielded the highest.

2.2.4 Critical Parameters in the Design of Volute

The cross-section area A_T is zero at the tongue and increases gradually with angle. It is a significant factor as it affects energy losses, such as friction loss and mixing loss in the volute. The volute cross-section area at a given polar position can be calculated from the flow rate and the volute design method. In the constant velocity method, the cross-section area is given by equation (2.6) while for the constant momentum method, the outer radius of the volute is governed by equation (2.7), and A_θ can be obtained based on the outer radius and volute height.

$$A_\theta = \frac{Q_\theta}{C_m} \quad (2.6)$$

$$C_u r_4 = K \quad (2.7)$$

The base circle radius r_3 is usually determined empirically based on the impeller outer radius r_2 . A minimum gap must be maintained between the impeller and volute tongue to prevent noise and vibration. Karassik [20] has proposed a gap of 6% of r_2 , while Lobanoff [39] suggested a 7% gap for macro-size pumps.

2.3 Centrifugal Pump Design

A centrifugal pump is mainly made up of an impeller and volute. The impeller influences the flow and pump characteristics of the pump at the design point, while the volute geometry is effective at off-design conditions. In order to improve the centrifugal pump performance, both impeller and volute geometry should be optimized. The design method of the macro-size impeller and volute will be summarized below.

2.3.1 Impeller Design Methods

For the design of impeller, there are two approaches, namely the direct (or traditional) approach and the inverse design approach.

The direct approach is based on trial and error changes to the blade shape. An initial blade shape is given based on empirical methods, and then modified subsequently [40, 41]. Besides experimental and numerical validation, the quasi-three-dimensional method offers a robust alternative, which is available in many software [42]. The initial blade shape is usually generated from pump traditional design approach, which is already well established [21, 29, 32, 34, 43]. Taking into consideration of flow separation, pump efficiency and the relationship among other parameters, Karassik [20] summarized and expanded the design procedure. Critical geometrical parameters listed in the above method include eye radius, inner radius, exit radius, exit height, hub exit radius, hub profile and blade profile.

The inverse design approach uses the flow information obtained from numerical simulations to compute the blade geometry directly based on aerodynamic principles with specified requirements [44]. 2-D inverse design approach [45, 46] is developed based on the pressure or velocity distribution on the pressure and suction surface. 3-D inverse design approach [47, 48] is based on either the blade loading or the circulation distribution together with thickness distribution. The velocity field, which includes circumferential averaged velocity and periodic velocity, is calculated from the vorticity, and the blade shape is determined when the inviscid slip condition is applied numerically. After the blade shapes are computed using an inviscid slip (or flow tangency) condition, viscous effects are introduced indirectly by using a viscous/inviscid procedure. This allows the generation of valuable design to generate

optimum designs very quickly even if the head, flow rates or rotational speed change for a given design.

Zangeneh et al. [49, 50] proposed a practical three-dimensional inverse design approach which can be effectively used to develop actual turbo machines. When the circulation distribution (rV_θ) and meridional shape (which are the design parameters that relate directly to performance) are given, an impeller shape with the desired performance may be obtained. Using the Navier-Stokes (N-S) Equation, a detailed flow pattern can be obtained, showing the effect of boundary layers and secondary flow on the pump performance. This method has been verified with several 3-D design examples [51]. The inverse design approach can also be easily used in the design of similar components, like compressor, aerofoil, diffuser [52], volute and other turbo-machine.

Direct approach and inverse design approach were compared by Zangeneh [53]. For three-dimensional inverse design techniques, although it can represent the flow accurately, it is more lengthy and complicated and requires detailed knowledge of the pressure distribution.

Besides the two approaches listed above, a new design method based on Medial Axis Transform (MAT) theory has been proposed [54]. Investigations were carried out on two new variables versus specific speed and results show that the meridional profile at any specific speed can be directly drawn through calculations if the specific speed ranges from 0.1 to 1.

2.3.2 Volute Design Methods

There are two basic approaches for volute profile design, namely the Constant Mean Velocity method and the Constant Angular Momentum method. These two methods

differ from different specification of the flow pattern in the volute.

The Constant Mean Velocity (CMV) approach for designing volute profile was first proposed by Stepanoff [43]. It assumes that a constant average velocity in the volute casing is most favorable for pump performance, especially for obtaining a uniform pressure distribution around the impeller. The volute cross sections will increase linearly from the volute tongue to the volute throat. The mean velocity is the velocity obtained by dividing the flow rate by the total area normal to the flow direction and is selected empirically. It is important to select a “correct” mean velocity value in the design process. For miniature pumps, the operating condition is different from traditional macro-size pumps and the Reynolds number is much lower. Hence, the present guideline of choosing a mean velocity value for macro-size pumps may not be applicable to centrifugal miniature pumps.

The Constant Angular Momentum (CAM) approach proposed by Anderson [55] is based on the principle of the conservation of angular momentum [20]. The circumferential component of the absolute velocity in the volute will decrease with increasing radial distance. This method yields a rather good performance based on experimental testing. The performance of CAM volute is also dependent on the selection of appropriate empirical constants.

Experimental measurements of the head generated by both methods are compared and shown in Fig. 2-10. Besides experimental investigations, many theoretical attempts have been conducted to obtain a better understanding of the flow characteristics in the volute in order to improve the volute design. However, the theoretical method that can reliably predict the complicated 3-D flow conditions in a centrifugal pump volute is still unavailable. This is mainly because many complex phenomenon, such as separation, turbulence and secondary flows occur, and these phenomena are amplified

at off-design conditions. Currently, theoretical analysis is only possible where the flow in the volute is assumed to be inviscid and irrotational.

(a) Constant Angular Momentum method; (b) Constant Mean Velocity method

Fig. 2-10 Comparison of the pressure distribution around the periphery of impeller for CAM and CMV volute [20]

Although the Constant Mean Velocity and Constant Angular Momentum methods are widely used for volute design, the dependency on empirical data limits their applications. For miniature pumps, the geometry and working conditions are quite different from commercial pumps and the operating Reynolds number of miniature pumps is much smaller. The flow field in the miniature pump is affected by the impeller and volute wall to a large extent. Thus, the commonly used design coefficients for commercial pumps may not be suitable for miniature pumps and there is a need to investigate the application of these two methods.

2.4 Research on Miniature Pumps

2.4.1 Design and Performance

Recently, miniature fluidic pumps have been in great demand in the fields of pharmaceutical devices [5], cardiac assist devices [6], liquid-cooling-devices [56],

liquid dispensing systems [7, 8] and fuel delivery systems for miniature fuel cells [9, 10]. In pharmaceutical applications, miniature infusion pumps have been developed to inject insulin into patients, but they need to be improved further to achieve better performances. Miniature pumps are also in great demand as cardiac assist device because existing cardiac pumps are still relatively big. The diameter of existing cardiac pumps is in the range of 24 mm to 40 mm. Water-cooling device, like miniature pumps, of central processing unit for computers are in great demand as the size of the chip decreases while the power density increases.

The application of miniature pumps is summarized by Rossetti [57]. Non-rotational miniature size pumps include piezoelectric pump [13, 58], electrostatic pump [15, 59], electro-osmotic pump [60], ultrasonic pump [17], membrane pump [61, 62], peristaltic pumps [63, 64], and reciprocation ball pump [65]. These devices are limited by the low flow rate and pressure generated capability [66]. Rotary pumps have reasonable performance compared with other pumps (displacement pump, membrane pump, ultrasonic pump piezoelectric pump, electrostatic pump and peristaltic pump), with higher flow rate, better performances, wider range of applications and operating conditions.

Due to extensive application of miniature pumps, the research on miniature pumps has received greater attention, not only in the development of new type of pumps, but also in the improvement of existing pumps. However, the design of miniature size rotary pump is different from macro-size pump, because the viscous force is significant at small scales, and results in a larger impedance [19]. The relatively larger viscous force will reduce the velocity, affect pressure distribution significantly, and hence reduces pumping efficiency. In addition, the operating Reynolds number is much lower and the

flow pattern will be different. Hence, the design of miniature pumps is more complex than that of macro-size pumps.

Al-Halhouli et al. [67] developed and tested a series of miniature spiral-channel viscous pumps with disk diameter of 12 mm to pump viscous fluid. The height of the channels ranges from 1 mm to 2 mm. The maximum flow rate achieved was 3.05 ml/min and the maximum pressure generated was 35,300 Pa at a rotational speed of 4138 rpm.

Abe et al. [68] developed a miniature undulation pump for artificial heart and investigated the performance using water. The diameter of the pump is 38 mm and the thickness of the pump is 11 mm. It can generate a pressure of 13,330 Pa with a flow rate of 5 l/min when the impeller is rotating at 3500 rpm.

Lei et al. [69] designed a centrifugal micro-pump for biomedical and chemical application, with the impeller geometry shown in Fig. 2-11. The diameter of the impeller is 4 mm, the length of each blade is 1 mm and the height of each blade is 0.15 mm. The pump is driven by a DC motor, whose voltage is linear to the impeller rotational speed. When the applied voltage is increased to 3 V, the maximum flow rate achieved is 2.45 ml/min, and when the voltage is 2 V, the maximum pressure generated is 7105 Pa.

Fig. 2-11 Impeller of the miniature pump designed by Lei [69]

Antaki et al. [70] developed a centrifugal miniature blood pump with a 46 mm impeller diameter, as shown in Fig. 2-12. The design operating condition is 2.7 l/min at a pressure generated of 13,330 Pa and at a speed of 3000 rpm. This blood pump can generate a maximum of 33,200 Pa and can handle a maximum flow rate of 3 l/min.

Fig. 2-12 Pump developed by Antaki and Diao [70]

Horiguchi et al. [71] designed a two-stage centrifugal blood pump with diameter 40 mm, blade height 3-5 mm, designed flow rate 3 l/min and head 500 mmHg (66661 Pa) when running at a speed of 4000 rpm. The designed model is investigated numerically and experimentally. The tip clearance is 1 mm, which is larger than traditional blood pump, can reduce the shear stress, and thrombus formation.

Table 2-1 Comparison of different miniature pumps

	Al-Halhouli et al. [67]	Abe et al. [68]	Lei et al. [69]	Antaki et al. [70]	Horiguchi et al. [71]
Diameter(mm)	12	38	4	46	40
Height(mm)	1 and 2	11	0.2	Not available	3-5
Maximum Pressure(Pa)	35300	13330	7105	33200	66661
Maximum Flowrate(l/min)	3.05×10^{-3}	5	2.45×10^{-3}	3	3
Rotational Speed(rpm)	4138	3500	Not available	3000	4000

The specifications of the pumps discussed above are listed in Table 2-1. The design of miniature pump still faces two key challenges: minimization of the size and improvement of the pump performance. It is difficult to reduce the pump size and yet achieve a good performance. Hence, further research on the design of miniature pumps is needed.

2.4.2 Pump Miniaturization Challenges

The problems encountered during the development of miniature pumps have impeded its progress. First of all, the main problem is the method of fabrication or assembly and the stringent accuracy required in fabricating the devices and the associated mechanisms [68]. While these devices are getting smaller, traditional machining method can no longer be employed. For these reasons, micromachining is developed and applied, including bulk micromachining, surface micromachining, LIGA technology [28] and 3-D printing. However, 3-D printing is not applicable for miniature pumps because of its coarse surface and low stress. 3-D printing is achieved using an additive process, where successive layers of material are laid down in different shapes. It is suitable for static model, where strength and rigidity are not crucial factors. For the miniature pump, the rotational speed is very high, and 3-D printing models will be subjected to very high stresses. Moreover, the surface roughness of 3-D printing models is typically in the order of a fraction of a millimeter, which is not suitable for miniature pumps.

Secondly, viscous force is significant at small scales, and this results in a large fluidic impedance [19]. This effect is undesirable, because viscous force will reduce the pumping efficiency significantly. In addition, when pumping some bio-fluid such as blood, red blood cells may be damaged when they are subjected to high shear stress

near the impeller and volute wall. All these problems have made the design of miniature pumps more complex than that of macro-size pumps.

2.5 Pump Scaling Effect

When a set of characteristic curves for a given pump stage is known, that machine can be used as a base model to design different pump sizes operating at similar conditions, like Reynolds number, Nusselt number and Prandtl number. This phenomenon is called pump affinity law, which means that the pump can be scaled up and down with the same dimensionless variables. The smallest size from published papers is 9 inch (23 cm) following affinity law. Scaling a given geometry to a different size means multiplying all dimensions of the model by the same scale factor, including all clearances and surface roughness elements, so that we have geometrically similar models instead of distorted models. The performance of the model is then scaled correspondingly by using dimensional analysis.

The pump affinity laws are accurate to predict pump performance with size and speed for large scale pumps, but are inadequate for miniature or micro scale pumps, because the hydraulic efficiency is greatly reduced [72].

In pump scaling research, the main approach to investigate the relationship between geometrical scaling and pump performance is through experimental studies. Based on analysis of Reynolds number similarity and experiments conducted, Lorenz et al. [73] found that the Reynolds number affects the deviation in similarity. A 45.4% error is observed between experiment and similarity analysis. A correction factor for Reynolds number was introduced to reduce the error into an acceptable range (7.5%). Kearney et al. [74] conducted experiments on three pumps and found that the pump scaling laws predict accurately the pump performance in downsizing a pump. However, big

deviation is observed on those pumps with all size scaling down except height in the axial direction. Wong et al. [75] found that scaling effect cannot be ignored for centrifugal blood pumps and the relationship between non-dimensional head and Re can be expressed as equation (2.8). Day et al. [72] found that at lower Reynolds number, flows did not adhere to traditional pump affinity laws, whereas higher Reynolds number flows scale according to pump affinity law as macro-size pumps operate at higher Reynolds number which is in the fully turbulent regime. Kearney et al. [76] investigated the performance of two miniature-scale centrifugal pumps, with diameter 34.3 mm and 17.15 mm. PIV results showed that geometric similarity differs as the scale is reduced. Gulich [77] provided a procedure to predict the roughness and Reynolds number effects on overall efficiency and hydraulic efficiency, including laminar flow and turbulent flow. The prediction results showed that surface roughness influence the hydraulic efficiency especially in small pumps. In addition, the surface roughness influences those pumps with low hydraulic efficiency. Obviously, the theoretical analysis is still in the developing stage.

$$\psi = \phi(Re_0/Re)^{0.25} (D_0/D)^{0.2} \quad (2.8)$$

Besides pump, the study of scaling is also applicable to fans. Classical fan scaling laws derived by Tanaka et al. [78] and Casey [79] indicated that two fan scaling parameters, namely, the fan size ratio and the fan speed ratio are independent. When classical fan scaling laws are used, $Q/ND^3=\text{const}$ should be maintained in fan performance prediction. However, from the Navier-Stokes equations, generalized fan scaling laws indicated that only one of the two fan scaling parameters is independent and change of one will influence another [80]. It is concluded that the Strouhal and the Reynolds numbers similarity must be maintained, implying that $ND^2/\nu=\text{const}$. By conducting experiments on a prototype fan, a 1/3 scaled and a 1/20 scaled fans, Grimes et al. [81]

concluded that a reduction in scale is accompanied by a reduction in efficiency. He also predicted that modifications must be made to the geometry when micro scale fans are compared with traditional geometric scaling of fan designs. Walsh et al. [82] have studied fans with diameters ranging from 15 to 30 mm and with numerous profile heights between 0.3 mm and 15 mm. The results showed that traditional scaling laws cannot be accurately applied when the blade profile alone is scaled.

2.6 Energy Models in Centrifugal Pumps

Energy is transferred from the shaft to the liquid through the impeller, and losses occur at the inlet, along the blade passage and in the volute. The ideal energy transferred to the liquid can be predicted by the Euler equation. The actual head is much lower than the Euler head because of energy losses.

2.6.1 Euler Head

The expression of the energy transfer between the impeller and fluid is based on one-dimensional energy transfer analysis, which is the Euler's Equation [83]. The Euler head is the ideal theoretical head developed by the impeller, and is defined as the rate of energy transfer for per unit mass of fluid flow. The Euler head is applicable for all kinds of flows, including compressible and incompressible, idealized and viscous flows. It is defined as the rate of energy transfer per unit mass of fluid flowing in meter as shown in equation (2.9).

$$E_{th} = \frac{T\omega}{gQ} = \frac{1}{g}(V_{u2}U_2 - V_{u1}U_1) \quad (2.9)$$

For centrifugal pumps, flow enters the impeller channels only in the radial direction, so the inlet circumferential velocity V_{u1} is zero. Hence, we have:

$$E_{th} = \frac{1}{4g} D_2^2 \omega^2 - \frac{\omega Q}{2\pi g b_2} \cot \beta \quad (2.10)$$

The determination of the hydraulic losses have been investigated by Eck [84], Oh [85] and Ntoko [86]. Optimization studies was conducted and the relationship between the maximum efficiency and specific speed, and other geometrical variables was obtained by Oh [85].

2.6.2 Slip Loss

Slip loss occurs in a centrifugal impeller due to incomplete guidance of fluid in the impeller passage [20], and it is more pronounced when the impeller has less number of blades. It is an inviscid phenomenon and is independent of the mechanical efficiency [87]. Generally, slip is mainly controlled by the blade geometry. However, other flow mechanisms will also play an important role on the slip phenomenon, such as the pressure gradient and boundary layer thickness near the blade walls, and the wake region [88]. However, these flow mechanisms are also induced by the blade geometry as well as impeller rotating condition. Studies have shown that the slip factor is a function of geometrical parameters, such as radius ratio r_1/r_2 , blade angle β and blade number Z .

Currently, there are two methods used to develop theoretical expression for the slip factor. The first method is based on eddy circle theory, which was first introduced by Stodola [89]. The slip factor was derived by assuming an eddy circle in the impeller passage. This method was subsequently modified and improved by other researchers, like Eck [84], Paeng and Chung [90] and von Backstrom [88]. The second method relies on two-dimensional potential flow theory. Busemann [91] obtained analytical expression for 2-dimensional impeller flow field and obtained values of slip factors

numerically. His result was subsequently improved by Wislicenus [92], Wiesner [93]. Visser et al. [94].

Besides geometrical factors, slip phenomenon is also influenced by the flow condition. Ohta's [95] experimental studies showed that operation at a higher Reynolds number will yield a larger slip. Memardefzouli's experiments [96] indicated that the slip factor is very sensitive to the flow coefficient. Boyce [31] summarized the effect of flow coefficient on slip factor. From Stahler's equation, the lower the flow coefficient, the higher the slip factor while for other equations the slip factor is constant with flow coefficient.

For a miniature impeller, the inner radius, blade angle and blade number have a significant effect on the flow field. To calculate the slip loss, Von Backstrom's [88] equation is selected, because all these geometrical parameters are considered. The slip factor is defined as:

$$\sigma_s = 1 - \frac{V_{\text{slip}}}{U_2} = 1 - \frac{1}{1 + 5 \frac{(1 - r_1/r_2)Z}{2\pi(\sin \beta)^{0.5}}} \quad (2.11)$$

The slip loss E_{slip} is:

$$E_{\text{slip}} = \frac{U_2^2}{g \left(1 + 5 \frac{(1 - r_1/r_2)Z}{2\pi(\sin \beta)^{0.5}} \right)} \quad (2.12)$$

2.6.3 Leakage Loss

There are two kinds of leakage loss. The first leakage flow occurs between the rotating disk and the stationary volute due to the pressure difference in the disk clearance between rotating impeller and stationary volute, causing a reverse flow. The second

one is the tip leakage loss, which is caused by the fluid flows from the pressure side to the suction side through the clearance between the volute and the blades. In centrifugal pumps, leakage flow will reduce the actual flow rate and hence the pump efficiency.

Leakage flow is influenced by many factors, such as blade geometry, rotating speed, and gap size. The research on leakage flow is mainly based on three methods, experimental investigation, CFD simulation and theoretical analysis.

To investigate the effect of the gap size and rotating speed on leakage flow, Anderson et al. [97] conducted numerical and experimental research using a magnetically suspended centrifugal pump, while some other researchers [98-100] conducted Particle Image Velocity (PIV) measurement. Chan's [101] results showed that leakage flow rate depends on a non-dimensional parameter, which is the ratio of the centrifugal forces to pressure generated by the pump theoretically. The washout hole is critical to balance the shaft force and reduce flow stagnation. Nishida et al. [102] and Wakisaka et al. [103] investigated the effects of washout holes to promote flow in the gap experimentally using Flow Visualization and Laser Doppler Velocimetry separately, while Triep et al. [104] established an optimal gap width and washout flow exist for different working conditions using CFD and PIV methods.

Will et al. [105] conducted CFD simulation on a complete pump and experimental testing on the pressure distribution. The energy loss due to leakage flow is investigated to conclude the effect of leakage on pump efficiency.

The theoretical method is mainly based on the flow in the gap between a stationary disk and a rotating disk. Exact cylindrically-symmetric solutions with closed plane streamlines were developed by Polyanin [106]. Chan [107] and Teo [108] also developed a symmetry solution based on Simplified N-S equations in cylindrical

coordinates. Turkyilmazoglu [109, 110] developed a new exact solution for 3-D Navier-Stokes equations for incompressible flow, which is different from Von Karman's axisymmetric similarity solution. Summarization of the above papers shows that non-dimensional analysis and the assumption of boundary conditions are critical to the solution.

Theoretical analysis of leakage flow conducted by Chan [107] and Teo [108] is used in this thesis, because it is derived based on Navier-Stokes equations. For gaps of constant height, the leakage flow rate is:

$$Q_{lk} = \frac{\pi \delta^3}{6\mu \ln(r_2 / r_1)} \left(\Delta P - \frac{3\rho\omega^2 (r_2^2 - r_1^2)}{20} \right) \quad (2.13)$$

where ΔP is the pressure difference between the trailing and leading edges. In the determination of the leakage loss, ΔP is assumed to be the pump head.

