INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps. Each original is also photographed in one exposure and is included in reduced form at the back of the book.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6” x 9” black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.
COMPUTATIONAL FLUID ANALYSIS OF A CENTRIFUGAL HEART PUMP

DISSERTATION

Presented in Partial Fulfillment of the Requirements for the Degree Doctor of Philosophy in the Graduate School of The Ohio State University

By

Wei Ding, M.S.

The Ohio State University

1998

Dissertation Committee:
Shoichiro Nakamura, Adviser
Mohammad Samimy
James N. Scott
William A. Smith

Approved by
Adviser
Department of Mechanical Engineering
ABSTRACT

The IVAS (Innovative Ventricular Assist System) pump, which is being developed at The Cleveland Clinic Foundation, is a non-pulsatile centrifugal blood pump. The primary objective of computational fluid dynamics (CFD) in this work is to aid in the design from the fluid mechanics point of view, i.e., to investigate the flow behavior inside the pump. A three-dimensional (3D) unsteady incompressible Navier-Stokes solver has been developed based on a new algorithm. The new flow solver has the following features: (1) accurate mass and momentum conservation; (2) little or no spurious spatial oscillation; (3) a small amount of numerical viscosity; (4) no numerical instability; (5) no difficulties in the pressure equation convergence; (6) robustness for high Reynolds number for the no turbulence model. There are three parts of the pump that need to be analyzed: (a) primary pump; (b) secondary pump; (c) journal bearing. In the investigation of the primary pump, 2-1/2-dimensional (2.5D) and 3-dimensional single passage models, and mean-line Pumpa code (one-dimensional (1D) model) developed by NASA Lewis Research Center are used. The 1D model, based on empirical data is simple and fast. The 2.5D single passage model has been tested as a feasible tool for a parametric study and the 3D model captures some flow structures which can not be seen through the 2.5D model. For the secondary pump, there is no conventional design tool available because of its unconventional features.
The 3D computational analysis investigates the flow patterns for different designs and the tip clearance effect on the pressure head. For the journal bearing design, the 3D simulation captures the flow patterns and a parametric study is performed. For all three parts, the shear stress distributions are calculated as a measure of the biological effects of the flow regimes to the blood. The new solvers including the 2.5D, the 3D, and the axisymmetric code work as useful tools in the designing of the IVAS. Simulation results provide designers with important information to improve the flow behavior inside the pump. Furthermore, these tools are not limited to the blood pump design, but can be used for other turbomachinery design as well.
Dedicated to my wife, Jian Tang
ACKNOWLEDGMENTS

I want to thank my adviser, Professor Shoichiro Nakamura, for intellectual support, encouragement, and enthusiasm in this project. I respect his work, and have learned a lot from him. I hope I can still work with him in the future.

I thank Dr. William A. Smith for his suggestions in the project.

I thank Professor Mohammad Samimy and Professor James Scott for discussing with me in various aspects of this dissertation.

I also thank Mr. James Brady for his help in editing the dissertation.

This research was supported by a grant from The National Institutes of Health through The Cleveland Clinic Foundation. I am also grateful for the award of computer time on the Cray T90 at The Ohio Supercomputer Center and for the RCL in Mechanical Engineering Department, without which the calculation would not have been possible.

I wish to thank my parents for their encouragement and help.

Finally, I want to express my special thanks to my wife. Without her encouragement and endearment I could not finish this dissertation.
VITA

November 22, 1965 .............................. Born - Nanjing, P. R. China

1988 ..................................................... B.S. Wuhan University of Water Transportation Engineering

1991 ..................................................... M.S. Wuhan University of Water Transportation Engineering

1991-1994 .............................................. Researcher, Nanjing College of Communications, Nanjing, P.R. China

1994-present ......................................... Graduate Research Associate, The Ohio State University

PUBLICATIONS

Research Publications


FIELDS OF STUDY

Major Field: Mechanical Engineering
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abstract</td>
<td>ii</td>
</tr>
<tr>
<td>Dedication</td>
<td>iv</td>
</tr>
<tr>
<td>Acknowledgments</td>
<td>v</td>
</tr>
<tr>
<td>Vita</td>
<td>vi</td>
</tr>
<tr>
<td>List of Tables</td>
<td>xi</td>
</tr>
<tr>
<td>List of Figures</td>
<td>xii</td>
</tr>
<tr>
<td><strong>Chapters:</strong></td>
<td></td>
</tr>
<tr>
<td>1. Introduction</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Organizations of the dissertation</td>
<td>1</td>
</tr>
<tr>
<td>1.2 A historical review of the artificial heart pumps</td>
<td>2</td>
</tr>
<tr>
<td>1.3 Why the centrifugal heart pumps</td>
<td>4</td>
</tr>
<tr>
<td>1.4 Some new designs of the centrifugal heart pumps</td>
<td>5</td>
</tr>
<tr>
<td>1.5 Objectives of the present study</td>
<td>8</td>
</tr>
<tr>
<td>2. Numerical solutions of the incompressible Navier-Stokes equations</td>
<td>11</td>
</tr>
<tr>
<td>2.1 The vorticity-stream function method</td>
<td>11</td>
</tr>
<tr>
<td>2.2 The MAC algorithm</td>
<td>13</td>
</tr>
<tr>
<td>2.3 The SMAC method</td>
<td>16</td>
</tr>
<tr>
<td>2.4 Pantanker's method</td>
<td>17</td>
</tr>
<tr>
<td>2.5 The PISO algorithm</td>
<td>22</td>
</tr>
<tr>
<td>2.6 The projection method (the fractional steps method)</td>
<td>24</td>
</tr>
</tbody>
</table>
2.7 The artificial compressibility method ................................................... 25

3. Fully conservative forms in the Navier-Stokes solver ................................. 26
   3.1 Governing equations ............................................................................ 27
   3.2 Discretization of the momentum equations ............................................. 27
      3.2.1 Non-conservative form .................................................................. 28
      3.2.2 Conservative form ....................................................................... 29
   3.3 The Rhie-Chow interpolation .................................................................. 30
   3.4 Boundary conditions ............................................................................. 36
   3.5 Derivation of the 3D governing equations ............................................. 38
      3.5.1 The SMAC method .................................................................... 38
      3.5.2 Coordinate transformation in a strong conservation law form ....... 40
      3.5.3 Differencing of the pressure correction equation ....................... 42

4. The journal Bearing .................................................................................... 45
   4.1 Circular shaft ........................................................................................ 46
   4.2 Non-circular shaft (more realistic model) ............................................. 59
   4.3 A parametric study ................................................................................ 67
      4.3.1 Group I: RPM effect ................................................................... 67
      4.3.2 Group II: pressure drop effect .................................................... 72
      4.3.3 Group III: the minimum gap effect ............................................ 76
   4.4 Investigation of hemolysis ...................................................................... 81
   4.5 Summary ............................................................................................... 82

5. The Secondary impeller ............................................................................. 84
   5.1 Torus cavity ........................................................................................ 84
   5.2 Secondary impeller cavity .................................................................... 93
      5.2.1 Straight blade design .................................................................... 93
      5.2.2 Wavy blade design ....................................................................... 106
      5.2.3 A newer design: B3548 ................................................................ 111
   5.3 Conclusions ........................................................................................... 131

6. The 2-1/2-dimensional single passage model for the primary pump ............. 166
   6.1 The reference case ................................................................................ 168
   6.2 Effect of the volute width ..................................................................... 169
   6.3 Effect of the volute height distribution ................................................. 169
   6.4 Pump flow rate ...................................................................................... 170
   6.5 RPM effect ............................................................................................ 171
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1 Parameters of the eccentric journal bearing</td>
<td>46</td>
</tr>
<tr>
<td>4.2 Some parameters of the non-circular shaft case</td>
<td>59</td>
</tr>
<tr>
<td>4.3 Group I parameters</td>
<td>67</td>
</tr>
<tr>
<td>4.4 Group II parameters</td>
<td>72</td>
</tr>
<tr>
<td>4.5 Group III parameters</td>
<td>76</td>
</tr>
<tr>
<td>5.1 Effects of tip clearance and blade angle</td>
<td>96</td>
</tr>
<tr>
<td>5.2 Parameters of the B3458 design</td>
<td>112</td>
</tr>
<tr>
<td>6.1 Parameters of the single passage model for the reference case</td>
<td>167</td>
</tr>
<tr>
<td>6.2 Other parameters of the reference case</td>
<td>168</td>
</tr>
<tr>
<td>6.3 Parameters of the B3452 design</td>
<td>174</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>The IVAS heart pump of The Cleveland Clinic Foundation</td>
<td>9</td>
</tr>
<tr>
<td>1.2</td>
<td>The IVAS heart pump assembly</td>
<td>10</td>
</tr>
<tr>
<td>2.1</td>
<td>A schema of a staggered grid</td>
<td>15</td>
</tr>
<tr>
<td>2.2</td>
<td>A finite volume with the SIMPLE notation</td>
<td>18</td>
</tr>
<tr>
<td>3.1</td>
<td>A control volume about point P</td>
<td>32</td>
</tr>
<tr>
<td>3.2</td>
<td>An illustration of the implementation of boundary condition</td>
<td>36</td>
</tr>
<tr>
<td>4.1</td>
<td>A gap between two cylindrical surfaces at eccentric positions</td>
<td>47</td>
</tr>
<tr>
<td>4.2</td>
<td>Pressure distribution versus theta ( z/H=0.5, \ j=6, \ H ) is the height of the cylinder, ( j=6 ) means the middle plane)</td>
<td>49</td>
</tr>
<tr>
<td>4.3</td>
<td>Velocity distribution at the middle level ( \text{RPM}=3,000, \ \Delta p=6,250 \text{ Pa}, \ z/H=0.5 )</td>
<td>50</td>
</tr>
<tr>
<td>4.4</td>
<td>Velocity profile at the maximum gap ( \text{RPM}=3,000, \ \Delta p=6,250 \text{ Pa}, \ z/H=0.5 )</td>
<td>51</td>
</tr>
<tr>
<td>4.5</td>
<td>Velocity profile at the minimum gap ( \text{RPM}=3,000, \ \Delta p=6,250 \text{ Pa}, \ z/H=0.5 )</td>
<td>51</td>
</tr>
<tr>
<td>4.6</td>
<td>Pressure contour plot at the middle level ( \text{RPM}=3,000, \ \Delta p=6,250 \text{ Pa}, \ z/H=0.5 )</td>
<td>52</td>
</tr>
</tbody>
</table>
4.7 Pressure contour on the middle plane (RPM=3,000, Δp=6,250 Pa, 
j=6, unit: bar) .......................................................... 53

4.8 Vertical velocity component at the inlet (RPM=3,000, Δp=6,250 Pa, 
z/H=1.0, top) .......................................................... 54

4.9 Vertical velocity component at the middle level (RPM=3,000, Δp=6,250 Pa, z/H=0.5, middle) .......................................................... 55

4.10 Vertical velocity component at the exit (RPM=3,000, Δp=6,250 Pa, 
z/H=0.0, bottom) .......................................................... 56

4.11 Velocity distributions at the top(inlet),middle, and bottom(exit) levels 
of the journal bearing .................................................. 57

4.12 Velocity distributions at the top(inlet),middle, and bottom(exit) levels 
of the journal bearing (3D presentation of Figure 4.11) .............. 58

4.13 MTF's film thickness versus theta ...................................... 61

4.14 Film thickness versus theta in our calculation ..................... 61

4.15 Computed pressure distribution versus theta (z/H=0.5, j=6) ..... 62

4.16 Pressure contour on the middle plane (RPM=3,000, Δp=6,250 Pa, 
j=6, unit: bar) .......................................................... 63

4.17 Velocity distributions at the top(inlet),middle, and bottom(exit) levels 
of the journal bearing .................................................. 64

4.18 Computed pressure distribution versus theta (mesh size effect) .. 65

4.19 Pressure contour on the middle plane (RPM=3,000, Δp=6,250 Pa, 
mesh size effect, unit: bar) ........................................... 66

4.20 Mass flow rate versus rotational speed ................................ 69

4.21 Pressure distributions versus theta (z/H = 0.5, j = 3) ............. 70
4.22 Pressure contour on the middle plane \((z/H = 0.5, j = 3)\) LT: \(RPM = 0\), RT: \(RPM = 1,500\), LB: \(RPM = 3,000\), RB: \(RPM = 4,500\) ... 71

4.23 Mass flow rate versus pressure drop .......................................................... 73

4.24 Pressure distributions versus theta \((z/H = 0.5, j = 3)\) .................................. 74

4.25 Pressure distribution on the middle plane \((z/H = 0.5, j = 3)\) LT: \(\Delta p = 3,125\), RT: \(\Delta p = 6,250\), LB: \(\Delta p = 12,500\) ............................... 75

4.26 Mass flow rate versus the minimum gap ..................................................... 78

4.27 Pressure distributions versus theta \((z/H = 0.5, j = 3)\) .................................. 79

4.28 Pressure distribution on the middle plane \((z/H = 0.5, j = 3\) for different minimum gap) LT: \(C/C_f = 0.8\), RT: \(C/C_f = 1.0\), LB: \(C/C_f = 1.2\) ... 80

4.29 Shear stress at the solid wall causing hemolysis in the journal bearing 83

5.1 Flow domain model for the secondary impeller cavity .......................... 85

5.2 Non-uniform velocity distribution at the inlet of the secondary impeller cavity ................................................................. 86

5.3 Cross sectional view of the flow patterns for the non-uniform flow from the journal bearing ................................................................. 88

5.4 Time history of the flow patterns with \(v_\theta/v_{ver} = 4:1\) (Re=100, time step is 100, 200, 400, 600, from the top to bottom) ........................................... 89

5.5 Time history of the flow patterns with \(v_\theta/v_{ver} = 1:1\) (Re=100, time step is 100, 200, 400, 600, from the top to bottom) ........................................... 90

5.6 Time history of the flow patterns with \(v_\theta/v_{ver} = 0.5:1\) (Re=100, time step is 100, 400, 600, from the top to bottom) ........................................... 91

5.7 Time history of the flow patterns with \(v_\theta/v_{ver} = 0:1\) (Re=100, time step is 100, 200, 400, 600, from the top to bottom) ........................................... 92

5.8 Straight blade design at CCF ................................................................. 94

xiv
5.9 Velocity distribution on the xy plane at different horizontal levels (Re=23,800, dt=0.01, step=9,000, H=0.2 mm) .............................................. 97

5.10 Velocity distribution on the xy plane at different horizontal levels (Re=23,800, dt=0.01, step=9,000, H=0.2 mm) .............................................. 98

5.11 Velocity distribution on the θz plane at different radii (Re=23,800, dt=0.01, step=9,000, H=0.2 mm) ............................................................. 99

5.12 Pressure distributions at different horizontal levels for case (1) . . . . 100

5.13 Streamline plot 1 ................................................................................ 101

5.14 Streamline plot 2 ................................................................................ 101

5.15 Streamline plot 3 ................................................................................ 101

5.16 Velocity distribution on the xy plane at different horizontal levels (Re=23,800, dt=0.01, step=9,000, H=0.5 mm) ....................................... 102

5.17 Velocity distribution on the xy plane at different horizontal levels (Re=23,800, dt=0.01, step=9,000, H=0.5 mm) ....................................... 103

5.18 Velocity distribution on the θz plane at different radii (Re=23,800, dt=0.01, step=9,000, H=0.5 mm) ............................................................. 104

5.19 Distribution of shear stress on selected grid planes in the secondary impeller cavity (D0 is the diameter of the inner cylinder) ........... 105

5.20 An example of the geometry modeling ............................................. 109

5.21 The geometry of the wavy blade design of the secondary impeller . 109

5.22 Pressure contour on the blade surface ............................................. 110

5.23 Streamline plot 1 ................................................................................ 110

5.24 Streamline plot 2 ................................................................................ 110
5.45 The reference case: \(i=52\) .................................................. 118
5.46 The reference case: \(i=56\) .................................................. 118
5.47 The reference case: \(i=58\) .................................................. 118
5.48 The reference case: \(j=2\) .................................................. 118
5.49 The reference case: \(j=6\) .................................................. 118
5.50 The reference case: \(j=10\) ................................................ 119
5.51 The reference case: \(j=14\) ................................................ 119
5.52 The reference case: \(j=18\) ................................................ 119
5.53 The reference case: \(j=22\) ................................................ 119
5.54 The reference case: \(j=26\) ................................................ 119
5.55 The reference case: \(j=30\) ................................................ 119
5.56 The reference case: \(j=34\) ................................................. 120
5.57 The reference case: \(j=36\) ................................................. 120
5.58 The reference case: \(j=38\) ................................................. 120
5.59 The reference case: \(j=40\) ................................................. 120
5.60 The reference case: \(j=42\) ................................................. 120
5.61 The reference case: \(j=44\) ................................................. 120
5.62 The reference case: \(j=48\) ................................................. 121
5.63 The reference case: \(j=52\) ................................................. 121
5.64 The reference case: \(j=56\) ................................................. 121

xvii
5.65 The reference case: j = 60 ........................................... 121
5.66 The reference case: j = 64 ........................................... 121
5.67 The reference case: j = 68 ........................................... 121
5.68 The reference case: k = 5 ........................................... 122
5.69 The reference case: k = 10 ........................................... 122
5.70 The reference case: k = 15 ........................................... 122
5.71 Shear stress distribution on the grid plane j = 36 (Unit: pa) ................. 122
5.72 Shear stress distribution on the grid plane j = 38 (Unit: pa) ................. 122
5.73 Shear stress distribution on the grid plane j = 40 (Unit: pa) ................. 122
5.74 Tip Gap increased case: i = 2 ........................................ 123
5.75 Tip Gap increased case: i = 6 ........................................ 123
5.76 Tip Gap increased case: i = 10 ....................................... 123
5.77 Tip Gap increased case: i = 14 ....................................... 123
5.78 Tip Gap increased case: i = 18 ....................................... 123
5.79 Tip Gap increased case: i = 22 ....................................... 123
5.80 Tip Gap increased case: i = 26 ....................................... 124
5.81 Tip Gap increased case: i = 30 ....................................... 124
5.82 Tip Gap increased case: i = 32 ....................................... 124
5.83 Tip Gap increased case: i = 36 ....................................... 124
5.84 Tip Gap increased case: i = 40 ....................................... 124

xviii
5.85 Tip Gap increased case: i = 44 ................................................................. 124
5.86 Tip Gap increased case: i = 48 ................................................................. 125
5.87 Tip Gap increased case: i = 52 ................................................................. 125
5.88 Tip Gap increased case: i = 56 ................................................................. 125
5.89 Tip Gap increased case: i = 58 ................................................................. 125
5.90 Tip Gap increased case: j = 2 ................................................................. 125
5.91 Tip Gap increased case: j = 6 ................................................................. 125
5.92 Tip Gap increased case: j = 10 ................................................................. 126
5.93 Tip Gap increased case: j = 14 ................................................................. 126
5.94 Tip Gap increased case: j = 18 ................................................................. 126
5.95 Tip Gap increased case: j = 22 ................................................................. 126
5.96 Tip Gap increased case: j = 26 ................................................................. 126
5.97 Tip Gap increased case: j = 30 ................................................................. 126
5.98 Tip Gap increased case: j = 34 ................................................................. 127
5.99 Tip Gap increased case: j = 36 ................................................................. 127
5.100 Tip Gap increased case: j = 38 ................................................................. 127
5.101 Tip Gap increased case: j = 40 ................................................................. 127
5.102 Tip Gap increased case: j = 42 ................................................................. 127
5.103 Tip Gap increased case: j = 44 ................................................................. 127
5.104 Tip Gap increased case: j = 48 ................................................................. 128

xix
5.105 Tip Gap increased case: $j=52$ .......................................................... 128
5.106 Tip Gap increased case: $j=56$ .......................................................... 128
5.107 Tip Gap increased case: $j=60$ .......................................................... 128
5.108 Tip Gap increased case: $j=64$ .......................................................... 128
5.109 Tip Gap increased case: $j=68$ .......................................................... 128
5.110 Tip Gap increased case: $k=5$ .............................................................. 129
5.111 Tip Gap increased case: $k=10$ ............................................................ 129
5.112 Tip Gap increased case: $k=15$ ............................................................ 129
5.113 Tip clearance on the pressure head .................................................. 129
5.114 Geometry of hang over part ............................................................... 133
5.115 Straight case: $i=3$ .......................................................... 133
5.116 Straight case: $i=4$ .......................................................... 133
5.117 Straight case: $i=5$ .......................................................... 133
5.118 Straight case: $i=6$ .......................................................... 133
5.119 Straight case: $i=7$ .......................................................... 133
5.120 Straight case: $i=8$ .......................................................... 134
5.121 Straight case: $i=9$ .......................................................... 134
5.122 Straight case: $i=10$ .......................................................... 134
5.123 Straight case: $i=12$ .......................................................... 134
5.124 Straight case: $i=14$ .......................................................... 134

xx
5.145 Straight case: \( j=19 \) .................................................. 138
5.146 Straight case: \( j=21 \) .................................................. 138
5.147 Straight case: \( j=23 \) .................................................. 138
5.148 Straight case: \( j=25 \) .................................................. 138
5.149 Straight case: \( j=27 \) .................................................. 138
5.150 Straight case: \( j=28 \) .................................................. 139
5.151 Straight case: \( j=29 \) .................................................. 139
5.152 Straight case: \( j=30 \) .................................................. 139
5.153 Straight case: \( j=31 \) .................................................. 139
5.154 Straight case: \( j=32 \) .................................................. 139
5.155 Straight case: \( j=33 \) .................................................. 139
5.156 Straight case: \( j=34 \) .................................................. 140
5.157 Straight case: \( j=35 \) .................................................. 140
5.158 Straight case: \( j=37 \) .................................................. 140
5.159 Straight case: \( j=39 \) .................................................. 140
5.160 Straight case: \( j=41 \) .................................................. 140
5.161 Straight case: \( j=43 \) .................................................. 140
5.162 Straight case: \( j=45 \) .................................................. 141
5.163 Straight case: \( j=47 \) .................................................. 141
5.164 Straight case: \( j=49 \) .................................................. 141

xxii
5.165 Straight case: j=51 ........................................ 141
5.166 Straight case: j=53 ........................................ 141
5.167 Straight case: j=55 ........................................ 141
5.168 Straight case: j=57 ........................................ 142
5.169 Straight case: j=59 ........................................ 142
5.170 Straight case: k=2 ........................................ 142
5.171 Straight case: k=4 ........................................ 142
5.172 Straight case: k=5 ........................................ 142
5.173 Straight case: k=6 ........................................ 142
5.174 Straight case: k=8 ........................................ 143
5.175 Straight case: k=10 ...................................... 143
5.176 Straight case: k=12 ...................................... 143
5.177 Straight case: k=13 ...................................... 143
5.178 Straight case: k=14 ...................................... 143
5.179 Straight case: k=15 ...................................... 143
5.180 Straight case: k=16 ...................................... 144
5.181 Straight case: k=17 ...................................... 144
5.182 Straight case: k=18 ...................................... 144
5.183 Straight case: k=19 ...................................... 144
5.184 Straight case: k=20 ...................................... 144

xxiii
5.185 Straight case: k=21 ............................................. 144
5.186 Straight case: k=22 ............................................. 145
5.187 Straight case: k=23 ............................................. 145
5.188 Straight case: k=24 ............................................. 145
5.189 Straight case: k=25 ............................................. 145
5.190 Straight case: k=26 ............................................. 145
5.191 Straight case: k=27 ............................................. 145
5.192 Straight case: k=28 ............................................. 146
5.193 Straight case: k=29 ............................................. 146
5.194 Straight case: k=30 ............................................. 146
5.195 Straight case: k=31 ............................................. 146
5.196 Straight case: k=32 ............................................. 146
5.197 Straight case: k=33 ............................................. 146
5.198 Straight case: k=34 ............................................. 147
5.199 Straight case: k=35 ............................................. 147
5.200 Straight case: k=37 ............................................. 147
5.201 Straight case: k=39 ............................................. 147
5.202 Straight case: k=41 ............................................. 147
5.203 Straight case: k=43 ............................................. 147
5.204 Straight case: k=45 ............................................. 148

xxiv
5.205 Straight case: k = 47 ................................................................. 148
5.206 Straight case: k = 49 ................................................................. 148
5.207 Straight case: k = 51 ................................................................. 148
5.208 Straight case: k = 53 ................................................................. 148
5.209 Straight case: k = 55 ................................................................. 148
5.210 Streamline plot 1 ................................................................. 149
5.211 Streamline plot 2 ................................................................. 149
5.212 Streamline plot 3 ................................................................. 149
5.213 Shear stress distribution on the grid planes j = 29, j = 33, and j = 35 (unit: Pascal) ................................................................. 149
5.214 Shear stress distribution on the outer radius grid plane (unit: Pascal) ................................................................. 149
5.215 Shear stress distribution on the bottom grid plane (unit: Pascal) ................................................................. 149
5.216 Slanted case: i = 2 ................................................................. 150
5.217 Slanted case: i = 3 ................................................................. 150
5.218 Slanted case: i = 4 ................................................................. 150
5.219 Slanted case: i = 5 ................................................................. 150
5.220 Slanted case: i = 6 ................................................................. 150
5.221 Slanted case: i = 7 ................................................................. 150
5.222 Slanted case: i = 8 ................................................................. 151
5.223 Slanted case: i = 9 ................................................................. 151
5.224 Slanted case: i = 10 ................................................................. 151

xxv
5.265 Slanted case: $j=47$ .................................................. 158
5.266 Slanted case: $j=49$ .................................................. 158
5.267 Slanted case: $j=51$ .................................................. 158
5.268 Slanted case: $j=53$ .................................................. 158
5.269 Slanted case: $j=55$ .................................................. 158
5.270 Slanted case: $j=57$ .................................................. 159
5.271 Slanted case: $j=59$ .................................................. 159
5.272 Slanted case: $k=2$ .................................................. 159
5.273 Slanted case: $k=4$ .................................................. 159
5.274 Slanted case: $k=5$ .................................................. 159
5.275 Slanted case: $k=6$ .................................................. 159
5.276 Slanted case: $k=8$ .................................................. 160
5.277 Slanted case: $k=10$ ................................................. 160
5.278 Slanted case: $k=12$ ................................................. 160
5.279 Slanted case: $k=13$ ................................................. 160
5.280 Slanted case: $k=14$ ................................................. 160
5.281 Slanted case: $k=15$ ................................................. 160
5.282 Slanted case: $k=16$ ................................................. 161
5.283 Slanted case: $k=17$ ................................................. 161
5.284 Slanted case: $k=18$ ................................................. 161

xxviii
5.305 Slanted case: k=43 .............................................................. 164
5.306 Slanted case: k=45 .............................................................. 165
5.307 Slanted case: k=47 .............................................................. 165
5.308 Slanted case: k=49 .............................................................. 165
5.309 Slanted case: k=51 .............................................................. 165
5.310 Slanted case: k=53 .............................................................. 165
5.311 Slanted case: k=55 .............................................................. 165

