BLOOD FLOW IN END-TO-SIDE ANASTOMOSES OF VENTRICULAR ASSIST DEVICES

A Dissertation in

Bioengineering

by

Ning Yang

©2009 Ning Yang

Submitted in Partial Fulfillment

of the Requirements

for the Degree of

Doctor of Philosophy

December 2009
The dissertation of Ning Yang was reviewed and approved* by the following:

Keefe B. Manning  
Assistant Professor of Bioengineering  
Dissertation Advisor  
Chair of Committee

Steven Deutsch  
Senior Scientist  
Professor Emeritus of Bioengineering

Eric G. Paterson  
Associate Professor of Mechanical Engineering

William J. Weiss  
Professor of Surgery and Bioengineering

Suzanne M. Shontz  
Assistant Professor of Computer Science and Engineering

Herbert H. Lipowsky  
Professor and Chairman of Bioengineering  
Head of the Department of Bioengineering

*Signatures are on file with the Graduate School.
Abstract

Although there are many studies that focus on the fluid mechanics within ventricular assist devices (VADs), the impact of VAD anastomoses on the aortic flow has been largely ignored. Hence, we study the blood flow within the end-to-side anastomoses of VADs to adult and pediatric aortas. The great vessels originating from the aortic arch are included to study flow splitting. A second-order accurate time-dependent flow solver based on finite volume method is used to simulate the flow. Monotone integrated large eddy simulation is used to resolve large scales of the resulting turbulent flow based on grid cut-off and to model the sub-grid scale (SGS) motions using non-linear built-in (implicit) SGS turbulence models. Hemolysis is modeled using an advection-reaction transport equation. The flow solver is validated against analytical and experimental results. The effect of blood viscoelasticity is examined experimentally using particle image velocimetry. Our results suggest that the VAD anastomosis significantly alters the flow in the aorta. A proximal anastomotic configuration diverts an impingement jet into the brachiocephalic artery and may increase the perioperative right-sided stroke rate, whereas a distal anastomotic configuration leads to a large stagnant flow region near the aortic valve and possible thrombosis. Turbulence occurs for both configurations during the complete VAD support. The continuous support significantly reduces the blood damage in the aorta when compared to the pulsatile support, although it could lead to end-organ failure as a result of the reduced pulsatility. This work may help to identify the risk of graft failure for patients with VAD assistance.
Contents

List of Figures xii

List of Tables xviii

Nomenclature xix

Acknowledgments xxxiii

1 Introduction and Background 1

1.1 Left Ventricular Assist Devices ............................... 1
1.2 Pediatric Ventricular Assist Devices .......................... 3
1.3 End-to-Side Anastomosis ...................................... 5
1.4 Summary of Previous Work .................................... 7
1.5 Turbulence Modeling .......................................... 9
1.6 Blood Viscoelasticity ........................................... 12
1.7 Hemolysis ..................................................... 12
1.8 Platelet Activation and Thrombosis ............................ 15
1.9 The Current Study ............................................. 16
    1.9.1 Methodology ............................................. 16
    1.9.2 Validation ............................................... 16
    1.9.3 Significance ............................................. 17
2 Methodology

2.1 Basic Equations

2.2 Turbulence

2.3 Large Eddy Simulation Principles and Modeling

2.3.1 Filtering

2.3.2 Governing Equations

2.3.3 Sub-grid Scale Models

2.3.3.1 Eddy-Viscosity-Based Models

2.3.3.2 Stress Transport Models

2.3.3.3 Scale Similarity and Linear Combination Models

2.3.3.4 Dynamic Models

2.3.3.5 Sub-grid Scale Model Choice

2.3.4 Wall Treatment

2.4 Monotone Integrated Large Eddy Simulation

2.4.1 Rational

2.4.2 Numerics

2.5 Hemolysis Modeling

2.5.1 Hyperbolic Transport Equation

2.5.2 Stress Equivalent

2.5.3 Hemolysis Indices

2.6 Hemodynamics Parameters

2.6.1 Surplus Hemodynamic Energy

2.6.2 Wall Shear Stress and Oscillatory Shear Index

2.6.3 Principal Reynolds Stress

2.7 Finite Volume Discretization

2.7.1 Domain Discretization

2.7.2 Spatial Discretization
2.9 Principle of Fluid Viscoelasticity Measurement .......................... 65
  2.9.1 Governing Equation .................................................. 65
  2.9.2 Impedance ............................................................. 66
  2.9.3 Maxwell Relaxation Time .............................................. 67
2.10 Experimental Setup ......................................................... 68
  2.10.1 Pediatric Mock Circulatory Loop ................................. 69
    2.10.1.1 Heart Simulator .............................................. 70
    2.10.1.2 Pediatric Aortic Model .................................... 71
    2.10.1.3 Resistance ................................................... 72
    2.10.1.4 Compliance .................................................. 72
    2.10.1.5 Fluid ........................................................ 72
  2.10.2 Flow Measurement .................................................... 74