For semi-open impellers, fluid flows from the pressure side to the suction side through the clearance between the volute and the blades, causing tip leakage loss. The pressure loss due to the tip clearance is mainly induced by the leakage flow through the tip [111]. The tip velocity is:

$$V_{slip} = \sqrt{\frac{4Q\omega \sin \beta}{b_2 Z}} \quad (2.14)$$

Hence, the leakage flow rate is:

$$Q_{tip} = Z \delta (r_2 - r_1) \sqrt{\frac{4Q\omega}{b_2 Z \sin \beta}} \quad (2.15)$$

If leakage occurs, the actual flow rate through the impeller channel will be increased, causing a lower Euler head, so the leakage loss and tip leakage loss are calculated as:

$$E_{lk} = \frac{\omega Q_{lk}}{2\pi g b_2} \cot \beta \quad (2.16)$$

$$E_{tip} = \frac{\omega Z \delta (r_2 - r_1) \cot \beta}{\pi g b_2} \sqrt{\frac{Q\omega}{b_2 Z \sin \beta}} \quad (2.17)$$

2.6.4 Friction Loss

The friction loss in a centrifugal pump will cause power loss and efficiency reduction, and can be classified as disk friction loss and wall friction loss respectively. The disk friction loss occurs on rear and front shrouds of the impellers [30], while the wall friction loss occurs near blade walls.

The effects of Reynolds number, surface roughness, direction and rate of leakage flow, the circumferential absolute velocity, and geometry [112] are critical to the friction loss. Many empirical equations have been put forth to predict the disk friction losses [112, 113], but there is no exact mathematical model to predict it. Research on the disk friction loss of actual pump is mainly conducted experimentally. Daily [113] and Fukuda [114] conducted experiments on disk friction losses with a simple disk rotating in a cylindrical casing, and empirical correlations of friction coefficient were concluded. Churchill [115] obtained friction coefficients for turbulent flow experimentally. Nemdili [116] conducted experiments to investigate the factors influencing the disk friction coefficient. His results showed that the Reynolds number and the volute will not affect the friction coefficient. The disk friction loss decreases with smaller disk width, higher clearance, blade number and blade angle.

Disk friction loss is caused by the shear stresses corresponding to the velocity gradient in the gap between the rotating disk and the stationary casing. The disk friction power loss of a rotating disk is defined as [30]:

$$P_{df} = \frac{1}{2} \rho C_f \omega^3 r_2^5 \left(1 - \left(\frac{r_1}{r_2} \right)^5 \right) \quad (2.18)$$

where C_f is the friction coefficient. For the rotating disk friction of miniature pumps, it can be expressed as [113]:

$$C_f = 1.85 \left(\frac{\delta}{r_2} \right)^{1/10} \text{Re}^{-1/2} \quad (2.19)$$

The disk friction loss is:

$$E_{df} = \frac{P_{df}}{\rho g Q_{lk}} = \frac{1}{2} \frac{C_f \omega^3 r_2^5}{g Q_{lk}} \left(1 - \left(\frac{r_1}{r_2} \right)^5 \right) \quad (2.20)$$

The wall friction loss along the impeller channel follows the standard pipe friction model [39]. The wall friction loss is:

$$E_{wf} = C_f \frac{(r_2 - r_1)(W_2 + W_1)^2}{4gD_h \sin \beta} \quad (2.21)$$

The hydraulic diameter D_h of the impeller blade passages is obtained by dividing the passage cross sectional area by the circumference. The default value of friction coefficient C_f is 0.005 for impeller channel. The hydraulic diameter is defined as:

$$D_h = \frac{b_2 (\pi D_2 / Z) \sin \beta}{b_2 + (\pi D_2 / Z) \sin \beta} \quad (2.22)$$

Hence, the wall friction loss can be expressed as:

$$E_{wf} = \frac{C_f (r_2 - r_1) Q^2}{16g\pi^2 b_2^2 \sin^3 \beta} \left(\frac{1}{2\pi r_2 \sin \beta / Z} + \frac{1}{b_2} \right) \left(\frac{1}{r_1} + \frac{1}{r_2} \right)^2 \quad (2.23)$$

2.6.5 Blockage Loss

Blockage loss is due to sudden contraction, which is caused by the flow stagnation and flow separation near the blade leading edge. This loss is relatively large for miniature pumps, because the blade is relatively thick. The flow near the blade leading edge can be treated as flow contraction.

$$E_{\text{blk}} = \frac{K_b}{2g} V_{\text{impeller}}^2 \quad (2.24)$$

where V_{impeller} is velocity after the impeller blockage, K_b is loss coefficient.

For the miniature pump in this study, the value of D_1/D_2 is between 1 to 1.22. A linear relationship can be selected between K_b and D_2/D_1 .

$$K_b = 0.45 \left(\frac{D_2}{D_1} - 1 \right) \quad (2.25)$$

So the blockage loss is:

$$E_{\text{blk}} = 0.45 \left(\frac{D_2}{D_1} - 1 \right) \frac{V_1^2}{2g} = \frac{0.225}{g} \left(\frac{Q}{\left(2\pi r_1 - \frac{Zt}{\sin \beta} \right) b_2} \right)^2 \left(\sqrt{\frac{1}{1 - \frac{Zt}{2\pi r_1 \sin \beta}}} - 1 \right) \quad (2.26)$$

2.6.6 Diffusion Loss

When fluid flows into impeller channels, expansion of the channels will cause a diffusion loss. The diffusion loss should be considered when the relative velocity ratio W_1/W_2 is larger than 1.4 [34], and the loss expression can be approximated by:

$$E_{\text{diff}} = \frac{0.25W_1^2}{2g} \quad (2.27)$$

2.6.7 Shock Loss

The direction of the relative flow may not coincide with the blade angle if the flow is changed [84], or the blade is relatively thick. The method for determining the shock loss coefficient is based on the application of the principles of superposition in the prediction of aerofoil drag. The drag coefficient of a blade at any incidence is considered to be the sum of the drag at zero incidence which is responsible for the profile loss [117] and the contribution due to the shock loss.

Ntoko [86] suggested an expression to calculate shock loss, while Gulich [30] obtained a simpler expression of shock loss when the $W_1/W_{m1} > 0.6$:

$$E_{shock} = \frac{0.3}{g} (W_1 - V_1)^2 \quad (2.28)$$

The theoretical pressure generated by the centrifugal impeller is:

$$E = E_{th} - E_{slip} - E_{lk} - E_{tip} - E_{df} - E_{wf} - E_{blk} - E_{diff} - E_{shock} \quad (2.29)$$

2.7 Effect of Blade Geometry on Pump Performance

The shape of the impeller blade is critical to the pump performance. The influence of the geometrical parameters was studied extensively by many researchers for macro-size pump [118], but no systematical research exists for miniature pump.

The effect of blade angle was first investigated by Kaminoto and Matsuoka [119] experimentally on a centrifugal impeller. Six models with blade profiles having constant blade angles of 30°, 50°, 70° and 90° respectively, were investigated. When running at 1750 rpm, the impeller with blade angle of 30° generated the best performance as smaller blade angle will guide the fluid flow better. Varley [120] conducted experiments on a double suction centrifugal pump at a speed of 1400 rpm.

The blade inlet angle was 36° while different discharge angles of 15° , 27° , 40° , 59° , 74° and 88° were selected to investigate the effect of discharge angle. The results showed that the pump head increased with increasing discharge angle and also increased with larger flow rate. Toyakura et al. [121] conducted experiments on a centrifugal pump at a rotational speed of 1400 rpm. Two discharge angles were investigated, 23° and 65° respectively. Similar to Varley, the pump head increased with increasing discharge angle, and the larger the flow rate, the higher the head generated. However, the effect of discharge angle on pump efficiency is different for water and oil. Tanaka and Ohashi [122] studied centrifugal pump impellers running with high-viscous oil and the radial blades had the highest pump head. Ohta et al. [123] experimentally studied the two-arc-shaped blade exit angles (10° , 20° , 40° , 60° , 73° , respectively) and the experimental results were similar to those of Toyakura et al. [121]. Jia et al. [124] developed a “C” shape blade to suppress secondary flow, and the simulation results showed that this shape is effective in suppressing secondary flow. The new designed blade shape has a large blade inlet angle and a small outlet angle. Li [125] investigated the effect of blade discharge angle on the performance. The analytical results showed that the blade discharge angle has a significant effect on the pump head, power and efficiency. The hydraulic loss increases with increasing blade exit angle. For fluids with different viscosity, the effect of discharge angle is different as shown in Fig. 2-13.

Fig. 2-13 Pump performance for fluid with different viscosity [125]

The blade radius is also critical to the pump performance. Luo et al. [126] investigated the effect of blade inner radius on cavitation in a 3-D centrifugal boiler feed pump numerically and experimentally. Five impellers with different blade inner radius were developed. The results showed that the pump cavitation performance can be significantly improved by properly extending the blade inner radius.

The blade number also plays an important role on pump performance. Li [127] investigated the influence of the blade number on the performance of a centrifugal pump using water and oil with varying viscosities. The effect of the number of blades on the pump performance is dependent on fluid viscosity. For fluids with higher viscosity, the effect of blade number was lower, while for fluids with lower viscosity, its effect was significant. The optimum number of blades was five when the kinematic viscosity of the liquid was less than 200 cSt. When the viscosity was larger than 200 cSt, the optimum number was three. Rababa [128] investigated the effects of blade number on flow field and characteristics of a centrifugal pump both experimentally and numerically. Models with 2, 3 and 4 blades were investigated. The results showed that the optimal flow rate, head and efficiency increased with increasing blade number. Liu [118] investigated pumps with 4, 5, 6 and 7 blades using FLUENT. The simulation results displayed that the blade number had significant effect on the low pressure region near blade inlet and jet-wake region. It was observed that the pump head increases with the number of blades.

Effect of tip gap size influences pump performance to some extent. Engeda [129] analyzed five different centrifugal pump impellers and concluded empirical correlation based on specific speed and tip clearance that predicts the tip clearance effects on pump efficiency. Bing et al. [130] investigated the hydraulic performance on models with different blade tip clearance, and their results showed that the efficiency

reduction varies linearly with the blade tip clearance. Cui et al. [131] investigated the effects of tip clearance on the flow and performance of low specific speed high-speed centrifugal pump with a semi-open impeller with tip clearances 0.5, 1.1 and 2.5 mm numerically and experimentally. The results show that larger clearance will generate a better flow field, but lower head and efficiency. Besides pumps, tip gap has also been studied on fans [132], compressors [133] and inducers [134].

2.8 Computational Fluid Dynamics

Physically, any fluid flow is governed by three fundamental principles, which are mass conservation, energy conservation and momentum conservation. These fundamental principles can be expressed mathematically using partial differential equations (PDEs). Using Computational Fluid Dynamics (CFD), the governing equations can be discretized and solved numerically.

The three numerical solutions of the PDEs are Finite Difference Method (FDM), Finite Element Method (FEM) and Finite Volume Method (FVM). The FDM is the most exact and direct method, but it is not easily to converge and not applicable for complex geometry. FEM is a Lagrangian-polynomial method, which is ideal for complex geometry, but not suitable for higher-order methods. FVM is the locally conservative method in the mass, momentum and energy aspects. Moreover, with FVM, unstructured grids and arbitrary geometries can easily be calculated numerically, but higher-order FVM is difficult to be realized.

Computational Fluid Dynamics has been applied to simulate fluid dynamics and heat transfer problems for many years. 2-D CFD method was first published in 1930s [135] and 3-D in 1967 [136]. After many years of development, CFD is used to support and complement experiments and theory. It is a convenient and economical tool to

investigate flow in complex geometries. One of the most important applications of CFD is in turbo-machines. It guides the design before fabrication and experiment, indicates any flaw in the design and solves existing problems economically and rapidly. In the design stage, CFD is used in two ways, which is direct design and inverse design. In the running stage, CFD can calculate and display some variables and phenomenon, which are very difficult to be measured and observed. In the post-processing stage, CFD is useful to check any fatigue failure of the mechanical components.

In turbo machinery applications, CFD is used in the design of impellers or diffusers in which the blades are modified based on simplified Quasi-3D flow calculations until the requirements are satisfied [30]. CFD is also used to predict the flow in pumps using 3-D Navier-Stokes method. Recently, the inverse design method which determines the geometry according to pressure distribution is developed in CFD. The application of CFD used to optimize pump design is summarized Fig. 2-14. The traditional pump design theory may not be suitable for specified conditions, but can be used as a guideline. Initial design is obtained, and according to CFD simulation results, the initial design is modified in order to satisfy specific conditions, like low shear stress, low turbulence and minimal flow separation.

Fig. 2-14 Flow chart of CFD simulation process [137]

CFD is used to simulate and optimize 3-D flow in pumps for many years, and results show that it can reflect actual situations accurately. Yuan [138] designed a mixed-flow pump and simulated it numerically using FLUENT. The results predicted the flow in the impeller of a mixed flow pump on the premise that the turbulent model and boundary conditions are similar to the actual situations. He also conducted experimental studies which showed that the numerical results were in good agreement with the experimental data.

Cong et al. [139] conducted simulations on a centrifugal pump using the standard $k-\varepsilon$ turbulence model. The internal flow and pressure distribution were analyzed and validated by experimental results. Many other researches such as Grapsas et al. [140] and Li et al. [141] have used CFD in their research on conventional macro-size pumps. In general, the numerical results are in good agreement with experimental results but CFD analysis of miniature pumps is challenging as the flow characteristics is unclear and a suitable turbulence model for low Reynolds number flow is not readily available.

2.9 Particle Image Velocimetry Method

The measurement of velocity can be classified into two types: intrusive measurement and non-intrusive measurement. The earliest instrument to measure fluid velocity is the Pitot-static tube, which was invented by Henri Pitot in the early 18th century and modified in the mid-19th century. In 1920, hot-wire anemometers appeared [142] and the measurement of velocity was improved with smaller size and better frequency response. These two measurement methods are intrusive, which will cause disturbances to the fluid flow.

In the 1960s, Laser-Doppler Anemometer (LDA) was introduced [143]. One laser beam is split into two beams, and the two beams interfere and generate a set of fringes.

When particles pass through the fringes, the reflected light is collected and post-processed to provide information such as velocity. One of the disadvantages is that LDA can only measure velocity at one specific point, which limited its application.

Two of the most popular and accurate velocity measurement methods are Particle Tracking Velocimetry (PTV) and Particle Image Velocimetry (PIV). Compared with previous methods, they are non-intrusive and will not affect the flow field. In addition, the velocity distribution in a wide face or volume can be measured simultaneously and instantaneously [142]. For PTV, each particle is tracked and analyzed individually, and the velocity can be calculated from the captured observations [144]. In this case, if the smallest scale of the flow is smaller than the distance between observations, the velocity cannot be obtained accurately. Hence, PTV is limited to laminar flow with low particle concentration.

For Particle Image Velocimetry (PIV), seeding particles are illuminated by a laser sheet, and the reflected light is captured by high-speed digital camera [142, 145]. Using a pulsed laser, the position of the particles is recorded as a pair of images, which are captured in a pair of laser flashes. Compared with the time taken for the particles to move through the image zone, the time interval between the two flashes is relatively short. The local velocity in any sub region of the acquired images can be calculated based on the time interval between two flashes and displacement of particles. This approach displays completely the instantaneous velocity field within the illuminated and captured region. The typical PIV system includes: an optically transparent test-section in which the flow is seeded with particles can be captured by a camera, a laser generator to illuminate the region of interest, an image recording hardware and a computer installed with software to process the recorded images. The typical PIV system is shown in Fig. 2-15.

Fig. 2-15 Basic requirements for a PIV system [142]

Particle Image Velocimetry is the most popular velocity measurement method to investigate velocity distribution in centrifugal pumps and numerous researches are conducted using PIV technology in rotating machines. Dong et al. [146] used PIV to study the effect of tongue and impeller geometry on the flow structure in a centrifugal pump.

2.10 Summary

In this chapter, a detailed literature review of pump design methods, the challenges in designing miniature pumps, pump scaling effect, energy losses, together with the effect of blade geometry are presented. In addition, the use of Computational Fluid Dynamics and Particle Image Velocimetry in analyzing fluid flow patterns is also discussed. The design of miniature pump is different from the design of macro-size pumps. The research on miniature size pumps shows that the two factors, the small pump size and large flow rate, cannot be achieved simultaneously. It is necessary to develop a small size pump with sufficient fluid flow rate. The energy model analysis for miniature pump is also reviewed, in order to be compared with the numerical and experimental results. The blade geometry of miniature pump is critical and the effect

of blade geometry should be re-evaluated. CFD and PIV are two methods to investigate miniature pump performance. Using CFD, the detailed pressure and velocity distribution can be displayed. The results of CFD need to be validated by experimental studies.

3. Design of Miniature Centrifugal Pump

In this chapter, the preliminary dimensions of the impeller and volute model will be established based on the design method for macro-size pump. Based on this preliminary model, new models will be developed based on consideration of miniature size and rational assumptions of geometrical parameters. These will allow a comparison of the effects of geometrical parameters on pump performances when these models are evaluated numerically in Chapter 4 and experimentally in Chapter 5. Selected models will be experimentally examined.

3.1 Impeller Design Procedure

The application requirements and operating conditions should be specified before design of the impeller geometry. Miniature pumps are usually used in limited space, which requires small size. So the impeller outer radius is usually limited by application requirement. The flow rate and the pump head are specified by the intended applications. The flow field is also a critical consideration in some medical applications. The running speed is usually limited by the parameters and size of the motor.

The type of pump and number of stages should be established in the first step of pump design. The selection of the pump type is based on the specific speed n_s as shown in the “Cordier” diagram (Fig. 2-5), while the selection of stages is based on the pump size [20]. To do this, the specific speed and specific diameter should be decided first. Based on the pump size and type, the specific speed of the pump is selected as 1 and from the “Cordier” diagram, the corresponding value of the specific diameter d_s is 3.

The outlet flow coefficient ϕ_i and head coefficient ψ are then determined accordingly with selected specific speed using Fig. 3-1. The impeller outer radius r_2 and blade

outlet height b_2 can be derived from equations (3.1) and (3.2) as:

$$r_2 = \frac{1}{N} \sqrt{\frac{g\Delta H}{\psi}} \quad (3.1)$$

$$b_2 = \frac{Q}{2\pi N r_2^2 \phi_i \varepsilon} \quad (3.2)$$

where ε is the blockage factor. Its value is generally between 0.8 and 0.9 and smaller values are recommended for small machines. A value of $\varepsilon=0.8$ is adopted for the design of miniature pump.

Fig. 3-1 Selection of outlet flow coefficient and head coefficient of pumps [20]

To simplify the geometry of the pump, the shaft radius can be neglected, and the impeller inner radius r_1 can be simplified to be equal to the eye radius. The flow coefficient ϕ is the ratio of the one-dimensional axial velocity entering the eye to the tangential speed of the impeller eye. It varies from 0.2 to 0.3 for impellers, and a larger value is recommended for smaller pumps. For miniature pump, ϕ can be selected as 0.29. The eye radius r_e can be estimated from equation (3.3).

$$r_e = \sqrt{\frac{Q}{N\pi\phi\left(1 - \frac{r_s^2}{r_e^2}\right)}} \quad (3.3)$$

The blade inlet angle β_1 and outlet angle β_2 are set to match the flow field, generally depending on the velocity triangles. The inlet velocity triangle enables the blades to receive the incoming fluid with minimum loss. The blade inlet angle β_1 is the angle between the mean camber line of the blade and the circumferential direction. It is usually set to be equal to or slightly higher (empirically 2-3° higher) than the fluid angle β_{f1} . The blade outlet angle β_2 is designed to transport the incoming fluid with minimum loss and avoid flow deviation along the impeller profile. Design experience shows that the impeller outlet angle β_2 is critical to pump performance [24]. The optimum impeller outlet angle from an efficiency point of view should be close to the inlet angle, which can be set as the same value.

The blade shape is determined by the inlet and outlet blade angle and the control equation of the blade shape. For constant blade angle ($\beta=\beta_1=\beta_2$), the blade is logarithmic-spiral, while for radial blade, the control equation of the blade shape is a straight line.

The number of blades Z should ensure minimal or no flow separation in the blade passage [20]. It is very complex to select an appropriate value of Z due to the interaction between the fluid and adjacent structure such as volute tongue or diffuser vane. Stepanoff [43] suggested a design equation for blade number as equation(3.4), while Pfleiderer's[147] proposed that Z be given by equation (3.5).

$$Z = \beta_2/3 \quad (3.4)$$

$$Z = Z = 6.5 \left[\frac{D_2 + D_1}{D_2 - D_1} \right] \sin \beta \quad (3.5)$$

3.2 Volute Design Procedure

The height of the volute b_3 is determined by the considerations of flow pattern in the volute, energy loss and accommodation of different size impellers. A relatively larger volute height will decrease the pressure variation along the volute circumference and hence a smoother flow field can be expected [148]. According to experimental data, the following guideline (Table 3-1) [43] is for the determination of the volute height b_3 in relationship with the pump's specific speed N_s . The definition of N_s is given as equation (3.6).

$$N_s = \frac{N(\text{rpm})\sqrt{Q(\text{gpm})}}{[\Delta H(\text{ft})]^{3/4}} = 2733n_s \quad (3.6)$$

Table 3-1 Guideline for volute height

Specific speed (N_s)(rpm(gpm) ^{0.5} /ft ^{0.75})	Volute height (b_3)
<1000	$2.0b_2$
1000 – 3000	$1.75b_2$
>3000	$1.6b_2$

The base circle is used to determine the starting point of the volute, and a suitable gap between the impeller and volute base circle is necessary to allow the flow from the impeller to exit smoothly into the volute. Small gap may result in damage to the impeller when pump is running at high speed, while a big gap will decrease pump efficiency. The first method of base circle optimization is through the definition of a fixed radial gap δ , which is defined as equation(3.7) [43]. Another approach is based on specific speed N_s and the impeller outer diameter D_2 , as shown in Table 3-2. For miniature pumps, the gap between the impeller outlet and the volute base circle will be too large if equation (3.7) is used, while using Table 3-2 will yield a very small value. A value of $1.2D_2$ is selected to ensure smooth flow in the gap.

$$\delta = \frac{D_3 - D_2}{D_2} \times 100\% \quad (3.7)$$

Table 3-2 Guideline for volute base circle

Specific speed (N_s)(rpm(gpm) ^{0.5} /ft ^{0.75})	Base circle diameter (D_3)
600 – 1000	$1.05 \times D_2$
1000 – 1500	$1.06 \times D_2$
1500 – 2500	$1.07 \times D_2$
2500—4000	$1.08 \times D_2$

3.2.1 Volute Profile with Constant Mean Velocity (CMV) Method

A Constant Mean Velocity is assumed in the volute passage, and hence the static pressure distribution around the impeller is uniform. In the case of a rectangular cross sectional shape, the cross section area $A_\theta = b_3(r_4 - r_3)$. With the mean velocity C_m , the volute cross-sectional area at any angular point A_θ can be calculated by equation (3.8). The mean velocity C_m is the most important parameter in the volute design procedure, and it can be expressed as a function of the pump pressure as given in equation (3.9) [149].

$$A_\theta = \frac{Q_\theta}{C_m} \quad (3.8)$$

$$C_m = K_3 \times \sqrt{2gH} \quad (3.9)$$

where g is the gravitational acceleration constant, H is the pump pressure head generated and K_3 is a design factor. The design factor K_3 is an empirical constant and is a function of the pump specific speed. The relationship between K_3 and specific speed is summarized empirically in Fig. 3-2. The volute outer profile is then defined by equation (3.10), which is an Archimedes spiral.