6.1 Geometry of the single passage model ........................................ 176
6.2 Grid for the calculation ............................................................. 176
6.3 Height distribution of the reference case ...................................... 177
6.4 Pressure contour for the reference case ....................................... 177
6.5 Streamline plot for the reference case ........................................ 178
6.6 Tangentially averaged static and total pressure distribution ......... 178
6.7 Three models of the volute height profile (plot of three height profiles including the reference case) .................................................... 179
6.8 Effect of volute height (plot of three cases of the tangentially averaged static and total pressure) ......................................................... 179
6.9 Pump power vs. flow rate ........................................................ 180
6.10 Pump rotor head vs flow rate .................................................... 180
6.11 Pressure contour plot for RPM=2000 ...................................... 181
6.12 Streamline plot for RPM=2000 ............................................... 181

xxx
6.13 Pressure contour plot for RPM=1000 ..................................................... 182
6.14 Streamline plot for RPM=1000 ............................................................... 182
6.15 Comparison of the tangentially averaged static and total pressure among
three RPMs ................................................................................................... 183
6.16 Power and rotor head vs. leading edge angle ......................................... 183
6.17 Power and rotor head vs. trailing edge angle ........................................ 184
6.18 A time sequence of pressure contour plots ............................................. 184
6.19 A time sequence of streamline plots ..................................................... 185
6.20 B3452 design at CCF ................................................................................. 185
6.21 The tangentially averaged static and total pressure versus radius . . . 186
6.22 Streamline plot of the B3452 design at 60,000 time step .................... 186
6.23 Velocity vector of the B3452 design at 60,000 time step ..................... 187
6.24 Pressure contour of the B3452 design at 60,000 time step ................. 187
6.25 A time sequence of pressure contour plots of the B3452 design ........ 188
6.26 A time sequence of velocity vector plots of the B3452 design .......... 188
7.1 Concept of the single passage model (seven passages are include in the
view ............................................................................................................. 191
7.2 A view of the grid for simulation ........................................................... 191
7.3 Cyclic boundary conditions in the single passage model .................... 192
7.4 Streamline plot ....................................................................................... 196
7.5 Velocity vectors on the grid plane k=5 .................................................... 196

xxxi
7.6 Velocity vectors on the grid plane k=9 .................................................. 196
7.7 Velocity vectors on the grid plane k=13 .................................................. 196
7.8 Velocity vectors on the grid plane k=17 .................................................. 196
7.9 Pressure contour on the grid plane k=5 .................................................. 196
7.10 Pressure contour on the grid plane k=9 .................................................. 197
7.11 Pressure contour on the grid plane k=13 ............................................... 197
7.12 Pressure contour on the grid plane k=17 ............................................... 197
7.13 Velocity vectors on the grid plane j=5 .................................................. 197
7.14 Velocity vectors on the grid plane j=12 .................................................. 197
7.15 Velocity vectors on the grid plane j=18 .................................................. 197
7.16 Velocity vectors on the grid plane j=25 .................................................. 198
7.17 Pressure contour on the grid plane j=5 .................................................. 198
7.18 Pressure contour on the grid plane j=12 .................................................. 198
7.19 Pressure contour on the grid plane j=18 .................................................. 198
7.20 Pressure contour on the grid plane j=25 .................................................. 198
7.21 Top: Rotor head comparison, Middle: Rotor head versus flow rate, Bottom: Tip clearance effect .................................................. 198
7.22 Shear stress contour on grid plane k=3 .................................................. 199
7.23 Shear stress contour on grid plane k=12 .................................................. 199
7.24 Shear stress contour on grid plane k=18 .................................................. 199

xxxii
7.25 B3452 design: Velocity vectors on the grid plane k=3 ................................ 202
7.26 B3452 design: Velocity vectors on the grid plane k=5 ................................ 202
7.27 B3452 design: Velocity vectors on the grid plane k=7 ................................ 202
7.28 B3452 design: Pressure contour on the grid plane k=3 ................................ 202
7.29 B3452 design: Pressure contour on the grid plane k=5 ................................ 202
7.30 B3452 design: Pressure contour on the grid plane k=7 ................................ 202
7.31 B3452 design: Velocity vectors on the grid plane k=9 ................................ 203
7.32 B3452 design: Velocity vectors on the grid plane k=11 ................................ 203
7.33 B3452 design: Velocity vectors on the grid plane k=13 ................................ 203
7.34 B3452 design: Pressure contour on the grid plane k=9 ................................ 203
7.35 B3452 design: Pressure contour on the grid plane k=11 ................................ 203
7.36 B3452 design: Pressure contour on the grid plane k=13 ................................ 203
7.37 B3452 design: Velocity vectors on the grid plane k=14 ................................ 204
7.38 B3452 design: Velocity vectors on the grid plane k=15 ................................ 204
7.39 B3452 design: Velocity vectors on the grid plane k=16 ................................ 204
7.40 B3452 design: Pressure contour on the grid plane k=14 ................................ 204
7.41 B3452 design: Pressure contour on the grid plane k=15 ................................ 204
7.42 B3452 design: Pressure contour on the grid plane k=16 ................................ 204
7.43 B3452 design: Velocity vectors on the grid plane k=17 ................................ 205
7.44 B3452 design: Velocity vectors on the grid plane k=19 ................................ 205

xxxiii
<table>
<thead>
<tr>
<th>No.</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.45</td>
<td>B3452 design: Velocity vectors on the grid plane j=5</td>
<td>205</td>
</tr>
<tr>
<td>7.46</td>
<td>B3452 design: Pressure contour on the grid plane k=17</td>
<td>205</td>
</tr>
<tr>
<td>7.47</td>
<td>B3452 design: Pressure contour on the grid plane k=19</td>
<td>205</td>
</tr>
<tr>
<td>7.48</td>
<td>B3452 design: Pressure contour on the grid plane j=5</td>
<td>205</td>
</tr>
<tr>
<td>7.49</td>
<td>B3452 design: Velocity vectors on the grid plane j=7</td>
<td>206</td>
</tr>
<tr>
<td>7.50</td>
<td>B3452 design: Velocity vectors on the grid plane j=9</td>
<td>206</td>
</tr>
<tr>
<td>7.51</td>
<td>B3452 design: Velocity vectors on the grid plane j=11</td>
<td>206</td>
</tr>
<tr>
<td>7.52</td>
<td>B3452 design: Pressure contour on the grid plane j=7</td>
<td>206</td>
</tr>
<tr>
<td>7.53</td>
<td>B3452 design: Pressure contour on the grid plane j=9</td>
<td>206</td>
</tr>
<tr>
<td>7.54</td>
<td>B3452 design: Pressure contour on the grid plane j=11</td>
<td>206</td>
</tr>
<tr>
<td>7.55</td>
<td>B3452 design: Velocity vectors on the grid plane j=13</td>
<td>207</td>
</tr>
<tr>
<td>7.56</td>
<td>B3452 design: Velocity vectors on the grid plane j=15</td>
<td>207</td>
</tr>
<tr>
<td>7.57</td>
<td>B3452 design: Velocity vectors on the grid plane j=17</td>
<td>207</td>
</tr>
<tr>
<td>7.58</td>
<td>B3452 design: Pressure contour on the grid plane j=13</td>
<td>207</td>
</tr>
<tr>
<td>7.59</td>
<td>B3452 design: Pressure contour on the grid plane j=15</td>
<td>207</td>
</tr>
<tr>
<td>7.60</td>
<td>B3452 design: Pressure contour on the grid plane j=17</td>
<td>207</td>
</tr>
<tr>
<td>7.61</td>
<td>B3452 design: Velocity vectors on the grid plane j=19</td>
<td>208</td>
</tr>
<tr>
<td>7.62</td>
<td>B3452 design: Velocity vectors on the grid plane j=21</td>
<td>208</td>
</tr>
<tr>
<td>7.63</td>
<td>B3452 design: Velocity vectors on the grid plane j=23</td>
<td>208</td>
</tr>
<tr>
<td>7.64</td>
<td>B3452 design: Pressure contour on the grid plane j=19</td>
<td>208</td>
</tr>
</tbody>
</table>
7.65 B3452 design: Pressure contour on the grid plane j=21 .......................... 208
7.66 B3452 design: Pressure contour on the grid plane j=23 .......................... 208
7.67 B3452 design: Velocity vectors on the grid plane j=25 .......................... 209
7.68 B3452 design: Pressure contour on the grid plane j=27 .......................... 209
7.69 B3452 design: Velocity vectors on the grid plane j=27 .......................... 209
7.70 B3452 design: Angular velocity contour on the grid plane j=15 ............... 209
7.71 B3452 design: Streamlines in the inlet viewed from the rotating frame .......................... 209
7.72 B3452 design: Streamlines in the inlet flow viewed from the fixed frame .......................... 209
CHAPTER 1

INTRODUCTION

The non-pulsatile blood pump, which is being developed at The Cleveland Clinic Foundation (CCF), is a centrifugal blood pump. The fluid dynamics of a blood pump plays an important role in its design and operation. Stagnation flow can cause thrombus formation, and high fluid shear rates will damage red blood cells. Also, a blood pump should have high hydraulic and mechanical efficiency so that a small power source can be used.

1.1 Organizations of the dissertation

This Chapter is an introduction and includes a historical review of the artificial heart pumps, the advantages of the centrifugal heart pumps, and the objectives of computational fluid dynamics (CFD). After that, a discussion and comparison of numerical methods for incompressible flow are made in Chapter 2. Chapter 3 introduces the fundamentals of the current work. Chapter 4 through Chapter 7 present details of the simulations for the journal bearing, the secondary pump, and the 2.5-dimensional (2.5D) and 3-dimensional (3D) models of the primary pump. Finally, conclusions will be made and some recommendations for the future work will be described in Chapter 8.
1.2 A historical review of the artificial heart pumps

It is estimated that 25,000 to 30,000 people suffer from weak or dying hearts annually in the United States, according to The National Heart, Lung and Blood Institutes (NHLBI). The mechanical heart pump can serve as a bridge for the patients awaiting heart transplantation.

Actually, the need for the mechanical cardiovascular support started with the beginning of open heart surgery. In 1952, a temporary support for blood circulation was achieved by the development of a cardiopulmonary bypass[1]. The stimulus for the development of a heart pump started in 1963, with The National Heart Institute beginning an artificial heart pump program in 1964[2]. The first total artificial heart (TAH) in patients, which was not for permanent implantation, was developed and implanted in Houston by Cooley and Liotta in 1969, and again in 1981. The first permanent implantation of a TAH was in Salt Lake City by DeVries and Joyce in 1982. The Symbio Jarvik-7 TAH[3] was developed and used in patients by mid 80's. By 1993, more than 240 TAHs have been used as bridges in transplantation applications[4].

Another type of devices which has been commonly used is ventricular assist system (VAS). In 1963, a centrifugal blood pump of VAS was first successfully used by Spencer. With the emphasis shifted to the left ventricular assist systems and blood compatible materials, VAS blood pumps became smaller, lighter, more acceptable and clinically successful. An implantable left heart bypass was used successfully in 1970[5]. By mid-1970s, pulsatile ventricular assist devices were being tested as bridges in transplantation. As well as the centrifugal pumps, the axial pumps have been used
with varying degrees of success[6][7]. However, such devices were to be used for a limited duration. An intra-corporeal left ventricular assist system (LVAS) as a bridge in transplantation was achieved by Kornos and Griffith in Pittsburgh in 1990 and by Frazier in Houston in 1992. Extensive clinical trials demonstrated improved pump reliability and clinic success.

The Institute of Medicine Review[8] gave a summary of the artificial heart devices in 1991: Short-term ventricular assist pumps include the ABIOMED system, the Biomedicus pump, the Novacot system, and the Sarns Centrimed system. Short-term TAH is the Penn State artificial heart. Long-term assist heart pump devices are the ABIOMED, the Nimbus, the Baxter, and the Penn State. Long-term TAHs are the ABIOMED, the Nimbus, and the Penn State with a support from The National Institutes of Health.

A mechanical pulsatile heart pump provides almost normal physiological conditions. However, it has the disadvantages of complicated drive mechanism and the need for artificial heart valves. On the other hand, a non-pulsatile pump is simple and small and does not require an artificial valve. But a non-pulsatile pump was not available for a long-term use. Moreover, it was not clear yet whether non-pulsatile flow can maintain the circulation physiologically for various demands. Besides, the problems associated with a non-pulsatile pump include hemolysis by the rotating impellers, thrombus formation behind impellers, and sealing a rotating shaft.

The Innovative Ventricular Assist System (IVAS) being developed at The Cleveland Clinic Foundation is a non-pulsatile centrifugal pump for a long-term use. Its
careful design intends to eliminate or reduce hemolysis and thrombus formation with high hydraulic efficiency.

1.3 Why the centrifugal heart pumps

The artificial heart pumps can be classified into two types, namely turbo and displacement types. A displacement pump, such as a diaphragm pump, is used for ventricular assist devices. While a turbo, a centrifugal pump for example, is used for extra-corporeal circulation. Furthermore, a turbo pump does not need valves. The problems are in the shaft bearing and seals.

According to the turbomachinery literature[9], the specific speed, \( n_s \), is defined as

\[
  n_s = \frac{NQ^{1/2}}{(gH)^{3/4}}
\]

where \( N(RPM) \) is the rotational speed of the impeller, \( Q(m^3/min.) \) the flow rate, \( H(m) \) the pump head, and \( g \) is the gravitational acceleration.

For a left heart, if the required pump head is about 100mmHg and the flow rate is about 5L/min, equation 1.1 indicates that the specific speed is proportional to the rotational speed. If a rotor is set to 1,500 – 6,700(RPM), the specific speed is from 90 to 270. From the maximum efficiency point of view, the centrifugal pumps are considered to be suitable for \( 80 < n_s < 1,400 \), while the axial pumps are suitable for \( n_s > 600 \). Therefore, the centrifugal pumps are more suitable for the above specific speed.

The pressure head of the centrifugal pumps, without loss, is given by

\[
  H_{th} = \frac{u_2v_{u2}}{g}
\]
where \( u_2 \) is the impeller velocity, \( v_{u2} \) is the tangential velocity component of the fluid at the impeller tip, and \( g \) is the gravitational acceleration.

For the axial flow pumps, the pressure head is

\[
H_{th} = \eta_{th} \frac{u_2 v_{u2}}{g}, \quad 1/2 \leq \eta_{th} \leq 1
\]

(1.3)

The ratio of the fluid velocity to the impeller velocity is mainly determined by discharge angle. For the centrifugal pumps, \( \frac{v_{u2}}{u_2} \leq 1 \), while for the axial-flow pumps, \( \frac{v_{u2}}{u_2} \leq \frac{1}{2} \). Therefore, the upper limits of the theoretical pump head are for the centrifugal pumps

\[
H_{th} \leq u_2^2 / g
\]

(1.4)

for the axial-flow pumps

\[
H_{th} \leq u_2^2 / 2g
\]

(1.5)

From the foregoing equations, we see that the axial pumps generate only one half of the head the centrifugal pumps generate with the same conditions. If the same pressure head is needed, the axial-flow pumps should be operated 41 percent faster than the centrifugal pumps. From hemolysis point of view, the centrifugal pumps seem to be safer than the axial-flow pumps.

### 1.4 Some new designs of the centrifugal heart pumps

The primary concern in designing a centrifugal blood pump is biocompatibility as related to blood trauma and coagulation. The next concerns are the performance of pump, safety and durability as well as ease of handling and control. Ordinary
centrifugal heart pumps have leakage, heat generation of the bearing and seals, as well as dead water zones and poor hydraulic efficiency, none of which are acceptable for a heart pump.

The Department of Cardiovascular Surgery in Germany developed a seal-less centrifugal blood pump with a magnetically suspended rotor[10] in 1995. This design intended to use a radial magnetic drive to eliminate a central shaft and seals.

Nippon Medical School [11] also exploited the magnetic drive to avoid bearing and seals. There is a washout hole under the impeller to avoid a stagnation zone under the vane in this design. This compact centrifugal pump system was developed and tested clinically to evaluate its fundamental properties and effectives.

Kyoto University Hospital and NTN Corporation developed a centrifugal pump with a magnetically suspended impeller[12]. In their study, the shape of the impeller, the number of blades, and the gap between the impeller discs and the pumping housing were investigated to increase efficiency and to decrease hemolysis. Hemolysis tests were carried out to find the optimal vane profile and gap clearance.

MITI and University of Tsukuba proposed a new mechanism of a mono-pivot magnetic-suspension centrifugal pump[13]. In their design, no seals or ball bearings were used. Therefore, the heat generation due to seal friction or flow stagnation around the bearing was significantly reduced.

Terumo Corporation in Japan developed a straight path centrifugal pump[14]. The unique straight path design successfully reduced the pump rotational speed and the prime volume of the rotor.
Baylor College of Medicine developed a completely seal-less centrifugal pump, the Baylor Gyro Pump, for a long-term use[15]. Their pump design showed good hemolysis in vitro with a sufficient pumping capacity. However, the heat generation level by the motor was high, which was not good for hemolysis. Another design developed in Baylor College of Medicine was the Baylor-Nikkiso centrifugal pump which utilized a purging system[16]. In this design, the most common stagnation area behind the impeller was constantly washed to minimize thrombus formation.

To overcome the shaft bearing and leakage problems, a magnetic coupling was used to eliminate the shaft, which penetrates the pump housing[10][11][12][13][14][15][16]. However, blood coagulation at the hot spots still occurred. Waseda University[17] presented a new concept of non-rotating type centrifugal pump, a kind of oscillating disk-type centrifugal pump. The disk is mounted on the edge of the rod. Leakage from the gap between the rod and the pump housing is prevented by a seal membrane. Thus, the rod and disk do not rotate but they oscillate in the housing.

Another unique design coming from Baylor College of Medicine is called a free impeller Gyro pump[20]. In their design, the rotational shaft and the mechanical bearing were completely eliminated to prevent stagnation around the shaft, particularly underneath the impeller. The impeller is lifted off the bottom casing and floated in the middle of the pump housing by employing a magnetic coupling. Because of low blood trauma as well as thrombus formation, this design can be used as a long-term centrifugal ventricular assist device.

The IVAS centrifugal pump being developed at The Cleveland Clinic Foundation [18][19] does not avoid the bearing. However, the lubricant is blood itself. In order
to prevent the dead water region in the secondary cavity, a secondary impeller is employed to compensate the pressure drop of the blood from the bearing channel so that the bearing blood is pumped up to the volute of the primary impeller. With all these features, this centrifugal pump should have low hemolysis and less thrombus formation. This design is shown in Figures 1.1 and 1.2.

1.5 Objectives of the present study

There are conventional ways to design the centrifugal pumps based on the turbo-machinery principles. However, there are some unconventional aspects in the IVAS pump such as the size of the entire pump; it weights 6 ounces including the motor and measures only \(1\frac{1}{2}\) inches by \(1\frac{3}{4}\) inches; about 50cc/min of blood flows through the journal bearing and the secondary impeller. In this study, CFD will play an important role in proposing improvement to the overall flow behavior of the pump.

Typically, pump performance is defined in terms of pressure head and energy consumption, which depends on flow rate, impeller size, rotational speed, and the working fluid properties. For a blood pump, however, another consideration is the physiological factor, namely, the damage of the blood due to the fluid motions, direct mechanical forces, and the exposure time to the non-biocompatible surfaces. It is believed that hemolysis is caused by high shear stress and long residence times within the artificial system lead to thrombus formation.

Therefore, the main objectives of CFD are to understand the flow behavior in the pump, to examine the existing design, and to help improve the flow behavior by eliminating or reducing the region of the reversed and stagnation flow in the pump at its operating conditions.
Figure 1.1: The IVAS heart pump of The Cleveland Clinic Foundation

There are three crucial parts in the IVAS pump: the journal bearing, the secondary pump, and the primary pump. Using blood as a lubricant is unique. The primary pump helps the heart’s left ventricle circulate blood through the body, while the secondary pump forces the blood through the journal bearing back to the main circulation.
Figure 1.2: The IVAS heart pump assembly
CHAPTER 2

NUMERICAL SOLUTIONS OF THE INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

For incompressible flow, the pressure field is implicitly governed by the momentum equations and the continuity equation, unlike for compressible flow, in which the density appears explicitly in the continuity equation and the pressure is determined by a suitable kinetic equation of state[21]. To overcome such a difficulty, two approaches can be adopted:

(1) The continuity constraint is exactly satisfied by rearranging of the equations. The vorticity-stream function method is an example.
(2) The continuity is enforced through iteration: MAC, SMAC, SIMPLE and PISO belong to this category.