3 Validation 77
  3.1 Two-dimensional Decaying Taylor-Green Vortex ..................... 77
    3.1.1 Analytical Solution ............................................. 77
    3.1.2 Numerical Simulation ........................................... 79
      3.1.2.1 Schemes and Procedure .................................. 79
      3.1.2.2 Results ................................................... 79
  3.2 Fully-Developed Turbulent Channel Flow ............................. 81
    3.2.1 Computational Setup ............................................ 82
      3.2.1.1 Geometry .................................................. 82
      3.2.1.2 Numerical Schemes ...................................... 82
      3.2.1.3 Fluid Properties ........................................ 83
      3.2.1.4 Boundary Conditions and Flow Initialization .......... 83
    3.2.2 Model Comparison ............................................... 84
      3.2.2.1 Velocity .................................................. 84
      3.2.2.2 Three-dimensional Vortex Structures ................... 85
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.3</td>
<td>Laminar Pulsatile Pipe Flow</td>
</tr>
<tr>
<td>3.3.1</td>
<td>Analytical Solution</td>
</tr>
<tr>
<td>3.3.2</td>
<td>Numerical Solution</td>
</tr>
<tr>
<td>3.3.2.1</td>
<td>Computational Setup</td>
</tr>
<tr>
<td>3.3.2.2</td>
<td>Numerical Results</td>
</tr>
<tr>
<td>3.4</td>
<td>Pediatric Aortic Flow</td>
</tr>
<tr>
<td>3.4.1</td>
<td>Experiment</td>
</tr>
<tr>
<td>3.4.1.1</td>
<td>Flow Conditions</td>
</tr>
<tr>
<td>3.4.1.2</td>
<td>The Effect of Viscoelasticity</td>
</tr>
<tr>
<td>3.4.2</td>
<td>Numerical Simulation</td>
</tr>
<tr>
<td>3.4.2.1</td>
<td>Numerical Setup</td>
</tr>
<tr>
<td>3.4.2.2</td>
<td>Comparison of Results</td>
</tr>
<tr>
<td>3.5</td>
<td>Conclusion</td>
</tr>
<tr>
<td>4</td>
<td>Blood Flow in the Adult Left Ventricular Assist Device Anastomoses</td>
</tr>
<tr>
<td>4.1</td>
<td>Methods</td>
</tr>
<tr>
<td>4.1.1</td>
<td>Engineering Model Geometry</td>
</tr>
<tr>
<td>4.1.2</td>
<td>Governing Equations and Non-dimensionalization</td>
</tr>
<tr>
<td>4.1.3</td>
<td>Grid Generation</td>
</tr>
<tr>
<td>4.1.4</td>
<td>Flow Solvers and Numerical Algorithms</td>
</tr>
<tr>
<td>4.1.5</td>
<td>Fluid Properties and Computational Parameters</td>
</tr>
<tr>
<td>4.1.6</td>
<td>Boundary Conditions</td>
</tr>
<tr>
<td>4.2</td>
<td>Results</td>
</tr>
<tr>
<td>4.2.1</td>
<td>Grid Study</td>
</tr>
<tr>
<td>4.2.2</td>
<td>Mean Primary Flow Field</td>
</tr>
<tr>
<td>4.2.3</td>
<td>Mean Secondary Flow Field</td>
</tr>
<tr>
<td>4.2.4</td>
<td>Turbulence</td>
</tr>
<tr>
<td>4.2.5</td>
<td>Mean Wall Shear Stress</td>
</tr>
</tbody>
</table>
4.2.6 Flow Splitting in the Great Vessels ......................... 145
4.3 Discussion ....................................................... 146
  4.3.1 Anastomotic Flow ........................................... 146
  4.3.2 Limitations .................................................. 148
4.4 Conclusion ....................................................... 149

5 Blood Flow in the Pulsatile Pediatric Ventricular Assist Device Anastomosis 150

  5.1 Methods ......................................................... 151
    5.1.1 Geometrical Model ........................................ 151
    5.1.2 Turbulence Model ......................................... 152
    5.1.3 Boundary Conditions ..................................... 153
    5.1.4 Grid Generation ........................................... 154
    5.1.5 Flow Solver and Numerical Algorithm .................... 154
    5.1.6 Fluid Properties and Computational Parameters ........ 155

  5.2 Results ......................................................... 156
    5.2.1 Solution Verification ...................................... 156
    5.2.2 Flow Splitting ............................................ 159
    5.2.3 Velocity Contours ........................................ 161
    5.2.4 Principal Reynolds Stress ................................ 166
    5.2.5 Wall Shear Stress Distributions ......................... 167

  5.3 Discussion ....................................................... 174
    5.3.1 Flow Field ................................................ 174
    5.3.2 Wall Shear Stress and Oscillatory Shear Index .......... 175
    5.3.3 Potential Sites for Intimal Hyperplasia ................. 176
    5.3.4 Limitations .............................................. 178

  5.4 Conclusions ..................................................... 180
6 The Effect of Pumping Mode

6.1 Methods ......................................................... 181
   6.1.1 Geometrical Model ....................................... 181
   6.1.2 Mathematical Model ....................................... 182
   6.1.3 Boundary Conditions ..................................... 182
   6.1.4 Grid Generation ........................................... 184
   6.1.5 Flow Solver and Numerical Algorithms ................. 184
   6.1.6 Fluid Properties and Computational Parameters ...... 184

6.2 Results ......................................................... 185
   6.2.1 Pulsatile Hemodynamic Performance ................... 185
   6.2.2 Flow Splitting ............................................ 189
   6.2.3 Velocity Contours ....................................... 190
   6.2.4 Principal Reynolds Stress ............................... 193
   6.2.5 Hemolysis ................................................ 194
   6.2.6 Wall Shear Stress Distribution ......................... 195

6.3 Discussion ...................................................... 199
   6.3.1 Pulsatility ............................................... 199
   6.3.2 Flow Splitting ............................................ 199
   6.3.3 Flow Field ................................................ 200
   6.3.4 Hemolysis ................................................ 201
   6.3.5 Wall Shear Stress and Oscillatory Shear Index ....... 202