Fig. 3-2 K_3 as function of the specific speed

$$r_4 = \frac{A_\theta}{b_3} + r_3 = \frac{Q \times \frac{\theta}{360}}{b_3 C_m} + r_3 = \frac{Q \times \theta}{360 \times b_3 \times C_m} + r_3 \quad (3.10)$$

3.2.2 Volute Profile with Constant Angular Momentum (CAM) Method

In the Constant Angular Momentum method, the fluid is assumed to have constant angular momentum in the volute, following the equation $C_u r = K$. For a rectangular cross-section volute, the flow rate through an infinitesimal cross-section located at an angular position θ can be given by equation (3.11):

$$dQ_\theta = b_3 C_u dr \quad (3.11)$$

The flow rate at any angular position θ can be integrated as:

$$Q_\theta = \int_{r_3}^{r_4} b_3 K \frac{dr}{r} = b_3 K \ln \frac{r_4}{r_3} \quad (3.12)$$

Finally, the outer radius of the volute can be expressed as:

$$r_4 = r_3 e^{\left(\frac{\theta}{360} \frac{Q}{K b_3} \right)} \quad \text{where } (0^\circ \leq \theta \leq 360^\circ) \quad (3.13)$$

Equation (3.13) gives a logarithmic spiral profile. To determine a suitable value of K , the empirical method developed by Anderson [24] which is based on $A_{\text{impeller}}/A_{\text{throat}}$ is employed as in Fig. 3-3. The throat area and the impeller outlet area are defined as equations (3.14) and (3.15).

Fig. 3-3 Empirical data curve of Anderson [24]

$$A_{\text{throat}} = b_3 \times (r_{4(\theta=360)} - r_{4(\theta=0)}) = b_3 r_3 \left(e^{\left(\frac{Q}{Kb_3} \right)} - 1 \right) \quad (3.14)$$

$$A_{\text{impeller}} = \pi D_2 b_2 \quad (3.15)$$

3.3 Design Example of Miniature Pump

According to the original design method mentioned above, geometrical parameters of both the impeller and volute will be obtained in this section. Both Constant Mean Velocity method and the Constant Angular Momentum method are employed respectively.

3.3.1 Specifications

The objective is to design a miniature pump delivering a flow rate of 2 l/min and has an impeller diameter of 20 mm. To achieve the best performance and the highest efficiency, a specific speed n_s of 1 is selected and the corresponding specific diameter d_s as 3 according to Cordier diagram. The pump head generated and rotational speed can be obtained from the definition equations of n_s and d_s . The calculated results and the selection of some parameters are shown in Table 3-3. Other volute parameters are selected and listed in Table 3-4.

Table 3-3 Pump requirements and selected empirical parameters

Flow rate Q (L/min)	Head ΔP (Pa)	Speed ω (rad/s)	Blockage factor ε	flow coefficient ϕ
2	562.5	112.5	0.8	0.29

Table 3-4 Parameters of volute

Volute cross section shape	rectangular
Volute tongue shape	semi-circle
Base circle radius r_3 (mm)	12
Design factor of CMV K_3	0.32
Ratio of throat area and the impeller outlet area	1.26
Volute height b_3	$1.75b_2$
Basic diameter D_3	$1.2D_2$

3.3.2 Original Design of Impeller

Based on the design procedure listed in sections 3.1 and 3.2, the geometrical parameters of both of the impeller and volute can be obtained. The impeller inner radius and height are given by equations (3.16) and (3.17). To specify the blade angles, the velocity triangles of the impeller inlet and outlet are investigated. For the impeller

inlet, the radial and tangential velocities can be calculated from the flow rate and rotational speed, while for the impeller outlet, the outlet flow coefficient and head coefficient are employed. The impeller outlet flow coefficient ϕ_i is 0.1715 from Fig. 3-1. For logarithmic blade with constant blade angle [32], the calculation procedure is summarized in Table 3-5. Blade inlet and outlet angles are same.

$$r_1 = r_e = \sqrt[3]{\frac{Q}{N\pi\phi_e \left(1 - \frac{r_s^2}{r_e^2}\right)}} = 6.88 \text{ mm} \quad (3.16)$$

$$b_2 = \frac{Q}{2\pi r_2^2 N \phi_e \varepsilon} = 3.44 \text{ mm} \quad (3.17)$$

Table 3-5 Velocity triangles of impeller inlet and outlet of the original model

Calculation equations	Value
$U_1 = r_1 \omega$	0.774 m/s
$V_1 = Q / A_e$	0.224 m/s
$\beta_{f1} = \tan^{-1}(V_1 / U_1)$	16.1° (17° is selected)
$W_1 = V_1 / \sin \beta_{f1}$	0.808 m/s
$U_2 = r_2 \omega$	1.125 m/s
$V_{m2} = \phi_i U_2 = 0.1715 U_2$	0.193 m/s
$W_2 = V_{m2} / \sin \beta_2$	0.660 m/s
$V_{\theta 2} = U_2 - W_2 \cos \beta_2$	0.494 m/s

The velocity triangles in Fig. 3-4 show that both the blade inlet and outlet angles are 17°, which is relatively small when compared with commercial pumps. Generally, for backward-curved impellers, the energy transfer from the rotor to the fluid is low as compared with radial and forward-curved ones [29]. The smaller the blade angle, the lesser energy will be transferred, resulting in a lower pump efficiency. However, smaller blade angle will ensure flow with less flow separation, which is desirable in some applications.

Fig. 3-4 Inlet and outlet velocity triangles of the originally designed impeller

Based on that odd number of blades is usually used for pumps [150], 5 is selected, which is the nearest odd number of the average of the minimum number of blades Z according to equations (3.4). The blade thickness t is selected as $0.04 r_2$, which is 0.4 mm. The number of blades Z is validated using the solidity σ , which is defined as equation (3.18). σ is 1.2 when the blade number is 5, which is in the range of smooth and attached flow as described by Karassik [20]. The profile of the impeller is shown as in Fig. 3-5.

$$\sigma = \frac{Zl}{2\pi r_2} \quad (3.18)$$

Fig. 3-5 Profile of the originally designed impeller (actual size)

3.3.3 Modified Models of Impeller

The design of miniature size rotary pump is different from macro-size pump, because the viscous force is significant at small scales, and results in larger impedance[97]. The relatively larger viscous force will reduce the velocity, affect pressure distribution significantly, and hence reduce pumping efficiency. In addition, the operating Reynolds number is much lower and the flow pattern will be different. Hence, the design of miniature pumps is more complex than that of macro-size pumps.

The original model developed using design method of macro-size pump may be inefficient and need to be modified in order to achieve better performance. The impeller inner radius r_1 of 6.88 mm is relatively big when compared with the outer radius r_2 , and the blade angle β is too small. Besides blade radius and angle, the number of blades Z can be reduced, to reduce blockage and wall effect.

In the design procedure of the original model, the flow coefficient plays an important role, and different values of flow coefficient will be chosen to satisfy different impeller inner radius and angle. For macro-size pump, the suggested flow coefficient is 0.29. To investigate the effect of flow coefficient on miniature pump, ten flow coefficients are selected as listed in Table 3-6. The blade inner radius r_1 , blade angle β are calculated accordingly. The blade number is 5 and blade thickness is 0.4 mm. To investigate the effect of blade angle, models 11, 12 and 13 with different blade angle listed in Table 3-7 are examined. Other geometrical parameters, such as blade thickness t and blade number Z are also investigated with models 14 to 16 and 17 to 21. To investigate the applicability of similarity laws, 10 mm diameter models that are geometrically similar are generated and examined too. The specifications of the 10 mm diameter models are listed in Table 3-8.

Table 3-6 Modified impeller models with different flow coefficient

No.	ϕ	r_1 (mm)	$\beta(^{\circ})$	Z	t (mm)
1	3.49	3	74	5	0.4
2	2.99	3.16	72	5	0.4
3	2.49	3.36	68	5	0.4
4	1.99	3.62	63	5	0.4
5	1.47	4	56	5	0.4
6	1.29	4.18	53	5	0.4
7	0.90	4.72	42	5	0.4
8	0.75	5	37	5	0.4
9	0.44	6	24	5	0.4
10	0.29	6.88	17	5	0.4

Table 3-7 Modified impeller models with different geometrical parameters

No.	r_1 (mm)	$\beta(^{\circ})$	Z	t (mm)
11	4	24	5	0.4
12	4	63	5	0.4
13	4	90	5	0.4
14	4	56	5	0.2
15	4	56	5	0.6
16	4	56	5	0.8
17	4	56	3	0.4
18	4	56	7	0.4
19	4	56	9	0.4
20	4	56	11	0.4
21	4	56	13	0.4

Table 3-8 10 mm diameter models

No.	r_1 (mm)	$\beta(^{\circ})$	Z	t (mm)
22	1.58	72	5	0.2
23	3.44	17	5	0.2
24	2	56	5	0.2

25	2	63	5	0.2
26	2	90	5	0.2
27	2	56	11	0.2
28	2	56	13	0.2

To examine the effects of other variables, impeller model 5 is taken as the reference model, where the inner radius is fixed at 4 mm. The details of model 5 are shown in Table 3-9.

Table 3-9 Velocity triangles of impeller inlet and outlet of model 5

Calculation equations	Value
$U_1 = r_1 \omega$	0.450 m/s
$V_1 = Q/A_e$	0.663 m/s
$\beta_{f1} = \tan^{-1}(V_1/U_1)$	55.8°(56° is selected)
$W_1 = V_1/\sin \beta_{f1}$	0.802 m/s
$U_2 = r \omega_2$	1.125 m/s
$V_{m2} = \phi U_2 = 0.1715 U_2$	0.193 m/s
$W_2 = V_{m2}/\sin \beta_2$	0.233 m/s
$V_{\theta 2} = U_2 - W_2 \cos \beta_2$	0.995 m/s

In general, comparing Fig. 3-6 and Fig. 3-4, it is noted that the blade angle of the modified impellers is larger than the originally designed model. Hence the pressure generated should be higher too. The blade thickness t and the blade height b_2 do not have any explicit relationship with the inner radius and blade numbers, so t is maintained at 0.4 mm and b_2 is 3.44 mm. The actual size of the impeller model 5 is sketched in Fig. 3-7.

Fig. 3-6 Inlet and outlet velocity triangles of impeller model 5

Fig. 3-7 Profile of the impeller model 5(actual size)

3.3.4 Original Design of Volute

For the Constant Mean Velocity method, the average velocity for all cross-sections is assumed to be constant, so the pressure distribution is uniform. Based on the selection of the empirical parameter K_3 , the constant mean velocity C_m , the volute height b_3 and the volute outside profile can be derived accordingly from equations (3.19) to (3.21). The volute profile is shown in Fig. 3-8.

$$C_m = K_3 \times \sqrt{2gH} = 0.68 \text{ m/s} \quad (3.19)$$

$$b_3 = 1.75b_2 = 6 \text{ mm} \quad (3.20)$$

$$r_4 = \frac{Q \times \theta}{360 \times b_3 \times C_m} = 0.0227\theta + 12 \quad (3.21)$$

Fig. 3-8 Volute profile of the Constant Mean Velocity method (actual size)

For Constant Angular Momentum method, the selection of impeller and throat area are based on Fig. 3-3. When the specific speed N_s is specified as 2733 (n_s is 1), $A_{\text{impeller}}/A_{\text{throat}}$ is 1.26. The parameter K is 0.00436. The volute profile is given by equation (3.22) and Fig. 3-9.

$$r_4 = r_3 e^{\left(\frac{\theta}{360} \frac{Q}{K b_3} \right)} = 12 e^{0.00354\theta} \quad (3.22)$$

Fig. 3-9 Volute profile of the Constant Angular Momentum method (actual size)

3.3.5 Modified Models of Volute

The volute design mainly depends on the selection of the mean velocity C_m and constant momentum K . The selection of C_m and K for the original design is mainly based on macro-size pump. For the design of miniature pumps, these two values should be investigated further to achieve the best pump performance. Several volutes are obtained based on different value of mean velocity C_m and constant momentum K , as illustrated in Table 3-10 and Table 3-11. For both CMV volute and the CAM volute, the outlet width is calculated by equation (3.23).

$$d = r_4(\theta = 360^\circ) - r_4(\theta = 0^\circ) - 2 \quad (3.23)$$

Table 3-10 Volute with different mean velocity C_m

No.	C_m	Volute profile (mm)	Outlet width (mm)
1	0.8	$r_4=12+0.0193\theta$	11.8
2	0.68	$r_4=12+0.0227\theta$	14.4
3	0.5	$r_4=12+0.0309\theta$	20.2
4	0.45	$r_4=12+0.0340\theta$	22.4
5	0.38	$r_4=12+0.0400\theta$	26.0
6	0.35	$r_4=12+0.0441\theta$	28.8

Table 3-11 Volute with different mean momentum K

No.	K	Volute profile (mm)	Outlet width (mm)
7	0.00646	$r_4=12\exp(0.00239\theta)$	14.4
8	0.00530	$r_4=12\exp(0.00291\theta)$	20.2
9	0.00500	$r_4=12\exp(0.00309\theta)$	22.4
10	0.00460	$r_4=12\exp(0.00336\theta)$	26.0
11	0.00436	$r_4=12\exp(0.00354\theta)$	28.8
12	0.00400	$r_4=12\exp(0.00386\theta)$	34.2

3.4 Summary

In this chapter, both impeller and volute are designed based on the traditional macro-size pump design method and are subsequently modified. For the impeller, the impeller inner radius r_1 , blade height b_2 , blade angle β , blade thickness t , blade number Z are designed with the consideration of less flow separation, high pump head, and ease of fabrication. Models with different flow coefficients and geometric parameters are developed, in order to investigate their effect. For the volute, the selection of the empirical parameters, mean velocity C_m of the Constant Mean Velocity method and constant momentum K for Constant Angular Momentum method, are critical. Different values of the C_m and K are selected and corresponding values of volute outlet width are described. The effect of the empirical parameters and outlet width will be investigated. The designed impeller and volute models will be numerically analyzed and compared in chapter four. The optimized impeller and volute will then be fabricated to validate the computed results, in order to provide further insights on the effects of geometrical parameters on the performance of miniature pumps.

4. Computational Fluid Dynamics Modeling

Computational Fluid Dynamics (CFD) provides an economical, effective and rapid approach to solve fluid flow problems. CFD can also provide detailed information of complex flow patterns, which is almost impossible to be measured experimentally and solved analytically. CFD is designed to solve practical problems using computer science and discrete mathematics. The basic principle of CFD is to solve nonlinear differential equations, including continuity equations, momentum equations and energy equations, to predict the flow field, heat transfer, mass transfer and combustion process [151].

This chapter presents the numerical techniques employed in the simulation of the flow pattern in a miniature centrifugal pump. Issues such as choice of turbulence models, grid independence and convergence will also be discussed.

4.1 Introduction of the Software FLUENT

FLUENT can handle various flow problems with its powerful and broad calculation capability. Flows with two and three dimensional geometries under a variety of conditions can be accurately simulated, including various materials (Newtonian and non-Newtonian), different flow models (inviscid, laminar and turbulent) and different time-dependence (steady and transit). An unstructured CFD solver for both structured and unstructured meshes that uses the finite volume method is employed to solve the governing equations. FLUENT can simulate convective, coupled and radiative heat transfer, and it can account for the mixing and reaction of chemical species, and for arbitrary sources of heat, mass, turbulence and momentum. Various rotating frames and a variety of meshes can also be employed.

4.2 Turbulent Models

FLUENT has various flow models and the two equations turbulent models are most commonly employed, including the k - ε model, k - ω model and Reynolds Stress Model.

The standard k - ε model is the most popular model and has the widest applications because of its stability, economy and high accuracy. From the k and ε equations, the turbulent viscosity μ_t can be obtained. Finally, the Reynolds stress can be solved with Boussinesq's assumption. The isotropy of turbulent flow is assumed in the Standard k - ε model, which limits its application to swirl flow. The RNG k - ε model and Realizable k - ε model take into consideration of anisotropy, and are more accurate.

The RNG k - ε model is similar to the standard k - ε model, but has some additional improvement features. An additional term in the ε equation between turbulence dissipation and mean shear is introduced. Moreover, the effect of swirl on turbulence, an analytical formula for the turbulent Prandtl number and a differential formula for effective viscosity are considered. All these improvements make this model more accurate in flows with high velocity gradient and transitional flows.

The realizable k - ε model has the same k equation as the standard k - ε model, but with an improved ε equation. The empirical parameter C_μ is a variable, which makes this model suitable for anisotropic flows. The realizable k - ε model has better performance in the model of planar and round jets, rotational flows, boundary layers under strong adverse pressure gradients or separation.

The k - ω turbulent model is a low-Reynolds number turbulent model, which takes into consideration of low-Reynolds-number, compressibility and the shear flow spreading

[152]. With these considerations, the $k-\omega$ model is suitable for calculation of wake flow, mixed layer, mixed jet and free shear flow affected by wall limitation.

The Shear Stress Transport $k-\omega$ model (SST $k-\omega$ model) combines the $k-\omega$ model and $k-\varepsilon$ model together with a blending function. In the near-wall region, the blending function is 1, meaning that the model is $k-\varepsilon$. In the far field, the blending function is 0, meaning that the model is $k-\omega$. In this case, the SST $k-\omega$ model can achieve better performance for both near-wall zone and far field. Moreover, the SST $k-\omega$ model has additional advantages which make it more applicable and accurate for the present pump model. The lateral dissipation derivative is considered in SST $k-\omega$ model.

For the Reynolds Stress Model, turbulent viscosity is not isotropic. This assumption is more effective and accurate for flows with swirl, rapid changes in strain rate or substantial streamline curvature. The viscosity term in the RANS equation is solved directly with 6 additional equations. This model requires more computing source and higher computer speed.

The Reynolds number is the key factor in the determination of an appropriate turbulence model and is defined in equation (4.1). In this project, the Reynolds number is about 45,000. For rotary machines, it is commonly accepted from studies that fully turbulent flow is achieved for a Reynolds number of more than 100,000 [153]. Hence, the laminar model is selected. For models running at twice the speed of the designed speed, the flow is no longer laminar and the SST $k-\omega$ is selected in the first 4000 steps, and Reynolds Stress Model is selected in the next 4000 steps. The first 4000 steps of SST $k-\omega$ is to ensure that the iteration is converged and stable, while the second 4000 steps of Reynolds Stress Model is to ensure an accurate final result.

$$\text{Re} = \frac{\rho \omega D_2^2}{\mu} \quad (4.1)$$

4.3 Governing Equations

The energy equation can be neglected since the temperature variation in this problem is insignificant. As the residence time of the fluid particle is relatively short inside the pump, the effect of the temperature fluctuations on the fluid density and viscosity are very small. Therefore, the change of the flow velocity distribution within the field caused by temperature fluctuation can be neglected in the study.

The governing equations that are applicable for pump flow are the conservation equations for mass and momentum. Using Cartesian coordinates, the governing equations for incompressible Newtonian fluid are:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (4.2)$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_j} + \frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_j} + F_i \quad (4.3)$$

The shear stress for turbulent flow is defined as:

$$\tau_{ij}^T = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \overline{\rho u_i u_j} \quad (4.4)$$

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

$$F_i = g_i - 2 \left(\omega \times \vec{U} \right)_i - \left[\vec{\omega} \times \left(\vec{\omega} \times \vec{r} \right) \right]_i + E_i \quad (4.5)$$

4.3.1 k - ω Model

For k - ω model, based on the Boussinesq's assumption, for incompressible turbulent flow, the relationship between the Reynolds stress and average velocity can be expressed as equation (4.6).

$$\overline{\rho u_i u_j} = -\mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \frac{2}{3} \left(\rho k + \mu_t \frac{\partial u_i}{\partial x_i} \right) \delta_{ij} \quad (4.6)$$

$$\delta_{ij} = \begin{cases} 1 & i = j \\ 0 & i \neq j \end{cases} \quad i, j = 1, 2, 3 \quad (4.7)$$

The unknown variable μ_t is introduced in the control equations. Turbulence models are needed to determine μ_t in terms of known quantities. For the SST k - ω model, the turbulent viscosity μ_t is expressed as equation (4.8).

$$\mu_t = \frac{\rho k}{\omega_t} \frac{1}{\max \left[\frac{1}{a^*}, \frac{\Omega F_2}{a_1 \omega_t} \right]} \quad (4.8)$$

where k is the turbulent kinetic energy and ω_t is turbulent kinetic energy dissipation rate.

To solve these variables, the k and ω equations should be added to the governing equation group. They are expressed as:

$$\rho \frac{\partial k}{\partial t} + \rho \frac{\partial (k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \quad (4.9)$$

$$\rho \frac{\partial \omega_t}{\partial t} + \rho \frac{\partial (\omega_t u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega_t}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (4.10)$$

where the turbulent generation terms G_k and G_ω , diffusion terms Y_k, Y_ω and divergence term D_ω are defined empirically.

4.3.2 Reynolds Stress Model

For the Reynolds stress model, the calculation of the Reynolds stress is different from the $k-\omega$ model.

$$\frac{\partial}{\partial t}(\overline{u_i' u_j'}) = P_{ij} + D_{ij} - \varepsilon_{ij} + \Pi_{ij} + \Omega_{ij} \quad (4.11)$$

where P_{ij} is the rate of production, D_{ij} is the transport by diffusion, ε_{ij} is the rate of dissipation, Π_{ij} is the transport due to turbulent pressure-strain interactions and Ω_{ij} is the transport due to rotation. Equation (4.11) describes six partial differential equations. The five terms in the right-hand-side of the equation (4.11) can be modeled as follows.

$$P_{ij} = - \left(\overline{u_i' u_m'} \frac{\partial u_j}{\partial x_m} + \overline{u_j' u_m'} \frac{\partial u_i}{\partial x_m} \right) \quad (4.12)$$

$$D_{ij} = \frac{\partial J_{ijk}}{\partial x_k} \quad (4.13)$$

where

$$J_{ijk} = \overline{u_i' u_j' u_k'} + \overline{p' (\delta_{jk} u_i' + \delta_{ik} u_j')} \quad (4.14)$$

$$\varepsilon_{ij} = 2\mu \overline{\frac{\partial u_i'}{\partial x_k} \frac{\partial u_j'}{\partial x_k}} \quad (4.15)$$

$$\Pi_{ij} = - \overline{p' \left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i} \right)} \quad (4.16)$$

$$\Omega_{ij} = - 2\omega_k \left(\overline{u_j' u_m' e_{ikm}} + \overline{u_i' u_m' e_{jkm}} \right) \quad (4.17)$$

where p is the pressure, e_{ijk} is -1, 0 or 1 depending on the indicators, ω_k is the rotational vector.