In this chapter, several numerical methods dealing with the incompressible Navier-Stokes equations will be reviewed.

2.1 The vorticity-stream function method

In order to avoid the difficulties of solving a pressure field, the vorticity-stream function method can be used, in which the vorticity and stream function are the dependent variables rather than velocities and pressure[22][23].
For a two-dimensional case, the pressure terms in the momentum equations are eliminated by differentiating $u$ momentum equation with respect to $y$, differentiating $v$ momentum equation with respect to $x$, and subtracting one from the other. The vorticity is defined by

$$\omega = \frac{\partial u}{\partial y} - \frac{\partial v}{\partial x} \quad (2.1)$$

The vorticity equation is obtained as

$$\frac{\partial \omega}{\partial t} = -\vec{V} \cdot \nabla \omega + \nu \nabla^2 \omega \quad (2.2)$$

where $\vec{V}$ is the vector of $u$ and $v$, and $\nu$ is the kinematic viscosity.

If the stream function is defined by

$$\frac{\partial \phi}{\partial y} = u, \quad \frac{\partial \phi}{\partial x} = -v \quad (2.3)$$

then equation 2.1 becomes a Poisson equation:

$$\nabla^2 \phi = -\omega \quad (2.4)$$

The implementation of the boundary conditions is complicated, however. Thorough reviews of boundary conditions can be found in Roache[24], Gupta and Manohar[25], Baker[26], Quartapelle and Napolitano[27], Gunzburger[28], and Gresho[29].

The extension of the vorticity-stream function to three dimensions is called the vector potential method. The vector potential is defined as

$$\vec{\psi} = \nabla \times \vec{\phi} \quad (2.5)$$

Therefore, the vorticity transport equation is

$$\frac{\partial \omega_i}{\partial t} - \omega_j \frac{\partial u_i}{\partial x_j} + \frac{\partial (u_j \omega_i - \frac{1}{Re} \frac{\partial \omega_i}{\partial x_j})}{\partial x_j} = 0 \quad (2.6)$$
The second term in the foregoing equation represents the three-dimensional "vortex stretching."

The Poisson equation is obtained by taking the curl of equation 2.6. Using the vector identity $\nabla \times (\nabla \times \vec{V}) = \nabla(\nabla \cdot \vec{V}) - \nabla^2 \vec{V}$, we get

$$\nabla^2 \vec{V} = -\omega \quad (2.7)$$

Aziz and Hellums[30] used the ADI method to solve the vorticity transport equation, while the Poisson equation was solved by the SOR method. Arebesola[31] and Richardson[32] proposed vorticity with vector and scalar potential formulation to overcome the difficulties in the specified boundary conditions.

The vorticity with velocity method was first introduced by Fasel[33]. Dennis[34] extended it to three dimensions. The contributions to the vorticity-velocity method are continuing. Agarmal[35] investigated a third-order-upwind scheme for the advective terms using a non-staggered mesh. Guj[36] employed a staggered mesh and the ADI iterative method to calculate 2D flows. Napolitano[37] calculated both 2D and 3D flows.

### 2.2 The MAC algorithm

The marker-and-cell (MAC) method is one of the most well known approaches for incompressible flow with a free surface, which was originally developed at Los Alamos Scientific Laboratory[38][39]. The markers are massless particles that move with the free surface to trace the location of the free surface. In this method, a pressure Poisson equation is derived and the continuity is assumed to be satisfied after the velocity field is updated. The modified pressure Poisson equation is obtained by
taking the divergence of the momentum equations. Consider a two-dimensional case as an example, the pressure equation is

$$P_{xx} + P_{yy} = \sigma - D_t$$  \hspace{1cm} (2.8)

where

$$\sigma = -(uu_x + uu_y)_x - (uv_x + vv_y)_y$$

$$D = u_x + v_y$$

Equation 2.8 is a second-order elliptic partial differential equation. The unsteady term is approximated by

$$D_t = \frac{D^{n+1} - D^n}{\delta t}$$  \hspace{1cm} (2.9)

$D^{n+1}$ is enforced to satisfy the continuity equation, which is an essential part of the MAC method.

One important feature of the original MAC is the staggered grid. Velocities are defined at the cell boundaries while the pressure is defined at the cell center shown in Figure 2.1.

After the pressure field is calculated, the velocities are advanced explicitly using a central difference discretization of spatial derivation in the momentum equations:

$$u_{i+1/2,j}^{n+1} = u_{i+1/2,j}^n + \delta t \left[ \frac{(u_{i,j}^n)^2 - (u_{i+1,j}^n)^2}{\delta x} + \frac{(uv)_{i+1/2,j-1/2}^n - (uv)_{i+1/2,j+1/2}^n}{\delta y} + \frac{1}{Re} \left( \frac{(u_{i+3/2,j}^n)^2 + (u_{i-1/2,j}^n)^2 - 2(u_{i+1/2,j}^n)^2}{\delta x^2} + \frac{(u_{i+1/2,j+1}^n)^2 + (u_{i+1/2,j-1}^n) - 2(u_{i+1/2,j}^n)^2}{\delta y^2} \right) \right]$$  \hspace{1cm} (2.10)
However, there are some drawbacks of the MAC method:

(1) A staggered grid can overcome a checker-board pressure distribution, but more efforts are necessary in programming, also it needs more memory space. Furthermore, artificial points just outside the real boundary are needed to implement boundary conditions\[40\].

(2) A compatibility condition must be met in order to make the pressure iteration convergent when all the pressure boundary conditions are of the homogeneous Neumann type. It has been shown that the compatibility condition is not automatically guaranteed on either a staggered or a non-staggered grid. Abdallah\[41\] proposed a modified MAC method based on a non-staggered grid to satisfy the compatibility condition.
2.3 The SMAC method

The simplified marker-and-cell method, SMAC[42], is a revised version of the MAC method. The motivation of the SMAC method was to eliminate the difficulties in applying a non-homogeneous Neumann boundary condition for the pressure calculation in the MAC method. If the intermediate velocity field is not mass conservative at wall, the vorticity production and diffusion at a rigid wall will not be correct. In the SMAC method, the velocity field from the previous time step is divergence free, so the vorticity production and diffusion at a wall is correct. The SMAC method contains two basic steps: (1) in the predictor step, the momentum equations are solved by the previous pressure field, and (2) in the corrector step, the velocity field is modified by setting the difference between the final and the predicted velocity equal to the finite difference gradient of a potential function. The potential function is the solution of a Poisson equation, for which the boundary conditions are homogeneous. In the original SMAC, a staggered grid is used.

More details are as follows: The intermediate velocity field \( u^{n+1} \) is obtained by the explicit manner from the previous time step, i.e.,

\[
\begin{align*}
  u^*_{i+1/2,j}^{n+1} &= u^n_{i+1/2,j} + \delta t \left[ \frac{(u^2_{i,j})^n -(u^2_{i+1,j})^n}{\delta x} ight. \\
  &\quad + \frac{(uv)_{i+1/2,j-1/2}^n -(uv)_{i+1/2,j+1/2}^n}{\delta y} + \frac{p_{i,j}^n - p_{i+1,j}^n}{\delta x} \\
  &\quad + \frac{1}{Re} \left( \frac{(u_{i+3/2,j})^n + (u_{i-1/2,j})^n}{\delta x^2} - 2(u_{i+1/2,j})^n \\
  &\quad \left. + \frac{(u_{i+1/2,j+1})^n + (u_{i+1/2,j-1})^n - 2(u_{i+1/2,j})^n}{\delta y^2} \right) \right]
\end{align*}
\]  
\( (2.11) \)
The mass-conservation is satisfied by the pressure correction equation, which is a Poisson equation. The final step is to update the velocity field to make the intermediate velocity field to satisfy the continuity equation.

Ikohag[44][45] applied the SMAC method to a general curvilinear coordinate grid so that the continuity condition can be satisfied strictly. He used contravariant velocities, a staggered grid and a modified QUICK scheme to overcome spurious oscillation and numerical instabilities.

2.4 Pantanker’s method

The SIMPLE (semi-implicit method for the pressure linked equations) algorithm of Patanker and Spalding has been widely used for incompressible flow problems[46][47]. Some modifications to the SIMPLE algorithm have been made. An early SIMPLE-like algorithm using the finite difference method was discussed by Roscoe[48]. The SIMPLER method (SIMPLE revised)[47] by Patanker and the SIMPLEC method (SIMPLE consistent) by Van Doormal and Raithbly[49] are the examples of such modifications and exhibit better behavior than the SIMPLE method. In these methods, a staggered grid technique is used to overcome the checker-board pressure difficulty. With a staggered grid, the scalar variables and pressure are stored at the grid points while the velocities are stored at the staggered locations as shown in Figure 2.1. A finite volume with the SIMPLE notations is shown in Figure 2.2.

Consider a 2D unsteady incompressible flow with constant properties, the finite difference equations are obtained by integrating the momentum equations over each staggered control volume and expressing the velocities in each coordinate direction.
Figure 2.2: A finite volume with the SIMPLE notation

by suitable profile approximations:

\[ (a_e + \frac{\rho \delta V}{\delta t}) u_e = \sum a_{nb} u_{nb} + A_e (p_P - p_E) + \frac{\rho \delta V}{\delta t} u^o_e \]  \hspace{1cm} (2.12) 

\[ (a_n + \frac{\rho \delta V}{\delta t}) v_n = \sum a_{nb} v_{nb} + A_n (p_P - p_N) + \frac{\rho \delta V}{\delta t} v^o_n \]  \hspace{1cm} (2.13) 

where the summation on the right side of the foregoing equations is the four neighbors and the superscript \( o \) denotes the previous time step values. The expressions for the coefficients \( a_e, a_n \) and \( a_{nb} \) are dependent on the choice of the profile approximations in each coordinate direction. \( A_e \) and \( A_n \) are the surface areas of the east and north face, respectively. \( \delta V \) is a volume of the control volume. \( \delta t \) is the time step.

For convenience, equations 2.12 and 2.13 are rewritten more compactly as

\[ a_{le} u_e = \sum a_{nb} u_{nb} + A_e (p_P - p_E) + b_e \]  \hspace{1cm} (2.14)
\[ a_{1n}v_n = \sum a_{nb}v_{nb} + A_n(p_P - p_N) + b_n \]  

(2.15)

where \( a_{1e} = a_e + \frac{\partial \phi}{\partial t}, \quad b_e = \frac{\partial \phi}{\partial t} u_e^0, \quad a_{1n} = a_n + \frac{\partial \phi}{\partial t}, \quad b_n = \frac{\partial \phi}{\partial t} v_n^0. \)

In the SIMPLE algorithm, the predicted velocities \( u^* \) and \( v^* \) satisfy

\[
 a_{1e}u_e^* = \sum a_{nb}u_{nb}^* + A_e(p_P^* - p_E^*) + b_e
\]  

(2.16)

\[
 a_{1n}v_n^* = \sum a_{nb}v_{nb}^* + A_n(p_P^* - p_N^*) + b_n
\]  

(2.17)

where \( p^* \) is a guessed pressure.

Subtracting equations 2.16 and 2.17 from 2.14 and 2.15, the fully implicit velocity correction equations are obtained as

\[
 a_{1e}u_e' = \sum a_{nb}u_{nb}' + A_e(p_P' - p_E')
\]  

(2.18)

\[
 a_{1n}v_n' = \sum a_{nb}v_{nb}' + A_n(p_P' - p_N')
\]  

(2.19)

where \( u' = u - u^* \), \( v' = v - v^* \), \( p' = p - p^* \)

Dropping the \( \sum a_{nb}u_{nb}' \) and \( \sum a_{nb}v_{nb}' \) terms, yields

\[
 u_e = u_e^* + d_e(p_P' - p_E')
\]  

(2.20)

\[
 v_n = v_n^* + d_n(p_P' - p_N')
\]  

(2.21)

where \( d_e = A_e/a_{1e}, \quad d_n = A_n/a_{1n} \)

To obtain the pressure correction equation, equations 2.20 and 2.21 are substituted into a discretized continuity equation:

\[
 (\rho_p - \rho_p^0) \frac{\partial u}{\partial t} + \rho_e u_e A_e - \rho_w u_w A_w + \rho_n u_n A_n - \rho_s u_s A_s = 0
\]  

(2.22)
The forgoing equation is rewritten as follows:

\[ a_p p'_p = a_E p'_E + a_W p'_w + a_N p'_N + a_S p'_S + b \]  

(2.23)

where \( a_E = \rho_e d_e A_e \), \( a_N = \rho_n d_n A_n \), and

\[ b = (\rho_p - \rho_p) \frac{\delta u}{\delta t} - \rho_e u^*_e A_e + \rho_w u^*_w A_w - \rho_n u^*_n A_n + \rho_s u^*_s A_s \]  

(2.24)

The SIMPLEC algorithm is quite similar to the SIMPLE algorithm. The primary difference of the former from the latter is that \( \sum a_{nb} u'_e \) is subtracted from both sides of equation 2.18. Therefore, for the SIMPLEC algorithm, we have

\[ (a_{1e} - \sum a_{nb}) u'_e = \sum a_{nb} (u'_{nb} - u'_e) + (p'_p - p'_E) A_e \]  

(2.25)

The term \( \sum a_{nb} (u'_{nb} - u'_e) \) on the right side of the foregoing equation is neglected. A similar operation is performed on the \( v' \) equation. The resulting velocity correction equations have the same form as equations 2.20 and 2.21 except for \( d_e \) and \( d_n \). In the SIMPLEC method, \( d_e \) and \( d_n \) are as follows:

\[ d_e = \frac{A_e}{a_{1e} - \sum a_{nb}} \]

\[ d_n = \frac{A_n}{a_{1n} - \sum a_{nb}} \]

The pressure correction equation is exactly the same as equation 2.23.

In the SIMPLER algorithm, two Poisson equations are solved: one is for the pressure, another is for the pressure correction. In the SIMPLE algorithm, the neglection of \( \sum a_{nb} u'_{nb} \) is regarded as the reason for the slower convergence of the pressure field. Thus, the equation for the pressure field is necessary.

In the SIMPLER, pseudo-velocity \( \hat{u}_e \) and \( \hat{v}_n \) are defined as

\[ \hat{u}_e = \frac{\sum a_{nb} u_{nb} + b_e}{a_{1e}}, \hat{v}_n = \frac{\sum a_{nb} u_{nb} + b_n}{a_{1n}} \]
Thus, equations 2.14 and 2.15 are rewritten as

\[ u_e = \hat{u}_e + d_e(p_P - p_E) \]  

(2.26)

\[ v_n = \hat{v}_n + d_n(p_P - p_N) \]  

(2.27)

Substituting equations 2.26 and 2.27 into the continuity equation leads to the pressure equation:

\[ a_P p_P = a_E p_E + a_W p_W + a_N p_N + a_S p_S + b \]  

(2.28)

where \( a_E, a_W, a_N, a_S, a_P \) are the same as those in the SIMPLE algorithm, but \( b \) is calculated with \( \hat{u} \) and \( \hat{v} \) rather than \( u^*, v^* \).

Actually, the most important distinction between the SIMPLE and SIMPLER method is in the fact that a better approximation is made in the derivation of the pressure equation for the SIMPLER method. If the correct velocity field is known, the pressure equation gives the correct pressure immediately. Connell and Stow\[50\] compared the convergence rate of the SIMPLE algorithm and the SIMPLER method.

The SIMPLE family (the SIMPLE, SIMPLEC, and SIMPLER algorithm) has been widely used on either a staggered grid or a non-staggered grid. The momentum-weighted interpolation (MWIN) method on a non-staggered grid by Aksoy and Chen\[51\] is very similar to another version called the pressure-weighted interpolation (PWIN) method, which is also on a non-staggered. A comparative assessment of the MWIN to the SIMPLER, the SIMPLEC, and the PWIN and a non-staggered grid version of the MAC method were made by Aksoy and Chen. J. Wu and H. J. Rath \[52\] proposed a difference scheme with a similar manner to the SIMPLE family to solve...
the incompressible Navier-Stokes equations with rotational and moving boundary in
a non-staggered grid. The Rhie-Chow interpolation is employed to avoid unrealistic
distribution of pressure. A. W. Date\[53\] developed a complete pressure correction
algorithm for incompressible flow.

2.5 The PISO algorithm

Issa and co-workers proposed the PISO (pressure-implicit with splitting of opera-
tors) algorithm\[54\]|\[55\], which is a non-iterative time-marching procedure for incom-
pressible and low-Mach number compressible flow. In the PISO method, during each
time step, there is a predictor step and one or more corrector steps. Its procedure is
as follows:

1) Predictor step for velocity—with the pressure field from previous time step

\[
a_e u^*_e = \sum a_{nb} u^*_{nb} + A_e (p^*_p - p^*_E) + b_e
\]  (2.29)

2) predictor step for pressure

\[
a_p p'_p = a_E p'_E + a_W p'_W + a_N p'_N + a_S p'_S + b
\]  (2.30)

which is the same as equation 2.23.

3) First corrector step for the velocity

\[
u^{**}_e = u^*_e + d_e [(p^*_p - p^*_p) - (p^*_E - p^*_E)]
\]  (2.31)

which is the same as the velocity correction in the SIMPLE method.

4) Corrector step for the pressure

\[
\nabla^2 p^{**} = -\nabla \cdot [\bar{u}^{**} \cdot \nabla \bar{u}^{**} - \frac{1}{Re} \nabla^2 \bar{u}^{**}]
\]  (2.32)
(5) Second corrector step for the velocity

\[
\begin{align*}
    u_e^{***} &= u_e^{**} + \left[ \sum a_{nb}(u_{nb}^{**} - u_{nb}^{*}) \right]/a_e + d_e[(p_p^{**} - p_p^*) - (p_e^{**} - p_e^*)] \\
\end{align*}
\] (2.33)

(6) Advance to the next time step

\[
\begin{align*}
    u_e^{n+1} &= u_e^{***} \\
    p^{n+1} &= p^{**} \\
\end{align*}
\] (2.34) (2.35)

In the foregoing equations, \( n \) represents the previous time step, \( * \) the present time step at predictor level, \( ** \) at first corrector level, and \( *** \) at the second corrector level.

A staggered grid is applied in the PISO method. Figure 2.1 can also be illustrated the discrete forms of the PISO method. From above, we see that the PISO method has two Poisson equations for the pressure, similar to the SIMPLER method. The second corrector velocity is taken as the next time step result, which is different from the SIMPLER method.

Jang et al. [56] made a comparative review of the performance of the PISO method, the SIMPLER method and the SIMPLEC method. They found that the PISO algorithm outperformed the SIMPLER and SIMPLEC in terms of computational effort. For strong coupling between the momentum and scalar transport problem, the PISO method did not show good convergence rate compared with the SIMPLER and the SIMPLEC method.

Braatan and Shyy [57] developed a multi step pressure-correction algorithm similar to the PISO method with extensions which included nonorthogonal curvilinear coordinates and a multi-grid solver for the Poisson equations.
2.6 The projection method (the fractional steps method)

Chorin[58] developed the fractional steps method for solving the time-dependent Navier-Stokes equations for incompressible flow. This method used the primitive variables, i.e., velocities and pressure, and is applicable to three dimensions.

Using Helmholtz decomposition theory, the momentum equations are decomposed into two parts: one is a rotational field \( F(U) \) and another is a irrotational field, \( -\frac{\mathbf{I}}{\rho} \nabla p \). \( F(U) \) is defined as follows:

\[
F(U) = -(U \cdot \nabla)U + \nu \nabla^2 U
\] (2.36)

Chorin proposed that the velocity field be advanced in two steps. In the first step, an auxiliary velocity field is obtained by

\[
\frac{\partial u_{i}^{aux}}{\partial t} = [F(u_i)]^n
\] (2.37)

In the second step, the previous velocity field is updated by

\[
\frac{\partial u_{i}^{n+1}}{\partial t} = -\frac{\partial p^{n+1}}{\partial x_i}
\] (2.38)

The Poisson pressure equation is derived with the requirement of continuity:

\[
\nabla \cdot u^{n+1} = 0
\] (2.39)

Thus the Poisson equation becomes

\[
\nabla^2 p^{n+1} = \frac{1}{\delta t} \nabla \cdot u^{aux}
\] (2.40)

After the pressure field is calculated, the final step in the projection method is to correct the auxiliary velocity by

\[
u^{n+1} = u^{aux} - \delta t \nabla p^{n+1}
\] (2.41)
Donea et al. implemented this method using the finite element approach\[59\]. Kawahara\[60\] and Rice\[61\] made further contributions.

### 2.7 The artificial compressibility method

The continuity equation of incompressible flow does not include the density derivative with respect to time. Chorin\[62\] introduced the concept of artificial compressibility and an artificial state equation:

$$\epsilon \frac{\partial \rho_a}{\partial t} + \frac{\partial u_i}{\partial x_i} = 0 \quad (2.42)$$

where $\epsilon$ is a very small value, and $\rho_a$ is the artificial compressibility.

The artificial state equation is written as

$$p = \frac{\rho_a}{\delta_a} \quad (2.43)$$

where $\delta_a$ is the coefficient of artificial compressibility.

Through such arrangement, the density is calculated from equation 2.42. After that, the pressure is obtained by equation 2.43. However, because of the very small value of $\epsilon$, the time step should be very small also. Some applications can be seen in \[63\][64][65].

25
CHAPTER 3

FULLY CONSERVATIVE FORMS IN THE NAVIER-STOKES SOLVER

The fundamentals of the numerical simulation in the present work are based on the new algorithm proposed by Nakamura [66][67]. The axisymmetric 2D model, the 2.5D model, and the 3D model are used in various investigations. The new Navier-Stokes solver is for unsteady incompressible flow written on the curvilinear coordinates and the SMAC method is adopted. A non-staggered grid is used. To prevent the spurious oscillations inherent with the non-staggered grid, the Rhie-Chow interpolation [68] is employed. The new Navier-Stokes solvers have the following properties:

(1) Accurate mass and momentum conservation;
(2) Little or no spurious spatial oscillation;
(3) A small amount of fourth-order numerical viscosity;
(4) No numerical instability;
(5) No difficulties to get the pressure equation converged;
(6) Robustness for high Reynolds number with no turbulence model.

Through various tests and applications[75][76][69], the above criteria are shown to be very well satisfied. Details of implementation are discussed in the following sections.
3.1 Governing equations

The governing equations for incompressible flow with constant properties are written as follows (non-dimensional form)[70]:

\[ \nabla \cdot \vec{v} = 0 \quad (3.1) \]

\[ \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla)\vec{v} = -\nabla p + \frac{1}{Re} \nabla^2 \vec{v} \quad (3.2) \]

where \( Re = \frac{\rho L U}{\mu} \), \( L \) is the characteristic length, \( U \) is the characteristic velocity, \( \rho \) is the density of the fluid, and \( \mu \) is the viscosity of the fluid.

A centrifugal pump is a rotational device. Most of the calculations in the present work are made on a rotating frame to make the analysis easy. Navier-Stokes equations on a rotating frame is expressed as follows[70]:

\[ \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla)\vec{v} + \Omega \times (\nabla \times \vec{R}) + 2\Omega \times \vec{v} = -\nabla p + 1/Re \nabla^2 \vec{v} \quad (3.3) \]

where \( \Omega \) is the angular velocity of the rotating frame, \( \vec{R} \) is the position in the rotating frame, and \( \vec{v} \) is the relative velocity to the rotating frame.