6.4 Conclusions .................................................... 202

7 Summary and Conclusions ....................................... 204

7.1 Summary ....................................................... 204

7.2 Conclusions ................................................... 205
   7.2.1 The Impact of Anastomotic Configurations ............. 205
   7.2.2 The Impact of VAD Pumping Mode ....................... 206
7.3 Recommendations for Future Work ........................................ 206

Bibliography ........................................................................... 207
List of Figures

1.1 Schematic of a LVAD anastomosis. ............................................. 3
1.2 Schematic of a PVAD anastomosis. ............................................. 5
1.3 Schematic of anastomoses: (a) end-to-end, (b) side-to-side, (c) end-to-side. 6
1.4 (a) Arterial graft, (b) AV graft. .................................................. 6
1.5 Schematic of a turbulent energy spectrum, showing (A) the region of very little energy production and no resolved dissipation, (B) the energy-containing region, (C) the initial subrange, (D) the dissipation region. E(k) is the kinetic energy spectral density, and k is the wavenumber. ... 10
1.6 Damage curves of red blood cells and platelets based on exposed stresses and exposure times. The region above and to the right of each curve represents the region with significant cell lysis of each type. ............... 13

2.1 Typical filter functions in real and Fourier space: (a) the Gaussian filter in real space; (b) the Gaussian filter in Fourier space; (c) the volume-averaged box filter in real space; (d) the volume-averaged box filter in Fourier space; (e) the Fourier cutoff filter in real space; (f) the Fourier cutoff filter in Fourier space. ....................................................... 22
2.2 CVs for the finite volume method. .................................................. 40
2.3 Face interpolation. ................................................................. 43
2.4 PISO procedure. ................................................................. 52
2.5 Experimental arrangement for PIV in a wind tunnel. ....................... 59
2.6 The three modes of particle image density: (a) low (PTV), (b) medium (PIV), (c) high image density (LSV) ........................................... 60

2.7 Schematic representation of geometric imaging. ........................................... 61

2.8 Cross correlation of interrogation areas of frames A and B: (a) the interrogation area of frame A, (b) the interrogation area of frame B, (c) the cross correlation function of the interrogation areas. ........................................... 62

2.9 Pictures of the in vitro mock circulatory loop composed of (1) heart simulator, (2) flow probe, (3) acrylic pediatric aortic model, (4) compliance chambers, (5) pressure transducer, (6) clamp, (7) resistance plate, (8) reservoir, and (9) air line. (a) The front view, (b) the back view, and (c) the enlarged view of the connection between heart simulator and pediatric aortic model. ........................................... 69

2.10 Schematic of the in vitro mock circulatory loop. Note that the red arrows represent the flow direction. ........................................... 70

2.11 The pediatric aorta: (a) the pediatric aortic model, (b) the acrylic model, (c) the split-up of the acrylic model. ........................................... 71

2.12 Viscosity and elasticity of the pediatric blood and blood analog at 40% HCT. ........................................... 74

2.13 An example of PIV images at different stages of processing: (a) raw image, (b) masked image, (c) background image, (d) pre-processed image after background subtraction. ........................................... 76

3.1 Analytical solution of the 2D Taylor-Green Vortex at time t = 1s: (a) velocity magnitude contour, (b) velocity vector, (c) pressure contour. ........................................... 78

3.2 Numerical velocity contours of the 2D Taylor-Green Vortex at time t = 1s using the TVD scheme: (a) 8 × 8 mesh, (b) 16 × 16 mesh, (c) 32 × 32 mesh, (d) 64 × 64 mesh. ........................................... 80
3.3 Normalized average velocity error over the computational domain versus normalized grid resolution for different numerical schemes at $t = 1s$. $|U_S - U_A|_{\text{mean}}$ is the mean velocity error between the simulation results ($U_S$) and analytical results ($U_A$) over the computational domain, $U_{\text{max}}$ is the maximum velocity magnitude, $\Delta x$ is the grid spacing, and $\Delta x_{\text{finest}}$ is the finest grid spacing.

3.4 Schematic of a fully-developed turbulent channel flow geometry. $x$ is the streamwise direction, $y$ is the cross-stream direction, and $z$ is the span-wise direction.

3.5 Mean streamwise velocity profile (a) and rms velocity fluctuations (b) in the fully-developed turbulent channel flow at $Re_\tau = 395$.

3.6 Instantaneous vortex structures in the fully-developed turbulent channel flow visualized through the isosurface contours of the second largest eigenvalue ($\lambda_2 < 0$) colored by normalized helicity: (a) LES, (b) MILES.

3.7 Schematic of the computational domain of a fully-developed laminar pipe flow.

3.8 Pipe inlet pressure and flow waveforms.

3.9 Radial cross-section of the pipe mesh.

3.10 Comparison of the analytical and numerical solutions in terms of the velocity composite Fourier coefficients.

3.11 Experimental flow waveforms on each boundary for the Newtonian and 40% HCT blood analogs.

3.12 Experimental aortic pressure waveforms for the Newtonian and 40% HCT blood analogs.

3.13 Experimental velocity contours in planes 12 mm, 10 mm and 12.8 mm at various time steps.

3.14 Plane locations.
3.15 Comparison of the experimental and numerical flow waveform at each boundary based on the Newtonian blood analog.

3.16 Comparison of the numerical and experimental flow field at different time steps.

4.1 (a) Healthy aorta, (b) proximal configuration with the cannula attached on the ascending aorta, (c) distal configuration with the cannula attached on the aortic arch.