4.4 Physical Model and Boundary Conditions

The simulation model of the fluid consists of four parts, which are inlet zone, gap zone, impeller zone and volute zone respectively, as illustrated in Fig. 4-1. In order to achieve fully developed flow, the velocity profiles at different positions are illustrated in Fig. 4-2 and Fig. 4-3. It can be concluded that the velocity profiles are similar when the dimensionless length z/d is larger than 4. So the inlet length is selected as 5 times of the inlet pipe diameter.

Fig. 4-1 Simulation model of the pump

Fig. 4-2 Velocity profile in the inlet

Fig. 4-3 Velocity vector in the inlet

The centrifugal pump model consists of an impeller partly shrouded with several backswept blades, a straight intake section, and a spiral volute casing. For the models in Tables 3-6, 3-7 and 3-8, the impeller blade has a constant blade angle throughout its profile. The flow from the impeller is discharged into a spiral volute casing with a mean circle diameter d_3 of 24 mm. The profile of volute is listed in Tables 3-10 and 3-11. The pump is designed to operate at 112.5 rad/s.

The boundary conditions are set according to the actual operating condition. Therefore, the flow rate through the pump and the rotational speed of the impeller are inputs to the simulation model. For a given flow rate, the pressure generated can be computed.

The fluid material is water in this study, which can be treated as a Newtonian fluid with constant viscosity.

Multiple Reference Frame (MRF) is used in this project. A new rotating reference frame is established in the impeller zone. The new frame is rotating at the speed of the motor speed. Walls of moving parts, like blade walls and impeller base walls are defined as stationary in the reference frame, which means that these walls are rotating with the same speed of the reference frame. Other walls are absolutely stationary, and can be defined as stationary in the absolute frame. For a given flow rate, the boundary condition at the inlet is the fluid velocity and the outlet gauge pressure is specified. All the walls in the rotational zone are rotating at the same speed of the shaft, and other walls are stationary. The definition of boundary conditions is as follows:

$$\text{Inlet face: } u = 0\mathbf{i} + 0\mathbf{j} - \frac{Q}{A_{inlet}}\mathbf{k}$$

$$\text{Outlet face: } P = 0$$

$$\text{Inlet wall and volute walls: } u = 0\mathbf{i} + 0\mathbf{j} + 0\mathbf{k}$$

In the rotating zone, both control equations and boundary conditions are described in cylindrical coordinates. All boundary conditions of solid walls (which include blade walls and impeller base walls) are listed as follows:

$$u = 0\hat{\mathbf{r}} + r\omega\hat{\boldsymbol{\phi}} + 0\hat{\mathbf{z}}$$

In the interface between rotating zone and stationary zones, the coordinates should be changed between Cartesian coordinates and cylindrical coordinates. From Cartesian coordinates to cylindrical coordinates:

$$u_r = u_x\mathbf{i} + u_y\mathbf{j} + u_z\mathbf{k} = (u_x \cos \varphi + u_y \sin \varphi)\hat{\mathbf{r}} + (-u_x \sin \varphi + u_y \cos \varphi)\hat{\boldsymbol{\phi}} + u_z\hat{\mathbf{z}}$$

where $\varphi = \tan^{-1}(y/x)$. From cylindrical coordinates to Cartesian coordinates:

$$\mathbf{u} = u_r \hat{\mathbf{r}} + u_\varphi \hat{\boldsymbol{\phi}} + u_z \hat{\mathbf{z}} = (u_r \cos \varphi - u_\varphi \sin \varphi) \mathbf{i} + (u_r \sin \varphi + u_\varphi \cos \varphi) \mathbf{j} + u_z \mathbf{k}$$

4.5 Grid Generation

The software GAMBIT is used to build and mesh the pump geometrical model. Unstructured triangular and tetrahedral meshes offer many advantages of moderately complex geometry especially in sharp corners. However, structured meshes are economical in some situations as they can provide higher-quality solutions with fewer cells, allowing a much larger aspect ratio than unstructured mesh, thus saving simulation time.

The mesh size is an important parameter in the grid generation procedure. In regions where the flow pattern is expected to be complicated, especially regions near to the sharp corners, greater nodal density and smaller mesh sizes are required in order to model the variation near such geometries accurately. Moreover, the total mesh numbers should also be considered, because very small mesh size will result in more meshes, which will require great computer resources and longer simulation time. The mesh in the boundary layer region is important and the boundary layer thickness should also be carefully controlled. To do this, a dimensionless variable y^+ is usually used and is defined as equation (4.18):

$$y^+ = \frac{\rho u_\tau y}{\mu} \quad (4.18)$$

The nature of viscous layer is classified by y^+ . It is the viscous sublayer when y^+ is in the range of 0 to 5, buffer layer when y^+ is in the range of 5 to 30, and log-law region when y^+ is larger than 30. y^+ is not useful for laminar flow and should be in a reasonable range for turbulent flow. For the present simulation of models with 20 mm

diameter impellers operating at 112.5 rad/s and 10 mm diameter impellers operating at 450 rad/s, a laminar model is used, and y^+ will not be evaluated. For models with diameter 20 mm operating at 225 rad/s, the Reynolds Stress Model is used. The Reynolds number in the impeller channel (calculated based on channel width and relative velocity) is 8688, and the recommended y^+ should be in the viscous sublayer or log-law region. For the present study, the pump size is small. If log-law region is used, only one or two meshes exist in some gaps. To have a clear view of the velocity in these gaps, more meshes are needed and y^+ should fall into the viscous sublayer. For the present calculation, y^+ , in general, is within viscous sublayer and the results show good agreement with experimental data.

Mesh quality parameters should also be considered in conjunction with the solution scheme chosen. The final accuracy of a procedure is always a function of the grid quality. Several parameters should be controlled in a limited range, e.g. equiangle skew, equisize skew and edge ratio.

For the models under consideration, 3-D meshes are generated based on 2-D meshes, which can be triangular or rectangular. For the inlet, volute and gap zones, 2-D meshes are regular rectangular meshes, while for the impeller zone where the flow is expected to be complex, triangular meshes are employed. The 3-D meshes employed are mainly wedge and cubic.

The numerical computation is carried out with a multiple reference frame (MRF) approach because the impeller flow field is within a rotating frame whereas the inlet flow, gap flow and volute casing flow are in the stationary frame. The meshes of four computational domains, the inlet zone, gap zones, impeller zone, and volute zone, are generated separately. The two connected zones will share the same surface, to ensure consistent meshes. The zones are meshed one by one. When the first zone is meshed,

the interface will be meshed simultaneously. The second zone will be meshed based on the existing meshes at the interface. Structured meshes with the wedge and cubic elements are used in all four computational domains. A localized refinement of mesh is employed in regions close to the volute tongue area, the impeller blade leading and trailing edges in order to accurately capture the flow field structure. The number of elements used in the numerical simulation is fixed after the grid independence study, which will be discussed in section 4.7.

4.6 Convergence

The residual is a measurement of how well the current solution satisfies the discrete form of each governing equation and whether the result has attained an acceptable value. FLUENT reports a residual for each governing equation solved, including velocities, continuity equation and turbulence. During the iteration procedure, residues will be monitored and recorded as the iteration process continues. In FLUENT, the residual for all parameters can be set to below 10^{-3} . The iteration process will continue until the residual for each equation falls to the prescribed value. An example of the residual plot is illustrated in Fig. 4-4.

Fig. 4-4 Example of simulation residual

Besides the residues, other considerations have to be taken to ensure convergence, such as parameters at specific faces. Normally, some important parameters, like the inlet and outlet pressure, the shear stress on the leading and trailing edge of blades, are monitored for convergence.

When the calculation has convergence difficulties, the first step is to check the grid quality. If the grid quality is too poor, a convergence problem may exist. Secondly, the boundary conditions should be examined to make sure they satisfy the actual situation. If the meshes and boundary conditions do not have any problems, it is necessary to examine the solution method. For uncomplicated models and models with convergence limited by the pressure-velocity coupling, SIMPLEC scheme can improve convergence when a suitable skewness correction is applied. Lower order discretization yields better convergence than the higher order. In order to achieve convergent and accurate results, many researchers use first order discretization for the initial computation and switch to higher order when the solution process is stable.

4.7 Grid Independency

The grid independence study is to check if the converged results obtained are independent of the grids employed. Results obtained from grid dependent solutions may be not accurate and hence erroneous design decisions can be made using the potentially faulty information. Therefore, grid independence should be viewed as an extremely important step of any CFD study.

The mesh generation technique and grid convergence were evaluated to establish grid independence using a relative error measure of hydrodynamic parameter magnitude between the coarse and fine solutions. Successive refinements of each mesh have been considered to better resolve regions of significant velocity gradients encountered in the flow domain.

There are two ways to check grid independence. In the first method, the mesh size is refined until the change of the monitored parameter is in an acceptable range. The second method is a novel method, called Grid Convergence Index (GCI). Using GCI, the exact results can be obtained from simulation results of different size. However, GCI is difficult to manipulate with unstructured meshes [154] and 3-D meshes. Hence the first method is employed.

To check grid independence, the grid adaption feature of FLUENT which is more efficient than uniformly doubling up the cell count is used. There are many possible criteria to adapt meshes. The gradient of static pressure will be utilized because the pressure distribution in the fluid domain is of key concern. Before proceeding to adapt meshes, it is advisable to save the case and data file since mesh changes are irreversible.

Grid independence can be said to be attained when further refinement of meshes yields almost no or insignificant changes in the final results. For the present grid-independent studies, three models with different counts of the meshes were checked for the grid independence, namely 0.75 million, 1 million and 1.25 million meshes separately, with the flow rate increasing from 0 l/min to 4 l/min as illustrated in Fig. 4-5. The maximum difference between 0.75 million meshes and 1.25 million meshes is 10.8%, while the value is 4.7% between 1 million meshes and 1.25 million meshes. The difference of 1 million meshes is acceptable in the simulation process and hence is selected for all subsequent studies.

Simultaneously, the computer running time of the three models is recorded and compared. The model with 0.75 million meshes takes 15.2 hours to converge, while it takes 19.8 hours and 23.5 hours for 1 million meshes and 1.25 million meshes

respectively. Taking consideration of the accuracy and running costs, 1 million meshes are selected for all future simulations.

Fig. 4-5 Grid independence check

4.8 Simulation Procedure

1. Analyze the problem and describe the problem with mathematical models, including mass, energy and momentum conservation.
2. Find fluid properties, which are constant or variable with time and space. If the fluid properties are variable, find the expression of the properties.
3. Simplify the problem with suitable assumptions, in order to make the problem trackable.
4. Define initial conditions, which will not result in backflow and limitation of other parameters.
5. Set boundary conditions based on the actual physical conditions.
6. Run calculation until the results are converged.

5. Experimental Setup

In this chapter, the test model and measurement setup will be presented. Experimental studies will be conducted to measure the performance of the pump prototype. Effects of blade geometry such as inner radius, blade angle, blade number and blade thickness will be examined. A scaled model is also fabricated and tested to study the effects of the scaling for miniature pumps. The pump characteristics will be measured and will be used to validate the CFD results. Velocity measurement using Particle Image Velocimetry will also be described.

5.1 Testing Models

7 selected models are fabricated and investigated experimentally, as shown in Fig. 5-1. The experimental results will be compared with the numerical and theoretical results. For PIV measurements, the impeller model 13 with radial blades is chosen as fabrication of the curved blades is more costly and complicated. Transparent material acrylic is selected for this model. Volute model 8 is selected to match with the fabricated impeller models. For the 10 mm diameter impeller, model 26 which is geometrically similar to model 13 is fabricated as shown in Fig. 5-2.

Fig. 5-1 Testing models of 20 mm diameter

Fig. 5-2 Testing model of 10 mm diameter

The experimental model is shown in Fig. 5-3. Both the volute and impeller are made of acrylic. The volute is made of four parts for ease of assembly. Volute 2 and 4 are symmetrical with the interface, while volutes 1 and 3 are not. This is to ensure that the interface will not overlap with the middle plane of the volute, as measurements using the laser will be made through the middle plane.

Fig. 5-3 Setup of the radial blade model

5.2 2-D PIV Measurement System

The Particle Image Velocimetry used in the present study is the DANTEC DynamicStudio PIV system as shown in Fig. 5-4. The illumination system is a Nd:YAG laser generator with Wavelength of 532 nm and maximum power of 1.8 W.

The CCD camera is the Dantec Flow Sense PIV camera with Nikon lens (Micro-Nikkor lens 60mm/2.8D) with a maximum recording frequency of 15 Hz and an image resolution of 1600×1200 pixels.

The synchronization board is used to synchronize the pulsing laser sheet and the CCD camera. The input buffer with personality module receives image maps from the camera, and the correlator unit processes image maps into vectors. The control software is the Dynamic Studio v 1.45, and the size of the interrogation area is 16×16 pixels.

Fig. 5-4 PIV measurement system

5.2.1 Tracer Particles

The selection of the tracer particles is important to ensure that clear images and accurate velocity vectors are obtained. The tracer particles should be easily visible, homogeneously and uniformly distributed and should not influence the original fluid flow. Factors such as particle density, particle size and concentration, are critical to obtaining correct data.

The particle density is mainly defined by the material selected. Some of the commonly-used particles for liquid and gas PIV measurements are listed in Table 5-1 and Table 5-2. The selected particle should be neutrally buoyant in the fluid. Hollow glass spheres with a density of 1.10 ± 0.05 g/c are selected in the present experiments. The properties of the particles are shown in Appendix.

Table 5-1 Seeding materials for liquid flows [155]

Type	Material	Mean diameter in μm
Solid	Polystyrene	10-100
	Aluminum flakes	2-7
	Hollow glass spheres	10-100
	Granules for synthetic	10-500
Liquid	Different oils	50-500
Gaseous	Oxygen bubbles	50-1000

Table 5-2 Seeding materials for gas flows [155]

Type	Material	Mean diameter in
Solid	Polystyrene	0.5-10
	Alumina Al_2O_3	0.2-5
	Titania TiO_2	0.1-5
	Glass micro-spheres	0.2-3
	Glass micro-balloons	30-100
	Granules for synthetic	10-50

The particle size is selected to ensure that the captured images can be processed easily and correctly. The diameter of the particles must be small enough not to disturb the flow being measured, but big enough to reflect sufficient light to be captured by the recording system. According to the manual of FlowMap PIV, the camera images of seeding particles should have a diameter of at least 2 pixels, and preferably 3 pixels or

more. Equation (5.1) is used to calculate the particle image size, from which the particle size d_p can be easily obtained.

$$d_{\text{image}} \approx \left[\frac{d_p^2}{S^2} + \left(2.44 \left(1 + \frac{1}{S} \right) f_{\#} \lambda \right)^2 \right]^{1/2} \quad (5.1)$$

where $f_{\#}$ is the recording aperture, S is the scalar factor, and λ is the light wavelength.

The particle concentration should be uniform and should be in an appropriate range. When the particle concentration is too low, the flow field cannot be represented accurately, while when the particle concentration is too high, it is difficult to observe individual particles in an image. In order to have a good signal-to-noise ratio, a minimum of 5 particles per interrogation area of the cross-correlation should be observed. The signal-to-noise ratio of the image is defined as the ratio of the peak image intensity of an average in focus particle, divided by the average background intensity. A good ratio indicates that the magnitude of the signal should be sufficiently larger than the noise.

5.2.2 Illumination System

The illumination system includes pulsed lasers, laser shutters and lens system. The lasers used are doubled-pulsed Nd:YAG laser, which are sensitive to HDD cameras. The specifications of the lasers are shown in Table 5-3.

Table 5-3 Laser specifications

Variables	Specification
Max output	1200 ms
Pulse duration	4 ms
Wave length	532 nm
Laser medium	Nd:YAG

The power of the laser sheet has a great impact on the measurement, because a higher power can illuminate the particles clearly, and make them sufficiently bright to be captured by the high-speed camera. However, if the laser power is too large, the particles will be too bright and the high-speed camera cannot differentiate their exact position, causing measurement error. The light power is controlled by the laser control panel as shown in Fig. 5-5. It is regulated to ensure that the intensity of the scattered light from the seeding particle can be captured by the HDD camera clearly. When the PIV experiment is started, the power of the laser is set at a low level. Initial testing image is conducted to ensure clear images and data is recorded. Adjustments are carried out until satisfactory results are obtained.

The repetition rate should be such that the flow field does not change significantly. It can be controlled by the control panel (Repetition rate as shown in Fig. 5-5) or the software. The repetition rate is set as 15 Hz in the present experiment.

Fig. 5-5 Control panel for PIV system

For the present system, two light pulses are fired in one flash lamp pulse, which is Q-switched in order to generate sufficient high-energy as shown in Fig. 5-6. The time interval between the double pulses should be adjusted to satisfy both space and time

conditions. For space considerations, the particle image displacement ΔX during the time interval should be in the interval given by the resolution of the system and the maximum allowable particle displacement. With regard to time consideration, the power of the second pulse should be high enough to illuminate tracking particles, which depends on the time interval to accumulate enough energy. To accumulate enough energy for the second pulse, the YAG-crystal requires a minimum time interval to receive energy from the flash lamp after the first pulse. The minimum time interval 50 μs is used in the present experiment.

Fig. 5-6 Double-Q-switching mechanism [156]

The duration of the light pulse should be sufficiently short to ensure that the particles do not move a significant distance as compared to the particle size [157]. The spherical tracking particles will appear to be elliptical in the captured images if the light pulse is fired too long. The time duration is about 5-10 ns in the present experiment.

5.2.3 CCD Camera

The CCD camera system includes Dantec Dynamic Flowsense 2Megapixels PIV camera and Nikon AF Micro-Nikkor lens 60mm /2.8D. The maximum recording frequency is 15 double-frame images with an image resolution of 1600×1200 pixels. The camera system is mainly used to capture the initial and final positions of particles,

from which the velocity vector can be calculated. An example of the pulse captured images from a CCD camera is shown in Fig. 5-7. The CCD camera is synchronized with the illumination system using the DynamicStudio processor.

(a)

(b)

Fig. 5-7 Image pairs captured by the CCD camera

Fig. 5-8 Flowsense 2M camera

5.2.4 Measurement Principle

The PIV system consists of seeding particles, a double-pulsed laser sheet, light sheet lens, a high-speed camera, an image digitization hardware and a computer for data storage and analysis. The procedure for velocity field measurement using PIV technology is illustrated in Fig. 5-9.

Fig. 5-9 Elements and processes in a 2-D particle image velocimeter system [158]

The Particle Image Velocimetry used in the present study is the DANTEC Dynamic Studio PIV system. This system consists of an illumination system, a CCD camera, the Dynamic Studio system hub (including correlator units, input buffers and synchronization unit) and a PC with the software installed (Dantec Dynamic Studio V1.45). The synchronization unit connects the illumination system and the CCD camera, in order to synchronize the pulsed light sheet and the recording of the camera images.

The laser generated by the illumination system is reflected and converted into a laser sheet by lenses. The tracking particles are illuminated by the laser sheet in a dark room where the test rig is located. The illuminated particles are captured by a CCD camera during each light pulse in separate image frames. The two image frames are compared and analyzed to obtain the velocity field. Once a sequence of two light pulses is recorded, the images are divided into small subsections called Interrogation Areas (IA). The interrogation areas of each image frame are cross-correlated with each other pixel by pixel. The correlation produces a signal peak, identifying the common particle displacement ΔX . The velocity vectors can be derived based on the particle displacement ΔX and the pulse duration Δt as given in equation (5.2).

$$\bar{V} = \frac{\Delta X}{\Delta t} \quad (5.2)$$

A velocity vector map over the whole target area is obtained by repeating the cross-correlation for each interrogation area over two image frames captured by the camera.

5.2.5 Experimental Procedure

1. Adjustment of the CCD camera

Place the camera on a tripod and adjust the height to match the laser and expected objectives. Zoom and focus the camera manually until a clear view is obtained in the desired plane. Refocus the camera on the particles to minimize the particle size on the images. Select a suitable aperture; if necessary compensate the loss light by increasing the laser power.

2. Calibration and scaling

A ruler is fastened in the same plane of the laser sheet. With the flow system exposed to room light, focus the camera on the ruler and capture an image of the ruler for calibration purpose in the DynamicStudio software. Then scale and specify two points in the image, input the ruler value, and click scale. Save the calibration. If the acquired images are distorted, the images need a correction based on a spatial calibration. All the captured size in the future PIV measurement will be scaled to the actual size.

3. Adjustment of lighting path

All lights in the experimental room are switched off, and all metal surfaces which will contact laser are painted black. The position and angle of the light lens are adjusted to obtain a light sheet in the desired location. In the present case, the light sheet should overlap the middle plane of the impeller blades.

4. Adjustment of lighting power

The power of the laser sheet affects the measurements significantly. Increase the power from a low level gradually, until the particles in captured images are clear.

5. Seeding the fluid

Select particles material, size and concentration. Add the particles into water, and mix them using an ultrasonic vibration machine. Pour some of the mixed fluid into the reservoir, and add more water into the reservoir, in order to dilute the mixed fluid to a

suitable concentration. Switch the camera in double frame model, and take an image with low seeding density. Adjust the concentration until suitable results are obtained.

6. Running motor and setting DynamicStudio software

Set the speed of the motor at desired speed (112.5 rad/s) and stir the water reservoir in order to mix tracking particles evenly. Open the DynamicStudio software, and create a new project. Connect the laser and CCD camera and make sure that they are synchronized. Double-frame is selected, and 50 μ s is set as the time interval. 100 images are collected.

7. Processing data

Cross-correlation is selected to process raw vector maps. Interrogation areas are set as 32 pixels $\times$ 32pixels, and horizontal and vertical overlap is 50%. Filtering is used to improve the signal to noise ratio. The processed velocity vectors can be exported numerically for further process.