3.2 Discretization of the momentum equations

The implicit Euler scheme is employed to solve the momentum equations. The Euler implicit scheme may be written as

\[ \vec{v}^{n+1} = \vec{v}^n + \Delta t G'(\vec{v}^{n+1}, p^n) \quad (3.4) \]

where prime of \( G \) indicates that the velocity terms in the coefficients of the convection terms use the previous time step value, i.e.,

\[ (\vec{v}^{n+1} \cdot \nabla)\vec{v}^{n+1} = (\vec{v}^n \cdot \nabla)\vec{v}^{n+1} \quad (3.5) \]
To increase the time differencing accuracy, the modified Euler method can be used as follows:

$$\vec{V}^{n+1} - \vec{V}^n = \Theta \Delta t G'(\vec{V}^{n+1}, \vec{p}^n) + \frac{1 - \Theta}{\Theta} f_{\vec{v}}$$  \hspace{1cm} (3.6)$$

where $\Theta = 0.5$, $f_{\vec{v}}$ comprises the convective and diffusion terms calculated by previous time step. Both non-conservative form and conservative form are used in the momentum equations.

### 3.2.1 Non-conservative form

For simplicity of explanation, two-dimensional equations are used. The two-dimensional Navier-Stokes equations in the generalized curvilinear coordinate are expressed by

$$u_t + [(u_y - u_x)u_x + (v_x - u_y)v_y]/J = -(y_np_x - y_xp_y)/J + (1/Re)\nabla_{\xi\eta}^2 u$$  \hspace{1cm} (3.7)$$

$$v_t + [(u_y - u_x)v_x + (v_x - u_y)v_y]/J = -(x[np_x - x_xp_y)/J + (1/Re)\nabla_{\xi\eta}^2 v$$  \hspace{1cm} (3.8)$$

where

$$\nabla_{\xi\eta}^2 A = \alpha A_{\xi\xi} - 2\beta A_{\eta\eta} + \gamma A_{\eta\eta}$$

$$+ [(\alpha x_{\xi\xi} - 2\beta x_{\eta\eta} + \gamma x_{\eta\eta})(y_x A_{\eta} - y_{\eta} A_x)$$

$$+(\alpha y_{\xi\xi} - 2\beta y_{\eta\eta} + \gamma y_{\eta\eta})(x_x A_{\eta} - x_{\eta} A_x)]/J^3$$  \hspace{1cm} (3.9)$$

with

$$J = x_{\xi}y_{\eta} - x_{\eta}y_{\xi}$$

$$\alpha = x_{\eta}^2 + y_{\eta}^2$$

28
\[ \beta = x_\xi x_\eta + y_\xi y_\eta \]
\[ \gamma = x_\xi^2 + y_\xi^2 \]

The third-order-upwind scheme is used in the convective terms[73]:

\[ a u_\xi = a - \frac{u_{i+2} + 8(u_{i+1} - u_{i-1}) + u_{i-2}}{12} \]
\[ +|a| \frac{u_{i+2} - 4u_{i+1} + 6u_{i} - 4u_{i-1} + u_{i-2}}{12} \quad (3.10) \]

### 3.2.2 Conservative form

With the no conservative form, the balance of momentum may be lost or gained in each time step, and such an effect can accumulate as the time advances. While in the conservative form, the conservation law is enforced in the whole domain.

The conservative form of Navier-Stokes equations are as follows (in the generalized curvilinear coordinates):

\[ J u_t + (U u)_\xi + (V u)_\eta = -y_\eta p_\xi + y_\xi p_\eta + \]
\[ Re^{-1}[A(u_\xi)_\xi - (B u_\eta)_\xi - (B u_\xi)_\eta + (C u_\eta)_\eta] \quad (3.11) \]
\[ J v_t + (U v)_\xi + (V v)_\eta = -x_\eta p_\xi + x_\xi p_\eta + \]
\[ Re^{-1}[A(v_\xi)_\xi - (B v_\eta)_\xi - (B v_\xi)_\eta + (C v_\eta)_\eta] \quad (3.12) \]

where \( U, V \) are the contravariant velocities defined by \( U = y_\eta u - x_\eta v, V = -y_\xi u + x_\xi v, \)

\( J \) is the Jacobian, \( J = x_\xi y_\eta - x_\eta y_\xi, \) and \( A = \frac{x_\xi^2 + y_\eta^2}{J}, B = \frac{x_\xi x_\eta + y_\xi y_\eta}{J}, C = \frac{x_\xi^2 + y_\xi^2}{J}. \)

The convective term in the conservative form is written as

\[ (U u)_\xi = U_{i+1/2} u_{i+1/2} - U_{i-1/2} u_{i-1/2} + \alpha (g_{i+1/2} - g_{i-1/2}) \quad (3.13) \]
with
\[ u_{i+1/2} = \left( u_i + u_{i+1} \right)/2 \]
\[ U_{i+1/2} = \left( U_i + U_{i+1/2} \right)/2 \]
where \( g_{i+1/2} \) is interpreted as a numerical viscosity term and \( \alpha \) is the intensity parameter of the numerical viscosity.

For the second-order numerical viscosity, \( g \) term is set:
\[ g_{i+1/2} = U_{i+1/2}(u_i - u_{i+1})/2 \]  \hspace{1cm} (3.14)

For the fourth-order numerical viscosity, \( g \) term is set:
\[ g_{i+1/2} = U_{i+1/2}(-u_{i-1} + 3u_i - 3u_{i+1} + u_{i+2})/4 \]  \hspace{1cm} (3.15)

The advantage of using the fourth-order numerical viscosity is that the artificial viscosity effect is smaller than the second-order so that more detailed flow activity can be captured. Moreover, if the outside point, say \( u_o \) is to be extrapolated from the inside using quadratic polynomial, the numerical flux will vanish. So, the numerical viscosity does not affect the balance of momentum over the entire domain[67].

### 3.3 The Rhie-Chow interpolation

The Rhie-Chow interpolation is used for a non-staggered grid to overcome spurious pressure oscillation. In the non-staggered grid, the central difference of pressure would cause a checker-board pressure distribution because of lacking the connection between two neighboring points for the pressure.

The pressure correction equation is written as
\[ \nabla^2 \delta p = \frac{\nabla \cdot \vec{u}}{\delta t} \]  \hspace{1cm} (3.16)
or equivalently,

\[
(A(\delta p)_{\xi})_{\xi} - (B(\delta p)_{\eta})_{\eta} - (B(\delta p)_{\eta})_{\xi} + (C(\delta p)_{\eta})_{\eta} = \frac{\nabla \cdot \tilde{v}}{\delta t}
\]  

(3.17)

where

\[
A = \frac{x_n^2 + y_n^2}{J}, B = \frac{x_\xi x_n + y_\xi y_n}{J}, C = \frac{x_\xi^2 + y_\xi^2}{J}
\]

Integrating equation 3.17 over a control volume, we get

\[
A_{i+1/2,j}(\delta p_{i+1,j} - \delta p_{i,j}) - A_{i-1/2,j}(\delta p_{i,j} - \delta p_{i-1,j}) +
\]

\[
C_{i,j+1/2}(\delta p_{i,j+1} - \delta p_{i,j}) - C_{i,j-1/2}(\delta p_{i,j} - \delta p_{i,j-1})
\]

\[
= U_{i+1/2,j} - U_{i-1/2,j} + V_{i,j+1/2} - V_{i,j-1/2}
\]

where

\[
A_{i+1/2,j} = 0.5(A_{i,j} + A_{i+1,j})
\]

\[
C_{i,j+1/2} = 0.5(C_{i,j} + C_{i,j+1})
\]

The Rhie-Chow interpolation can be expressed in the following two ways:

(1) Invariant with time:

The advantage of this scheme is that the coefficients of the Rhie-Chow interpolation are calculated from the metrics. Therefore, the coefficients need to be calculated only once. Some CPU time is saved. The interpolation is expressed by

\[
U_{i+1/2,j} = 0.5(\dot{U}_{i,j} + \dot{U}_{i+1,j}) - A_{i+1/2,j}(p_{i+1,j} - p_{i,j})
\]

(3.19)

\[
V_{i,j+1/2} = 0.5(\dot{V}_{i,j} + \dot{V}_{i,j+1}) - C_{i,j+1/2}(p_{i,j+1} - p_{i,j})
\]

(3.20)

\[
\dot{U}_{i,j} = U_{i,j} + A_{i,j}\frac{p_{i+1,j} - p_{i-1,j}}{2}
\]

(3.21)
Figure 3.1: A control volume about point P

\[
\dot{V}_{i,j} = V_{i,j} + C_{ij} \frac{P_{i,j+1} - P_{i,j-1}}{2}
\]  

(3.22)

(2) Momentum-weighted interpolation:

A compact form of the u momentum equation is written as follows:

\[
ap u_p = \sum a_{nb} u_{nb} - \left(\xi_x \frac{\partial p}{\partial \xi} + \eta_y \frac{\partial p}{\partial \eta}\right) J \Delta \xi \Delta \eta
\]  

(3.23)

where \( a_p \) denotes the coefficient of the calculated point \( P \), \( a_{nb}, u_{nb} \) represent the coefficients and the velocities of the neighboring points of \( P \), respectively, and \( J \) is the Jacobian. A control volume in the discretization is shown in Figure 3.1.

If the pressure contribution in the \( \eta \) direction is ignored, the u momentum equation about point \( e \) is written as

\[
ap e u_e = \left(\sum a_{nb} u_{nb}\right)_e - \left(\xi_x \frac{\partial p}{\partial \xi}\right)_e J \Delta \xi \Delta \eta
\]  

(3.24)
Similarly, we have the \( v \) momentum equation about \( e \):

\[
a_x u_e = \left( \sum a_{nb} v_{nb} \right)_e - \left( \xi_y \frac{\partial p}{\partial \xi} \right)_e J \Delta \xi \Delta \eta \tag{3.25}
\]

i.e.

\[
u_e = \frac{\sum a_{nb} v_{nb}}{a_e} - \xi_x J \Delta \xi \Delta \eta \left( p_E - p_p \right) \tag{3.26}
\]

\[
v_e = \frac{\sum a_{nb} v_{nb}}{a_e} - \xi_y J \Delta \xi \Delta \eta \left( p_E - p_p \right) \tag{3.27}
\]

Using the definition of contravariant velocity, we get

\[
U_e = (u_e \xi_x + v_e \xi_y) J_e = \left( \sum a_{nb} v_{nb} \xi_x \right)_e + \left( \sum a_{nb} v_{nb} \xi_y \right)_e
\]

\[-\left( \frac{\xi_x^2 J \Delta \xi \Delta \eta}{a_e} \right)_e (p_E - p_p) - \left( \frac{\xi_y^2 J \Delta \xi \Delta \eta}{a_e} \right)_e (p_E - p_p) \tag{3.28}
\]

Set

\[
f_u^\xi = \frac{J \Delta \xi \Delta \eta \xi_x}{a_p}
\]

\[
f_v^\xi = \frac{J \Delta \xi \Delta \eta \xi_y}{a_p}
\]

\[
\left( \sum a_{nb} v_{nb} \xi_x J \right)_e = A_e
\]

\[
\left( \sum a_{nb} v_{nb} \xi_y J \right)_e = B_e
\]

The equation 3.28 becomes

\[
U_e = A_e + B_e - 0.5 \left( J_p \left[ (\xi_x (f_u^\xi)_p + (\xi_y (f_v^\xi)_p) \right] + J_E \left[ (\xi_x (f_u^\xi)_E + (\xi_y (f_v^\xi)_E) \right] \frac{p_E - p_p}{\Delta \xi} \right) \tag{3.29}
\]

\( A_e, B_e \) can be considered as the average of the \( A_p, B_p \) and \( A_E, B_E \). From the momentum equations, we have

\[
u_p = \frac{\sum a_{nb} v_{nb}}{a_p} - \frac{\xi_x J \Delta \xi \Delta \eta p_E - p_W}{2 \Delta \xi} \tag{3.30}
\]
\[ v_P = \frac{\sum a_n b_n u_n}{a_P} - \frac{\xi_y J \Delta \xi \Delta \eta p_E - p_W}{2\Delta \xi} \] (3.31)

i.e.

\[
\frac{\sum a_n b_n u_n}{a_P} = u_P + f_u \frac{p_E - p_W}{2\Delta \xi} \] (3.32)

\[
\frac{\sum a_n b_n v_n}{a_P} = v_P + f_v \frac{p_E - p_W}{2\Delta \xi} \] (3.33)

Set

\[ u_P^\xi = \frac{\sum a_n b_n u_n}{a_P} \] (3.34)

\[ v_P^\xi = \frac{\sum a_n b_n v_n}{a_P} \] (3.35)

Equations 3.34 and 3.35 become

\[ u_P^\xi = u_P + f_u \frac{p_E - p_W}{2\Delta \xi} \] (3.36)

\[ v_P^\xi = v_P + f_v \frac{p_E - p_W}{2\Delta \xi} \] (3.37)

Using the foregoing equations, we get

\[ A_e = \frac{1}{2}(J_P \xi_x u_P^\xi + J_E \xi_x u_E^\xi) \] (3.38)

\[ B_e = \frac{1}{2}(J_P \xi_y v_P^\xi + J_E \xi_y v_E^\xi) \] (3.39)

By rearranging the above equations, we obtain the expression for \( U_e \):

\[ U_e = U_e^\xi - m \frac{p_E - p_P}{\Delta \xi} \] (3.40)
where

\[ U_E^\xi = \frac{1}{2} \left[ J_P (\xi_x u_E^\xi + \xi_y v_E^\xi) + J_E (\xi_x u_E^\xi + \xi_y v_E^\xi) \right] \]  \hspace{1cm} (3.41)

\[ m_e = 0.5 (J_P [(\xi_x (f_E^\xi)_P + (\xi_y (f_E^\xi)_P)]
\]
\[ + J_E [(\xi_x (f_E^\xi)_E + (\xi_y (f_E^\xi)_E))] \]  \hspace{1cm} (3.42)

Similarly, we have

\[ f_u^\eta = \frac{J \Delta \xi \Delta \eta}{a_p} \eta_x \]
\[ f_v^\eta = \frac{J \Delta \xi \Delta \eta}{a_p} \eta_y \]

\[ u_p^\eta = u_P + f_u^\eta \frac{N - p_S}{2 \Delta \xi} \]  \hspace{1cm} (3.43)

\[ v_p^\eta = v_P + f_v^\eta \frac{N - p_S}{2 \Delta \xi} \]  \hspace{1cm} (3.44)

\[ V_n = V_n^\eta - m_n \frac{p_N - p_P}{\Delta \eta} \]  \hspace{1cm} (3.45)

where

\[ V_n^\eta = \frac{1}{2} \left[ J_P (\eta_x u_N^\eta + \eta_y v_N^\eta) + J_N (\eta_x u_N^\eta + \eta_y v_N^\eta) \right] \]  \hspace{1cm} (3.46)

\[ m_n = 0.5 (J_P [(\eta_x (f_N^\eta)_P + (\eta_y (f_N^\eta)_P)]
\]
\[ + J_N [(\eta_x (f_N^\eta)_N + (\eta_y (f_N^\eta)_N))] \]  \hspace{1cm} (3.47)
3.4 Boundary conditions

One advantage of the SMAC method is that the homogeneous boundary condition, \( \frac{\partial p}{\partial n} = 0 \), is applied to the whole physical domain where the pressure is unknown or else \( \delta p = 0 \) where the pressure is specified. According to the Gauss theory, the integration over the whole computational domain is converted to the integration on the surface of the domain. To make the pressure correction equation convergent, it is required that the total mass balance on the surface be satisfied. When this is satisfied, the convergence error of the pressure equation can reach a machine error level.

The implementation of \( \frac{\partial p}{\partial n} = 0 \) on the generalized curvilinear coordinates is as follows (Figure 3.2 as an illustration):
On the computational domain, the boundary condition becomes

\[ \frac{\partial p}{\partial n} = c_1 \frac{\partial p}{\partial \xi} + c_2 \frac{\partial p}{\partial \eta} \quad (3.48) \]

where

\[ c_1 = \frac{(n_x y_\eta - n_y x_\eta)}{J} \]

\[ c_2 = \frac{(-n_x y_\xi - n_y x_\xi)}{J} \]

where \( n_x, n_y \) are \( x, y \) components of unit vector normal outward at the boundary.

Consider a boundary along the \( \eta \) coordinate, assuming \((x_\eta, y_\eta)\) is the tangential to \( \xi = \text{constant} \), we get

\[ \mathbf{n} \cdot (x_\eta, y_\eta) = 0 \quad (3.49) \]

i.e.

\[ (n_x, n_y) \cdot (x_\eta, y_\eta) = n_x x_\eta + n_y y_\eta = 0 \quad (3.50) \]

or

\[ n_x = -\frac{n_y y_\eta}{x_\eta} \quad (3.51) \]

so, \( c_1, c_2 \) become

\[ c_1 = \frac{-n_y y_\eta^2 - n_y x_\eta}{x_\eta} = \frac{-n_y x_\eta y_\eta^2 + x_\eta^2}{x_\eta} \quad (3.52) \]

\[ c_2 = \frac{n_y y_\xi y_\eta - n_y x_\xi}{x_\eta} = \frac{n_y y_\xi y_\eta + x_\eta x_\xi}{x_\eta} \quad (3.53) \]

Along a \( \xi = \text{constant} \) line, the pressure boundary condition become

\[ \frac{y_\eta y_\xi + x_\eta x_\xi}{J} \frac{\partial p}{\partial \eta} - \frac{y_\eta^2 + x_\eta^2}{J} \frac{\partial p}{\partial \xi} = 0 \quad (3.54) \]

Similarly for a \( \eta = \text{constant} \) line, we have

\[ \frac{y_\eta y_\xi + x_\eta x_\xi}{J} \frac{\partial p}{\partial \xi} + \frac{y_\eta^2 + x_\eta^2}{J} \frac{\partial p}{\partial \eta} = 0 \quad (3.55) \]
3.5 Derivation of the 3D governing equations

3.5.1 The SMAC method

Three-dimensional incompressible Navier-Stokes equations in conservative forms are written as

\[ u_x + v_y + w_z = 0 \]

\[ u_t + (uu)_x + (vu)_y + (wu)_z = -p_x + Re^{-1} \nabla^2 u \]

\[ v_t + (uv)_x + (vv)_y + (wv)_z = -p_y + Re^{-1} \nabla^2 v \]

\[ w_t + (uw)_x + (vw)_y + (ww)_z = -p_z + Re^{-1} \nabla^2 w \quad (3.56) \]

The implicit difference equations are

\[ \frac{u^{k+1} - u^k}{\Delta t} + (uu^{k+1})_x + (vu^{k+1})_y + (wu^{k+1})_z = -p_x^{k+1} + Re^{-1} \nabla^2 u^{k+1} \]

\[ \frac{v^{k+1} - v^k}{\Delta t} + (uv^{k+1})_x + (vv^{k+1})_y + (wv^{k+1})_z = -p_y^{k+1} + Re^{-1} \nabla^2 v^{k+1} \]

\[ \frac{w^{k+1} - w^k}{\Delta t} + (uw^{k+1})_x + (vw^{k+1})_y + (ww^{k+1})_z = -p_z^{k+1} + Re^{-1} \nabla^2 w^{k+1} \quad (3.57) \]

The SMAC algorithm is a two-step scheme. The first step is called predictor. The equations are given by

\[ \frac{\bar{u} - u^k}{\Delta t} + (u\bar{u})_x + (v\bar{u})_y + (w\bar{u})_z = -p_x^k + Re^{-1} \nabla^2 \bar{u} \]

\[ \frac{\bar{v} - v^k}{\Delta t} + (u\bar{v})_x + (v\bar{v})_y + (w\bar{v})_z = -p_y^k + Re^{-1} \nabla^2 \bar{v} \]
\[
\frac{\dot{w} - w^k}{\delta t} + (u\dot{w}) + (v\dot{w}) + (w\dot{w}) = -p_z^k + Re^{-1} \nabla^2 \dot{w}
\]  

(3.58)

where \(\dot{u}, \dot{v}, \) and \(\dot{w}\) are the predictors for \(u^{k+1}, v^{k+1},\) and \(w^{k+1}\), respectively.