4.2 The effect of grid size on mean velocity magnitude near the distal cannulation site under operating condition III: (a) Position I: \( r^* = \frac{R_0^* + R_i^*}{2}, \theta = 45^\circ \);
(b) Position II: \( r^* = \frac{R_0^* + R_i^*}{2}, \theta = 15^\circ \).

4.3 The effect of grid size on the temporal energy spectra at probe 1 for the distal configuration under operating condition II.

4.4 Summary of the plane and probe locations used in this study.

4.5 Mean velocity contour: (a) condition I, (b) condition II, proximal configuration, (c) condition III, proximal configuration, (d) condition II, distal configuration, (e) condition III, distal configuration.

4.6 Mean secondary flow fields for the proximal configuration under flow condition III.

4.7 Mean secondary flow fields for the distal configuration under flow condition III.

4.8 Power spectra of the fluctuating velocities at probe 2 for the distal configuration.

4.9 The maximum principal normal Reynolds stress at planes \( z^* = 0, z^* = -0.3, \) and \( z^* = 0.3 \) for flow condition II: (a) proximal configuration, (b) distal configuration.
4.10 Mean WSS magnitude contour: (a) condition I; (b) condition II, proximal configuration; (c) condition III, proximal configuration; (d) condition II, distal configuration; (e) condition III, distal configuration.

5.1 (a) Healthy pediatric patient-specific aortic model, (b) proximal anastomotic model, (c) enlarged distal view of the graft junction indicated by the back square in (b).

5.2 Effect of grid on the pressure and flow waveforms at each boundary. Note that the three different grids are differentiated by line type.

5.3 Effect of grid on WSS in six locations. Note that the three different grids are differentiated by line type.

5.4 Pressure and flow waveforms at each boundary for 50% and 100% bypass. Note that the pressure and flow waveforms are differentiated by line type.

5.5 Planes used in this study for velocity contours.

5.6 Velocity magnitude contour (unit: m/s) for the healthy pediatric aorta.

5.7 Velocity magnitude contour (unit: m/s) for 50% bypass.

5.8 Velocity magnitude contour (unit: m/s) for 100% bypass.

5.9 Magnitude contour of Reynolds stress (unit: dynes/cm²) at peak systole for 100% bypass: (a) maximum principal normal stress; (b) maximum shear stress. Note that the flow waveform shown above is the cannula outflow.

5.10 WSS magnitude (unit: dynes/cm²) contour at peak systole: (a) healthy pediatric aorta, (b) 50% bypass, (c) 100% bypass.

5.11 Pressure contour (unit: mmHg) at peak systole for 100% bypass. Note that the flow waveform shown above is the cannula outflow.

5.12 Contour of time-averaged WSS (unit: dynes/cm²): (a) healthy pediatric aorta, (b) 50% bypass, (c) 100% bypass.
5.13 Contour of OSI: (a) healthy pediatric aorta, (b) 50% bypass, (c) 100% bypass. ................................................................. 173
5.14 Regions prone to the development of IH. ............................. 178

6.1 Flow waveforms at the boundaries for all five cases. ............... 186
6.2 Pressure waveforms at the boundaries for all five cases. .......... 187
6.3 Comparison of percentage change in SHE from the healthy aortic flow (case 1) for the 50% and 100% support in the PM and CM. ......... 189
6.4 Velocity magnitude contour (unit: m/s) for case 3. ................... 192
6.5 Velocity magnitude contour (unit: m/s) for case 5. ................. 193
6.6 Magnitude contour of maximum principal normal Reynolds stress (unit: dynes/cm²) for 100% PVAD support: (a) case 4 (PM) at peak systole, (b) case 5 (CM). ................................................. 194
6.7 Linear hemolysis fraction $H_l$ contour for 100% PVAD support: (a) case 4 (PM) at peak systole, (b) case 5 (CM). .................... 195
6.8 Time-averaged WSS magnitude (unit: dynes/cm²) contour: (a) anterior view, (b) posterior view. ................................. 196
6.9 Posterior view of pressure contour (unit: mmHg) for 100% bypass: (a) case 4 (PM) at peak systole, (b) case 5 (CM). .................. 197
6.10 OSI contour: (a) anterior view, (b) posterior view. ............... 198
List of Tables

2.1 Chemical compositions of the blood analogs. .................................. 73

3.1 Flow splitting in the aorta. Flow percentage is the ratio between the flow rate and the CO. ................................................................. 96

4.1 Mesh non-orthogonality and skewness. ............................................. 128

4.2 Summary of operating flow conditions applied for both anastomotic configurations. ................................................................. 130

4.3 Flow splitting in the great vessels. Flow percentage is the ratio between the flow rate and the CO. ................................................................. 146

5.1 Summary of flow conditions. ............................................................. 153

5.2 Flow splitting in the great vessels. Flow percentage is the ratio between the flow rate and the CO. ................................................................. 161

6.1 Summary of flow conditions. ............................................................. 183

6.2 Comparison of pulsatile hemodynamic performance parameters between the pulsatile and continuous PVAD support. ......................... 188

6.3 Flow splitting in the great vessels. Flow percentage is the ratio between the flow rate and the CO. ................................................................. 190