5.3 Pump Performance Measurement System

The front and back view of the experimental setup is shown in Fig. 5-10. The prototype is driven by a motor (Maxon EC 118889). The pressure generated by the pump is measured by the pressure transducer (Setra 230), and the voltage signal of the transducer is converted into pressure by an INFINITY C Process meter (INFCP1) with a reading range of 0 to 9999 Pa. The flow rate is measured by a flowmeter (ABB FLX 5000). The flow field is measured using the Dantec PIV system DynamicStudio V 1.45. The laser is released from the laser generator and reflected by the mirror into the laser beam diverging lens to produce the laser sheet. The CCD camera was located beside the tested pump to capture the flow illuminated by the laser sheet.

Front view

Back view

Fig. 5-10 Prototype of the front and back view of the testing rig

The closed flow loop is designed and fabricated as shown in Fig. 5-11. The models, water tank and measurement facilities are connected with PVC pipe FEP flexible pipe. Two valves are located at the upstream and downstream of the water reservoir. When

the experiment is initiated, the upstream valve is completely open and the downstream valve is used to control the flow rate. The flow rate is zero when the downstream valve is completely closed. The flow rate is gradually increased with the opening of the downstream valve.

To obtain the average pressure between the pump inlet and outlet, four measurement points are drilled evenly into the inlet and outlet pipe respectively, and then connected with a cylinder. The measured flow rate signal is recorded on a PC, and the averaged value is taken after 1 minute.

Due to the losses in the system, the designed pumps can only operate up to a certain flow rate. Taking model 13 in Table 3-7 with radial blade for example, a maximum measured flow rate of 1.7 l/min, with a pressure rise 448 Pa can be recorded. To measure the complete characteristics of the designed pumps, an additional pump is added into the system to overcome the system losses at a higher flow rate. This allows us to measure the complete pump characteristics. Hence, a whole pump performance curve can be obtained, with the lowest flow rate of 0 l/min and the lowest pressure rise is 0 Pa.

The experimental procedure is as follows:

1. Before the experiment is started, the reservoir is completely filled with water.
The upstream valve is completely open and the downstream valve is closed.
The motor, flow meter and pressure transducer are then calibrated.
2. Switched on the power supply of the flow meter, motor, and ensure that the readings of flow meter and pressure transducer are zero.
3. Set the motor at the design running speed, which is connected and controlled by a PC.

4. Read and record the pressure and flow rate; open the downstream valve slightly and record the reading of corresponding pressure and flow rate.
5. Open the downstream valve gradually and repeat step 4 until the downstream valve is completely open.
6. The reverse sequence is then performed. The downstream valve is closed gradually and measurements are made until the downstream valve is completely closed.
7. Repeat steps 4 to 6 for 5 times to ensure that the results are consistent and repeatable.

Fig. 5-11 Actual testing rig

6. Results and Discussions

In this chapter, the CFD results of impeller models in Table 3-6 and Table 3-7, as well as volute models in Table 3-10 and Table 3-11 are presented to demonstrate the effects of the geometry of impeller and volute on the performance of miniature centrifugal pumps. The impeller models are summarized in Table 6-1. Theoretical and experimental investigations on selected models are also conducted.

Table 6-1 Geometrical parameters and conditions of models

No.	ϕ_e	$r_2(\text{mm})$	$r_1(\text{mm})$	$\beta(^{\circ})$	Z	$t(\text{mm})$	Examine effect of()
1	3.49	20	3	74	5	0.4	ϕ
2	2.99	20	3.16	72	5	0.4	ϕ
3	2.49	20	3.36	68	5	0.4	ϕ
4	1.99	20	3.62	63	5	0.4	ϕ
5	1.47	20	4	56	5	0.4	$\Phi, \beta, Z, \text{scale}$
6	1.29	20	4.18	53	5	0.4	ϕ
7	0.90	20	4.72	42	5	0.4	ϕ
8	0.75	20	5	37	5	0.4	ϕ
9	0.44	20	6	24	5	0.4	ϕ
10	0.29	20	6.88	17	5	0.4	ϕ
11	0.44	20	4	24	5	0.4	β
12	1.99	20	4	63	5	0.4	β
13	∞	20	4	90	5	0.4	β, t, scale
14	∞	20	4	90	5	0.2	t
15	∞	20	4	90	5	0.6	t
16	∞	20	4	90	5	0.8	t
17	1.47	20	4	56	3	0.4	Z
18	1.47	20	4	56	7	0.4	Z

19	1.47	20	4	56	9	0.4	Z
20	1.47	20	4	56	11	0.4	Z
21	1.47	20	4	56	13	0.4	Z
22	2.99	10	1.58	72	5	0.2	ϕ
23	0.29	10	3.44	17	5	0.2	ϕ
24	1.47	10	2	56	5	0.2	$\phi, \beta, scale$
25	1.99	10	2	63	5	0.2	β
26	∞	10	2	90	5	0.2	$\beta, scale$
27	1.47	10	2	56	11	0.2	Z
28	1.47	10	2	56	13	0.2	Z

Results obtained from the theoretical, numerical and experimental studies will be presented sequentially. A comparison of the results will then be made next. The effect of geometrical parameters and flow coefficient on the pump performance will be discussed. Comparison of the impeller designed using conventional macro-size techniques and the modified impellers will be made. Highlight of the inadequacies of the original impeller will be presented. The effect of the volute geometry will also be discussed. Lastly, the flow field in original and modified impeller will be compared.

The impeller models are operating under steady state and water at ambient conditions is used as the test fluid. The flow field distributions will be presented for the middle plane. For the 20 mm diameter and 10 mm diameter models, the geometrical parameters are as discussed in Chapter 3. Model 26, together with models 2, 5, 10, 12, 13, 20, 21 are fabricated and investigated experimentally, as listed in Chapter 5. The operating conditions of 20 mm diameter models and 10 mm diameter models are listed in Table 6-2. Model 13 with radial blades is selected to do PIV measurement. The experimental results will be used to validate the numerical results.

Table 6-2 Design requirements of models

Variables	10 mm diameter models	20 mm diameter models
Outer radius	10 mm	5 mm
Designed flow rate	2 l/min	1 l/min
Rotational speed	112.5 rad/s	450 rad/s

Dimensional analysis is an important and effective method to simplify the analysis process, especially to estimate the design and performance changes between two pumps of similar design. To estimate the pump performance, the variables including flow rate and pressure generated should be selected, which are the flow coefficient and pressure coefficient. The definitions of the two variables are given in equations (2.4) and (2.5). The flow coefficient and pressure coefficient of 20 mm diameter models together with the corresponding 10 mm diameter impeller models will be calculated based on the numerical and experimental data. The 10 mm diameter models at a speed of 450 rad/s and 20 mm diameter models at a speed of 112.5 rad/s and 225 rad/s are analyzed and compared for selected models.

6.1 Analysis of Uncertainty

Error bars are used on graphs to indicate the error or uncertainty in a measurement. They give a general idea of how accurate a measurement is, or how far the reported data is away from the true value.

The standard uncertainty can be obtained where errors from the source will be contained within stated limits. For uniformly distributed errors contained within symmetrical tolerance offsets of $\pm L$, the standard uncertainty U_s can be found by the following equation [159]:

$$U_s = \frac{L}{\sqrt{3}} \quad (6.1)$$

The pressure is measured by the pressure transducer Setra 230 with an accuracy of 0.25% of full scale (± 86 Pa). The flow rate is measured by a flowmeter ABB FLEX 5000, with an accuracy of 0.21% of full scale (± 0.021 l/min). The uncertainties of the pressure and flow rate measurements are calculated as:

$$U_p = \frac{86}{\sqrt{3}} = 50 \text{ Pa} \quad (6.2)$$

$$U_Q = \frac{0.021}{\sqrt{3}} = 0.012 \text{ l/min} \quad (6.3)$$

The numerical simulation is repeated for three times on the 20 mm diameter radial model at speeds of 112.5 rad/s and 225 rad/s. The simulation results are almost identical for the 3 trials.

6.2 Pump Performance Predicted by Theoretical Model

The energy transferred from the motor to the fluid can be theoretically analyzed based on Euler's head and energy losses, as described in Chapter 2. In addition, slip loss, leakage loss, tip leakage loss, disk friction loss, impeller friction loss, shock loss, blockage loss and diffusion loss are taken into consideration. The calculation equations are summarized in Table 6-3. In this section, each of the energy losses will be investigated, to find out the effect of them and which can be neglected. This can guide future work on theoretical analysis of miniature pumps.

Table 6-3 Energy equations

Energy	Equation
Euler head	$E_{th} = \frac{1}{4g} D_2^2 \omega^2 - \frac{\omega Q}{2\pi g b_2} \cot \beta$
Slip loss	$E_{slip} = \frac{U_2^2}{g \left(1 + 5 \frac{(1 - r_1/r_2)Z}{2\pi (\sin \beta)^{0.5}} \right)}$
Leakage flow rate	$Q_{lk} = \frac{\pi \delta^3}{6\mu \ln(r_2/r_1)} \left(\Delta P - \frac{3\rho \omega^2 (r_2^2 - r_1^2)}{20} \right)$
Tip leakage loss	$E_{tip} = \frac{\omega Z \delta (r_2 - r_1) \cot \beta}{\pi g b_2} \sqrt{\frac{Q\omega}{b_2 Z \sin \beta}}$
Disk friction loss	$E_{df} = \frac{1}{2} \frac{C_f \omega^3 r_2^5}{g Q_{lk}} \left(1 - \left(\frac{r_1}{r_2} \right)^5 \right)$
Wall friction loss	$E_{wf} = C_f \frac{(r_2 - r_1)(W_2 + W_1)^2}{4g D_h \sin \beta}$
Blockage loss	$E_{blk} = 0.45 \left(\frac{D_2}{D_1} - 1 \right) \frac{V_1^2}{2g}$
Diffusion loss	$E_{diff} = \frac{0.25 W_1^2}{2g}$
Shock loss	$E_{shock} = \frac{0.3}{g} (W_1 - V_1)^2$
Theoretical head	$E = E_{th} - E_{slip} - E_{lk} - E_{tip} - E_{df} - E_{wf} - E_{blk} - E_{diff} - E_{shock}$

In the calculation of the leakage flow, the pressure difference ΔP between the trailing and leading edges is unknown. This will cause uncertainty in the determination of the leakage loss and disk friction loss. An iteration method is employed. The theoretical pressure without leakage loss and disk friction loss is used as an approximation to predict the leakage flow. The theoretical pressure is updated to provide a second approximation and the difference between the initial value and the revised value is monitored. The process continues until the difference is less than 5%.

The energy losses versus blade angle are illustrated in Fig. 6-1 and Fig. 6-2 for 20 mm

and 10 mm diameter models respectively. A very small blade angle is not desirable because of high energy loss and low theoretical head. So the blade angle range is selected from 30° to 90° . For the 20 mm diameter impeller models, the impeller inner radius is fixed as 4 mm while the flow rate is 2 l/min and rotational speed is 112.5 rad/s. The disk friction loss, wall friction loss, leakage loss and tip leakage loss are so low that they can be neglected. Taking the 80° blade angle for example, the values of the four losses are 1.4 Pa, 5.0 Pa, 7.2 Pa and 7.2 Pa respectively. The theoretical head, Euler head, slip loss and blockage loss increase with increasing blade angle, while the diffusion loss and shock loss decrease with increasing blade angle. The slip loss, diffusion loss and blockage loss account for the three most important losses, while the shock loss is significant when the blade angle is smaller than 45° . The shock loss is 9% of the theoretical head when the blade angle is 45° and is 70% when the blade angle is 30° .

Fig. 6-1 Energy losses versus blade angle for 20 mm diameter impellers at 112.5 rad/s

For the 10 mm diameter impeller models as shown in Fig. 6-2, the inner radius is 2 mm, the flow rate is 1 l/min and the rotational speed is 450 rad/s. Compared with the results of the 20 mm diameter models Fig. 6-1, the disk friction loss is a little higher, and the trends of other losses are similar. It is indicated that the friction loss is higher in small models. The wall friction loss, leakage loss and the tip leakage loss are still negligible. 20 mm and 10 mm diameter models with a blade angle of 90° and 56° have been selected to examine the effects of radius ratio and flow rate.

Fig. 6-2 Energy losses versus blade angle for 10 mm diameter impellers at 450 rad/s

The energy losses versus radius ratio r_1/r_2 for 90° and 56° blade angle of 20 mm diameter models are listed in Fig. 6-3 and Fig. 6-4. It is observed that, the slip loss is the most significant and increases with radius ratio. The wall friction loss, tip leakage loss and shock loss are negligible. The disk friction loss, blockage loss and diffusion loss are considerable when the radius ratio is small. They decrease with increasing

radius ratio. This is because a smaller radius ratio means a larger blockage at the impeller inlet and a longer impeller channel, thus causing larger wall friction loss, blockage loss and diffusion loss. For the radial impeller, the leakage loss is negligible, while for the 56° impeller, the leakage loss increases with radius ratio. The leakage flow rate is 15% of the total flow rate when the radius ratio is 0.4 and is 52% when the radius ratio is 0.9. The theoretical head is a maximum when the radius ratio is 0.54 for both 90° and 56° impellers. It is indicated that an optimized value of the radius ratio exists.

Fig. 6-3 Energy losses versus radius ratio for 20 mm diameter radial impeller at 112.5 rad/s

Fig. 6-4 Energy losses versus radius ratio for 20 mm diameter impeller with blade angle 56° at 112.5 rad/s

The effect of the r_1/r_2 for 90° and 56° blade angle of 10 mm diameter models is shown in Fig. 6-5 and Fig. 6-6. Similar to the 20 mm diameter models, the slip loss is the most significant loss. The wall friction loss, tip leakage loss and shock loss are negligible. The disk friction loss, blockage loss and diffusion loss decrease with radius ratio. Leakage loss is very significant for the 56° model when the radius ratio is large. The optimum value of the radius ratio to achieve the highest pressure generated is 0.54 which is same as the 20 mm diameter model.

Fig. 6-5 Energy losses versus radius ratio for 10 mm diameter radial impeller at 450 rad/s

Fig. 6-6 Energy losses versus radius ratio for 10 mm diameter impeller with blade angle 56° at 450 rad/s

The effects of flow rate on the pressure generated are shown in Fig. 6-7 and Fig. 6-8. As expected, the theoretical head decreases with increasing flow rate. The disk friction loss, blockage loss and diffusion loss increase with the flow rate, while the wall friction loss, shock loss, leakage loss and tip leakage loss are negligible. The slip loss is independent of the flow rate. This is mainly because that, in the calculation equation of slip loss, no variable related to flow rate exists. This phenomenon is consistent with many reference papers using theoretical prediction. However, according to the principle of the slip phenomenon, the flow rate will affect the velocity passing through the impeller channel, and thus the slip loss. This phenomena is proved by the experimental study of Memardezfouli [96].

Fig. 6-7 Energy losses versus flow rate for 20 mm diameter radial impeller at 112.5 rad/s

Fig. 6-8 Energy losses versus flow rate for 20 mm diameter impeller with blade angle 56° at 112.5 rad/s

Fig. 6-9 Energy losses versus flow rate for 10 mm diameter radial impeller at 450 rad/s

Fig. 6-10 Energy losses versus flow rate for 10 mm diameter impeller with blade angle 56° at 450 rad/s

For the 10 mm diameter models, the effect of the radius ratio on energy losses is similar to the 20 mm diameter models, as shown in Fig. 6-9 and Fig. 6-10. The disk friction loss, blockage loss and diffusion loss increase with the flow rate, while the wall friction loss, shock loss, leakage loss and tip leakage loss are negligible. The disk friction loss is significant with a smaller blade angle and higher flow rate.

Based on the above analysis, some of the energy losses can be neglected in the analysis procedure, like wall friction loss and tip leakage loss, while the slip loss is the most significant one. The blockage loss, diffusion loss and disk friction loss increase with flow rate, and decrease with the radius ratio. The leakage loss is only considered when the radius ratio is very large, and the shock loss is considered when the blade angle is very small. This will guide the future energy analysis of the miniature pumps. The theoretical head increases with the blade angle, decreases with the flow rate. In

addition, there is an optimal radius ratio where the pressure generated is a maximum. This value is the same for both the 20 mm and 10 mm diameter models, which is 0.54. The slip loss is the most significant loss for typical blade angle and radius ratio. A large blade angle and moderate value of radius ratio is desirable to have a high pressure generated.

6.3 Pump Performance Predicted by Numerical Studies

The turbulent models are investigated in this section, and suitable models will be selected for future research. CFD validation of the pump performance and similarity will also be discussed in this section.

6.3.1 Turbulent Models

Different turbulent models are verified for the 20 mm diameter radial models when the rotational speed is 225 rad/s, which has a design flow rate of 4 l/min. As discussed in Chapter 4, for 20 mm diameter models running at 112.5 rad/s and 10 mm diameter models running at 450 rad/s, the Reynolds number is 45,000, and a laminar model is selected. However, for the 20 mm diameter models running at a speed of 225 rad/s, the flow is neither laminar nor fully turbulent. Hence, it is necessary to select a suitable turbulence model to numerically simulate the flow. Three turbulent models, namely, laminar, SST $k-\omega$ and Reynolds Stress Models are selected to test their suitability. The pump performance curves obtained using these models, together with the experimental results are shown in Fig. 6-11. It is noted that the pressure generated using the SST $k-\omega$ model is the lowest, and differs significantly from the experimental results. Hence, for subsequent studies of the 20 mm diameter models, the SST $k-\omega$ model is not used. The pump characteristics obtained using the laminar and Reynolds Stress Models have similar trend with magnitudes comparable with the experimental results. When the

flow rate is less than 2 l/min, it is observed that these 2 models over predict the pressure generated. For flow rates much smaller than the design flow rate, a large zone of flow separation is expected and the numerical models cannot simulate the flow accurately. The simulation results as shown in Fig. 6-11 show that the results from both Reynolds Stress Model and laminar model are reasonable. When 3-D double precision and 4 processors are used in running FLUENT 14.0, it will take about 20 hours to run the Reynolds Stress Model.

Fig. 6-11 Different turbulent models for 20 mm diameter models with radial blades

The discrepancy between CFD results and experimental data in Fig. 6-11 is due to the fact that at very low flow rates, large scale flow separation is expected within the impeller channels and numerical simulations may not be able to simulate the losses accurately. For very low flow rates near to the shut off point, it is observed that the

pressure generated drops and then remains almost constant when the flow rate is less than 3 l/min. This is a characteristic of radial impellers operating at low flow rates [20].

The relative velocity field of the Reynolds Stress Model and laminar model are verified in order to differentiate them, as shown in Fig. 6-12 and Fig. 6-13. The rotational direction is in the clockwise direction. The result from the laminar model is more complex with more vortices in the impeller channels, especially in the downstream zone near the pressure side. This explains the relatively poor accuracy of the laminar model for larger flow rates. For Reynolds Stress Model, a stable separated region or wake on the suction side and a main stream or jet near the pressure side of the blades exists and this is called a jet-wake flow. This phenomenon is observed in the impeller channels near the suction side. Hence, the Reynolds Stress Model is selected for the 20 mm diameter model running at 225 rad/s.

Fig. 6-12 Relative velocity distribution of radial impeller with Reynolds Stress Model

Fig. 6-13 Relative velocity distribution of radial impeller with laminar model

6.3.2 Pump Performance Study

The performance curves of the 20 mm and 10 mm diameter models are characterized numerically in Fig. 6-14 to Fig. 6-16. For the 20 mm diameter models, two speeds, 112.5 rad/s and 225 rad/s are selected, while for the 10 mm diameter models, the rotational speed is 450 rad/s. The design flow rate for 112.5 rad/s of 20 mm diameter model is 2 l/min, while 4 l/min for the 225 rad/s and 1 l/min for the 10 mm diameter model running at 450 rad/s.

Fig. 6-14 Numerical results of pump performance of 20 mm diameter models at 112.5 rad/s (Laminar model)

The pressure generated decreases with flow rate, while the performance curve is higher with larger blade angle at the design point. The model with 90° blade angle generates the highest pressure, while the model with blade angle 17° generates the lowest. However, when the flow rate is far away from the design point, the radial impeller does not have a better performance. For the radial impeller model, the pressure generated decrease with flow rate. This phenomenon agrees with the recorded pump performance of macro-size pumps. The 20 mm diameter radial impeller at a speed of 225 rad/s has a very significant high pressure generated when the flow rate is larger than 4 l/min. The difference between Fig. 6-14 and Fig. 6-16 is that the models with 72° and 90° have highest pressure for 20 mm diameter models, while the model with 56° is superior when the flow rate is larger than 1.3 l/min for 10 mm diameter

models. This is mainly because when the blade angle is large, the fluid is hard to be guided by the blade, and causes more slip loss.

Fig. 6-15 Numerical results of pump performance of 20 mm diameter models at 225 rad/s (Reynolds Stress Model)

The radial impeller is not always the best selection in order to have the best performance. The curve of the radial impeller crosses other curves, indicating that the radial impeller is superior near the design point. The larger blade angle will generate large flow separation and vortices, causing higher energy losses. So a suitable value of blade angle should be chosen based on the requirements of the pressure and flow field.

Fig. 6-16 Numerical results of pump performance of 10 mm diameter models at 450 rad/s (laminar)

6.3.3 Pump Similarity

The pump similarity studies of the 20 mm diameter model at 112.5 rad/s and 225 rad/s, together with the 10 mm diameter model at 450 rad/s are shown in Fig. 6-17 to Fig. 6-21. For the radial models, the difference between the three curves are not significant, while for other models, the 20 mm diameter model at a speed of 225 rad/s is lower than the other two curves. The difference increases with the decrease of blade angle. The difference between the 20 mm diameter model at a speed of 112.5 rad/s and the 10 mm diameter model at a speed of 450 rad/s is not significant for all 5 models. Generally speaking, it can be concluded that for the miniature pump with the impeller size in the order of centimeters, pump similarity law is quite well expected, but different speeds may yield slightly different results. This conclusion will be demonstrated by the experimental results in the next section.

Fig. 6-17 Numerical results of pump similarity of models with radial impeller

Fig. 6-18 Numerical results of pump similarity of models with blade angle 72°

Fig. 6-19 Numerical results of pump similarity of models with blade angle 63 °

Fig. 6-20 Numerical results of pump similarity of models with blade angle 56 °

Fig. 6-21 Numerical results of pump similarity of models with blade angle 17°

6.4 Pump Performance (Experimental Studies)

The experimental results will be investigated in this section. The measurement method has been presented in chapter 5.