Subtracting each term of equation 3.57 from the corresponding one of the equation 3.58, and ignoring the differences between equation 3.57 and equation 3.58 in the convection terms and viscous terms, yield

\[
\frac{u^{k+1} - \dot{u}}{\delta t} = -\delta p_x
\]

(3.59)

\[
\frac{v^{k+1} - \dot{v}}{\delta t} = -\delta p_y
\]

\[
\frac{w^{k+1} - \dot{w}}{\delta t} = -\delta p_z
\]

where \(\delta p = p^{k+1} - p^k\)

Taking divergence of the foregoing equation, we obtain

\[
\frac{\text{div}(\vec{v}^{k+1}) - \text{div}(\vec{v})}{\delta t} = -\nabla^2 \delta p
\]

(3.60)

where

\[
\begin{bmatrix}
\vec{v}
\end{bmatrix} =
\begin{bmatrix}
u \\
v \\
w
\end{bmatrix}
\]

If we require \(\vec{v}^{k+1}\) to satisfy the continuity, equation 3.60 becomes

\[
\nabla^2 \delta p = \frac{\text{div}(\vec{v})}{\delta t}
\]

(3.61)

The foregoing equation is solved by an iterative scheme. Once the solution for \(\delta p\) is obtained. \(u^{k+1}, v^{k+1},\) and \(w^{k+1}\) are computed by equation 3.59.
3.5.2 Coordinate transformation in a strong conservation law form

The continuity equation in the conservative form on curvilinear coordinates is written as

$$U_{\xi} + V_\eta + W_\zeta = 0$$  \hspace{1cm} (3.62)

where $U$, $V$, and $W$ are the contravariant velocities, defined by

$$U = (\xi_x u + \xi_y v + \xi_z w)J$$

$$V = (\eta_x u + \eta_y v + \eta_z w)J$$

$$W = (\zeta_x u + \zeta_y v + \zeta_z w)J$$  \hspace{1cm} (3.63)

where $J$ is the Jacobian given by

$$J = x_\xi(y_\eta z_\zeta - y_\zeta z_\eta) - x_\eta(y_\xi z_\zeta - z_\xi y_\zeta) + x_\zeta(y_\xi y_\eta - z_\xi y_\eta)$$  \hspace{1cm} (3.64)

The momentum equations become

$$J \phi_t + (U \phi)_\xi + (V \phi)_\eta + (W \phi)_\zeta - (Re^{-1} J(q_{11} \phi_\xi + q_{12} \phi_\eta + q_{13} \phi_\zeta))_\xi$$

$$-(Re^{-1} J(q_{21} \phi_\xi + q_{22} \phi_\eta + q_{23} \phi_\zeta))_\eta$$

$$-(Re^{-1} J(q_{31} \phi_\xi + q_{32} \phi_\eta + q_{33} \phi_\zeta))_\zeta = -JS(\xi, \eta, \zeta)$$  \hspace{1cm} (3.65)

where

$$[\phi] = \begin{bmatrix} u \\ v \\ w \end{bmatrix}$$
\[
\begin{bmatrix}
\dot{S}(\xi, \eta, \zeta) = \\
\end{bmatrix}
= \begin{bmatrix}
\xi_x p_\xi + \eta_x p_\eta + \zeta_x p_\zeta \\
\xi_y p_\xi + \eta_y p_\eta + \zeta_y p_\zeta \\
\xi_z p_\xi + \eta_z p_\eta + \zeta_z p_\zeta
\end{bmatrix}
\]

\[q_{11} = \xi_x^2 + \eta_x^2 + \zeta_x^2 \quad q_{12} = \xi_x \eta_x + \xi_y \eta_y + \xi_z \eta_z \]
\[q_{13} = \xi_x \zeta_x + \xi_y \zeta_y + \xi_z \zeta_z \quad q_{22} = \eta_x^2 + \eta_y^2 + \eta_z^2 \]
\[q_{23} = \eta_x \zeta_x + \eta_y \zeta_y + \eta_z \zeta_z \quad q_{33} = \zeta_x^2 + \zeta_y^2 + \zeta_z^2 \]
\[q_{21} = q_{12} \quad q_{31} = q_{13} \quad q_{32} = q_{23} \]

The equations corresponding to equations 3.60 become

\[u^{k+1} = \bar{u} - \delta t[(\xi_x \delta p)_\xi + (\eta_x \delta p)_\eta + (\zeta_x \delta p)_\zeta] \]

\[v^{k+1} = \bar{v} - \delta t[(\xi_y \delta p)_\xi + (\eta_y \delta p)_\eta + (\zeta_y \delta p)_\zeta] \]

\[w^{k+1} = \bar{w} - \delta t[(\xi_z \delta p)_\xi + (\eta_z \delta p)_\eta + (\zeta_z \delta p)_\zeta] \quad (3.66) \]

We require

\[\text{div}(\vec{\nu}^{k+1}) = 0 \quad (3.67) \]

By introducing equation 3.66 into equation 3.67, yields

\[\begin{align*}
[q_{11}(\delta p)_\xi]_\xi + [q_{12}(\delta p)_\eta]_\xi + [q_{13}(\delta p)_\zeta]_\xi + [q_{12}(\delta p)_\xi]_\eta + [q_{22}(\delta p)_\eta]_\eta \\
+ [q_{23}(\delta p)_\zeta]_\eta + [q_{23}(\delta p)_\xi]_\zeta + [q_{33}(\delta p)_\eta]_\zeta + [q_{33}(\delta p)_\xi]_\zeta = \frac{\text{div}(\vec{\nu})}{\delta t} \quad (3.68)
\end{align*} \]

The right hand side of the foregoing equation is

\[\frac{\text{div}(\vec{\nu})}{\delta t} = \bar{U}_\xi + \bar{V}_\eta + \bar{W}_\zeta \quad (3.69) \]
The spatial derivatives on the transformed coordinates are approximated by the central difference approximations except for the convention terms of the momentum equations. The difference approximation of the convective terms is written by

\[ (Uu)_{i} = f_{i+1/2} - f_{i-1/2} \] (3.70)

where \( f \) is the numerical flux given by

\[
\begin{align*}
    f_{i+1/2} &= \frac{-(Uu)_{i-1} + 7(Uu)_{i} + 7(Uu)_{i+1} - (Uu)_{i+2}}{12} \\
    &\quad - \frac{|a_{i+1/2}| - u_{i-1} + 3u_{i} - 3u_{i+1} - u_{i+2}}{12}
\end{align*}
\] (3.71)

where

\[
a_{i+1/2} = \frac{(Uu)_{i} + (Uu)_{i+1}}{u_{i+1} - u_{i}}
\]

### 3.5.3 Differencing of the pressure correction equation

Using the Rhie-Chow method to eliminate the spatial oscillation, the difference approximation for equation 3.68 is written as

\[
\begin{align*}
    (q_{11})_{i+1/2,j,k} &= (\delta p_{i+1,j,k} - \delta p_{i,j,k}) - (q_{11})_{i-1/2,j,k} (\delta p_{i,j,k} - \delta p_{i-1,j,k}) \\
    &+ (q_{22})_{i+1/2,j,k} (\delta p_{i+1,j,k} - \delta p_{i,j,k}) - (q_{22})_{i-1/2,j,k} (\delta p_{i,j,k} - \delta p_{i-1,j,k}) \\
    &+ (q_{33})_{i,j+1/2,k} (\delta p_{i,j,k+1} - \delta p_{i,j,k}) - (q_{33})_{i,j-1/2,k} (\delta p_{i,j,k} - \delta p_{i,j,k-1}) \\
    &+ \frac{1}{4} \left[ (q_{12})_{i+1,j,k} (\delta p_{i+1,j,k+1} - \delta p_{i+1,j,k+1}) - (q_{12})_{i-1,j,k} (\delta p_{i-1,j,k+1} - \delta p_{i-1,j,k+1}) \right] \\
    &+ \frac{1}{4} \left[ (q_{13})_{i,j+1,k} (\delta p_{i+1,j,k+1} - \delta p_{i+1,j,k+1}) - (q_{13})_{i-1,j,k} (\delta p_{i-1,j,k+1} - \delta p_{i-1,j,k+1}) \right] \\
    &+ \frac{1}{4} \left[ (q_{12})_{i,j+1,k} (\delta p_{i+1,j,k+1} - \delta p_{i+1,j,k+1}) - (q_{12})_{i,j-1,k} (\delta p_{i+1,j,k+1} - \delta p_{i+1,j,k+1}) \right] \\
    &+ \frac{1}{4} \left[ (q_{23})_{i,j+1,k} (\delta p_{i+1,j,k+1} - \delta p_{i+1,j,k+1}) - (q_{23})_{i,j-1,k} (\delta p_{i+1,j,k+1} - \delta p_{i+1,j,k+1}) \right]
\end{align*}
\]

42
\[
\frac{1}{4}[(q_{13})_{i,j,k+1}(\delta p_{i+1,j,k+1} - \delta p_{i-1,j,k-1}) - (q_{13})_{i,j,k-1}(\delta p_{i+1,j,k-1} - \delta p_{i-1,j,k-1})]
\]
\[
+\frac{1}{4}[(q_{23})_{i,j,k+1}(\delta p_{i,j+1,k+1} - \delta p_{i,j-1,k+1}) - (q_{23})_{i,j,k-1}(\delta p_{i,j+1,k-1} - \delta p_{i,j-1,k-1})]
\]
\[
= \frac{\tilde{U}_{i+1/2,j,k} - \tilde{U}_{i-1/2,j,k} + \tilde{V}_{i,j+1/2,k} - \tilde{V}_{i,j-1/2,k} + \tilde{W}_{i,j,k+1/2} - \tilde{W}_{i,j,k-1/2}}{\delta t J(i,j,k)} \quad (3.72)
\]

In the foregoing equation, the following computations of the coefficients and the terms on the right side are used:

\[
(q_{11})_{i+1/2,j,k} = 0.5[(q_{11})_{i,j,k} + (q_{11})_{i+1,j,k}] \quad (3.73)
\]

\[
(q_{22})_{i,j+1/2,k} = 0.5[(q_{22})_{i,j,k} + (q_{22})_{i,j+1,k}] \quad (3.74)
\]

\[
(q_{33})_{i,j,k+1/2} = 0.5[(q_{33})_{i,j,k} + (q_{33})_{i,j,k+1}] \quad (3.75)
\]

and

\[
\tilde{U}_{i+1/2,j,k} = 0.5(U^{\xi}_{i,j,k} + U^{\xi}_{i+1,j,k}) - \delta t(q_{11})_{i+1/2,j,k}(p_{i+1,j,k} - p_{i,j,k}) \quad (3.76)
\]

\[
\tilde{V}_{i,j+1/2,k} = 0.5(V^{\eta}_{i,j,k} + V^{\eta}_{i,j+1,k}) - \delta t(q_{22})_{i,j+1/2,k}(p_{i,j+1,k} - p_{i,j,k}) \quad (3.77)
\]

\[
\tilde{W}_{i,j,k+1/2} = 0.5(W^{\zeta}_{i,j,k} + W^{\zeta}_{i,j,k+1}) - \delta t(q_{33})_{i,j,k+1/2}(p_{i,j,k+1} - p_{i,j,k}) \quad (3.78)
\]

where

\[
U^{\xi} = (\xi_x u^{\xi} + \xi_y v^{\xi} + \xi_z w^{\xi}) J
\]

\[
V^{\eta} = (\eta_x u^{\eta} + \eta_y v^{\eta} + \eta_z w^{\eta}) J
\]

\[
W^{\zeta} = (\zeta_x u^{\zeta} + \zeta_y v^{\zeta} + \zeta_z w^{\zeta}) J
\]
where

\[ u_{i,j,k}^{\xi} = u_{i,j,k} + \delta t \xi_x(i, j, k) \frac{P_{i+1,j,k} - P_{i-1,j,k}}{2} \]

\[ v_{i,j,k}^{\xi} = v_{i,j,k} + \delta t \eta_x(i, j, k) \frac{P_{i+1,j,k} - P_{i-1,j,k}}{2} \]

\[ w_{i,j,k}^{\xi} = w_{i,j,k} + \delta t \zeta_x(i, j, k) \frac{P_{i+1,j,k} - P_{i-1,j,k}}{2} \]

\[ u_{i,j,k}^{\eta} = u_{i,j,k} + \delta t \xi_y(i, j, k) \frac{P_{i,j+1,k} - P_{i,j-1,k}}{2} \]

\[ v_{i,j,k}^{\eta} = v_{i,j,k} + \delta t \eta_y(i, j, k) \frac{P_{i,j+1,k} - P_{i,j-1,k}}{2} \]

\[ w_{i,j,k}^{\eta} = w_{i,j,k} + \delta t \zeta_y(i, j, k) \frac{P_{i,j+1,k} - P_{i,j-1,k}}{2} \]

\[ u_{i,j,k}^{\zeta} = u_{i,j,k} + \delta t \xi_z(i, j, k) \frac{P_{i,j,k+1} - P_{i,j,k-1}}{2} \]

\[ v_{i,j,k}^{\zeta} = v_{i,j,k} + \delta t \eta_z(i, j, k) \frac{P_{i,j,k+1} - P_{i,j,k-1}}{2} \]

\[ w_{i,j,k}^{\zeta} = w_{i,j,k} + \delta t \zeta_z(i, j, k) \frac{P_{i,j,k+1} - P_{i,j,k-1}}{2} \]

The central differencing is used in the foregoing equation. For the pressure points along a boundary, it is replaced by a forward or backward differencing scheme.
CHAPTER 4

THE JOURNAL BEARING

The journal bearing is one of the critical parts in the IVAS pump, where the blood is used as a lubricant. The analysis of the flow patterns in the eccentric journal bearing cavity between the shaft and the rotor is very important both from the physiological and mechanical points of view. Computational fluid dynamics analysis can provide valuable information.

A sketch of the journal bearing part is shown in Figure 4.1. One half of the surface of the shaft is circular, the other half is elliptic. The outer cylinder rotates, while the shaft is stationary. The features of this part are as follows: (1) the design total flow rate through the journal bearing is 50 cc/min; (2) the design rotational speed is 3,000 RPM; (3) the minimum gap of the journal bearing is 0.0635 mm while the maximum gap is 0.127 mm. The shear stress which causes hemolysis should be under the critical value of the shear stress for hemolysis.

The objectives of the CFD in this study for the journal bearing design are as follows:

(1) Visualize the flow patterns in the bearing passage;

(2) Provide flow quantities such as pressure distributions, velocity profiles to the
designers to review their design;

(3) Investigate the shear stress.

The flow in the journal bearing was computed by the 3D Navier-Stokes solver. The simulation includes four separate studies: (1) circular shaft; (2) non-circular shaft (more realistic model); (3) parametric study; (4) hemolysis investigation.

The Reynolds number in the calculation is defined as follows:

\[ Re = \frac{L \cdot V \cdot \rho}{\mu} \]  \hspace{1cm} (4.1)

where \( L \) is the characteristic length and equal to the diameter of the outer cylinder; \( V \) is the characteristic velocity equal to the tangential velocity of the outer cylinder, \( \nu_g \); \( \rho \) is the density of the blood, 1,000 \( kg/m^3 \); \( \mu \) is the viscosity of the blood, 3.5 cp = 0.0035 \( kg/m*sec \).

4.1 Circular shaft

The goal of this section is to investigate the flow patterns in the journal bearing. The geometrical parameters are shown in table 4.1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter of the inner shaft</td>
<td>( D_i = 1.6129(cm) )</td>
</tr>
<tr>
<td>The minimum gap width</td>
<td>( C = 0.00635(cm) )</td>
</tr>
<tr>
<td>The maximum gap width</td>
<td>( C_{max} = 2C = 0.0127(cm) )</td>
</tr>
<tr>
<td>Diameter of outer cylinder</td>
<td>( D_o = 1.63195(cm) )</td>
</tr>
<tr>
<td>Length of the cylinder</td>
<td>( H = 1.8(cm) )</td>
</tr>
<tr>
<td>Flow area</td>
<td>( A = 0.048549(cm^2) )</td>
</tr>
</tbody>
</table>

Table 4.1: Parameters of the eccentric journal bearing
The Reynolds number calculated by equation 4.1 is 11,947. The pressures at the inlet and exit are both set to constants, with a difference of 0.0625 bar. The mesh size is as follows: 11 points in the radial direction, 81 points in the circumferential direction, and 11 points in the vertical direction.

Figure 4.2 shows the pressure distribution, which is at the middle level in the vertical direction and on the middle plane in the journal bearing gap, computed by the 3D solver in comparison with the pressure predicted by The Mechanical Technology Incorporated (MTI). The flow rate obtained by the 3D solver is 18.75cc/min, which agrees well with the result of MTI, 18.97cc/min. The velocity distribution along the middle level is shown in Figure 4.3. Figure 4.4 and Figure 4.5 show the velocity profiles at the minimum and the maximum gaps. The pressure contour at
the middle level in Figure 4.6 indicates that the pressure is almost invariant in the radial direction. The pressure distribution on the middle plane is plotted in Figure 4.7, where \( \theta \) is the azimuthal angle measured from a reference point\(^1\). The maximum pressure of 0.15 bar (relative to the pressure at the exit) occurs at 220 degrees of the circumferential direction and at \( z/H = 0.55 \)\(^2\) of the vertical direction. The minimum pressure of -0.86 bar occurs at 330 degrees and at \( z/H = 0.44 \). Figure 4.9 is the vertical velocity component along the middle level. It indicates that at this level the blood flows downward; at around 270 degrees, the velocity is low; and at around 90 degrees, the velocity is high. Figure 4.8 and Figure 4.10 show that a strong reversed flow occurs both at the top and at the bottom ends of the journal bearing. Figure 4.11 is a presentation of the circumferential distribution of the velocity at the inlet, the middle and the exit levels. Figure 4.12 is the 3D velocity plot of Figure 4.11. From Figure 4.11, we see that despite the pressure drop from the top to the bottom, suction and ejection both occur at the top as well as the bottom ends of the journal bearing passage. That is, the fluid is sucked into the journal bearing where the gap width increases to the tangential direction of the rotation, and conversely, the fluid is ejected where the gap width decreases in the tangential direction.

\(^1\)0 degree is assumed from the minimum gap 90 degrees counterclockwise

\(^2\)z is the vertical distance from the bottom
Figure 4.2: Pressure distribution versus theta (\(z/H=0.5, j=6\), \(H\) is the height of the cylinder, \(j=6\) means the middle plane)
Figure 4.3: Velocity distribution at the middle level (RPM=3,000, Δp=6,250 Pa, 
z/H=0.5)
Figure 4.4: Velocity profile at the maximum gap (RPM=3,000, Δp=6,250 Pa, z/H=0.5)

Figure 4.5: Velocity profile at the minimum gap (RPM=3,000, Δp = 6,250 Pa, z/H = 0.5)
Figure 4.6: Pressure contour plot at the middle level (RPM=3,000, Δp=6,250 Pa, z/H=0.5)
Figure 4.7: Pressure contour on the middle plane (RPM=3,000, Δp=6,250 Pa, j=6, unit: bar)
Figure 4.8: Vertical velocity component at the inlet (RPM=3,000, Δp=6,250 Pa, z/H=1.0, top)
Figure 4.9: Vertical velocity component at the middle level (RPM=3,000, Δp=6,250 Pa, z/H=0.5, middle)
Figure 4.10: Vertical velocity component at the exit (RPM=3,000, Δp=6,250 Pa, z/H=0.0, bottom)
Figure 4.11: Velocity distributions at the top(inlet), middle, and bottom(exit) levels of the journal bearing
Figure 4.12: Velocity distributions at the top (inlet), middle, and bottom (exit) levels of the journal bearing (3D presentation of Figure 4.11)
4.2 Non-circular shaft (more realistic model)

The shaft of the IVAS rotor assembly is a hybrid of circular and elliptic. Figure 4.13 is the film thickness versus theta provided by MTI. Figure 4.14 shows the film thickness versus theta for our simulation. They match well except at about $\theta=260$ degrees\(^3\). What should be mentioned is that in the MTI's case 0 degree is defined the same as that in our case, but the region from $-90$ degrees to 0 degree is corresponding to the region from 270 degrees to 360 degrees in our case. Some important parameters are shown in the following Table:

<table>
<thead>
<tr>
<th>Length, Diameter, Clearance (mm)</th>
<th>18</th>
<th>16.129</th>
<th>0.0855</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start, End Pressure (bar)</td>
<td>0.0625</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>Viscosity (cp)</td>
<td>3.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rotation Speed (RPM)</td>
<td>3,000</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 4.2: Some parameters of the non-circular shaft case

The pressures at the inlet and the exit are the same as those in section 4.1. A total of 11 points are used in the radial direction, 81 points in the circumferential direction, and 11 points in the vertical direction. The computation was performed on a SGI workstation. The CPU is approximately 10 minutes. Figure 4.15 shows the pressure distribution computed by the 3D solver and that by MTI. The mass flow rate calculated by our code is 48.89 cc/min, which is close to MTI's result, 50.60 cc/min. Figure 4.16 shows the pressure contour on the $j=6$ plane (the middle plane). The highest pressure of approximately 0.15 bar occurs at $\theta=210$ degrees, while the lowest $^3$0 degree is assumed from the minimum gap 90 degrees counterclockwise
pressure of -0.1 bar occurs at θ=280 degrees. Figure 4.17 shows the circumferential velocity distribution at the inlet, the middle level and the exit. A similar flow pattern as in the last section is observed.

To investigate the effect of the mesh size, the mesh size is increased to 81 × 21 × 31: 81 points in the circumferential direction, 21 points in the radial direction and 31 in the axial direction. The simulation is run on a SGI workstation. It takes approximately 90 minutes CPU time to reach the steady state. Figure 4.18 through Figure 4.19 are some results. From Figure 4.19 we notice that there is no significant difference when the mesh size is increased, that is, the original mesh size is good.
Figure 4.13: MTI’s film thickness versus theta

Figure 4.14: Film thickness versus theta in our calculation
Figure 4.15: Computed pressure distribution versus theta (z/H=0.5, j=6)
Figure 4.16: Pressure contour on the middle plane (RPM=3,000, Δp=6,250 Pa, j=6, unit: bar)
Figure 4.17: Velocity distributions at the top (inlet), middle, and bottom (exit) levels of the journal bearing
Figure 4.18: Computed pressure distribution versus theta (mesh size effect)
Figure 4.19: Pressure contour on the middle plane (RPM=3,000, Δp=6,250 pa, mesh size effect, unit: bar)
4.3 A parametric study

It is important to investigate the effects of parameters such as the rotational speed, the pressure drop, and the minimum gap on the flow in the journal bearing. The reference case is chosen as follows: the pressure drop $\Delta P_f = 6,250$ pa, $RPM_f = 3,000$, the minimum gap $C_f = 0.0635$ cm. The geometry is kept the same as the case used in section 4.1 (circular film distribution in the theta direction). The parametric study is categorized into three groups. In group I, the RPM effect will be investigated. The pressure drop effect and the minimum gap effect will be discussed in group II and III.

4.3.1 Group I: RPM effect

It is necessary to investigate the RPM effect because of the existence of off-design conditions in the IVAS pump. In this study, the pressure drop and the minimum gap are the same as the reference case. The RPM is changed from 0 to 1.75 $RPM_f$. The summary of the parameters are shown in the Table 4.3:

<table>
<thead>
<tr>
<th>Cases</th>
<th>RPM</th>
<th>Pressure Drop (pa)</th>
<th>Minimum Gap (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0RPM_f$</td>
<td>0</td>
<td>6,250</td>
<td>0.00635</td>
</tr>
<tr>
<td>$0.5RPM_f$</td>
<td>1,500</td>
<td>6,250</td>
<td>0.00635</td>
</tr>
<tr>
<td>$RPM_f$</td>
<td>3,000</td>
<td>6,250</td>
<td>0.00635</td>
</tr>
<tr>
<td>$1.25RPM_f$</td>
<td>3,750</td>
<td>6,250</td>
<td>0.00635</td>
</tr>
<tr>
<td>$1.5RPM_f$</td>
<td>4,500</td>
<td>6,250</td>
<td>0.00635</td>
</tr>
<tr>
<td>$1.75RPM_f$</td>
<td>5,250</td>
<td>6,250</td>
<td>0.00635</td>
</tr>
</tbody>
</table>

Table 4.3: Group I parameters
Figure 4.20 shows that the mass flow rate does not change much (approximately 3 percentage) when the RPM is changed from 0 to 5,250. It also indicates that the mass flow rate decreases and a step decrease occurs near the reference RPM with an increase of RPM. After that the mass flow rate increases with a further increment of RPM.

From Figure 4.21, we see the circumferential locations of the maximum pressure and the minimum pressure at the middle level are not affected by different rotational speeds. The maximum pressure occurs at 330 degrees. Furthermore, the maximum pressure becomes higher when the RPM increases, and the minimum pressure is even lower with higher RPM.

The left top plot in Figure 4.22 shows that without rotation the pressure distribution along the vertical direction is linear as theory predicts.

The right top figure in Figure 4.22 shows that the maximum pressure of the reference case occurs at about 0.55 ($z/H = 0.55$) level from the bottom, while the minimum pressure occurs at about 0.44 level from the bottom. The vertical and circumferential locations of the maximum and the minimum pressure of the middle plane do not change much with different RPMs.

$^4$LT, RT, LB, and RB in the caption of Figure 4.22 stand for the left top, the right top, the left bottom, and the right bottom plot in Figure 4.22, respectively.
Figure 4.20: Mass flow rate versus rotational speed
Figure 4.21: Pressure distributions versus theta ($z/H = 0.5, j = 3$)
Figure 4.22: Pressure contour on the middle plane \((z/H = 0.5, j = 3)\) LT: \(RPM = 0\), RT: \(RPM = 1,500\), LB: \(RPM = 3,000\), RB: \(RPM = 4,500\)
4.3.2 Group II: pressure drop effect

In this group the pressure drop effect is investigated. The RPM and the minimum gap are the same but the pressure drop is changed from $0.5\Delta P_f$ to $0.2\Delta P_f$. The summary of the parameters are shown in the Table 4.4:

<table>
<thead>
<tr>
<th>Cases</th>
<th>RPM</th>
<th>Pressure Drop (Pa)</th>
<th>Minimum Gap (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0.5\Delta P_f$</td>
<td>3,000</td>
<td>3,125</td>
<td>0.00635</td>
</tr>
<tr>
<td>$\Delta P_f$</td>
<td>3,000</td>
<td>6,250</td>
<td>0.00635</td>
</tr>
<tr>
<td>$2.0\Delta P_f$</td>
<td>3,000</td>
<td>12,500</td>
<td>0.00635</td>
</tr>
</tbody>
</table>

Table 4.4: Group II parameters

Figure 4.23 shows that the mass flow rate increases linearly with an increase of the pressure drop.

Figure 4.24 shows that the angular location of the maximum pressure and that of the minimum pressure at the middle level remain the same regardless to the changes in the pressure drop. Both the maximum pressure and the minimum pressure increase with the increase of the pressure drop.