6.4 Comparison of MIH for all five cases. .............................................. 195
Nomenclature

Roman Symbols

\( A \)  
surface of a control volume

\( A^+ \)  
constant

\( a_C \)  
central coefficient

\( A_f \)  
shared face

\( A_n \)  
Fourier coefficient of velocity

\( a_N \)  
matrix coefficient corresponding to the neighbor cells

\( A_o \)  
outlet surface

\( B \)  
combined stress

\( B_n \)  
Fourier coefficient of velocity

\( C \)  
cell centroid

\( C_\epsilon \)  
constant

\( C_k \)  
constant

\( C_n \)  
Fourier coefficient of pressure

\( C_{sgs} \)  
cross stress
\( C_s \) Smagorinsky constant

\( C_t \) matrix coefficient corresponding to the last time step

\( D \) diameter

\( D_a \) van Driest damping function

\( d_c \) vector connecting the centroids of two adjacent cells

\( D_d \) topology vector connecting neighboring control volume

\( D_n \) Fourier coefficient of pressure

\( D_e \) Deborah number

\( E \) kinetic energy spectral density

\( E_{TVD} \) error of TVD schemes when compared to the linear scheme

\( F \) cross point of centroid vector on the shared face

\( f \) any flow variable

\( f_{cutoff} \) cut-off frequency

\( F_i \) factor of in-plane particle loss

\( F_o \) factor of out-of-plane particle loss

\( G \) filter function

\( H \) plasma free hemoglobin fraction

\( h \) channel half height

\( H_l \) linear hemolysis variable

\( H_t \) total blood hemoglobin concentration of the plasma
\( He \)  
\( I \) unit tensor  
\( I_1 \) first invariant of Reynolds stress tensor  
\( I_2 \) second invariant of Reynolds stress tensor  
\( I_3 \) third invariant of Reynolds stress tensor  
\( I_A \) intensity distribution of image frame A  
\( I_B \) intensity distribution of image frame B  
\( J_0 \) zero-order Bessel function of the first kind  
\( J_1 \) first-order Bessel function of the first kind  
\( K \) complex constant  
\( k \) wavenumber  
\( k_c \) constant  
\( k_{first} \) first-order reaction coefficient  
\( k_{s gs} \) subgrid scale kinetic energy  
\( k_{spring} \) spring constant of the Maxwell model  
\( k_{zero} \) zero-order reaction coefficient  
\( L \) length  
\( L_{s gs} \) Leonard stress  
\( M \) off-diagonal matrix components  
\( m \) constant
\( m_p \) mass of a particle

\( N \) centroid of a neighbor cell

\( n \) normal unit vector on a surface

\( n_f \) unit normal vector on a shared face

\( n_w \) unit normal vector on the wall

\( p \) pressure

\( p_0 \) base pressure

\( Q \) volumetric flow rate

\( q \) turbulence kinetic energy

\( R \) radius of a pipe

\( r \) coordinate in the radial direction

\( R_{\Pi} \) cross correlation function

\( R_l \) hemolysis reaction coefficient

\( r_p \) equivalent radius of a particle

\( R_{sgs} \) Reynolds-like stress

\( Re \) Reynolds number

\( Re_f \) friction Reynolds number

\( S \) strain rate tensor

\( s \) shifting vector in the correlation plane

\( S_\psi \) source term
\( St \)  \( \text{Stokes number} \)

\( T \)  \( \text{period of flow cycle} \)

\( t \)  \( \text{time} \)

\( t_b \)  \( \text{bulk flow local time scale} \)

\( U \)  \( \text{velocity vector} \)

\( u \)  \( \text{velocity in the x direction} \)

\( u_x \)  \( \text{friction velocity} \)

\( U_A \)  \( \text{analytical velocity vector} \)

\( U_{\text{mean}} \)  \( \text{mean velocity magnitude} \)

\( U_S \)  \( \text{numerical velocity vector} \)

\( V \)  \( \text{flow domain} \)

\( v \)  \( \text{velocity in the y direction} \)

\( W \)  \( \text{anti-symmetric part of the velocity gradient tensor} \)

\( Wo \)  \( \text{Womersley number} \)

\( X \)  \( \text{coordinates in the image plane} \)

\( x \)  \( \text{x coordinate} \)

\( x_1 \)  \( \text{random position in the x coordinate} \)

\( y \)  \( \text{y coordinate} \)

\( y^+ \)  \( \text{non-dimensional wall distance} \)

\( y_w \)  \( \text{normal distance from the wall} \)
Z  impedance
z  z coordinate

Greek Symbols

\( \alpha \)  constant
\( \beta \)  constant
\( \chi \)  resistance
\( \delta \)  difference or error
\( \delta_{bias} \)  bias error
\( \delta_{rms} \)  uncertainty error
\( \delta_s \)  step function
\( \delta_{tot} \)  total error
\( \epsilon \)  dissipation rate
\( \eta \)  viscoelasticity
\( \eta'' \)  elasticity
\( \eta' \)  viscosity
\( \eta_{dash} \)  dash constant of the Maxwell model
\( \eta_K \)  Kolmogorov length scale
\( \gamma \)  constant
\( \Gamma_\psi \)  diffusivity
\( \Gamma_{num} \)  numerical diffusivity tensor
\( \kappa \) von Karman constant