6.4.1 Effect of Blade Geometry on Pump Performance

The pressure generated and flow rate measured for models 2, 5, 10, 12 and 13 are illustrated in Fig. 6-22 and Fig. 6-23. Due to the limitation of the measurement range of the flow meter, flow rates less than 0.5 l/min cannot be captured. Similar to the numerical results, the radial impeller generates the highest pressure at the design point and the pressure generated has a decreasing trend. These are similar to the numerical results. The experimental results are smooth except for the model 10 with blade angle 17° . An obvious drop of the pressure is noticed when the flow rate is around 1.2 l/min for 112.5 rad/s and 2.4 l/min for 225 rad/s. This phenomenon is mainly caused by the choked flow in the impeller passage.

Fig. 6-22 Experimental results of pump performance of 20 mm diameter models at 112.5 rad/s

Fig. 6-23 Experimental results of pump performance of 20 mm diameter models at 225 rad/s

Choked flow occurs when the fluid is obstructed when passing through restrictions, causing a high velocity and low pressure. When the pressure is below the liquid vapor

pressure at the corresponding temperature, the liquid will partially flash into vapor and cavitation appears. In this situation, increasing the pressure or flow rate will not have an effect on the other. The choked flow has a significant damage to the equipment, so it should be avoided in the application.

When the impeller model 10 is running at 225 rad/s with a flow rate of 2.26 l/min, the pressure at the narrowest point of the impeller channel is same as the water vapor pressure, which is 3172 Pa. This flow rate agrees with the experimental value as in Fig. 6-23.

6.4.2 Pump Similarity

The pump similarity of the radial model at 112.5 rad/s and 225 rad/s is listed in Fig. 6-24. The experimental data show that the trends of the three curves are similar. The magnitude of the 20 mm diameter model at 112.5 rad/s is different from the 10 mm diameter model at speed 450 rad/s and 20 mm diameter model at 225 rad/s, indicating that both the size and the speed do not follow the similarity law in the centimeter scale.

The similarity analysis of other models is listed below in Fig. 6-25 to Fig. 6-28. Similar to the radial blade model, the 20 mm diameter model running at 112.5 rad/s is higher than the speed of 225 rad/s. The smaller the blade angle, the smaller is the deviation between the two speeds. For the 17° model, they almost overlap, and the similarity law is effective. This is mainly because the smaller the blade angle, the flow is more close to laminar flow, and the flow separation is less serious, and similarity may be effective. Taking the 17° model for example, the flow field is smooth without any flow separation as in Fig. 6-50.

Fig. 6-24 Experimental results of pump similarity of radial models 13 and 26

Fig. 6-25 Experimental results of pump similarity of impeller model 2 with blade angle

72°

Fig. 6-26 Experimental results of pump similarity of impeller model 12 with blade angle 63°

Fig. 6-27 Experimental results of pump similarity of impeller model 5 with blade angle 56°

Fig. 6-28 Experimental results of pump similarity of impeller model 10 with blade angle 17°

6.5 Comparison of the Theoretical, Numerical and Experimental Results

The performance curves of the 20 mm and 10 mm diameter models with radial blades (model 13 and 26) are characterized theoretically, numerically and experimentally.

The pump performance curves for the 20 mm radial model running at a speed of 112.5 rad/s is illustrated in Fig. 6-29. The trends of the experimental, numerical and theoretical results are similar, while the magnitude of the theoretical pressure is higher than the other two. This is because some energy losses are not considered, including energy losses in the inlet and volute zone. The numerical pressure is a little higher than the experimental pressure, and this deviation decreases with increasing flow rate. At the design point when the flow rate is 2 l/min, the pressure generated for the experimental, numerical and theoretical results is 536 Pa, 581 Pa and 606 Pa respectively. However, the designed pressure as in chapter 3 is 562.5 Pa. The differences between the designed value and the experimental value, and between the

designed value and numerical value are 5% and 3%, indicating that the approach adopted is sufficiently effective. The results show that the numerical and theoretical model can be used to predict the trend of the pump performance.

Fig. 6-29 Pump performance of 20 mm diameter radial model (model 13) at 112.5 rad/s

The performance curve for the 20 mm diameter radial model at a speed of 225 rad/s is illustrated in Fig. 6-30. The numerical and the experimental results are very similar, indicating that the numerical results can be used to predict the magnitude and trend of the pump performance. When the rotational speed is doubled from 112.5 rad/s to 225 rad/s, the pressure generated is supposed to be 4 times of the low-speed one, which is 2250 Pa with a double flow rate. From Fig. 6-30, the experimental and numerical pressures at the flow rate of 4 l/min are 1842 Pa and 1645 Pa. The difference is 18% between the designed and experimental pressure. It can be concluded that when the speed is changed, the measured pressure will be far away from the designed value. This is mainly caused by the inapplicability of the similarity law at different rotational speeds, as discussed in section 6.3.

Fig. 6-30 Pump performance of 20 mm diameter radial (model 13) at 225 rad/s

The pump performance curves for the 10 mm diameter radial model are illustrated in Fig. 6-31. The trends of the three data are similar to the model, while the magnitude of the theoretical results is higher than the experimental and numerical results.

Fig. 6-31 Pump performance of 10 mm diameter model with radial blades

The Reynolds number of the 10 mm diameter model operating at the speed of 450 rad/s will be same as the 20 mm diameter model at the speed of 112.5 rad/s, indicating that the two size pumps should follow the affinity law. The flow coefficient and pressure coefficient will be the same, and the flow rate and pressure generated should follow the equations. The design point for the 20 mm diameter model is 2 l/min with a pressure rise of 562.5 Pa as in chapter 3. The corresponding flow rate and pressure generated are 1 l/min and 2250 Pa for the 10 mm diameter model. The experimental pressure rise is 1091 Pa at a flow rate of 1 l/min. The difference between designed and experimental pressure is 52%, indicating that the macro-size design point cannot be achieved and the macro-size pump design method is not applicable for the 20 mm diameter model.

6.6 Effect of Blade Geometry on Pump Performance

6.6.1 Effect of the Blade Angle on Pump Performance

The blade angle plays an important role in the pump design. To investigate the effect of the blade angle, other parameters such as flow rate, rotational speed, blade inner radius, blade number and blade thickness are fixed as in Table 6-4. The effect of blade angle is shown in Fig. 6-32 and Fig. 6-33. For the 20 mm diameter models as shown in Fig. 6-32, the pressure generated increases with blade angle. For the current study, it is a maximum when the blade angle is 90° . It is noted that the theoretical analysis and the numerical simulation agree reasonably well with the experimental data. The trends of the three curves are similar, while the magnitude of the numerical and theoretical results is higher than the experimental results. This is mainly because the energy losses in the pump inlet and volute have not been considered for the numerical and theoretical prediction. Moreover, the surface roughness of the model used in the

experiments also causes relatively inaccurate prediction of the turbulent model for low Reynolds number flow in the impeller. For the 10 mm diameter models shown in Fig. 6-33, the theoretical pressure is higher than numerical and experimental results.

Table 6-4 Flow conditions and geometrical parameters

Model	Flow rate Q	Speed ω	Inner radius r_1	Number Z	Thickness t
20 mm	2 l/min	112.5 rad/s	4 mm	5	0.4 mm
10 mm	1 l/min	450 rad/s	2 mm	5	0.2 mm

Fig. 6-32 Effect of blade angle of 20 mm diameter models

Fig. 6-33 Effect of blade angle of 10 mm diameter models

Nevertheless, some conclusions can be drawn from these results. In general the pressure generated increases with blade angle, but the rate of increase decreases with increasing blade angle. The theoretical prediction of 10 mm diameter models is much higher than the numerical and experimental results. From the present results, the radial blade model generates the highest pressure generated. This is consistent with Euler pump theory. However, it is expected that flow separation will occur in the blade passages with larger blade angle. This is due to the increasing difference between the blade angle and the inlet flow angle. Furthermore, the performance of the impeller running at off-design point will be different, causing a disadvantage for models with larger blade angle. Hence, the selection of the appropriate blade angle depends on the nature of application and requirements.

6.6.2 Effect of the Blade Inner Radius on Pump Performance

The blade inner radius r_1 is also an important parameter that affects the pump performance, and its effect is illustrated in Fig. 6-34 and Fig. 6-35. The impellers used are radial with flow rate at design point. For the 20 mm diameter models, the flow rate is 2 l/min while 1 l/min for 10 mm diameter models. Both numerical and theoretical results showed that an optimized inner radius exists. For the 20 mm diameter models, the value is 4 mm for the numerical simulation, while it is 5.4 mm for the theoretical results. Experiments are conducted on model 13, and the measured pressure is 10% lower than the numerical pressure. The result of the 10 mm diameter models is illustrated in Fig. 6-35. The optimum radius for numerical results is 2 mm while it is 2.7 mm for theoretical prediction. The experimental pressure generated of model 26 is almost the same as the numerical result. This indicates that our numerical simulations can give reasonably accurate predictions. For both of the 20 mm and 10 mm models, the optimum radius ratio r_1/r_2 is 0.4 and 0.47 numerically and experimentally.

Fig. 6-34 Effect of blade inner radius r_1 of 20 mm diameter models

Fig. 6-35 Effect of blade inner radius r_1 of 10 mm diameter models

6.6.3 Effect of the Blade Thickness on Pump Performance

The effect of blade thickness is shown in Fig. 6-36 and Fig. 6-37 and all the models have radial blades. For miniature pump, blade thickness will affect the performance as it contributes significantly to blockage loss. It is noted that the pressure generated decreases linearly with increasing blade thickness. A thinner blade is needed to generate a higher pressure. However, the thickness is usually limited by material property and fabrication technology. The difference between the numerical and theoretical results for the 20 mm diameter models is very small, while it is very large for the 10 mm diameter models. This is mainly because the energy loss of the smaller models is larger.

Fig. 6-36 Effect of blade thickness of 20 mm diameter models **Fig. 6-37 Effect of blade thickness of 10 mm diameter models**

6.6.4 Effect of the Blade Number on Pump Performance

The effects of the blade number of 20 mm and 10 mm diameter models with blade angle 56° are illustrated in Fig. 6-38 and Fig. 6-39. The theoretical pressure is lower than the numerical pressure, and this is caused by inaccuracy of the present theoretical models. For the 20 mm diameter models, the numerical and experimental results show that the pressure generated increases with the number of blades, while an optimum blade number of 13 exists for theoretical results. For the 10 mm diameter models, both the numerical and theoretical results show that a maximum pressure generated exists. For numerical results, this maximum pressure generated appears when the blade number is 11, while it is 15 for the theoretical results. This is mainly because more blades will have higher frictional loss, resulting in a drop in the pressure generated. Furthermore, the effect of the blade thickness will be more significant when the blade number is large. Theoretically speaking, the pressure generated will increase with the blade number if the blade thickness is negligible, because more blades will provide more guidance and cause more smooth flow and less energy loss. However, the thickness of the blade will block the flow and cause more blockage loss and shock loss, and will cause a drop in pressure generated.

Fig. 6-38 Effect of blade number of 20 mm diameter models

Fig. 6-39 Effect of blade number of 10 mm diameter models

To validate this trend, models 5, 20 and 21 are investigated numerically and experimentally as in Fig. 6-40 and Fig. 6-41. The 5 bladed model generates the lowest pressure generated, while 13 bladed model generates the highest. However, the difference between 11 bladed model and 13 bladed model is very small. Taking the

design point for example, when models are running at 112.5 rad/s, the designed flow rate is 2 l/min. The experimental pressure generated for 11 blades is 438Pa, while 457Pa for 13 blades. The difference is about 4%. When the models are running at 225 rad/s, the designed flow rate is 4 l/min. The experimental pressure generated for 11 blades is 1513 Pa, while 1573 Pa for 13 blades model. The difference is also 4%. More blades could not be investigated experimentally, because it is not possible to fabricate impellers with more than 13 blades. For the numerical results as in Fig. 6-40 and Fig. 6-41, the trend is similar to the experimental results. However, the pressure generated increases slightly when the flow rate is about 1.5 l/min for the 112.5 rad/s and 4.5 l/min for the 225 rad/s. Taking consideration of the numerical and theoretical results in Fig. 6-39, it can be concluded that an optimum value of blade number exists. However, the blade number is usually limited by many factors, such as material, fabrication technology, convenience of installation and price.

Fig. 6-40 Pump performance of 20 mm diameter models at 112.5 rad/s with different Z

Fig. 6-41 Pump performance of 20 mm diameter models at 225 rad/s with different Z

6.7 Effect of the Flow Coefficient on Pump Performance

The flow coefficient is important in the design procedure, and the recommended value for macro-size pump is not suitable for miniature pumps. When the flow coefficient is varied for a given outer radius and operating speed, the maximum pressure is generated when the flow coefficient is at its optimum value.

Fig. 6-42 presents the variation of numerical pressure generated with flow coefficient for models 1 to 10. From Table 3-6, as the flow coefficient increases from 0.29 to 3.49, the blade inner radius decreases from 6.88 mm to 3 mm and the blade angle increases from 17° to 74° . The pump model 10 with a flow coefficient of 0.29, which is recommended for macro-size pump, will generate a negative pressure, indicating that the macro-size pump design method is not suitable. As the flow coefficient increases, the pressure generated increases to a maximum value and then subsequently decreases

to a relatively stable value. This is because for models with very small flow coefficient, the inner radius is very large. Hence the blade channel length is very short, and fluid particles do not spend sufficient time in the blade passages to receive energy from the impeller, while for models with very large flow coefficient, the inner radius and blade angle are very small, causing large blockage loss.

Fig. 6-42 Effect of flow coefficient on pump performance of 20mm diameter models

From the numerical results, the optimum flow coefficient ranges from 1.47 to 2 with an inner radius ranging from 3.62mm to 4 mm. The flow coefficient of 1.47 is selected for the future study with the consideration of the flow coefficient and radius ratio. According to the numerical results, it is noted that the optimal flow coefficient is about 5 times that of the value recommended by the macro design approach. This will serve as a guide for future optimization of the impeller design.

6.8 Comparison of Original Designed Impeller with Modified Impeller

6.8.1 The Original-design Model 10

Model 10 is the original-design model with recommended flow coefficient of 0.29 and the performance curves are shown in Fig. 6-43 and Fig. 6-44. According to the experimental results, an obvious drop occurs for both speeds. This is mainly caused by choked flow. This phenomenon does not exist for numerical results, because only liquid phase of water is considered in the simulation process, and the vaporization of the water due to low pressure could not be estimated accurately. At the design point which is 2 l/min, the pressures generated by both experimental and numerical results are negative, indicating that the device is working as turbine. The flow will be driven by pressure force from inlet to outlet.

Fig. 6-43 Pump performance of original impeller model 10 at 112.5 rad/s

Fig. 6-44 Pump performance of original impeller model 10 at 225 rad/s

6.8.2 The Modified Model 5

Model 5 is the model with the optimum flow coefficient 1.47 and its performance curves are listed in Fig. 6-45 and Fig. 6-46. For the speed of 112.5 rad/s, the numerical pressure is higher than the experimental pressure. For the speed of 225 rad/s, the numerical pressure generated is in general similar to the experimental result except for very high or low flow rates. This is caused by the inaccuracy of the turbulent model.

Fig. 6-45 Pump performance of impeller model 5 at 112.5 rad/s

Fig. 6-46 Pump performance of impeller model 5 at 225 rad/s

6.9 Effect of the Volute Geometry on Pump Performance

The volute outlet is assumed to be rectangular with constant volute height. The volute

outlet width is varied based on the control equation of r_4 . Impeller model 5 with blade angle 56° is selected to operate with the designed volutes and the running speed is 112.5 rad/s at a flow rate of 2 l/min. The effect of the volute outlet width is illustrated in Fig. 6-47. It can be observed that the two lines of CMV and CAM almost overlap each other. The influence of the volute outlet width is important and is analyzed. As the volute outlet width increases, the pressure generated increases, but the influence is decreasing as the outlet width is increasing. It means that the pressure will tend to a steady value for increasing outlet width. For both CAM and CMV volutes, when the outlet width is bigger than 20.2 mm, the increase in pressure generated is negligible. For example, the difference pressure between 20.2 mm width and 28.8 mm width is 50.7 Pa for CAM volute and 50.3 Pa for CMV volute. Hence, a wide of 20.2 mm is selected for future study.

Fig. 6-47 pump performance of CMV and CAM volute with different volute outlet width

There are three kinds of energy losses in the pump, namely, mechanical loss, volumetric loss and hydraulic loss which will affect the pump performance. When the

pump model with the same impeller is running at the same working condition, the difference in energy losses between CMV and CAM models is caused by hydraulic loss. The hydraulic loss includes friction loss, loss caused by acceleration and retardation of the fluid. For the same volute inlet and outlet, the acceleration and retardation loss will be similar. Hence, the only difference is the friction loss. Taking the volute models with outlet width 20.2 mm for example, the difference of the volute length is only 1%. Using the same material, volute inlet and height, the difference of the friction loss is negligible.

The pump performance of the volutes designed by CMV and CAM method are investigated. The volute outlet width is selected as 20.2 mm. The pump performance curves are illustrated as Fig. 6-48. It can be observed that the CAM volute is superior to the CMV volute when the flow rate is less than 1.4 l/min. If the flow rate is higher than 1.4 l/min, the difference between the two volutes is small enough to be neglected.

Fig. 6-48 Pump performance of pumps with CMV and CAM volutes (outlet width 20.2 mm)

6.10 Flow Field Investigation

The 20 mm diameter model with radial blades is investigated using PIV technology. The velocity distributions inside the blade channels and near volute tongue are critical and flow separation and vortex are obvious in these zones. Hence they are selected to be compared between numerical results and experimental results.

6.10.1 Flow Field of 20 mm Diameter Model with the Originally Designed Impeller

To compare and evaluate the performances of models 10 (original designed model) and 5 (optimal model), the static pressure distributions in the middle plane of the blades are compared. The static pressure of model 10 is shown in Fig. 6-49. The static pressure is relative to the pump outlet port, which is set to zero. At a flow rate of 2 l/min with the impeller rotating at 112.5 rad/s, the average pressure rise of model 10 is -22 Pa between the pump inlet and outlet. This indicates that the device is operating as a turbine where high pressure flow is directed into the impeller passages. The negative pressure rise is mainly caused by the short channel passage. The ratio of the inner radius and outer radius is 0.688, and the distance between them is only 1.56 mm, which means that fluid particles entering the blade passage do not have sufficient time to receive energy from the impeller blade. In this case, no energy can be transferred from impeller to fluid.

Fig. 6-49 Static pressure distribution in the middle plane of model 10

The relative velocity distribution of model 10 is illustrated in Fig. 6-50. The flow pattern in the blade passages is smooth without any flow separation. The magnitude of the velocity in the impeller passage is almost uniform along the blade meridional direction. As the blade passages are relatively short, little or no flow separation is expected.

Fig. 6-50 Relative velocity distribution in the middle plane of model 10

6.10.2 Flow Field of 20 mm Diameter Model with the Modified Impeller

For model 5 when running at the same conditions as model 10, the impeller produces a head of 536 Pa, which is close to the designed pressure, 562.5 Pa. The static pressure and relative velocity are shown in Fig. 6-51 and Fig. 6-52. The static pressure increases from the pump eye through the blade channel to the pump volute gradually, and is further increased when the flow enters the volute. The lowest static pressure occurs at the leading edge of the impeller blade suction side, while the highest static pressure appears near the volute tongue. The flow is not entirely smooth and the flow pattern in the blade passage varies with its location relative to the volute. In each blade passage, there is a main stream near the pressure side of the blades. Near the suction side of the blade, the fluid has a lower velocity and flow separation can be observed. It is noted that some flow reversal occurs along the suction side of the blades and near to the blade tips.

Fig. 6-51 Static pressure distribution in the middle plane of model 5

Fig. 6-52 Relative velocity distribution in the middle plane of model 5

6.10.3 Flow Field of the Radial Impeller Model

The numerical and experimental velocity fields near the volute tongue are shown in Fig. 6-53. The rotating speed is 225 rad/s and the flow rate is 4 l/min. It is observed that the results obtained from the two methods agree reasonably. The flow separates near the tongue, and the velocity magnitude near walls are smaller than the main stream.

Fig. 6-53 Velocity distribution near the volute tongue

For the flow in the middle cutting plane of the impeller, zones 1 and 2 as shown in Fig. 6-54 are selected for the comparison of PIV and CFD results. The zone near the impeller inlet is blocked by the inlet tube, and the zone near the blade outlet tips are affected by the reflection light as illustrated in Fig. 6-55. So the PIV results in these zones are not accurate. For the velocity field in Fig. 6-55 at 225 rad/s, the numerical and experimental results agree reasonably well. It is observed that both results predict the existence of a vortex near to the suction side. The velocity vectors near suction and pressure side wall follow the wall direction for the experimental results. For the velocity field in Fig. 6-57 at 112.5 rad/s, The general trend of the velocity field is similar for numerical and PIV results. However, a small vortex is observed in the numerical data near the pressure side while no obvious vortex existed in the PIV results. This is mainly because the vortex is near the impeller inlet, and the measurement results are influenced by the blockage. Generally speaking, the results show that flow separation is inevitable when a radial blade impeller is used.

Fig. 6-54 Sketch of the impeller channel zone

Fig. 6-55 Captured picture of impeller zone 1

Fig. 6-56 Relative velocity distribution of the experimental and numerical results in impeller channel 1 at 225 rad/s

Fig. 6-57 Relative velocity distribution of the experimental and numerical results in impeller channel 2 at 112.5 rad/s

6.11 Summary

In this chapter, the numerical, analytical and experimental results are summarized and analyzed as follows:

1. Different turbulent models for 20 mm diameter model at 225 rad/s are compared with experimental results, and the Reynolds stress model is selected.
2. Energy models are expressed and the effects of the blade angle, radius ratio and flow rate are analyzed. Results show that the slip loss is most significant, while the wall friction loss and tip leakage loss are negligible. The blockage loss, diffusion loss and disk friction loss increase with flow rate, and decrease with the radius ratio. The leakage loss is significant when the radius ratio is larger than 0.4, and the shock loss should be considered when the blade angle is very smaller than 45° .
3. The effects of the blade geometry, including blade angle, inner radius, thickness, blade number are studied. The trends of the numerical, analytical

and experimental results are similar, while the magnitude of the analytical pressure is the highest. Increasing blade angle and decreasing blade thickness will increase the pressure generated. Optimized values of inner radius and blade number exist and are obtained.