From Figure 4.25, we conclude that the vertical location of the maximum pressure moves up as the pressure drop increases and the vertical location of the minimum pressure moves down as the pressure drop increases.

$^5$LT, RT, and LB in the caption in Figure 4.25 stand for the left top, the right top, and the left bottom plot in Figure 4.25, respectively.
Figure 4.23: Mass flow rate versus pressure drop
Figure 4.24: Pressure distributions versus theta ($z/H = 0.5, j = 3$)
Figure 4.25: Pressure distribution on the middle plane \((z/H = 0.5, j = 3)\) LT: \(\Delta p = 3,125\), RT: \(\Delta p = 6,250\), LB: \(\Delta p = 12,500\)
4.3.3 Group III: the minimum gap effect

As the shaft moves in the journal bearing cavity, the minimum gap changes. The effect of the minimum gap is the target of this section. In the calculation, the pressure drop and the RPM are the same but the minimum gap is changed from $0.8C_f$ to $1.2C_f$ (assuming the $C_{max} = 2C$, always). The summary of the parameters are shown in the Table 4.5:

<table>
<thead>
<tr>
<th>Cases</th>
<th>RPM</th>
<th>Pressure Drop (Pa)</th>
<th>Minimum Gap (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0.8C_f$</td>
<td>3,000</td>
<td>6,250</td>
<td>0.00508</td>
</tr>
<tr>
<td>$C_f$</td>
<td>3,000</td>
<td>6,250</td>
<td>0.00635</td>
</tr>
<tr>
<td>$1.2C_f$</td>
<td>3,000</td>
<td>6,250</td>
<td>0.00762</td>
</tr>
</tbody>
</table>

Table 4.5: Group III parameters

Figure 4.26 shows that the mass flow rate increases with an increase of the minimum gap in the journal bearing.

From Figure 4.27 we notice that the angular location of the maximum pressure and that of the minimum pressure at the middle level are almost the same. The maximum pressure becomes lower with an increase of the minimum gap and the minimum pressure gets higher when the minimum gap increases. We also see that when the minimum gap is $0.8C_f$ the maximum pressure does not change very much compared with the reference case. However, when the minimum gap increases to $1.2C_f$, the maximum pressure and the minimum pressure change much.
From Figure 4.28, we see that the vertical location of the maximum pressure and that of the minimum pressure on the middle plane do not change much when the minimum gap is $0.8C_f$ and $C_f$ (the left top and the right top figure in Figure 4.28). When the minimum gap increases to $1.2C_f$, however, the vertical location of the maximum pressure goes up, while that of the minimum pressure goes down.

$^6$LT, RT, and LB in the caption in Figure 4.28 stand for the left top, the right top, and the left bottom plot in Figure 4.28, respectively.
Figure 4.26: Mass flow rate versus the minimum gap
Figure 4.27: Pressure distributions versus theta ($z/H = 0.5, j = 3$)
Figure 4.28: Pressure distribution on the middle plane ($z/H = 0.5, j = 3$ for different minimum gap) LT: $C/C_f = 0.8$, RT: $C/C_f = 1.0$, LB: $C/C_f = 1.2$
4.4 Investigation of hemolysis

Hemolysis is one of the hazards in the artificial heart. How to reduce the hemolysis is a big concern for the engineers. There are some effects on the hemolysis such as shear stress, solid surface interaction, and viscous heating[77][78]. Experiment is one way to investigate, but the above aspects intervene each other. So it needs extra efforts to get rid of the other influences. Numerical calculation can possibly predict not only the local status of hemolysis, but also can easily focus on one aspect at a time. Reduction of shear stress is one way to decrease hemolysis. Some experiments show that there is a threshold shear stress of 150N/m², above which the damage of red blood cells is linearly dependent on the exposure time. Below this point, there is little damage due to shear stress[79][80].

The shear stress is defined by[82]

\[ \tau_{\text{max}} = 0.5(\sigma_{\text{max}} - \sigma_{\text{min}}) \] (4.2)

where \( \sigma_{\text{max}}, \sigma_{\text{min}} \) are the maximum and minimum values of the principal stresses, which are the roots of the following equation:

\[ \sigma_n^3 - I_1 \sigma_n^2 + I_2 \sigma_n - I_3 = 0 \] (4.3)

where

\[ I_1 = \sigma_{xx} + \sigma_{yy} + \sigma_{zz} \] (4.4)

\[ I_2 = \sigma_{xx} \sigma_{yy} + \sigma_{xx} \sigma_{zz} + \sigma_{yy} \sigma_{zz} - \tau_{xy}^2 - \tau_{xz}^2 - \tau_{yz}^2 \] (4.5)

\[ I_3 = \sigma_{xx} \sigma_{yy} \sigma_{zz} - \sigma_{xx} \tau_{yz}^2 - \sigma_{yy} \tau_{xz}^2 - \sigma_{zz} \tau_{xy}^2 + 2\tau_{xy} \tau_{yz} \tau_{xz} \] (4.6)
The shear stress contour on the solid wall is plotted in Figure 4.29 and the range of the shear stress is from 60 to $137N/m^2$. Its peak occurs at $\theta = 300$ degrees. From this calculation, we conclude that the journal bearing would not cause blood damage in the current design.

4.5 Summary

Through the computational analysis for the flows in the journal bearing of the IVAS heart pump, we conclude: (1) The flow from the journal bearing to the secondary impeller is not only non-uniform but also reversed in some portion of the passage. (2) The pressure does not change in the radial direction. (3) The maximum pressure of 0.15 bar occurs at $\theta = 210$ degrees of the circumferential direction and at $z/H = 0.558$ of the vertical direction, the minimum pressure of -0.1 bar occurs at $\theta = 280$ degrees and at $z/H = 0.44$. (4) The flow patterns do not change neither for the change in the pressure drop between the inlet and the exit nor the change in RPM. (5) The highest shear stress in the journal bearing is $137N/m^2$ that occurs at $\theta = 300$ degrees at the middle level, which is below the threshold of hemolysis.
Figure 4.29: Shear stress at the solid wall causing hemolysis in the journal bearing

Shear stress contour on the outer surface (N/m^2)

Vertical direction (cm)

Theta direction (degree)
CHAPTER 5

THE SECONDARY IMPELLER

The secondary impeller is another key part of the IVAS pump. The purpose of the secondary impeller is to pump the blood from the journal bearing back to the main flow. A careful design is important from the physiological requirement point of view. A good design should reduce hemolysis caused by rotating impeller, avoid dead water zone to prevent thrombus formation, and provide an enough pressure head to pump the blood from the journal bearing.

A torus cavity model without impeller is first used to investigate the influence of the flow from the journal bearing on the flow in the cavity. The following three sections discuss three different designs: straight blade design, wavy blade design and an improved design.

5.1 Torus cavity

The investigation of the flow through the journal bearing indicates that a non-uniform axial flow distribution exists at the exit of the journal bearing. To investigate the effects of the non-uniform inlet flow pattern on the secondary impeller, a three-dimensional torus cavity is adopted as a model geometry, as shown in Figure 5.1.
Figure 5.1: Flow domain model for the secondary impeller cavity

Figure 5.2 shows the non-uniform velocity distribution at the inlet. The inlet opening is assumed to be the bottom of the journal bearing. The top of the cavity is assumed to rotate at the rotational speed of the impeller. The 3D Navier-Stokes solver is applied to simulate the flow in the torus cavity. The mesh size is 41x81x11, namely, 41 points in the radial direction, 81 points on the circumferential direction, and 11 points in the vertical direction.

Figure 5.3 illustrates the flow patterns at three different $r-z$ view planes from the three-dimensional computation. At $\theta = 0$ degree, the flow is sucked into the journal bearing, at $\theta = 90$ degrees the flow is from the journal bearing, and at $\theta = 180$ degrees, the trend is the same but the velocity is much higher than at $\theta = 90$ degrees. In each plane, boundary layers are formed at both the top and the bottom boundaries. In the top boundary layer the fluid moves outward because of the centrifugal force,
Figure 5.2: Non-uniform velocity distribution at the inlet of the secondary impeller cavity

while in the bottom boundary layer the fluid moves inward. The fluid flow from and to the journal bearing have little influence on the bulk flow patterns in the torus cavity. An additional examination of the three-dimensional results indicates that the plane view of the flow patterns at different angular positions are essentially the same. This means, the flow is nearly axisymmetric despite the uneven flow pattern from the journal bearing gap[75].

Because of the axisymmetric property, the flow patterns are further investigated using the axisymmetric code based on the same algorithm[66][67] so that the computational time is much less than that for the 3D calculation. The governing equations of the axisymmetric geometries are written as follows[71]:

$$\frac{\partial u}{\partial r} + r \frac{\partial w}{\partial z} = 0$$  \hspace{1cm} (5.1)
The effects of the ratio of the vertical velocity to the tangential velocity at the inlet on the flow patterns are investigated next. For simplicity, the Reynolds number in the calculation is fixed to 100, based on the radius of the inner surface and the tangential velocity of the inner surface. The ratio of the tangential velocity to the vertical velocity at the inlet is set to 4:1, 1:1, 0.5:1, and 0:1, respectively.

Figure 5.4 shows the results for the ratio of 4:1. In this case, the flow moves outward at the top rotating surface. A strong vortex appears near the outer radius of the cavity and the flow pattern shows a unsteady behavior. Figure 5.5 shows a similar phenomenon with a smaller tangential velocity. The flow becomes more stable as time increases. Figure 5.6 shows the results for the ratio of 0.5:1. In this case, the flow at the inlet looks like a jet. Because of the centrifugal force, the flow leaves the exit with a very weak vortex. Figure 5.7 shows the time history of the flow pattern developed for the case of no rotation. From this figure, we see that a strong vortex moves outward and stays near the exit.
Figure 5.3: Cross sectional view of the flow patterns for the non-uniform flow from the journal bearing
Figure 5.4: Time history of the flow patterns with $v_{\theta}/v_{\text{ver}} = 4 : 1$ (Re=100, time step is 100, 200, 400, 600, from the top to bottom)
Figure 5.5: Time history of the flow patterns with $u_g/u_wer = 1:1$ (Re=100, time step is 100, 200, 400, 600, from the top to bottom)
Figure 5.6: Time history of the flow patterns with $v_{\theta}/v_{\text{ver}} = 0.5 : 1$ (Re=100, time step is 100, 400, 600, from the top to bottom)
Figure 5.7: Time history of the flow patterns with $u_0/u_{ver} = 0:1$ (Re=100, time step is 100, 200, 400, 600, from the top to bottom)
5.2 Secondary impeller cavity

There are three different designs for the IVAS secondary pump: (1) straight blade design; (2) wavy blade design; and (3) an improved design. The investigations of the flow patterns around the secondary impeller and the effects of the tip clearance and the blade angles constitute the purposes of this section.

According to the previous study, the flow rate from the journal bearing to the secondary impeller cavity is set to zero in the following investigations. The periodic pattern of the secondary impeller is taken as an advantage. That is, only one blade with its surrounding space as a unit geometry is considered. Similar approaches are often used in the flow analysis for turbomachinery. Cyclic boundary conditions are imposed on the boundary of the adjacent spaces. The coordinate system is on the rotating blade.

5.2.1 Straight blade design

For the straight blade design, the angle of the blade to the radial direction is set to 18.12 degrees which is smaller than the original IVAS design. The impeller blade is replaced by a thin plate in the computation. The design of the straight blade is shown in Figure 5.8.

In the three-dimensional simulation, two cases are considered: Case (1) tip clearance of $H = 0.2\,mm$, and Case (2) tip clearance of $H = 0.5\,mm$. The computational grid uses 41 points in the radial direction, 43 in the circumferential direction, and 31 in the vertical direction. The rotor speed is 3,000 RPM.
Figure 5.9, 5.10 and 5.11 are the velocity plots on the $xy$ and $\theta z$ planes for Case (1). The velocities are viewed from the rotating frame. The blade can be recognized in the most plots by a white area in the middle. Cyclic boundary conditions are imposed at the boundaries in the circumferential direction. In Figures 5.9 and 5.10, $k$ is the plane number of the grid counted from the bottom, while $i$ in Figure 5.11 is the grid plane counted from the inner cylinder surface. Several flow patterns can be recognized from these figures. Looking at $k=4$ and $k=6$ in Figures 5.9, it can be found that the flow near the bottom plate is inward. This may be interpreted as the Ekman layer effect. The flow near the inner and outer cylindrical surfaces are affected by the moving boundaries. At the pressure side of the blade and at levels $k=14$, and above (Figure 10), a strong trend of reversal flow is observed. On the suction side, separations occur near the leading edge as well as the trailing edge at levels $k=14$. 

Figure 5.8: Straight blade design at CCF
and above. In Figure 5.11, a strong recirculation pattern is recognized near the inner wall (i=5 and i=10). This trend is the result of the flow between the leading edge and the inner wall. Another area of reversal flow is observed near the bottom at i=5 through 25.

Pressure distributions at different levels for Case (1) are plotted in Figure 5.12. Sample streamlines in Case (1) are illustrated through Figure 5.13 to Figure 5.15. These figures show very complicated flow patterns in the secondary impeller cavity.

The tip gap for Case (2) is hypothetically increased to magnify the effect of the tip gap. Figure 5.16, 5.17 and Figure 5.18 show the results for Case (2). Essentially all the flow patterns observed in Case (1) are also found in Case (2). The effect of increased tip gap is, however, that the amount of flow through the tip gap is increased significantly. Consequently, separation, recirculation and reversed flow on the suction side in this case is much stronger than Case (1).

To investigate the effect of the tip clearance, the pressure heads for Case (1) and Case (2) are compared. Also, to investigate the effect of the blade angle, we reverse the rotational direction of Case (1) and Case (2), which will be referred to as Case (3) and Case (4). Table 5.1 indicates that the increase in the tip clearance decreases the pressure head in the secondary impeller. On the other hand, the reversed rotation, which is equivalent to flipping the blade angle to -18.12 degrees with no change of rotational direction, has almost no effects on the pressure head.
The shear stress distributions are calculated in the same way as discussed in Chapter 4. We investigate the shear stress for case (1). Figure 5.19 shows the distributions of shear stress for two selected cross sections, which indicate that the major peaks of shear stress occur near the solid surface. The highest values of the shear stress is in the range of 20 to 80 $N/m^2$ and significantly lower than the shear stress in the journal bearing. Minor peaks of shear stress occur in the flow away from the solid surfaces but they are much weaker than those near the solid surfaces.
Figure 5.9: Velocity distribution on the xy plane at different horizontal levels (Re=23,800, dt=0.01, step=9,000, H=0.2 mm)
Figure 5.10: Velocity distribution on the xy plane at different horizontal levels (Re=23,800, dt=0.01, step=9,000, H=0.2 mm)
Figure 5.11: Velocity distribution on the θz plane at different radii (Re=23,800, dt=0.01, step=9,000, H=0.2 mm)
Figure 5.12: Pressure distributions at different horizontal levels for case (1)
Figure 5.13: Streamline plot 1

Figure 5.14: Streamline plot 2

Figure 5.15: Streamline plot 3
Figure 5.16: Velocity distribution on the xy plane at different horizontal levels
(Re=23,800, dt=0.01, step=9,000, H=0.5 mm)
Figure 5.17: Velocity distribution on the xy plane at different horizontal levels (Re=23,800, dt=0.01, step=9,000, H=0.5 mm)
Figure 5.18: Velocity distribution on the $\theta z$ plane at different radii ($Re=23,800$, $dt=0.01$, step=9,000, $H=0.5$ mm)
Figure 5.19: Distribution of shear stress on selected grid planes in the secondary impeller cavity (D0 is the diameter of the inner cylinder)
5.2.2 Wavy blade design

The wavy blade design was proposed by Joseph P. Veres[83]. The idea is as follows. The possibility of flow separation and recirculation will be decreased by reducing the flow area. Besides, much smoother blades would reduce the vortex activities.

Geometrical modeling

To construct the surface of the wavy blade for the flow simulation, the following partial differential equation is solved:

\[-\nabla^2 T + aT = s\] (5.5)

where \( T \) represents the height of the blade, \( a \) is the coefficient of the convection term, and \( s \) is source term.

Figure 5.21 illustrates the result of this method. Figure 5.22 is the shape of the wavy blade (using periodic boundary condition only one part of the geometry is considered).

Grid generation

The grid generation is a prerequisite for the calculation. Here the hybrid of the elliptic and hyperbolic grid generation method[84][85] is adopted. The three-dimensional hyperbolic grid generation scheme is based on two orthogonality relations and an additional equation to control the inverse Jacobian equation to a specified quantity of \( V \) which represents the volume of the grid cell:

\[x_\xi x_\zeta + y_\xi y_\zeta + z_\xi z_\zeta = 0\] (5.6)

\[x_\eta x_\zeta + y_\eta y_\zeta + z_\eta z_\zeta = 0\] (5.7)
\[
\frac{\partial (x, y, z)}{\partial (\xi, \eta, \zeta)} = V
\]

where \(\xi, \eta, \zeta\) are the coordinates on the computational domain, for which the grid indices of \(i, j, k\), respectively, are used. It is assumed that \(\zeta\) is the direction nearly perpendicular to the surface to which the grid is desired to maintain orthogonality. The \(V\) determines the size of the mesh cell and is a predetermined function in space.

**Simulation**

Computational analysis of the wavy blade design is performed for one unit of the cavity with periodic boundary conditions. The Reynolds number based on the inner diameter of the rotor (\(D_i = 1.869\) cm) is 31,354. The cavity height is 2 mm, and the blade tip clearance is 0.2 mm. The blade angle is approximately 47 degrees, which is much larger than the straight blade design (blade angle is about 18 degrees). The grid for the calculation is 61x72x21: 61 points in the radial direction, 72 points in the circumferential direction, and 21 points in the vertical direction. The computation was carried out on a SGI workstation. Figure 5.23 shows the pressure contours at the middle grid level. The value in the figure is non-dimensional. From this figure we see that pressure increases toward the outer radius. Figure 5.24 through Figure 5.26 show the streamline plots. From Figure 5.24, we see that the sample streamlines are selected passing a vertical line at \(i = 38\) location and the streamlines in the upper layers move inward. The twisted ribbons indicate that a vortex exists especially on the suction side of the blade. Figure 5.25 shows the sample streamlines near the inner surface of the cylinder. The movements of those streamlines are fairly smooth. Figure 5.26 presents the streamlines near the bottom. From this figure, we notice that
a complicated flow pattern occurs at the suction side of the blade. At the pressure side, however, such a flow pattern could not be seen.
Figure 5.20: An example of the geometry modeling

Figure 5.21: The geometry of the wavy blade design of the secondary impeller
Figure 5.22: Pressure contour on the blade surface

Figure 5.23: Streamline plot 1

Figure 5.24: Streamline plot 2

Figure 5.25: Streamline plot 3
5.2.3 A newer design: B3548

Two changes to the secondary impeller design have been proposed. The first is to make the blade perpendicular to the rotational direction. The second is to make the impeller surface smoothly curved by rounding off sharp edges or filling in sharp concave corners. The impeller still has the trailing edge part that hangs over the impeller hub. This "hang-over" is intended to cause some turbulence in the corner of the secondary impeller cavity in order to minimize thrombosis.

The objectives of the present computational simulation are two-fold, first, to visualize the flow around the smoothly curved secondary impeller blades, and second, to analyze the effect of the hang-over on the flows near the corner of the secondary impeller cavity.

In the remainder of the flow analysis, we consider two separate geometrical models: Geometry 1 and Geometry 2 as indicated in Figure 5.27. In the first, we consider a closed torus cavity with the secondary impeller rotating, where the flows from the journal bearing to the annulus are ignored. In the second geometry, one might imagine a vertically long torus cavity in which the inner surface with bars sticking out into the cavity rotates. The inner surface is set at the inner radius of the annulus (the outer radius of the assembly), while the outer surface is set at the outer radius of the annulus.

Simulation of the pure wavy blade part: Geometry 1

The secondary impeller has nine identical blades. Assuming that the flow distributions around each blade are identical, we consider a unit domain that includes only one blade. Cyclic boundary conditions are imposed at the boundaries with adjacent
flow domains. Figure 5.29 illustrates the grid planes in the simulated flow domain and explains the grid numbering systems. The number of grid points used for this geometry is 61x72x21. The velocities are defined relative to the coordinates fixed on the rotor. On the relative coordinates, the blade is stationary, while the other three walls rotate to the direction opposite to the rotor motion. The key parameters are illustrated in Table 5.2.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total flow rate</td>
<td>0 L/min</td>
</tr>
<tr>
<td>Rotation speed</td>
<td>3,000 RPM</td>
</tr>
<tr>
<td>Number of blades</td>
<td>9</td>
</tr>
<tr>
<td>Fluid</td>
<td>blood</td>
</tr>
<tr>
<td>Inner surface radius</td>
<td>0.3835 in</td>
</tr>
<tr>
<td>Outer surface radius</td>
<td>0.625 in</td>
</tr>
<tr>
<td>Height of cavity</td>
<td>0.0700 in</td>
</tr>
<tr>
<td>Tip gap</td>
<td>0.0164 in</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>8512.6 (based on the radius of the inner surface)</td>
</tr>
</tbody>
</table>

Table 5.2: Parameters of the B3458 design

Figure 5.213 is the pressure contour (non-dimensional) on the blade surface. The pressure increases toward the outer radius direction, but it does not change much in the vertical direction. Figure 5.34 through 5.48 are the velocity vectors on the grid planes in the i direction. Figure 5.49 through 5.68 are the velocity vectors on the grid planes in the j direction. Figure 5.69 through 5.71 are the velocity vectors on the grid planes in the k direction. From these figures we see that vortices appear on the suction side of the blade. From the R-Z views, we notice a boundary layer along the top wall. The direction of the flow near the top wall (stationary) is inward,
which may be considered as a reversed Ekman layer. In the cross section near the blade, there is a stronger circulation in the inner radius region. Figure 5.31 through 5.33 shows streamlines at a higher level, middle level and lower level, respectively. In these figures, streamlines start from the right and move to the left. Twisted ribbons indicate the vortical motions. On the pressure side, reversed flow and vortices are observed.

**Tip clearance effect**

To investigate the tip clearance effect, the tip gap is hypothetically increased to 0.0251 inches. Compared with the corresponding velocity vectors plots, we find similar flow patterns. However, with an increase of the tip gap, the flow volume becomes large. Consequently, more flows pass through that gap. In the outer radius region, the size of vortex becomes larger and the intensity becomes stronger. Figure 5.75 through 5.113 are the velocity vectors on different grid planes for the tip gap increased case.

With the gap increased to 0.0251 inches, the pressure head decreases, from 0.64 (non-dimensional) of the reference case to 0.59 (non-dimensional). Figure 5.114 shows the pressure head versus tip gap.