\( \lambda \) relaxation time

\( \lambda_e \) tensor eigenvalue

\( \mu \) dynamic viscosity

\( \mu_{eff} \) effective dynamic viscosity

\( \mu_{sgs} \) subgrid scale eddy viscosity

\( \Omega \) control volume

\( \omega \) radian frequency

\( \omega_{\nu or} \) vorticity

\( \phi \) flux

\( \Psi \) flux limiter

\( \psi \) scalar property

\( \rho \) density

\( \sigma \) principal normal stress

\( \sigma_R \) principal normal Reynolds stress

\( \tau \) stress

\( \tau_{REq} \) Reynolds stress equivalent

\( \tau_R \) Reynolds stress

\( \tau^P_R \) principal Reynolds shear stress

\( \tau_{sgs} \) subgrid scale stress
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\tau^\text{MILES}_{\text{sgs}}$</td>
<td>implicit subgrid scale stress based on monotone integrated large eddy simulation</td>
</tr>
<tr>
<td>$\tau_s$</td>
<td>Reynolds stress threshold for hemolysis</td>
</tr>
<tr>
<td>$\tau_T$</td>
<td>stress equivalent in Tresca form</td>
</tr>
<tr>
<td>$\tau_{vE}q$</td>
<td>viscous shear stress equivalent</td>
</tr>
<tr>
<td>$\tau_{vM}$</td>
<td>stress equivalent in von Mises form</td>
</tr>
<tr>
<td>$\tau^{\text{norm}}_{vM}$</td>
<td>stress equivalent in normalized von Mises form</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>wall shear stress</td>
</tr>
<tr>
<td>$\Theta$</td>
<td>distribution of particle locations in an interrogation volume</td>
</tr>
<tr>
<td>$\theta$</td>
<td>viscosity angle</td>
</tr>
<tr>
<td>$\Upsilon$</td>
<td>magnitude of complex pressure waveform for sinusoidal flows</td>
</tr>
<tr>
<td>$v$</td>
<td>kinematic viscosity</td>
</tr>
<tr>
<td>$\varphi$</td>
<td>distance ratio</td>
</tr>
<tr>
<td>$\zeta$</td>
<td>shear rate</td>
</tr>
<tr>
<td>$\xi$</td>
<td>local ratio of consecutive gradients of a scalar variable</td>
</tr>
</tbody>
</table>

**Superscripts**

<table>
<thead>
<tr>
<th>Superscript</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$'$</td>
<td>variable in sub-grid scales</td>
</tr>
<tr>
<td>$^*$</td>
<td>dimensionless form</td>
</tr>
<tr>
<td>$\dagger$</td>
<td>complex form</td>
</tr>
<tr>
<td>$h$</td>
<td>high order</td>
</tr>
</tbody>
</table>
\( l \)  low order
\( n \)  time step
\( tr \)  transpose of a matrix

**Subscripts**

\( \Delta \)  non-orthogonal part
\( C \)  centroid
\( d_c \)  orthogonal part
\( f \)  face index
\( N \)  centroid at a neighbor cell

mean  mean
rms  root mean square of a variable
time-averaged  cycle average

**Other Symbols**

\( \bar{\cdot} \)  filtered in real space
\( \Delta \)  filter width
\( \cdot \)  dot product
\( \frac{D}{Dt} \)  material derivative
\( \hat{\cdot} \)  filtered in Fourier space
\( \Im \)  imaginary part
\( \int \)  integration
\((\triangle t)_{pl}\) \quad \text{time pulse delay}

\((N_p)_A\) \quad \text{total number of particle in frame A}

\((N_p)_{eff}\) \quad \text{effective particle image pair density}

\([\psi_C]\) \quad \text{vector of variable values at all cell centroids}

\([A]\) \quad \text{sparse matrix}

\([b]\) \quad \text{source vector determined by the previous time steps}

\([M]\) \quad \text{preconditioner}

\(\langle \rangle\) \quad \text{spatial average}

\(\mathcal{O}\) \quad \text{order}

\(\nabla\) \quad \text{gradient}

\(\nabla \cdot\) \quad \text{divergence}

\(\Re\) \quad \text{real part}

\(\sum\) \quad \text{sum}

\(\triangle D\) \quad \text{displacement vector of each particle}

\(\triangle p\) \quad \text{pressure drop}

\(\triangle t\) \quad \text{time step interval}

\(\triangle x\) \quad \text{grid spacing}

\(\triangle x_C\) \quad \text{mesh related length scale around the centroid C}

\(\triangle x_{\text{finest}}\) \quad \text{finest grid spacing}

\(\triangle x_p\) \quad \text{particle displacement in the x direction}
$\Delta y_p$ particle displacement in the y direction

**Acronyms**

- 2D two-dimensional
- 3D three-dimensional
- ADP adenosine diphosphate
- AV arteriovenous
- BD backward differencing
- BICCG incomplete Cholesky preconditioned biconjugate gradient
- BPM beats per minute
- CCD charge-coupled device
- CD central differencing
- CFD computational fluid dynamics
- CHD congenital heart disease
- CM continuous mode
- CO cardiac output
- CPB cardiopulmonary bypass
- CV control volume
- CVD cardiovascular disease
- DAG diacylglycerol
- DES detached eddy simulation
<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DILU</td>
<td>diagonal-based incomplete lower upper</td>
</tr>
<tr>
<td>DNS</td>
<td>direct numerical simulation</td>
</tr>
<tr>
<td>ECMO</td>
<td>extracorporeal membrane oxygenation</td>
</tr>
<tr>
<td>EEP</td>
<td>energy equivalent pressure</td>
</tr>
<tr>
<td>FCT</td>
<td>flux corrected transport</td>
</tr>
<tr>
<td>FDA</td>
<td>Food and Drug Administration</td>
</tr>
<tr>
<td>FVM</td>
<td>finite volume method</td>
</tr>
<tr>
<td>GAMG</td>
<td>generalised geometric/algebraic multi-grid</td>
</tr>
<tr>
<td>Gp</td>
<td>glycoprotein</td>
</tr>
<tr>
<td>GS</td>
<td>grid scale</td>
</tr>
<tr>
<td>HCT</td>
<td>hematocrit</td>
</tr>
<tr>
<td>HF</td>
<td>heart failure</td>
</tr>
<tr>
<td>HR</td>
<td>heart rate</td>
</tr>
<tr>
<td>ICCG</td>
<td>incomplete Cholesky preconditioned conjugate gradient</td>
</tr>
<tr>
<td>IH</td>
<td>intimal hyperplasia</td>
</tr>
<tr>
<td>ILES</td>
<td>implicit large eddy simulation</td>
</tr>
<tr>
<td>LDA</td>
<td>laser Doppler anemometry</td>
</tr>
<tr>
<td>LES</td>
<td>large eddy simulation</td>
</tr>
<tr>
<td>LSV</td>
<td>laser speckle velocimetry</td>
</tr>
<tr>
<td>LVAD</td>
<td>left ventricular assist device</td>
</tr>
</tbody>
</table>