4. The volute design method, Constant Mean Velocity (CMV) method and Constant Angular Momentum (CAM) method are compared. The difference between them is negligible at the design flow rate. The pressure generated increases with the volute outlet width, but the increasing rate is getting smaller and smaller. A value of 20.2 mm is selected for future study.
5. The effect of the flow coefficient is studied. This value is important in the design procedure, while the recommended value of macro-size pump may be not suitable for miniature pump. Results show that this value is in the range of 1.47 to 2.
6. Pump similarity is analyzed. The 20 mm diameter model at a speed of 225 rad/s generates the lowest pressure generated, indicating that the 20 mm diameter model do not follow the similarity law. When the flow rate is high, the 10 mm diameter model operating at 450 rad/s and 20 mm diameter model operating at 112.5 rad/s are similar. The lower the flow rate, the bigger the deviation between them.
7. The flow field of the 20 mm diameter model with radial blades is investigated numerically and experimentally. Three zones are selected, including the volute tongue, impeller channel 1 and 2. The two results agree reasonably well.
8. The flow fields of the original design model and modified model 5 are analyzed. The original design model generates negative pressure rise at the design point, and the flow pattern is very smooth. In this situation, the pump

works as a turbine. For the modified model, the pressure rise is 536 Pa, with some flow separation in the impeller channels.

7. Conclusions and Future Works

7.1 Conclusions

In this research, several impellers and volutes are designed based on traditional macro-pump design method and rational modifications. The 3-D models of these pumps are established and analyzed theoretically, numerically and experimentally. For the theoretical study, energy models based on previous study are selected. For the numerical study, the 3-D pumps with several turbulent models are investigated. For the experimental study, the pump performance and flow fields are measured and analyzed.

7.1.1 Impeller Design

Using traditional design method, an impeller inner radius of 6.88 mm, outer radius of 10 mm and blade angle of 17° is developed. The static pressure generated by this impeller is negative at the design point, which means that the traditional design method is not suitable for the miniature-size pump. Modifications are made to the inner radius and blade angles. With a smaller inner radius of 4 mm and bigger blade angle of 56° , the performance of the modified impeller is much better than the traditionally designed impeller.

Based on the study, it can be concluded that as the blade inlet and outlet angles are increased, the pressure generated is increased, but the flow pattern becomes more undesirable with flow separations occurring. For impellers with small blade inlet angles, the influence of the blade outlet angle is large. Similarly, for impellers with small blade outlet angles, the influence of the blade inlet angle is large. The radial impeller model generates the highest pressure but has the worse flow pattern.

Taking consideration of both the pressure generated and the flow pattern in the impeller passages, compromised values of blade inlet and outlet angles should be selected based on the requirement needed.

7.1.2 Volute Design

Following the traditional design method, the volute design procedure is listed in detail and modified in the current study. Two volutes are obtained using the volute design methods, including Constant Mean Velocity volute (CMV) and Constant Angular Momentum (CAM) volute. According to the design methods, the volute outlet width using CMV is about 14.4 mm, while that using CAM is about 28.8 mm. Additional volutes with different volute outlet width are derived when the empirical parameters are changed.

In general, the static pressure generated increases with the outlet width, but the influence of the volute outlet width decreases with increasing outlet width. It is noted from the simulation that the larger the outlet width, the higher the pressure generated. However, the rate of increase of the pressure generated decreases with increasing outlet width. The pressure tends to reach a relatively steady value when the outlet width is about 20.2 mm.

The results indicated that the smaller outlet generates a lower pressure, so a larger outlet is desirable. However, our objective is to design small pumps, which means that we cannot increase the volute outlet width excessively to obtain a high pressure. Our purpose is to find a suitable volute outlet width which can generate sufficiently high pressure and small size. From the present simulation results, it was observed that a volute outlet width of 20.2 mm is appropriate.

7.1.3 Energy Models

Energy models are developed and the effect of the blade angle, radius ratio and flow rate are analyzed. The results show that when the blade angle and radius ratio is not small, and flow rate not very high, the slip loss is most significant. Some of the energy losses can be neglected in the analysis procedure, like wall friction loss and tip leakage loss as they are much smaller in magnitude. The blockage loss, diffusion loss and disk friction loss increase with flow rate, and decrease with radius ratio. The leakage loss is only considered when the radius ratio is larger than 0.4, and the shock loss is considered when the blade angle is smaller than 30° . This will guide future energy analysis of miniature pumps. The theoretical head increases with the blade angle and decreases with the flow rate. In addition, there is an optimal radius ratio where the pressure generated is a maximum. This value is 0.54 for both 20 mm diameter models and 10 mm diameter models. The slip loss is most significant for typical blade angle and radius ratio. A large blade angle and moderate value of radius ratio is desirable to have a high pressure generated.

7.1.4 Numerical Models

For the 20 mm diameter models when running at 112.5 rad/s and the 10 mm diameter model running at 450 rad/s, laminar model are used, while for the 20 mm diameter models when running at 225 rad/s, the Reynolds Stress Model is used because the Reynolds number is no longer in the laminar flow region.

7.1.5 Effect of the Pump Geometry

The effects of the blade geometry, including blade angle, inner radius, thickness, blade number and volute outlet width are studied. The trends of the numerical, analytical and experimental results are similar, while the magnitude of the analytical pressure is the

highest. With increasing blade angle, and decreasing blade thickness, the pressure rise is increasing. Optimized values of inner radius and blade number exist.

Both numerical and theoretical results showed that an optimized inner radius and blade number exist. For the 20 mm diameter models, the optimum radius is 4 mm for the numerical simulation, while it is 4.9 mm for the theoretical results, and for the 10 mm diameter models, the optimum radius for numerical results is 2 mm while it is 2.7 mm for theoretical prediction. Similarly, the effect of blade number is also studied. For the 20 mm diameter models, the pressure generated increases with the number of blades, and the optimum blade number is 13 for theoretical prediction. For the 10 mm diameter models, the maximum pressure rise occurs when the blade number is 11, while it is 15 for the theoretical results. A large blade angle and moderate value of radius ratio is desirable to have a high pressure generated.

The volute design method, Constant Mean Velocity (CMV) method and Constant Angular Momentum (CAM) method are compared. The difference between them is negligible. The pressure generated increases with volute outlet width, and a value of 20.2 mm is selected for future study.

The effect of the flow coefficient is also studied. This value is important in the design procedure, and the recommended value for macro-size pump may be not suitable for miniature pump. Results show that an optimized value of flow coefficient exists. This value is in the range of 1.47 to 2 for numerical results.

The results obtained in the present study provide us guidelines on the design and performance study of centrifugal miniature pump.

7.1.6 Performance Characterization

The performance characterization of the model with radial blades, original design blades and optimum model 24 are conducted at speeds of 112.5 rad/s and 225 rad/s respectively. The flow fields of the original design model and modified model are analyzed. The original design model generates negative pressure at the design point, but the flow pattern is very smooth. In this situation, the pump works as a turbine. For the modified model, the pressure rise is 536 Pa, with some flow separation in the impeller channels.

7.1.7 Flow Field

The flow field of the 20 mm diameter model with radial blades is investigated numerically and experimentally. Two zones are selected, including the volute tongue and impeller channel 1. The two results agree reasonably well.

7.1.8 Pump Similarity

The pump similarity is analyzed. The 20 mm diameter model at a speed of 225 rad/s generates the lowest pressure rise, indicating that the speed of 20 mm diameter model do not follow the similarity law. When the flow rate is high, the 10 mm diameter model at 450 rad/s and 20 mm diameter model at 112.5 rad/s are similar.

7.2 Future Works

The present work focuses on the effect of geometrical parameters on miniature size pump, whose diameter is in the scale of centimeters. 20 mm diameter pump is investigated and half size model (10 mm diameter) is compared. Theoretical, numerical and experimental methods are employed and some conclusions are obtained. However, there are still some limitations of the present work.

The optimization of the flow coefficient, together with the blade inner radius and blade number should be investigated further by exploring ANOVA or Taguchi methods. These techniques may provide a clearer picture of how various variables will affect the pump performance.

For the theoretical analysis, the energy losses are based on empirical equation from previous researchers, which is derived for macro-size pump. They will be improved by reviewing the loss terms. A modified slip loss can be derived based on the physics of the slip velocity, and the leakage loss can be derived based on the N-S equations in the gap between rotating disk and stationary casing. The blockage loss will be derived based on the change of velocity field.

For the 10 mm diameter model, only the radial model is studied experimentally, which is not sufficient to validate the present conclusions for smaller size pumps. More impeller models should be studied and compared with the corresponding 20 mm diameter models. PIV measurements should be conducted to validate more models at the two diameters.

References

1. S. J. Kim and S. W. Lee, *Air Cooling Technology for Electronic Equipment*. 1996, Boca Raton: CRC Press.
2. R. Hannemann, *Thermal Control of Electronics: Perspectives and Prospects*. 2003, Massachusetts Institute of Technology.
3. A. Etemoglu, *A Brief Survey and Economical Analysis of Air Cooling for Electronic Equipments*. International Communications in Heat and Mass Transfer, 2007. **34**(1): p. 103-113.
4. P. Rodgers, V. Eveloy, and M. G. Pecht. *Extending the Limits of Air-Cooling in Microelectronic Equipment*. Proceedings of the 6th International Conference on Thermal, Mechanical and Multi-Physics Simulation and Experiments in Micro-Electronics and Micro-Systems - EuroSimE 2005, 2005.
5. A. Oki, M. Takai, H. Ogawa, Y. Takamura, T. Fukasawa, J. Kikuchi, Y. Ito, T. Ichiki, and Y. Horiike, *Healthcare Chip for Checking Health Condition from Analysis of Trace Blood Collected by Painless Needle*. Japanese Journal of Applied Physics, Part 1: Regular Papers and Short Notes and Review Papers, 2003. **42**(6 A): p. 3722-3727.
6. D. Kikugawa, T. Murakami, K. Endo, T. Fujiwara, and S. Takatani, *Evaluation of an Implantable Motor-driven Left Ventricular Assist Device*. Artificial Organs, 1999. **23**(3): p. 249-252.
7. S. Takahashi, H. Kitagawa, and Y. Tomikawa, *A Study of Liquid Dispensing Head Using Fluidic Inertia*. Japanese Journal of Applied Physics, Part 1: Regular Papers and Short Notes and Review Papers, 2002. **41**(5 B): p. 3442-3445.
8. S. Takahashi and Y. Tomikawa, *An Aspirating and Dispensing Device in the Nanoliter Range Using Piezoelectric Transducer*. Japanese Journal of Applied Physics, Part 1: Regular Papers and Short Notes and Review Papers, 2003. **42**(9 B): p. 6135-6138.
9. R. Luharuka, C. F. Wu, and P. J. Hesketh, *Design, Fabrication, and Testing of a Near Constant Pressure Fuel Delivery System for Miniature Fuel Cells*. Sensors and Actuators, A: Physical, 2004. **112**(2-3): p. 187-195.
10. X. Yang, Z. Zhou, H. Cho, and X. Luo, *Study on a PZT-actuated Diaphragm Pump for Air Supply for Micro Fuel Cells*. Sensors and Actuators, A: Physical, 2006. **130-131**(Spec. Iss.): p. 531-536.
11. D. Maillefer, S. Gamper, B. Frehner, P. Balmer, H. Van Lintel, and P. Renaud. *A High-performance Silicon Micropump for Disposable Drug Delivery Systems*. Proceedings of the IEEE Micro Electro Mechanical Systems (MEMS), 2001.
12. M. J. Capdeville and N. G. Smedira, *Advances and Future Directions for Mechanical Circulatory Support*. Anesthesiology Clinics, 2013. **31**(2): p. 321-353.
13. I. Ederer, P. Raetsch, W. Schullerus, C. Tille, and U. Zech, *Piezoelectrically Driven Micropump for On-demand Fuel-drop Generation in an Automobile*

- Heater with Continuously Adjustable Power Output*. Sensors and Actuators, A: Physical, 1997. **62**(1-3): p. 752-755.
14. H. K. Ma, B. R. Chen, J. J. Gao, and C. Y. Lin, *Development of an OAPCP-micropump Liquid Cooling System in a Laptop*. International Communications in Heat and Mass Transfer, 2009. **36**(3): p. 225-232.
 15. R. Zengerle, J. Ulrich, S. Kluge, M. Richter, and A. Richter, *A Bidirectional Silicon Micropump*. Sensors and Actuators, A: Physical, 1995. **50**(1-2): p. 81-86.
 16. R. Ardito, E. Bertarelli, A. Corigliano, and G. Gafforelli, *On the Application of Piezolaminated Composites to Diaphragm Micropumps*. Composite Structures, 2013. **99**: p. 231-240.
 17. T. Hasegawa, D. Koyama, K. Nakamura, and S. Ueha, *A Design of a Miniature Ultrasonic Pump Using a Bending Disk Transducer*. Journal of Electroceramics, 2008. **20**(3-4): p. 145-151.
 18. D. Schimelman, *Miniature Diaphragm Pumps Meet the Performance Challenge*. World Pumps, 2006(477): p. 44-46.
 19. D. Blanchard, P. Ligrani, and B. Gale, *Miniature Single-disk Viscous Pump (Single-DVP), Performance Characterization*. Journal of Fluids Engineering, Transactions of the ASME, 2006. **128**(3): p. 602-610.
 20. I. J. Karassik, J. P. Messina, P. Cooper, and C. C. Heald, *Pump Handbook*. 4th ed. 2008, New York: McGraw-Hill.
 21. P. Girdhar, *Practical Centrifugal Pumps: Design, Operation and Maintenance*. 2005, Oxford: Newnes.
 22. D. B. McWhorter and D. K. Sunada, *Ground-water Hydrology and Hydraulics*. 1977, Fort Collins: Water Resources Publications.
 23. Wikipedia. http://en.wikipedia.org/wiki/Denis_Papin.
 24. H. H. Anderson, *Centrifugal Pumps*. 3rd ed. 1980, Surrey: Trade & Technical Press.
 25. E. Meng, X. Q. Wang, H. Mak, and Y. C. Tai, *Check-valved Silicone Diaphragm Pump*. Proceedings of the IEEE Micro Electro Mechanical Systems (MEMS), 2000: p. 62-67.
 26. J. G. Smits, *Piezoelectric Micropump with Three Valves Working Peristaltically*. Sensors and Actuators, A: Physical, 1990. **21**(1-3): p. 203-206.
 27. D. J. Laser and J. G. Santiago, *A Review of Micropumps*. Journal of Micromechanics and Microengineering, 2004. **14**(6): p. R35-R64.
 28. N. T. Nguyen, X. Huang, and T. K. Chuan, *MEMS-micropumps: A Review*. Journal of Fluids Engineering, Transactions of the ASME, 2002. **124**(2): p. 384-392.
 29. R. S. R. Gorla and A. A. Khan, *Turbomachinery: Design and Theory*. 2003, New York: Marcel Dekker, Inc.
 30. J. F. Gulich, *Centrifugal Pumps*. 2nd ed. 2010, Berlin: Springer-Verlag Berlin Heidelberg.

31. M. P. Boyce, *Gas Turbin Engineering Handbook*. 4th ed. 2012, Waltham: Butterworth-Heinemann.
32. O. E. Balje, *Turbomachines: a Guide to Design Selection and Theory*. 1981, New York: Wiley.
33. Ph. Epple, F. Durst, and A. Delgado, *A Theoretical Derivation of the Cordier Diagram for Turbomachines*. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 2011. **225**(2): p. 354-368.
34. J. Tuzson, *Centrifugal Pump Design*. 2000, New York: John Wiley & Sons.
35. D. S. Hira and V. P. Vasandani, *Influence of Volute Tongue Length and Angle on the Pump Performance*. Journal of the Institution of Engineers (India): Mechanical Engineering Division, 1975. **56**(pt ME 1): p. 55-59.
36. E. Ayder, R. V. Braembussche, and J. J. Brasz, *Experimental and Theoretical Analysis of the Flow in a Centrifugal Compressor Volute*. Journal of Turbomachinery, 1993. **115**(3): p. 582-589.
37. Y. W. Wong, W. K. Chan, and W. Hu, *Effects of Tongue Position and Base Circle Diameter on the Performance of a Centrifugal Blood Pump*. Artificial Organs, 2007. **31**(8): p. 639-645.
38. F. Boehning, D. L. Timms, F. Amaral, L. Oliveira, R. Graefe, P. L. Hsu, T. Schmitz-Rode, and U. Steinseifer, *Evaluation of Hydraulic Radial Forces on the Impeller by the Volute in a Centrifugal Rotary Blood Pump*. Artificial Organs, 2011. **35**(8): p. 818-825.
39. V. S. Lobanoff and R. R. Ross, *Centrifugal Pumps : Design & Application* 2nd ed. 1992: Houston. Gulf Professional Publishing.
40. D. P. Sloteman, A. N. Saad, and P. Cooper. *Designing Custom Pump Hydraulics Using Traditional Methods*. Proceedings of the ASME Fluids Engineering Division Summer Meeting. 2003.
41. M. Asuaje, F. Bakir, S. Kouidri, and R. Rey, *Computer-aided Design and Optimization of Centrifugal Pumps*. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 2004. **219**: p. 7.
42. T. S. Luu, B. Viney, and L. Bencherif, *Inverse Problem Using S2-S1 Approach for the Design of the Turbomachine with Splitter Blades*. RFM, Revue Francaise de Mecanique, 1992(3): p. 209-220.
43. A. J. Stepanoff, *Centrifugal and Axial Flow Pumps: Theory, Design, and Application*. 1957, New York: Wiley.
44. M. Asuaje, F. Bakir, S. Kouidri, and R. Rey, *Inverse Design Method for Centrifugal Impellers and Comparison with Numerical Simulation Tools*. International Journal of Computational Fluid Dynamics, 2004. **18**(2): p. 101-110.
45. K. Murugesan and J. W. Raily, *Pure Design Method for Aerofoils in Cascade*. Journal of Mechanics Engineering Science, 1969. **11**(5): p. 454-467.
46. M. Hart and D. S. Whitehead, *Design Method for Two-dimensional Cascades of Turbomachinery Blades*. International Journal for Numerical Methods in Fluids, 1987. **7**(12): p. 1363-1381.

47. X. Zhao, C. Sun, and C. Wu, *Simple Method for Solving Three-dimensional Inverse Problems of Turbomachine Flow and the Annular Constraint Condition*. Journal of Engineering for Gas Turbines and Power, 1985. **107**(2): p. 293-300.
48. W. S. Ghaly, *Parametric Study of Radial Turbomachinery Blade Design in Three-dimensional Subsonic Flow*. Journal of Turbomachinery, 1990. **112**(3): p. 338-345.
49. M. Zangeneh, A. Goto, and T. Takemura, *Suppression of Secondary Flows in a Mixed-flow Pump Impeller by Application of Three-dimensional Inverse Design Method: Part 1: Design and Numerical Validation*. Journal of Turbomachinery, 1996. **118**(3): p. 536-543.
50. M. Zangeneth. *Inviscid-viscous Interaction Method for 3D Inverse Design of Centrifugal Impellers*. International Gas Turbine and Aeroengine Congress and Exposition of ASME. 1993.
51. M. Zangeneh, A. Goto, and H. Harada, *On the Role of Three-dimensional Inverse Design Methods in Turbomachinery Shape Optimization*. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering, 1999. **213**(1): p. 27-42.
52. A. Goto and M. Zangeneh. *Hydrodynamic Design of Pump Diffuser Using Inverse Design Method and CFD*. Proceedings of the ASME Fluid Engineering Division Summer Meeting. 1998.
53. M. Zangeneh, *Compressible Three-dimensional Design Method for Radial and Mixed Flow Turbomachinery Blades*. International Journal for Numerical Methods in Fluids, 1991. **13**(5): p. 599-624.
54. J. Zou, P. Wang, X. Ruan, and F. Xin, *A Statistical Method on Meridional Profiles of Centrifugal Pumps*. Journal of Fluids Engineering, Transactions of the ASME, 2012. **134**(2).
55. H. H. Anderson, *Modern Developments in the Use of Large Single-entry Centrifugal Pumps*. Proceedings of the Institution of Mechanical Engineers, 1955. **169**(6): p. 141-161.
56. L. Jiang, J. Mikkelsen, J. M. Koo, D. Huber, S. Yao, L. Zhang, P. Zhou, J. G. Maveety, R. Prasher, J. G. Santiago, T. W. Kenny, and K. E. Goodson, *Closed-loop Electroosmotic Microchannel Cooling System for VLSI Circuits*. IEEE Transactions on Components and Packaging Technologies, 2002. **25**(3): p. 347-355.
57. A. Rossetti, G. Pavesi, and G. Ardizzon, *A New Two Stage Miniature Pump: Design, Experimental Characterization and Numerical Analyses*. Sensors and Actuators, A: Physical, 2010. **164**(1-2): p. 74-87.
58. N. T. Nguyen and T. Q. Truong, *A Fully Polymeric Micropump with Piezoelectric Actuator*. Sensors and Actuators, B: Chemical, 2004. **97**(1): p. 137-143.
59. O. Francais and I. Dufour, *Dynamic Simulation of an Electrostatic Micropump with Pull-in and Hysteresis Phenomena*. Sensors and Actuators, A: Physical, 1998. **70**(1-2): p. 56-60.