**Shear stress estimation**

The maximum shear stresses around 200 pa occurs at the bottom corners of the blade. This is due to the high rotational speed and the narrow gap between the rotational surface and the stationary blade. Figure 5.72 through 5.74 shows the shear
stress distributions at different locations around the blade. An alarmingly high shear stress appears at the inner edge of the blade as well as around the blade.
Figure 5.26: Flow passage from journal bearing gap to the secondary impeller cavity

Figure 5.27: Concept of the wavy blade design

Figure 5.28: Grid illustration

Figure 5.29: Pressure contour on the blade surface

Figure 5.30: Streamlines at the higher lever

Figure 5.31: Streamlines at the middle lever
Figure 5.32: Streamlines at the lower lever

Figure 5.35: The reference case: \( i=14 \)

Figure 5.33: The reference case: \( i=2 \)

Figure 5.36: The reference case: \( i=18 \)

Figure 5.34: The reference case: \( i=10 \)

Figure 5.37: The reference case: \( i=22 \)
Figure 5.38: The reference case: $i=26$

Figure 5.41: The reference case: $i=36$

Figure 5.39: The reference case: $i=30$

Figure 5.42: The reference case: $i=40$

Figure 5.40: The reference case: $i=32$

Figure 5.43: The reference case: $i=44$
Figure 5.44: The reference case: $i=48$

Figure 5.45: The reference case: $i=52$

Figure 5.46: The reference case: $i=56$

Figure 5.47: The reference case: $i=58$

Figure 5.48: The reference case: $j=2$

Figure 5.49: The reference case: $j=6$
Figure 5.50: The reference case: $j=10$

Figure 5.53: The reference case: $j=22$

Figure 5.51: The reference case: $j=14$

Figure 5.54: The reference case: $j=26$

Figure 5.52: The reference case: $j=18$

Figure 5.55: The reference case: $j=30$
Figure 5.62: The reference case: \( j=48 \)

Figure 5.65: The reference case: \( j=60 \)

Figure 5.63: The reference case: \( j=52 \)

Figure 5.66: The reference case: \( j=64 \)

Figure 5.64: The reference case: \( j=56 \)

Figure 5.67: The reference case: \( j=68 \)
Figure 5.68: The reference case: $k=5$

Figure 5.69: The reference case: $k=10$

Figure 5.70: The reference case: $k=15$

Figure 5.71: Shear stress distribution on the grid plane $j=36$ (Unit: Pa)

Figure 5.72: Shear stress distribution on the grid plane $j=38$ (Unit: Pa)

Figure 5.73: Shear stress distribution on the grid plane $j=40$ (Unit: Pa)
Figure 5.74: Tip Gap increased case: i=2  
Figure 5.75: Tip Gap increased case: i=6  
Figure 5.76: Tip Gap increased case: i=10  
Figure 5.77: Tip Gap increased case: i=14  
Figure 5.78: Tip Gap increased case: i=18  
Figure 5.79: Tip Gap increased case: i=22
Figure 5.80: Tip Gap increased case: $i=26$

Figure 5.83: Tip Gap increased case: $i=36$

Figure 5.81: Tip Gap increased case: $i=30$

Figure 5.84: Tip Gap increased case: $i=40$

Figure 5.82: Tip Gap increased case: $i=32$

Figure 5.85: Tip Gap increased case: $i=44$
Figure 5.86: Tip Gap increased case: $i=48$

Figure 5.87: Tip Gap increased case: $i=52$

Figure 5.88: Tip Gap increased case: $i=56$

Figure 5.89: Tip Gap increased case: $i=58$

Figure 5.90: Tip Gap increased case: $j=2$

Figure 5.91: Tip Gap increased case: $j=6$
Figure 5.92: Tip Gap increased case: $j=10$  
Figure 5.95: Tip Gap increased case: $j=22$

Figure 5.93: Tip Gap increased case: $j=14$  
Figure 5.96: Tip Gap increased case: $j=26$

Figure 5.94: Tip Gap increased case: $j=18$  
Figure 5.97: Tip Gap increased case: $j=30$  

126
Figure 5.98: Tip Gap increased case: $j=34$
Figure 5.101: Tip Gap increased case: $j=40$

Figure 5.99: Tip Gap increased case: $j=36$
Figure 5.102: Tip Gap increased case: $j=42$

Figure 5.100: Tip Gap increased case: $j=38$
Figure 5.103: Tip Gap increased case: $j=44$
Figure 5.104: Tip Gap increased case: $j=48$  Figure 5.107: Tip Gap increased case: $j=60$

Figure 5.105: Tip Gap increased case: $j=52$  Figure 5.108: Tip Gap increased case: $j=64$

Figure 5.106: Tip Gap increased case: $j=56$  Figure 5.109: Tip Gap increased case: $j=68$
Figure 5.110: Tip Gap increased case: $k=5$

Figure 5.111: Tip Gap increased case: $k=10$

Figure 5.112: Tip Gap increased case: $k=15$

Figure 5.113: Tip clearance on the pressure head
Simulation of the hang over part: Geometry 2

The trailing edge of each secondary impeller blade, "hang-over," is designed to generate turbulent flow in the bottom corner of the cavity so that the potential of thrombosis is decreased. Figure 5.115 shows a perspective view of the surfaces of the flow space being considered. In all of the following velocity vector plots, the viewer is placed on the rotating coordinates. Two cases are considered. One is called straight case, in which the hang-overs are perpendicular to the bottom surface. In another case, the hang-overs are tilted 45 degrees with respect to the bottom surface, which is called slanted case.

First we consider the straight case. Figure 5.117 through 5.138 are the velocity vectors on the grid planes in the i direction. Figure 5.139 through 5.170 are the velocity vectors on the grid planes in the j direction. Figure 5.171 through 5.210 are the velocity vectors on the grid planes in the k direction. Figure 5.118 through 5.128 show a Karman vortex formation just behind the bar, and it tends to become weak with the increase of the radius. From the R-Z view we notice a large convective circulation. It starts at the hang-over level, moves upward along the outer wall, flows inward along the top wall, and comes down along the inner wall. There is also a circulation near the inner surface just under the bar. Multiple vortices tangle with each other in the tangential direction at the hang-over level. Look at the top view, at the suction side of the bar, some vortices interfere with each other. Such a phenomenon is much stronger at the middle levels.

Three streamline plots are given in Figure 5.211, 5.212 and 5.213. The flow starts at far end of the view space and flows toward the viewer. Figure 5.211 demonstrates
a very important flow pattern, that is, the flow that comes under the hang-over near the inner surface rotates in the vortical separation behind the hang-over. The fluid moves outward during the rotation and then leaves the hang-over area, but still keeps moving outward approaching the outer boundary. In general, the fluid in the separated area behind the hang-over gets more momentum in the tangential direction, and accordingly moves more violently in the radial outward direction.

Figures 5.214 through 5.216 show the shear stress distributions around the blade, on the bottom, and on the outer surfaces. A high shear stress always appear near the edge of the blades or solid surfaces. The lower part of the vertical solid wall has significantly higher shear stress because of the turbulence caused by the hang-over. Notice that the shear stress at the corner of the outer wall and the bottom is very low. This means there is still a danger of thrombosis at the corner.

To investigate the effect of the blade shape of the hang over part, a slanted case is considered next. Figure 5.217 through 5.240 are the velocity vectors on the grid planes in the i direction. Figure 5.241 through 5.272 are the velocity vectors on the grid planes in the j direction. Figure 5.273 through 5.312 are the velocity vectors on the grid planes in the k direction. We get the similar flow structures as in the straight hang-over case. Because of the tilted bar, the disturbance at the larger radius location seems stronger than that of the straight bar case. As far as the overall disturbance effect concerned, the straight bar case is better than the slanted case.

5.3 Conclusions

Through the computational analyses for the flows in the secondary impeller cavity, we conclude:
(1) For the main part of the secondary impeller there is a major recirculation, i.e., the direction of the flow near the top wall is inward, which may be considered as a reversed Ekman layer, while the flow near the bottom surface moves outward. Many small vortices appear in the cavity. The flows near the leading edge of the blade are noticed to be vortical and turbulent.

(2) For the hang-over part, two flow trends exist: (i) Karman vortex just behind the bars, and (ii) multiple vortex roll-ups in the tangential direction at the hang-over level.

(3) The flow patterns with the smoothed secondary impeller blades produces weaker vortex motions behind the blade. However, the flow passing by the edges of the blade is exposed to a high shear stress.

(4) The pressure head is affected by the tip gap width. As the tip gap increases the pressure head decreases.

(5) The hang-overs increase shear stress on the side and bottom surfaces of the secondary impeller cavity, which is desirable to decrease the potential of thrombosis. However, the shear stress at the corner is still very low.
Figure 5.114: Geometry of hang over part

Figure 5.115: Straight case: i=3

Figure 5.116: Straight case: i=4

Figure 5.117: Straight case: i=5

Figure 5.118: Straight case: i=6

Figure 5.119: Straight case: i=7
Figure 5.126: Straight case: $i=18$

Figure 5.129: Straight case: $i=24$

Figure 5.127: Straight case: $i=20$

Figure 5.130: Straight case: $i=26$

Figure 5.128: Straight case: $i=22$

Figure 5.131: Straight case: $i=28$
Figure 5.132: Straight case: i=30

Figure 5.133: Straight case: i=32

Figure 5.134: Straight case: i=34

Figure 5.135: Straight case: i=36

Figure 5.136: Straight case: i=38

Figure 5.137: Straight case: i=40
Figure 5.138: Straight case: \( j=5 \)

Figure 5.139: Straight case: \( j=7 \)

Figure 5.140: Straight case: \( j=9 \)

Figure 5.141: Straight case: \( j=11 \)

Figure 5.142: Straight case: \( j=13 \)

Figure 5.143: Straight case: \( j=15 \)
Figure 5.144: Straight case: $j=17$   
Figure 5.147: Straight case: $j=23$

Figure 5.145: Straight case: $j=19$   
Figure 5.148: Straight case: $j=25$

Figure 5.146: Straight case: $j=21$   
Figure 5.149: Straight case: $j=27$
Figure 5.150: Straight case: $j=28$

Figure 5.151: Straight case: $j=29$

Figure 5.152: Straight case: $j=30$

Figure 5.153: Straight case: $j=31$

Figure 5.154: Straight case: $j=32$

Figure 5.155: Straight case: $j=33$
Figure 5.156: Straight case: $j=34$

Figure 5.157: Straight case: $j=35$

Figure 5.158: Straight case: $j=37$

Figure 5.159: Straight case: $j=39$

Figure 5.160: Straight case: $j=41$

Figure 5.161: Straight case: $j=43$
Figure 5.162: Straight case: $j=45$

Figure 5.163: Straight case: $j=47$

Figure 5.164: Straight case: $j=49$

Figure 5.165: Straight case: $j=51$

Figure 5.166: Straight case: $j=53$

Figure 5.167: Straight case: $j=55$
Figure 5.168: Straight case: \( j = 57 \)

Figure 5.169: Straight case: \( j = 59 \)

Figure 5.170: Straight case: \( k = 2 \)

Figure 5.171: Straight case: \( k = 4 \)

Figure 5.172: Straight case: \( k = 5 \)

Figure 5.173: Straight case: \( k = 6 \)
Figure 5.174: Straight case: $k=8$

Figure 5.175: Straight case: $k=10$

Figure 5.176: Straight case: $k=12$

Figure 5.177: Straight case: $k=13$

Figure 5.178: Straight case: $k=14$

Figure 5.179: Straight case: $k=15$
Figure 5.180: Straight case: $k=16$

Figure 5.181: Straight case: $k=17$

Figure 5.182: Straight case: $k=18$

Figure 5.183: Straight case: $k=19$

Figure 5.184: Straight case: $k=20$

Figure 5.185: Straight case: $k=21$
Figure 5.186: Straight case: $k=22$

Figure 5.187: Straight case: $k=23$

Figure 5.188: Straight case: $k=24$

Figure 5.189: Straight case: $k=25$

Figure 5.190: Straight case: $k=26$

Figure 5.191: Straight case: $k=27$
Figure 5.192: Straight case: $k=28$

Figure 5.193: Straight case: $k=29$

Figure 5.194: Straight case: $k=30$

Figure 5.195: Straight case: $k=31$

Figure 5.196: Straight case: $k=32$

Figure 5.197: Straight case: $k=33$
Figure 5.210: Streamline plot 1

Figure 5.211: Streamline plot 2

Figure 5.212: Streamline plot 3

Figure 5.213: Shear stress distribution on the grid planes $j=29$, $j=33$, and $j=35$ (unit: Pascal)

Figure 5.214: Shear stress distribution on the outer radius grid plane (unit: Pascal)

Figure 5.215: Shear stress distribution on the bottom grid plane (unit: Pascal)
Figure 5.222: Slanted case: $i=8$

Figure 5.225: Slanted case: $i=12$

Figure 5.223: Slanted case: $i=9$

Figure 5.226: Slanted case: $i=14$

Figure 5.224: Slanted case: $i=10$

Figure 5.227: Slanted case: $i=16$
Figure 5.234: Slanted case: $i=30$

Figure 5.237: Slanted case: $i=36$

Figure 5.235: Slanted case: $i=32$

Figure 5.238: Slanted case: $i=38$

Figure 5.236: Slanted case: $i=34$

Figure 5.239: Slanted case: $i=40$
Figure 5.246: Slanted case: \( j=17 \)

Figure 5.249: Slanted case: \( j=23 \)

Figure 5.247: Slanted case: \( j=19 \)

Figure 5.250: Slanted case: \( j=25 \)

Figure 5.248: Slanted case: \( j=21 \)

Figure 5.251: Slanted case: \( j=27 \)
Figure 5.252: Slanted case: $j=28$  
Figure 5.253: Slanted case: $j=29$  
Figure 5.254: Slanted case: $j=30$  
Figure 5.255: Slanted case: $j=31$  
Figure 5.256: Slanted case: $j=32$  
Figure 5.257: Slanted case: $j=33$
Figure 5.270: Slanted case: $j=57$

Figure 5.271: Slanted case: $j=59$

Figure 5.272: Slanted case: $k=2$

Figure 5.273: Slanted case: $k=4$

Figure 5.274: Slanted case: $k=5$

Figure 5.275: Slanted case: $k=6$
Figure 5.276: Slanted case: $k=8$

Figure 5.279: Slanted case: $k=13$

Figure 5.277: Slanted case: $k=10$

Figure 5.280: Slanted case: $k=14$

Figure 5.278: Slanted case: $k=12$

Figure 5.281: Slanted case: $k=15$
Figure 5.282: Slanted case: $k=16$

Figure 5.283: Slanted case: $k=17$

Figure 5.284: Slanted case: $k=18$

Figure 5.285: Slanted case: $k=19$

Figure 5.286: Slanted case: $k=20$

Figure 5.287: Slanted case: $k=21$
Figure 5.288: Slanted case: $k=22$

Figure 5.290: Slanted case: $k=24$

Figure 5.291: Slanted case: $k=25$

Figure 5.292: Slanted case: $k=26$

Figure 5.293: Slanted case: $k=27$
Figure 5.300: Slanted case: $k=34$

Figure 5.301: Slanted case: $k=35$

Figure 5.302: Slanted case: $k=37$

Figure 5.303: Slanted case: $k=39$

Figure 5.304: Slanted case: $k=41$

Figure 5.305: Slanted case: $k=43$
Figure 5.306: Slanted case: $k=45$

Figure 5.309: Slanted case: $k=51$

Figure 5.307: Slanted case: $k=47$

Figure 5.310: Slanted case: $k=53$

Figure 5.308: Slanted case: $k=49$

Figure 5.311: Slanted case: $k=55$
CHAPTER 6

THE 2 – 1/2-DIMENSIONAL SINGLE PASSAGE MODEL FOR THE PRIMARY PUMP

The 2 – 1/2-dimensional single passage model was first used by Yano[86] and Velasquez[87] to analyze the flows between two adjacent primary blades of the IAVS pump. The advantage of this model is that the height of the pump is included in the governing equations, i.e., the third dimensional effect is considered. The computation time is much shorter than the 3D model. It can be used to investigate the validity of the single passage model and to perform a parametric study to provide an optimal design.

The single passage model includes an inlet, the passage between two blades, and a volute area. Figure 6.1 shows the geometry of the single passage model. The flow is assumed to be cyclical on the two blades and the blade is considered infinitely thin. The coordinates are fixed on the impeller. If the impeller is rotated clockwise, the volute outer boundary moves counterclockwise relatively to the impeller with the same rotational speed as that of the impeller. The flow enters through the boundary B1, and leaves through B5. B3 is the edge of the rotor.

The most uncertain part of the present modeling is how to treat the flow exit condition because no spiral volute can be considered in the model. Here, the volute
area is replaced by a concentric circular domain. The flow is assumed to exit through the circular outer boundary of the flow domain.

The inlet flow condition is assumed to be uniform in the radial direction on the stationary coordinates with no pre-swirl.

The velocity boundary conditions along the flow exit boundary are as follows: The tangential velocity is zero, and the radial velocity is constant when viewed from the stationary frame. On the relative coordinates, the tangential velocity at the exit is in the counter-clock direction at the same angular velocity of the rotor.

Whether the model is sensitive to the volute width and to the volute height are two critical questions to be answered first. If these factors do not influence much on the flow up to the trailing edge of the impeller, we can successfully predict the rotor performance. For this reason, examining the validity of this volute modeling is an important subject in this chapter.

In the following calculation, a reference case is considered. The parameters of the reference case are shown in table 6.1. Figure 6.2 shows the grid for calculation. The height distribution for the reference case is given in Figure 6.3. The other parameters

<table>
<thead>
<tr>
<th>Reference case</th>
<th>Inch</th>
<th>Non-dimensional (ND)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet boundary</td>
<td>0.1328</td>
<td>0.24</td>
</tr>
<tr>
<td>Leading edge</td>
<td>0.2089</td>
<td>0.37</td>
</tr>
<tr>
<td>Trailing edge</td>
<td>0.5675</td>
<td>1.0</td>
</tr>
<tr>
<td>Outer boundary</td>
<td>0.8920</td>
<td>1.57</td>
</tr>
</tbody>
</table>

Table 6.1: Parameters of the single passage model for the reference case
of the reference case are illustrated in table 6.2.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total flow rate</td>
<td>5 L/min</td>
</tr>
<tr>
<td>Rotation speed</td>
<td>3,000 RPM</td>
</tr>
<tr>
<td>Number of blades</td>
<td>6</td>
</tr>
<tr>
<td>Fluid</td>
<td>blood</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>37281.2 (based on the diameter of trailing edge)</td>
</tr>
</tbody>
</table>

Table 6.2: Other parameters of the reference case

6.1 The reference case

The reference case is first examined. Figure 6.4 and Figure 6.5 depict the pressure and the streamline plots, respectively. The number of time steps of the computation for the present results is $4 \times 10^4$. To confirm that the results are not affected by the computational initial conditions, the computation was further continued until $10^5$ time steps are reached. The result at this time step agrees well with that at the $10^4$th step. Figure 6.6 plots the tangentially averaged static pressure versus non-dimensional radius. We observe that both the static and the dynamic pressure tend to decrease toward the leading edge. This decrease is due to a small flow separation at the leading edge that can be found in Figure 6.4 (see the low pressure at the suction side of the leading edge). The static pressure increases to the exit boundary at $r = 1.57(ND)$, while the total pressure keeps rising up to the trailing edge and then decreases toward the flow exit boundary. The decrease of the total pressure between the blade trailing edge and the exit boundary indicates a loss of head due to the friction in the volute.
The increase of the static pressure in this interval is due to the conversion of a part of the rotational kinetic energy to the static pressure energy.

6.2 Effect of the volute width

Since the volute width and thickness are arbitrary parameters, an immediate question is how these parameters would affect the flow inside the rotor as well as the computed pressure distributions. To answer the questions, we change the volute width to 1/2 and 1/4 of the reference width. The tangentially averaged static and total pressure versus radius are compared also in Figure 6.6. It is found that the total pressure at \( r = 1 \) is not affected by the volute width.

6.3 Effect of the volute height distribution

The next topic to investigate is the height effect. We set the width to the reference value but the height distribution of the volute is changed to two different distributions as illustrated in Figure 6.7. The tangentially averaged static and total pressure distributions are compared among the three cases in Figure 6.8. This figure indicates that, first, the static pressure distribution is hardly affected by the height profiles, and second, the total pressure is not affected except in the vicinity of the exit boundary.

Therefore, we conclude that the flow distribution and the integrated quantities such as the tangentially averaged static pressure and total pressure for the rotor are hardly affected by the parameters in the volute model.
6.4 Pump flow rate

Pump flow rate is changed between 3 L/min and 9 L/min to investigate the performance of the pump (rotor head and power). The instantaneous power of the rotor is computed by

\[ P = N \int_{Blade Surface} \left[ p \bar{n} + \tau \right] \cdot \hat{V} \, ds \]

where \( N \) is the number of blades on the rotor, \( \bar{n} \) is a normal vector pointing outward from the blade surface, \( \hat{V}(r) \) is the velocity of the rotating blade surface at radius \( r \), and \( \tau \) is the shear stress on the blade surface, integration is extended over both surfaces of the blade. The value of \( P \) is averaged over approximately 30,000 time steps because the power fluctuates in time.

To compute the rotor head, we first compute the increase in the total pressure through the rotor, which is given by

\[ H_{rotor} = \frac{\int_{B1+B2} \left[ p + (u^2 + v^2)/2 \right] \hat{V} \, d\hat{S}}{\int_{B2} \hat{V} \, d\hat{S}} \]

where the denominator is the flow rate through one passage of the rotor. The foregoing \( H_{rotor} \) is then converted to head in mm Hg. An average of the rotor head over the same number of time steps as for the power is taken.

Figure 6.10 shows the rotor head calculated by the 2.5D model and by the Pumpa code, which is a 1D model from NASA Lewis Research Center[88]. Figure 6.9 is the comparison of the power by the CFD code and by the Pumpa code. The rotor head is over-predicted by the CFD code, because the CFD model does not consider the
leakage loss and turbulence loss, while in the empirical correlation such effects are taken into consideration.

6.5 RPM effect

Different RPM may change the flow patterns. For the present investigation, two cases of computation are considered: (1) \(RPM=1,000\); (2) \(RPM=2,000\). The flow rate is fixed to the reference value of 5 L/min. From Figure 6.4, 6.11 and 6.13, we can see that separation formed near the leading edge at 3,000 RPM becomes significantly smaller for 2,000 RPM and 1,000 RPM. Figures 6.5, 6.12 and 6.14 plot the streamline in the decreasing order of RPM. These figures indicate that the recirculation zone on the suction side disappears for RPM=2000 and 1000. The recirculation zone on the pressure side is observed in each plot, but the size diminishes as RPM decreases. Figure 6.15 shows the comparison of the tangentially averaged static and total pressure among three RPMs.

6.6 Leading and trailing edge angles effects

Leading and trailing edge angles are two important parameters to affect the performance of the pump. Two series of computations are performed to investigate the effects of the blade angles. In the first series, the leading edge angle is changed from 5 degrees to 20 degrees while the trailing edge angle is set to 25 degrees. In the second series, the trailing edge is changed from 25 degrees to 85 degrees while the leading edge angle is set to 15 degrees.

The power versus the leading edge angle, and the rotor head versus the same, are shown in Figure 6.16. The power and the rotor head versus the trailing edge
angle are shown in Figure 6.17. Examination of the streamline plot indicates that
the separation near the leading edge becomes smallest with the leading edge angle
of 5 degrees. In Figure 6.17, the power as well as the rotor head becomes smallest
at the trailing edge angle of 45 degrees. The streamline and pressure plots indicate
that among the four cases of trailing edge angles, 25, 45, 65, and 85 degrees, the flow
patterns for the 45 degrees become most unsteady, which will be illustrated in more
details in section 6.7.

The reason why the case of blade angles of leading edge angle of 15 degrees and
the trailing edge angle of 45 degrees becomes most unsteady is not clear at this time.
But it may be explained as follows. In general as the trailing edge angle increases, the
vortex shedding from the trailing edge is imminent. Once a vortex shedding starts,
the flow patterns inside and outside are significantly affected and strong unsteadiness
is likely to continue. In the present simulations, no such trend of vortex shedding
is observed for the trailing edge angles of 65 and 85 degrees, but probably some
disturbance of the flow due to an external cause, such as sudden fluctuation of inlet
flow, would onset vortex shedding. Another possible reason includes the effect of
grid for the trailing edge angle of 65 and 85 degrees that might have stabilized the
computed flow.