xxx
<table>
<thead>
<tr>
<th>Term</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>MCSD</td>
<td>mechanical circulatory support device</td>
</tr>
<tr>
<td>MIH</td>
<td>modified index of hemolysis</td>
</tr>
<tr>
<td>MILES</td>
<td>monotone integrated large eddy simulation</td>
</tr>
<tr>
<td>MRI</td>
<td>magnetic resonance imaging</td>
</tr>
<tr>
<td>MUSCL</td>
<td>monotone upstream-centered schemes for conservation laws</td>
</tr>
<tr>
<td>NHLBI</td>
<td>National Heart, Lung and Blood Institute</td>
</tr>
<tr>
<td>NIH</td>
<td>National Institute Health</td>
</tr>
<tr>
<td>NIHe</td>
<td>normalized index of hemolysis</td>
</tr>
<tr>
<td>OSI</td>
<td>oscillatory shear index</td>
</tr>
<tr>
<td>PBICG</td>
<td>preconditioned bi-conjugate gradient</td>
</tr>
<tr>
<td>pCAS</td>
<td>pediatric cardiopulmonary assist system</td>
</tr>
<tr>
<td>PISO</td>
<td>pressure implicit with splitting of operators</td>
</tr>
<tr>
<td>PIV</td>
<td>particle image velocimetry</td>
</tr>
<tr>
<td>PKC</td>
<td>protein kinase C</td>
</tr>
<tr>
<td>PM</td>
<td>pulsatile mode</td>
</tr>
<tr>
<td>PPM</td>
<td>piecewise parabolic method</td>
</tr>
<tr>
<td>PTV</td>
<td>particle tracking velocimetry</td>
</tr>
<tr>
<td>PVAD</td>
<td>pediatric ventricular assist device</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-average Navier-Stokes</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Definition</td>
</tr>
<tr>
<td>--------------</td>
<td>------------</td>
</tr>
<tr>
<td>REMATCH</td>
<td>randomized evaluation of mechanical assistance for the treatment of congestive heart failure</td>
</tr>
<tr>
<td>RI</td>
<td>refractive index</td>
</tr>
<tr>
<td>rms</td>
<td>root mean square</td>
</tr>
<tr>
<td>SGS</td>
<td>subgrid scale</td>
</tr>
<tr>
<td>SHE</td>
<td>surplus hemodynamic energy</td>
</tr>
<tr>
<td>SIMPLE</td>
<td>semi-implicit pressure linked equations</td>
</tr>
<tr>
<td>SV</td>
<td>stroke volume</td>
</tr>
<tr>
<td>TAH</td>
<td>total artificial heart</td>
</tr>
<tr>
<td>TAP</td>
<td>time-averaged pressure</td>
</tr>
<tr>
<td>TV</td>
<td>total variation</td>
</tr>
<tr>
<td>TVD</td>
<td>total variation diminishing</td>
</tr>
<tr>
<td>TxA2</td>
<td>thromboxane A2</td>
</tr>
<tr>
<td>UD</td>
<td>upwind differencing</td>
</tr>
<tr>
<td>VAD</td>
<td>ventricular assist device</td>
</tr>
<tr>
<td>vWF</td>
<td>von Willebrand factor</td>
</tr>
<tr>
<td>WSS</td>
<td>wall shear stress</td>
</tr>
<tr>
<td>WSSG</td>
<td>spatial wall shear stress gradient</td>
</tr>
</tbody>
</table>
Acknowledgments

I would like to first thank my thesis advisor Dr. Keefe B. Manning for all of his support during my tenure at Penn State. Without him, this thesis work would have not been possible. Thanks then go to Dr. Steven Deutsch and Dr. Eric G. Paterson for their guidance and assistance throughout this work. I also must thank Dr. William J. Weiss and Dr. Suzanne M. Shontz for serving on the committee and providing unique insight to this work.

I would like to thank my parents, Weigu Yang, and Chunhua Ma, for their love and support. They have been the constant supporter in my life. I feel extremely fortunate to have their love.

I would like to offer my gratitude to all my current and past fellow lab members in the Artificial Heart Lab at Penn State as well. In particular, I would like to acknowledge Varun Reddy, Breigh Roszelle, and Jason Nanna for their help with particle image velocimetry, Luke H. Herbertson for his help with laser Doppler velocimetry, and Mike Perone for his assistance with my experiments.