-
60. X. Wang, C. Cheng, S. Wang, and S. Liu, *Electroosmotic Pumps and their Applications in Microfluidic Systems*. Microfluidics and Nanofluidics, 2009: p. 1-18.
 61. A. Olsson, G. Stemme, and E. Stemme, *A Valve-Less Planar Fluid Pump with Two Pump Chambers*. Sensors and Actuators, A: Physical, 1995. **47**(1-3): p. 549-556.
 62. N. T. Nguyen and X. Huang, *Miniature Valveless Pumps Based on Printed Circuit Board Technique*. Sensors and Actuators, A: Physical, 2001. **88**(2): p. 104-111.
 63. L. S. Jang and W. H. Kan, *Peristaltic Piezoelectric Micropump System for Biomedical Applications*. Biomedical Microdevices, 2007. **9**(4): p. 619-626.
 64. C. S. Chao, P. C. Huang, M. K. Chen, and L. S. Jang, *Design and Analysis of Charge-recovery Driving Circuits for Portable Peristaltic Micropumps with Piezoelectric Actuators*. Sensors and Actuators, A: Physical, 2011. **168**(2): p. 313-319.
 65. C. F. Lieu, W. K. Chan, and K. T. Ooi, *Experimental Investigation of the Reciprocating Ball Pump (RBP)*. Medical Engineering and Physics, 2012. **34**(8): p. 1101-1108.
 66. X. Wang, C. Cheng, S. Wang, and S. Liu, *Electroosmotic Pumps and Their Applications in Microfluidic Systems*. Microfluid Nanofluidics, 2009. **6**(2): p. 145-162.
 67. A. T. Al-Halhouli, S. Demming, M. Feldmann, S. Buttgenbach, M. I. Kilani, and A. Al-Salaymeh, *Performance Characterization of a Miniature Spiral-channel Viscous Pump*. Sensors and Actuators, A: Physical, 2008. **142**(1): p. 256-262.
 68. Y. Abe, T. Ono, T. Isoyama, S. Mochizuki, K. Iwasaki, T. Chinzei, I. Saito, A. Kouno, and K. Imachi, *Development of a Miniature Undulation Pump for the Distributed Artificial Heart*. Artificial Organs, 2000. **24**(8): p. 656-658.
 69. K. F. Lei, R. H. W. Lam, J. H. M. Lam, and W. J. Li. *Polymer Based Vortex Micropump Fabricated by Micro Molding Replication Technique*. IEEE/RSJ International Conference on Intelligent Robots and Systems. 2004.
 70. J. F. Antaki, C. G. Diao, F. J. Shu, J. C. Wu, R. Zhao, and M. V. Kameneva, *Microhaemodynamics within the Blade Tip Clearance of a Centrifugal Turbodynamic Blood Pump*. Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine, 2008. **222**(4): p. 573-581.
 71. H. Horiguchi, T. Tsukiya, T. Nomoto, T. Takemika, and Y. Tsujimoto, *Research on the Development of Two-stage Centrifugal Blood Pump for Cardiopulmonary Support System*. Transactions of the Japan Society of Mechanical Engineers, Part B, 2013. **79**(800): p. 482-491.
 72. S. W. Day, P. P. Lemire, R. D. Flack, and J. C. McDaniel. *Effect of Reynolds Number on Performance of a Small Centrifugal Pump*. Proceedings of the ASME/JSME Joint Fluids Engineering Conference. 2003.

73. M. Lorenz and W. A. Smith, *Rotodynamic Pump Scaling*. ASAIO Journal, 2002. **48**(4): p. 419-430.
74. D. Kearney, J. Punch, and R. Grimes. *An Experimental and Theoretical Investigation of the Pumping Performance of Geometrically Similar Flow Fields within Miniature-scale Centrifugal Pumps*. Proceedings of the ASME/JSME Thermal Engineering Summer Heat Transfer Conference. 2007.
75. Y. W. Wong, W. K. Chan, S. C. M. Yu, and L. P. Chua, *Effects of Scaling on Centrifugal Blood Pumps*. Artificial Organs, 2002. **26**(11): p. 998-1001.
76. D. Kearney, R. Grimes, and J. Punch, *An Experimental Investigation of the Flow Fields within Geometrically Similar Miniature-scale Centrifugal Pumps*. Journal of Fluids Engineering, Transactions of the ASME, 2009. **131**(10): p. 1011011-10110110.
77. J. F. Gulich, *Effect of Reynolds Number and Surface Roughness on the Efficiency of Centrifugal Pumps*. Journal of Fluids Engineering, Transactions of the ASME, 2003. **125**(4): p. 670-679.
78. S. Tanaka and S. Murata, *Scale Effects in Cross-Flow Fans (Effects of Fan Dimensions on Performance Curves)*. JSME International Journal, Series B: Fluids and Thermal Engineering, 1994. **37**(4): p. 844-852.
79. M. V. Casey, *Effects of Reynolds Number on the Efficiency of Centrifugal Compressor Stages*. Journal of Engineering for Gas Turbines and Power, 1985. **107**(2): p. 541-548.
80. A. M. Sardar and W. K. George Jr. *Reconsideration of the Fan Scaling Laws. Part I: Theory*. Proceedings of the ASME/JSME Joint Fluids Engineering Conference. 2003.
81. R. Grimes, E. Walsh, D. Quin, and M. Davies, *Effect of Geometric Scaling on Aerodynamic Performance*. AIAA Journal, 2005. **43**(11): p. 2293-2298.
82. P. Walsh, V. Egan, R. Grimes, and E. Walsh, *Profile Scaling of Miniature Centrifugal Fans*. Heat Transfer Engineering, 2009. **30**(1-2): p. 130-137.
83. Sulzer Pumps Ltd, *Centrifugal Pump Handbook*. 3rd ed. 2010, Oxford: Butterworth-Heinemann.
84. B. Eck, *Fans Design and Operation of Centrifugal, Axial-flow, and Cross-flow Fans*. 1973, Oxford, New York: Pergamon Press.
85. H. W. Oh and M. K. Chung, *Optimum Values of Design Variables Versus Specific Speed for Centrifugal Pumps*. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 1999. **213**(3): p. 219-226.
86. N. M. Ntoko and K. Djidjou, *Basic for Teaching Centrifugal Pump Characteristics in an Undergraduate Course in Turbomachinery*. International Journal of Mechanical Engineering Education, 2005. **33**(4).
87. S. Yedidiah, *New Look at the Slip in a Centrifugal Impeller*. Proceedings of the Institution of Mechanical Engineers, 1974. **188**(47): p. 499-504.
88. T. W. von Backstrom, *A Unified Correlation for Slip Factor in Centrifugal Impellers*. Journal of Turbomachinery, 2006. **128**(1): p. 1-10.

89. A. Stodola, *Steam and Gas Turbines*. 1927, New York: McGraw-Hill Book Company.
90. K. S. Paeng and M. K. Chung, *A New Slip Factor for Centrifugal Impellers*. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 2001. **215**(5): p. 645-649.
91. A. Busemann, *Das Förderhöhenverhältniss Radialer Kreispumpen Mit Logarithmisch-Spiraligen Schaufeln*. Zeitschrift für angewandte mathematik und mechanik, 1928. **8**: p. 14.
92. G. F. Wislicenus, *Fluid Mechanics of Turbo Machinery*. 2nd ed. Vol. 1. 1965, New York: Dover.
93. F. J. Wiesner, *A Review of Slip Factors for Centrifugal Impellers*. Transactions of ASME: Journal of Gas Turbines Power, 1967. **89**: p. 558-572.
94. F. C. Visser, J. J. H. Brouwers, and R. Badie, *Theoretical Analysis of Inertially Irrotational and Solenoidal Flow in Two-dimensional Radial-flow Pump and Turbine Impellers with Equiangular Blades*. Journal of Fluid Mechanics, 1994. **269**: p. 107-142.
95. H. Ohta, *Effect of Reynolds Number on Slip Factor of Centrifugal Pump for High-Viscosity Liquids*. Transactions of the Japan Society of Mechanical Engineers, Part B, 1999. **65**(639): p. 3697-3704.
96. M. Memardefzouli and A. Nourbakhsh, *Experimental Investigation of Slip Factors in Centrifugal Pumps*. Experimental Thermal and Fluid Science, 2009. **33**(5): p. 938-945.
97. J. Anderson, H. G. Wood, P. E. Allaire, and D. B. Olsen, *Numerical Analysis of Blood Flow in the Clearance Regions of a Continuous Flow Artificial Heart Pump*. Artificial Organs, 2000. **24**(6): p. 492-500.
98. S. W. Day, J. C. McDaniel, H. G. Wood, P. E. Allaire, N. Landrot, and A. Curtas, *Particle Image Velocimetry Measurements of Blood Velocity in a Continuous Flow Ventricular Assist Device*. ASAIO Journal, 2001. **47**(4): p. 406-411.
99. K. Takiura, T. Masuzawa, S. Endo, Y. Wakisaka, E. Tatsumi, Y. Taenaka, H. Takano, T. Yamane, M. Nishida, B. Asztalos, Y. Konishi, Y. Miyazoe, and K. Ito, *Development of Design Methods of a Centrifugal Blood Pump with in Vitro Tests, Flow Visualization, and Computational Fluid Dynamics: Results in Hemolysis Tests*. Artificial Organs, 1998. **22**(5): p. 393-398.
100. T. Ikeda, T. Yamane, T. Orita, and T. Tateishi, *A Quantitative Visualization Study of Flow in a Scaled-up Model of a Centrifugal Blood Pump*. Artificial Organs, 1996. **20**(2): p. 132-138.
101. W. K. Chan and Y. W. Wong, *A Review of Leakage Flow in Centrifugal Blood Pumps*. Artificial Organs, 2006. **30**(5): p. 354-359.
102. M. Nishida, T. Yamane, T. Orita, B. Asztalos, and H. Clarke, *Quantitative Visualization of Flow through a Centrifugal Blood Pump: Effect of Washout Holes*. Artificial Organs, 1997. **21**(7): p. 720-729.
103. Y. Wakisaka, Y. Taenaka, K. Araki, K. I. Chikanari, T. Nakatani, Y. Baba, H. Anai, K. Eya, K. Toda, and H. Takano, *Effects of Self Washout Structure on the*

- Antithrombogenicity and the Hemolytic Properties of a Centrifugal Pump*. Artificial Organs, 1997. **21**(2): p. 148-153.
104. M. Triep, C. Brucker, W. Kerkhoffs, O. Schumacher, and O. Marseille, *Investigation of the Washout Effect in a Magnetically Driven Axial Blood Pump*. Artificial Organs, 2008. **32**(10): p. 778-784.
105. B. C. Will, F. K. Benra, and H. J. Dohmen, *Investigation of the Flow in the Impeller Side Clearances of a Centrifugal Pump with Volute Sasing*. Journal of Thermal Science. **21**(3): p. 197-208.
106. A. D. Polyanin, *Exact Solutions to the Navier-Stokes Equations with Generalized Separation of Variables*. Doklady Physics, 2001. **46**(10): p. 726-731.
107. W. K. Chan, T. Akamatsu, and H. D. Li, *Analytical Investigation of Leakage Flow in Disk Clearance of a Magnetically Suspended Centrifugal Impeller*. Artificial Organs, 2000. **24**(9): p. 734-742.
108. J. B. Teo, W. K. Chan, and Y. W. Wong, *Prediction of Leakage Flow in a Shrouded Centrifugal Blood Pump*. Artificial Organs, 2010. **34**(9): p. 788-791.
109. M. Turkyilmazoglu, *Exact Solutions for the Incompressible Viscous Magnetohydrodynamic Fluid of a Rotating-disk Flow with Hall Current*. International Journal of Non-Linear Mechanics, 2011. **46**(8): p. 1042-1048.
110. M. Turkyilmazoglu, *Exact Solutions for the Incompressible Viscous Magnetohydrodynamic Fluid of a Rotating Disk Flow*. International Journal of Non-Linear Mechanics, 2011. **46**(1): p. 306-311.
111. Y. Senoo and M. Ishida, *Pressure Loss Due to the Tip Clearance of Impeller Blades in Centrifugal and Axial Blowers*. Journal of Engineering for Gas Turbines and Power, 1986. **108**(1): p. 32-37.
112. J. F. Gulich, *Disk Friction Losses of Closed Turbomachine Impellers*. Forschung im Ingenieurwesen/Engineering Research, 2003. **68**(2): p. 87-95.
113. J. W. Daily and R. E. Nece, *Chamber Dimension Effects on Induced Flow and Friction Resistance of Enclosed Rotating Disks*. Journal of Basic Engineering, 1960. **82**: p. 217-232.
114. H. Fukuda, *The Effect of Surface Roughness on the Performance of a Francis Turbine*. JSME, 1964. **7**(26): p. 346-356.
115. S. W. Churchill, *Viscous Flows: The Practical Use of Theory*. 1988, London: Butterworth Ser Chem Engng.
116. A. Nemdili and D. H. Hellmann, *Investigations on Fluid Friction of Rotational Disks with and without Modified Outlet Sections in Real Centrifugal Pump Casings*. Forschung im Ingenieurwesen/Engineering Research 2007. **71**(1): p. 59-67.
117. P. M. Sforza, *Fluid Mechanics Thermodynamics of Turbomachinery*. 6th ed. 2012, Oxford: Academic Press. 684.
118. H. Liu, Y. Wang, S. Yuan, M. Tan, and K. Wang, *Effects of Blade Number on Characteristics of Centrifugal Pumps*. Chinese Journal of Mechanical Engineering (English Edition), 2010. **23**(6): p. 742-747.

119. G. Kaminoto and Y. Matsuoka, *On the Flow in the Impeller of Centrifugal Type Hydraulic Machinery*. Transactions of ASME, 1956. **22**(113): p. 55-59.
120. F. A. Varley, *Effects of Impeller Design and Surface Roughness on the Performance of Centrifugal Pumps*. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 1961. **175**(196): p. 955-989.
121. T. Toyakura, J. Kurokawa, and T. Keanemoto, *Performance Improvement for Centrifugal Pump Handling High Viscosity Oil*. Turbomachinery, 1979. **7**(2): p. 8-15.
122. K. Tanaka and H. Ohashi, *Performance of Centrifugal Pumps at Low Reynolds Number(1st report, experimental study)*. Transactions of JSME, 1984. **50**(449): p. 279-285.
123. H. Ohta and K. Aoki, *Study on Centrifugal Pump for High-Viscosity Liquids. (Effect of Impeller Blade Number on the Pump Performance)*. Transactions of the Japan Society of Mechanical Engineers, Part B, 1990. **56**(526): p. 1702-1707.
124. R. Jia and H. Xu, *Optimal Design of Low Specific Speed Mixed-flow Pump Impeller*. Journal of Drainage and Irrigation Machinery Engineering, 2010. **28**(2): p. 98-102.
125. W. G. Li, *Blade Exit Angle Effects on Performance of a Standard Industrial Centrifugal Oil Pump*. Journal of Applied Fluid Mechanics, 2011. **4**(3): p. 105-119.
126. X. Luo, Y. Zhang, J. Peng, and H. Xu, *Effect of Impeller Inlet Geometry on Centrifugal Pump Cavitation Performance*. Journal of Tsinghua University, 2008. **48**(5): p. 836-839.
127. W. G. Li, *Influence of the Number of Impeller Blades on the Performance of Centrifugal Oil Pumps*. World Pumps, 2002(427): p. 32-35.
128. K. S. Rababa, *The Effect of Blades Number and Shape on the Operating Characteristics of Groundwater Centrifugal Pumps*. European Journal of Scientific Research, 2011. **52**(2): p. 243-251.
129. A. Engeda, *Correlation and Prediction of Efficiency of Centrifugal Pumps Due to Tip Clearance Effects*. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 1995. **209**(2): p. 111-114.
130. H. Bing, S. L. Cao, C. L. He, and L. Lu, *Experimental Study of the Effect of Blade Tip Clearance and Blade Angle Error on the Performance of Mixed-flow Pump*. Science China Technological Sciences, 2013. **56**(2): p. 293-298.
131. B. Cui, D. Huang, P. Shi, and Q. Jin, *Effect of Tip Clearance on Performance of Low Specific Speed Centrifugal Pump with Semi-open Impeller*. Journal of Drainage and Irrigation Machinery Engineering, 2012. **30**(3): p. 283-288.
132. T. Engin, M. Gur, and R. Scholz, *Effects of Tip Clearance and Impeller Geometry on the Performance of Semi-open Ceramic Centrifugal Fan Impellers at Elevated Temperatures*. Experimental Thermal and Fluid Science, 2006. **30**(6): p. 565-577.

133. Z. Y. Liu, X. D. Wang, and S. Kang, *CFD Simulations of Uncertain Tip Clearance Effect on Compressor Performance*. Kung Cheng Je Wu Li Hsueh Pao/Journal of Engineering Thermophysics, 2013. **34**(4): p. 628-631.
134. L. Torre, A. Pasini, A. Cervone, G. Pace, P. Miloro, and L. D'Agostino, *Effect of Tip Clearance on the Performance of a Three-bladed Axial Inducer*. Journal of Propulsion and Power, 2011. **27**(4): p. 890-898.
135. L. M. Milne-Thomson, *Theoretical Aerodynamics*. 1973, New York: Dover Publications. 430.
136. J. L. Hess and A. M. O. Smith, *Calculation of Potential Flow about Arbitrary Bodies*. Progress in Aerospace Sciences, 1967. **8**(0): p. 1-138.
137. G. W. Burgreen, J. F. Antaki, Z. J. Wu, and A. J. Holmes, *Computational Fluid Dynamics as a Development Tool for Rotary Blood Pumps*. Artificial Organs, 2001. **25**(5): p. 336-340.
138. C. Yuan, *Numerical Calculation of Turbulent Flow and Performance Experiment of Mixed-flow Pump Impeller*. Transactions of the Chinese Society of Agricultural Machinery, 2008. **39**(3): p. 52-55.
139. X. Cong, G. Wang, D. Yuan, and Z. Jia, *Numerical Calculation and Optimal Design of Centrifugal Pump with Space Guide Vanes*. Journal of Drainage and Irrigation Machinery Engineering, 2010. **28**(6): p. 488-491.
140. V. A. Grapsas, J. S. Anagnostopoulos, and D. E. Papantonis, *Flow Measurements and Simulation in a Model Centrifugal Pump Impeller*. International Journal of Fluid Mechanics Research, 2010. **37**(2): p. 149-161.
141. J. Li, Y. Zeng, X. Liu, and H. Wang. *Optimum Design on Impeller Blade of Mixed-flow Pump Based on CFD*. 1st International Conference on Advances in Computational Modeling and Simulation. 2011.
142. A. K. Prasad, *Particle Image Velocimetry*. Current Science, 2000. **79**(1): p. 51-60.
143. Y. Yea and H. Z. Cummings, *Localized Fluid Flow Measurements with an He-Ne Laser Spectrometer*. Applied Physics Letters, 1964. **4**: p. 176-178.
144. N. A. Malik and T. Dracos, *Lagrangian PTV in 3D Flows*. Applied Scientific Research, 1993. **51**(1-2): p. 161-166.
145. I. Grant, *Particle Image Velocimetry: A Review*. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 1997. **211**(1): p. 55-76.
146. R. Dong, S. Chu, and J. Katz, *Effect of Modification to Tongue and Impeller Geometry on Unsteady Flow, Pressure Fluctuations, and Noise in a Centrifugal Pump*. Journal of Turbomachinery, 1997. **119**(3): p. 506-515.
147. C. Pfleiderer, *Die Kreiselpumpen für Flüssigkeiten und Gase*. 1961, Berlin: Springer.
148. W. Wesche, *Measurement on the Volute Casing of a Centrifugal Pump*. World Pumps, 1987: p. 321-324.
149. R. T. Turton, *Principles of Turbomachinery*. 2nd ed. 1995, London: Chapman & Hall. 265.

-
150. P. Dorfler, M. Sick, and A. Coutu, *Flow-Induced Pulsation and Vibration in Hydroelectric Machinery*. 2013, London: Springer London.
 151. C. A. J. Fletcher, *Computational Techniques for Fluid Dynamics: Fundamental and General Techniques*. 2nd ed. 1991, New York: Springer Verlag.
 152. Fluent Inc., *FLUENT 6.2 Tutorial Guide*. 2005.
 153. B. Lakshminarayana, *Fluid Dynamics and Heat Transfer of Turbomachinery*. 1996, New York: Wiley.
 154. M. G. Davies, A. B. Lumsden, W. E. Kline, and I. Kakadiaris, *Pumps and Pipes: Proceedings of the Annual Conference*. 2011, Boston: Springer Science+Business Media.
 155. M. Raffel, J. Kompenhans, S. T. Wereley, and C. E. Willert, *Particle Image Velocimetry: A Practical Guide* 2nd ed. Experimental Fluid Mechanics. 2007, Berlin: Heidelberg.
 156. Dentec Dynamics, *Flowmanager Software and Introduction to PIV Instrumentation*. 2002.
 157. C. Tropea, J. F. Foss, and A. L. Yarin, *Springer Handbook of Experimental Fluid Mechanics*. 2007, Berlin, Heidelberg : Springer Science+Business Media, 2007.
 158. R. J. Adrian and Westerweel. J., *Particle Image Velocimetry*. 2011, New York: Cambridge University Press.
 159. K. Birch. *Measurement Good Practice Guide No 36. Estimating Uncertainties in Testing*. 2001.

Appendix

Potters Industries Inc.
an affiliate of PQ Corporation

ENGINEERED GLASS MATERIALS DIVISION

HOLLOW MICROSPHERES

SPHERICEL® GENERAL PRODUCT CHARACTERISTICS

Composition: Fused Borosilicate Glass (ASTM C169)
Shape: Hollow non-porous microspheres
Color: White (min. 90.0 L value on dry powder)
Hardness: 6 Moh's scale
Alkalinity: Max 0.5 meg/gr (ASTM D3100)
Loss on Drying: Max. 0.5 weight % (10 min. @ 250°F)

	<u>SPHERICEL®</u>			
	<u>110P8</u>	<u>60P18</u>	<u>34P30</u>	<u>25P45</u>
<u>DENSITY</u> (ASTM D2840) grams/cc	1.10 ± 0.05	0.60 + 0.05	0.34 + 0.05	0.25 ± 0.02
<u>STRENGTH, psi</u>	10,000	8,000	3,000	750 psi
<u>OIL ABSORPTION</u> (ASTM D281)* g oil/100 g g oil/100cc	32 36	48 30	44 14	
<u>PARTICLE SIZE, µm</u> Average Size	9 – 13	16 – 20	29 – 31	40 – 60
<u>SIZE DISTRIBUTION</u> (microns)				
90% finer than	19 – 21	28 – 32	44 – 46	55 – 75
50% finer than	9 – 11	15 – 19	26 – 30	40 – 60
10% finer than	4 – 5	6 – 10	15 – 18	15 – 30

*Sphericel® hollow glass microspheres do not absorb oil. The value reported is the amount of oil necessary to wet the surface of the glass.

PQ Corporation is a privately held global enterprise operating in 20 countries, with annual revenues in excess of \$500 million. PQ is a leading producer of silicate, zeolite, and other performance materials serving the detergent, pulp and paper, chemical, petroleum, catalyst, water treatment, construction, and beverage markets.

Potters Industries, PQ's wholly owned subsidiary, is a leading producer of engineered glass materials serving the highway safety, polymer additive, fine abrasive, and conductive product markets.