6.7 Unsteadiness of flows

The unsteadiness of the flows is affected by both leading edge and trailing edge
angles. Although the separations at both leading edge and trailing edges increases
unsteadiness of the flows, their roles are somewhat different. The separated flow
(recirculation zone) near the leading edge is more stable than the separated domain
near the trailing edge, because the former tends to stay without shedding vortices, while the latter tends to shed vortices that cause unsteadiness in the flow through the blade passage. However, when a leading edge separation exists, complexity of the streamline patterns in the blade passage increases and the flow downstream becomes more unstable.

In general, intensity of separation increases as the trailing edge angle becomes larger. Figure 6.18 shows a sequence of the pressure contour plots for the leading edge angle of 15 degrees and the trailing edge angle of 45 degrees. Figure 6.19 illustrates the streamline plots corresponding to Figure 6.18. In the top left figure in Figure 6.18, a pressure depression is observed above the trailing edge of the blade. This is a vortex originally shed from the trailing edge of the blade. This vortex disappears from the left flow boundary but reappears, as seen in the top right plot in Figure 6.18, from the right flow boundary because the left and right boundaries are cyclically connected. The vortex moves as time elapses to the relative flow direction in the volute and further moves to the left as seen in the lower right plot in Figure 6.18. The vortex is originally shed from the trailing edge and appears a few time in the same computational domain until it is washed away to the flow exit. In a real volute, a vortex shed from the trailing edge would flow in a real volute passing by trailing edges of several other blades.

The vortex moving in the volute space has significant effects on the flow patterns inside the rotor as demonstrated by streamline plots in Figure 6.19. The plots in Figure 6.19 correspond exactly to those in Figure 6.18. Note that the vortex is not
clearly seen in the streamline plots because there is a strong bulk flow in the volute to the counterclockwise direction.

6.8 Simulation of the B3452 blade design with the 2.5D model

The current design of the primary impeller at The Cleveland Clinic Foundation is shown in Figure 6.20. The parameters of the rotor are shown in table 6.3. The height distribution is the same as the reference case, shown in Figure 6.3.

<table>
<thead>
<tr>
<th>B3452</th>
<th>Non-dimensional (ND)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet boundary</td>
<td>0.1328 In. 0.21</td>
</tr>
<tr>
<td>Leading edge</td>
<td>0.240 In. 0.38</td>
</tr>
<tr>
<td>Trailing edge</td>
<td>0.624 In. 1.0</td>
</tr>
<tr>
<td>Outer boundary</td>
<td>0.8920 In. 1.43</td>
</tr>
<tr>
<td>Total flow rate</td>
<td>5 L/min</td>
</tr>
<tr>
<td>Rotation speed</td>
<td>3,000 RPM</td>
</tr>
<tr>
<td>Number of blades</td>
<td>7</td>
</tr>
<tr>
<td>Fluid</td>
<td>blood</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>156,566</td>
</tr>
<tr>
<td>Leading edge angle</td>
<td>15 degrees</td>
</tr>
<tr>
<td>Trailing edge angle</td>
<td>85 degrees</td>
</tr>
</tbody>
</table>

Table 6.3: Parameters of the B3452 design

The tangentially averaged static and total pressure versus radius is plotted in Figure 6.21. Figure 6.22 through 6.24 show the streamline, velocity vector and pressure contour, respectively, at time step 60,000. From these figures, we find that a separation occurs at the leading edge and a circulation exists in the middle region of two adjacent blades. Vortices are also shed from the trailing edge. The flow patterns are
more active in the volute area than that in the reference case due to the enlarged flow region. The flow patterns at different time steps are shown in Figures 6.25 and 6.26, corresponding to velocity vector and pressure contour plots.

6.9 Conclusions

The single passage model uses only a crude approximation for the volute. An early part of this chapter found, however, that arbitrary parameters for the volute hardly affect the flow and rotor performance results.

The comparison of the present computational model to the Pumpa code indicates that the pump power computed by the CFD model is possibly underestimated, while the rotor head computed is significantly overestimated. The errors in the CFD results for the rotor head are due to the limitations in the two-dimensional flow simulation, and thus are thought to be inevitable. The cause of possible errors in the power need to be investigated because they may be related to numerical errors, including the effects of pressure boundary condition, and resolution of the boundary layers near the blade surfaces.

Through the analysis of the effects of rotational speed, blade angles and flow rate, we demonstrated how the pump parameters affect the flow and pressure distributions.

Finally we note that the Pumpa code and the present CFD model augment each other very well. The former gives reliable prediction for the pump performance based on empirical data but gives no fluid dynamical explanation. The computational model, on the other hand, simulates the flows and visualizes the flow quantities that answer many questions.
Figure 6.1: Geometry of the single passage model

Figure 6.2: Grid for the calculation
Figure 6.3: Height distribution of the reference case

Figure 6.4: Pressure contour for the reference case
Flow rate: 5 L/min.
RPM: 3,000
Volute width: 0.8920 in. (1.57 ND)
\[ \Delta t = 0.001 \]
Time step: 42,000

Figure 6.5: Streamline plot for the reference case

Figure 6.6: Tangentially averaged static and total pressure distribution
Figure 6.7: Three models of the volute height profile (plot of three height profiles including the reference case)

Figure 6.8: Effect of volute height (plot of three cases of the tangentially averaged static and total pressure)
Figure 6.9: Pump power vs. flow rate

Figure 6.10: Pump rotor head vs flow rate
Figure 6.11: Pressure contour plot for RPM=2000

Figure 6.12: Streamline plot for RPM=2000
Flow rate: 5 L/min.
RPM: 1,000
Volute width: 0.8920 in. (1.57 ND)
\[ \delta t = 0.001 \]
Time step: 42,000

Figure 6.13: Pressure contour plot for RPM=1000

Flow rate: 5 L/min.
RPM: 1,000
Volute width: 0.8920 in. (1.57 ND)
\[ \delta t = 0.001 \]
Time step: 42,000

Figure 6.14: Streamline plot for RPM=1000
Figure 6.15: Comparison of the tangentially averaged static and total pressure among three RPMs

Figure 6.16: Power and rotor head vs. leading edge angle
Figure 6.17: Power and rotor head vs. trailing edge angle

Figure 6.18: A time sequence of pressure contour plots
Figure 6.19: A time sequence of streamline plots

Figure 6.20: B3452 design at CCF
Figure 6.21: The tangentially averaged static and total pressure versus radius

Figure 6.22: Streamline plot of the B3452 design at 60,000 time step

186
Figure 6.23: Velocity vector of the B3452 design at 60,000 time step

Figure 6.24: Pressure contour of the B3452 design at 60,000 time step
Figure 6.25: A time sequence of pressure contour plots of the B3452 design

Figure 6.26: A time sequence of velocity vector plots of the B3452 design
CHAPTER 7

THE THREE-DIMENSIONAL SINGLE PASSAGE MODEL FOR THE PRIMARY PUMP

The function of the primary impeller of the IVAS pump is to pump enough blood (5 L/min) for human body. A good design consists of low shear stress due to hemolysis restriction, no dead water zone, and high hydraulic efficiency. The objectives of this chapter are (1) to visualize 3D flow distributions in the blade passage, (2) to compare with the 2.5D as well as with the results of the Pumpa code, (3) to investigate the tip clearance effect and the pre-swirl effect, and (4) to estimate the shear stress distribution as a measure of the biological effects of the flow regimes to the blood.

The 3D single passage model is an extension of the 2.5D single passage model which has been successfully used to predict the fluid flows in the IVAS heart pump. However, the need of 3D simulation has become imminent to analyze such effects as the tip clearance, leakage and pre-swirl effects that cannot be investigated by the 2.5D model.

Experiment is one method to investigate three-dimensional viscous flow structures in turbomachines. However, the detailed analysis of flow structures in the flow passage and tip gap can also be studied by CFD. Hah[89] predicted the overall effects of tip clearance in axial compressor rotors by solving the Reynolds-averaged Navier-Stokes

7.1 Preliminary simulation

The single passage model assumes that flow distributions in all blade passages are identical and therefore only one passage is considered for computation. The present single passage model includes (1) an inlet duct, (2) a flow domain between two adjacent blades, and (3) a simplified volute space. In this model, the real spiral volute is replaced by a concentric circular domain. Figure 7.1 illustrates a geometry that includes seven single passages. The Reynolds number of the flow simulation is approximately 20,000 based on the rotor diameter and the tangential speed of the flow at the rotor edge. The grid for computation is shown in Figure 7.2 as the first effort on the 3D analysis.

7.1.1 Boundary conditions

Cyclic boundary conditions are imposed along the boundaries between two adjacent blades as illustrated in Figure 7.3. The flow enters the inlet opening and leaves
Figure 7.1: Concept of the single passage model (seven passages are included in the view)

Figure 7.2: A view of the grid for simulation
through the volute outer boundary. Because the coordinates are fixed on the rotating blade, the tangential velocity at the outer boundary of the volute is set in the opposite direction relative to the rotor.

7.1.2 Flow analysis

The computed flow field in the single passage is visualized here by (1) streamlines, (2) velocity vector plots, and (3) pressure contour over different grid planes. Figure 7.4 is a streamline plot. It can be observed that the fluid entering the blade passage at a lower level near the rotor hub flows more smoothly than the fluid that enters at a higher level near the top surface. The flow exiting the rotor almost suddenly alter the flow direction to the tangential direction.
Figures 7.5 through 7.8 show the top view of the flow velocity vectors at different grid levels. The complicated flow patterns seen in these figures first indicate that the flows in the blade passages are significantly more agitated than what the 2.5D computations predict. Complexity of the flow patterns increases toward the top surface of the flow passage. This may be attributed to the effects of the tip gap and stationary top surface plus the diminishing centrifugal force. The flow near the top boundary is reversed. The flow patterns at the outer edge of the rotor, where the flow passage narrows near the volute, appear to be nearly turbulent. Pressure contours corresponding to Figures 7.5 through 7.8 are plotted in Figure 7.9 through Figure 7.12. Figures 7.9 and 7.12 show that a separation occurs near the leading edge. The numerous small peaks and bottoms of the pressure along the outer edge of the rotor match the observation from the velocity vectors in the area.

Figures 7.13 through 7.16 are the side views of the velocity vectors. A strong separation is observed in the inlet channel near the top of the leading edge. Recirculation patterns are found in the entrance to the volute as well as within the volute. Figures 7.17 through 7.20 are the pressure plots corresponding to Figures 7.13 through 7.16.

7.1.3 Comparison to the 2.5D and the 1D models

The differences between the 3D and 2.5D results are attributed to two reasons: the first is the geometrical simplification in the 2.5D particularly in the inlet geometry as well as the volute domain, and second is the three-dimensional flow effect. A comparison between the 3D and the 2.5D results indicates that the effect of the second cause is overwhelmingly important.
The rotor heads computed by the 1D (Pumpa code), the 2.5D and the 3D model, respectively, are shown in the upper part of Figure 7.21. The 2.5D model over-predicts the rotor head, while the 3D is closer to the result of the 1D model. The middle part of Figure 7.21 shows the rotor head versus flow rate for three different models. The 2.5D and the 3D have the similar tendency, but there is a difference at a low flow rate range compared with the 1D model. The 2.5D model is approximately 25 percent higher than the 1D model. For the 3D there is a 10 percent discrepancy from the 1D model.

7.1.4 Tip clearance effect

To investigate the influence of the tip clearance at the shroud, another case is calculated with a tip clearance ratio, \( \frac{c}{\text{chord}} \), equal to 28 percent. The lower part of Figure 7.21 plots the rotor head versus the tip clearance change. With the tip clearance ratio increases from 7 percent to 28 percent, the static pressure difference drops from 57.4 mmHg to 45.1 mmHg.

7.1.5 Shear stress distributions

For the clinic use, the pump should satisfy the physiological criterion for the rate of hemolysis. As a first step of investigation, the shear stress is investigated. The definition of the shear stress is the same as that in chapter 4. Figure 7.22 through 7.24 are the shear stress distributions on grid planes \( k=3 \), \( k=12 \), and \( k=18 \), respectively.

The maximum shear stress in the entire domain is found to be 1500 dyne/cm\(^2\). At the lowest grid level, the maximum shear stress is 970 dyne/cm\(^2\) at the entrance of the volute. At the middle grid level it is 330 dyne/cm\(^2\) at the outer boundary of
the volute. At the highest grid level it is 1500 dyne/cm² around the trailing edge of the blade.
Figure 7.4: Streamline plot

Figure 7.5: Velocity vectors on the grid plane $k=5$

Figure 7.6: Velocity vectors on the grid plane $k=9$

Figure 7.7: Velocity vectors on the grid plane $k=13$

Figure 7.8: Velocity vectors on the grid plane $k=17$

Figure 7.9: Pressure contour on the grid plane $k=5$

196
Figure 7.10: Pressure contour on the grid plane k=9

Figure 7.11: Pressure contour on the grid plane k=13

Figure 7.12: Pressure contour on the grid plane k=17

Figure 7.13: Velocity vectors on the grid plane j=5

Figure 7.14: Velocity vectors on the grid plane j=12

Figure 7.15: Velocity vectors on the grid plane j=18
Figure 7.16: Velocity vectors on the grid plane $j=25$

Figure 7.17: Pressure contour on the grid plane $j=5$

Figure 7.18: Pressure contour on the grid plane $j=12$

Figure 7.19: Pressure contour on the grid plane $j=18$

Figure 7.20: Pressure contour on the grid plane $j=25$

Figure 7.21: Top: Rotor head comparison, Middle: Rotor head versus flow rate, Bottom: Tip clearance effect
Figure 7.22: Shear stress contour on grid plane k=3

Figure 7.23: Shear stress contour on grid plane k=12

Figure 7.24: Shear stress contour on grid plane k=18
7.2 The B3452 design

In this section the actual IVAS geometry is used for investigation. We obtain a similar flow behavior as in the previous calculation. Figure 7.25 through Figure 7.47 are the velocity vectors and the corresponding pressure contour on the grid plane counted from the bottom. Separations always occur at the leading edge of the blade. The flow pattern becomes more complicated near the top surface of the passage. At the entrance of the volute, the flow seems turbulent. Recirculation is found at the pressure side of the blade, and it becomes stronger with the grid plane increasing to the top surface of the flow passage. Figure 7.48 through Figure 7.69 are the velocity vectors and the corresponding pressure contour on the grid plane in the theta direction. Recirculation can be observed in the volute area as well as at the entrance of the volute. Strong reversed flow appears in the inlet channel at the leading edge. Figure 7.70 shows the angular velocity contour at grid plane j=15. The upper part of the leading edge is exposed to the pre-swirled fluid, while the lower part of the blade leading edge is passed by the non-swirling inlet flow. Figure 7.71 is the streamlines in the inlet flow viewed from the rotating frame. The streamlines in the inlet flow viewed from the fixed frame is plotted in Figure 7.72. The streamlines near the center line are fairly straight, while others turn rapidly when approaching the blades.

7.3 Summary

The simulation results for the design operating conditions indicate:
(1) A large flow separation exists in the inlet channel before the impeller. This separation is caused by the backward flow along the top surface above the impeller.

(2) The lower part of the blade leading edge is passed by the non-swirling inlet flow, while the upper part of the leading edge is exposed to the pre-swirled fluid.

(3) The flow in the volute has strong vortical motions.

(4) In the middle region of the adjacent blades, flow exiting the impeller and entering the volute looks like a jet.

(5) The computed rotor head agrees well with the prediction of the Pumpa code.

(6) With the tip clearance ratio (c/chord) increases form 7 percent to 28 percent, the rotor head drops from 57.4 mmHg to 45.1 mmHg.

(7) The maximum shear stress is approximately 150 pa and is within the safety criterion for hemolysis. It occurs at the entrance of the volute near the bottom and at the trailing edge of the impeller near the shroud,
Figure 7.25: B3452 design: Velocity vectors on the grid plane $k=3$

Figure 7.26: B3452 design: Velocity vectors on the grid plane $k=5$

Figure 7.27: B3452 design: Velocity vectors on the grid plane $k=7$

Figure 7.28: B3452 design: Pressure contour on the grid plane $k=3$

Figure 7.29: B3452 design: Pressure contour on the grid plane $k=5$

Figure 7.30: B3452 design: Pressure contour on the grid plane $k=7$
Figure 7.31: B3452 design: Velocity vectors on the grid plane k=9

Figure 7.34: B3452 design: Pressure contour on the grid plane k=9

Figure 7.32: B3452 design: Velocity vectors on the grid plane k=11

Figure 7.35: B3452 design: Pressure contour on the grid plane k=11

Figure 7.33: B3452 design: Velocity vectors on the grid plane k=13

Figure 7.36: B3452 design: Pressure contour on the grid plane k=13
Figure 7.37: B3452 design: Velocity vectors on the grid plane \(k=14\)

Figure 7.38: B3452 design: Velocity vectors on the grid plane \(k=15\)

Figure 7.39: B3452 design: Velocity vectors on the grid plane \(k=16\)

Figure 7.40: B3452 design: Pressure contour on the grid plane \(k=14\)

Figure 7.41: B3452 design: Pressure contour on the grid plane \(k=15\)

Figure 7.42: B3452 design: Pressure contour on the grid plane \(k=16\)
Figure 7.43: B3452 design: Velocity vectors on the grid plane $k=17$

Figure 7.46: B3452 design: Pressure contour on the grid plane $k=17$

Figure 7.44: B3452 design: Velocity vectors on the grid plane $k=19$

Figure 7.47: B3452 design: Pressure contour on the grid plane $k=19$

Figure 7.45: B3452 design: Velocity vectors on the grid plane $j=5$

Figure 7.48: B3452 design: Pressure contour on the grid plane $j=5$
Figure 7.49: B3452 design: Velocity vectors on the grid plane j=7

Figure 7.50: B3452 design: Velocity vectors on the grid plane j=9

Figure 7.51: B3452 design: Velocity vectors on the grid plane j=11

Figure 7.52: B3452 design: Pressure contour on the grid plane j=7

Figure 7.53: B3452 design: Pressure contour on the grid plane j=9

Figure 7.54: B3452 design: Pressure contour on the grid plane j=11
Figure 7.55: B3452 design: Velocity vectors on the grid plane j=13

Figure 7.56: B3452 design: Velocity vectors on the grid plane j=15

Figure 7.57: B3452 design: Velocity vectors on the grid plane j=17

Figure 7.58: B3452 design: Pressure contour on the grid plane j=13

Figure 7.59: B3452 design: Pressure contour on the grid plane j=15

Figure 7.60: B3452 design: Pressure contour on the grid plane j=17

207
Figure 7.61: B3452 design: Velocity vectors on the grid plane j=19
Figure 7.64: B3452 design: Pressure contour on the grid plane j=19

Figure 7.62: B3452 design: Velocity vectors on the grid plane j=21
Figure 7.65: B3452 design: Pressure contour on the grid plane j=21

Figure 7.63: B3452 design: Velocity vectors on the grid plane j=23
Figure 7.66: B3452 design: Pressure contour on the grid plane j=23

208
Figure 7.67: B3452 design: Velocity vectors on the grid plane j=25

Figure 7.70: B3452 design: Angular velocity contour on the grid plane j=15

Figure 7.68: B3452 design: Pressure contour on the grid plane j=27

Figure 7.71: B3452 design: Streamlines in the inlet viewed from the rotating frame

Figure 7.69: B3452 design: Velocity vectors on the grid plane j=27

Figure 7.72: B3452 design: Streamlines in the inlet flow viewed from the fixed frame
CHAPTER 8

CONCLUSIONS AND RECOMMENDATIONS FOR THE FUTURE WORK

A 3D unsteady incompressible Navier-Stokes solver was developed. Both the momentum and pressure correction equations are discretized in the fully conservative forms, based on Nakamura's new algorithm [66][67]. The conservative differencing and homogeneous Neumann boundary condition for the pressure correction equation make the pressure equation converge to machine errors. An extremely small amount of fourth-order numerical viscosity helps the solver to capture unsteady and small vortex activities even with a relatively coarse mesh. To prevent the spurious oscillations inherent with the non-staggered grid, the Rhie-Chow correction is adopted.

In the analysis of the journal bearing, the computational results agreed well with the results from the Reynolds equation (MTI’s results). The flow from the journal bearing to the secondary impeller is not only non-uniform but also reversed in some portion of the passage. The pressure does not change in the radial direction. The maximum pressure of 0.15 bar occurs at $\theta = 210$ degrees in the circumferential direction and at $z/H = 0.55$ in the vertical direction. The minimum pressure of -0.1 bar occurs at $\theta = 280$ degrees and at $z/H = 0.44$. Through a parametric study we find that the flow patterns are not changed neither for the change in the pressure
drop between the inlet and the exit nor the change in the RPM. The maximum shear stress is below the threshold of hemolysis.

Through the computational analysis for the flows in the secondary impeller cavity, we observe a reversed Ekman layer flow pattern. Besides, many small vortices are distributed in the cavity. The pressure distribution varies both in space and time. In fact, the time-wise fluctuation indicates existence of strong unsteady vortical flows around the blades. The tip gap affects the pressure head. With an increase in the tip gap, the pressure head decreases. The flow patterns with the smoothed secondary impeller blades produces weaker vortex motions behind the blade. For the hang-over part of the improved design, Karman vortices are noticed. The hang-overs increase the shear stress on the side and bottom surfaces of the secondary impeller cavity, which is desirable to decrease the potential of thrombosis. However, the shear stress at the corner is still very low.

A single passage model is used to investigate the flow behavior of the primary impeller. The validity of the model has been confirmed through a parametric study of the 2.5D model. The rotor head is over-predicted when compared to the results of the Pumpa code. This is because the tip clearance loss and turbulence loss are considered in the Pumpa code while in the 2.5D such effects are not built in. The Pumpa code generally gives reliable prediction for the pump performance based on empirical data but gives no fluid dynamical explanation. The computational model, on the other hand, simulates the flows and visualizes the flow quantities that answer many questions.
The 3D single passage model is an extension of the 2.5D single passage model. It can investigate the effect of tip clearance, the effect of leakage, and the effect of pre-swirl, which can not be calculated by the 2.5D model. For the design operating conditions, a large flow separation exists in the inlet channel before the impeller. This separation is caused by the backward flow along the top surface above the impeller. The lower part of the leading edge is passed by the non-swirling inlet flow, while the upper part of the leading edge is exposed to the pre-swirled fluid. The flow in the volute has strong vortical motions. In the middle region of the adjacent blades, flow exiting the impeller and entering the volute looks like a jet. The computed rotor head agrees well with the prediction of the Pumpa code. Tip clearance effect on the rotor head is quantitatively investigated. The maximum shear stress is approximately 150 pa, which occurs at the entrance of the volute near the bottom and at the trailing edge of the impeller near the shroud, and is within the safety criterion for hemolysis.

8.1 Recommendations for the future work

Following are some suggestions for the future work:

(1) The computational results need to be verified with experimental data. Quantitative comparisons are quite critical.

(2) A journal bearing is a dynamic procedure, i.e., the shaft movement in the bearing gap is time dependent. Simulation of such a moving boundary is quite challenging, however.

(3) For the shear stress exceeding 150 pa, the effect on hemolysis will need more investigation.

(4) Evaluating the shear stress caused by numerical viscosity.
(5) Because the flow shows transient behavior, it is necessary to investigate the unsteady phenomenon of shear stress.

(6) The single passage model can be extended to handle the whole pump geometry so that the water cut effect, volute effect can be investigated.
BIBLIOGRAPHY