I must acknowledge Jason Holmes and Jeff Nucciarone of the High Performance Computing Group at Penn State for their technical support with PC clusters. Thanks are also due to Dr. Ajit Yoganathan and Mr. Chris Haggerty from the Department of Biomedical Engineering at the Georgia Institute of Technology for providing the pediatric aortic model as well as David Porzio for his help with building the pediatric anastomotic model.

Last but not the least, I would like to give special thanks to the Bioengineering De-
partment and the administrative assistants for their help with my thesis work. Especially, I would like to thank the department chair, Dr. Herbert H. Lipowsky, for his support during my time at Penn State.

This work was funded in part by NIH NHLBI Grant HV 48191.
Chapter 1

Introduction and Background

It is estimated that 80,000,000 adults in America have one or more types of cardiovascular disease (CVD), of whom 5,700,000 have heart failure (HF) [1]. The estimated direct and indirect cost of HF in the United States for 2009 is $37.2 billion [1]. Transplantation has been an effective treatment for most end-stage heart failure patients, but it is severely limited by an inadequate supply of donor hearts. The increasing prevalence of HF has stimulated the research and development of ventricular assist devices (VADs) and total artificial hearts (TAHs), with an underlying aim of improving the survival and the quality of life for patients with end-stage heart failure. While VAD and TAH designs have improved significantly over the last few decades, they still are susceptible to hemolysis, thrombosis, tissue overgrowth and material erosion. In this chapter, we describe the evolution of VADs, their anastomoses (surgical connections of two structures) to the aorta, and their hemodynamics. Mechanisms for flow-induced blood damage is also described.

1.1 Left Ventricular Assist Devices

Since the inception of the artificial heart program at the National Institute Health (NIH) in 1964, a primary goal has been the development of VADs and TAHs for both temporary
and long-term use [2]. Left ventricular assist devices (LVADs) were initially designed to aid failing left hearts as a bridge to transplant for adult patients. In 2001, the Randomized Evaluation of Mechanical Assistance for the Treatment of Congestive Heart failure (REMATCH) study compared the treatment of patients, not eligible for transplant, using optimal medical management or LVAD implantation [3]. The study found that patients receiving LVADs were 48% less likely to die within the initial two-year period. The Food and Drug Administration (FDA) has approved several LVADs for use as a bridge-to-transplant in adult patients with end-stage HF [4]. The demand for use of LVADs has been increasing due to limited heart donor availability. For instance, there are 550,000 new patients diagnosed with HF in the United States annually, but fewer than 3,000 donor organs available worldwide every year [1, 3, 5]. As a result, LVADs have been increasingly considered as an option for destination therapy. Only one device (HeartMate VE) has been approved for this purpose [4].

Both 70 and 50 ml stroke volume (SV) LVADs have been developed at Penn State. The 70 ml pneumatic pump was first implanted clinically in 1985 [6], and is now the Thoratec® VAD. A completely implantable 70 ml LVAD [7], LionHeart®, with an electrically driven pusher plate to achieve fluid compression, has been found to provide satisfactory hemodynamic performance with little thrombus formation for adult patients with body weight over 70 kilograms. A 50 cc pump, designed for small adults and adolescents, entered the development phase in 1998 [8].

The LVAD is typically connected to the native heart between the apex of the left ventricle by the LVAD inflow cannula and to the ascending aorta through the LVAD outflow cannula (Figure 1.1). Depending on the state of the native heart, the LVAD can work in serial or in parallel to it. Soon after surgery, for example, most blood flow moves through the LVAD, short-circuiting the aortic valve. As the heart recovers, however, it starts to pump blood through the native aortic valve, working in parallel with the LVAD. After the heart recovers, or the patient is ready for transplant, blood flow through the
LVAD is reduced, and eventually the device is explanted.

Figure 1.1: Schematic of a LVAD anastomosis.

1.2 Pediatric Ventricular Assist Devices

In contrast to adult patients, there are no mechanical circulatory support devices (MCSDs) approved in the United States for pediatric patients. Congenital heart disease (CHD) is the leading cause of infant death from birth defects in the United States [9]. Over 35,000 babies are born with heart defects annually in the United States, with a quarter of these patients requiring invasive treatment [10, 11]. Transplantation, an excellent option for
these patients, is limited by the inadequate number of donor organs [12]. For this reason, the NIH initiated The National Heart, Lung and Blood Institute (NHLBI) Pediatric Circulatory Support Program in 2004 [11]. Five contracts were awarded with the task of creating MCSDs for children ranging from 2 to 25 kg. Three of the five MCSDs are designed to deliver continuous blood flow. The fourth pump is a pediatric cardiopulmonary assist system (pCAS), which in addition to continuous flow can deliver pulsatile flow. The last, the Penn State pediatric ventricular assist device (PVAD), is a pulsatile, seamless blood sac design based on the successful, pneumatic Pierce-Donachy/Thoratec adult device [13].

The Penn State PVADs are being developed with 12 and 25 ml nominal SVs. The 12 ml pump is designed for patients less than a year old, weighing 2-7 kg, and having 0.5-1.4 L/min of cardiac output (CO), while the 25 ml pump is for patients 2-6 months to 5-9 years old, weighing 6-23 kg, and having 1.3-3 L/min of CO. The anastomoses of PVADs are very similar to those of adult LVADs. In some cases the PVAD inflow cannula is inserted into the left atrium instead of the apex of the left ventricle, as shown in Figure 1.2.
1.3 End-to-Side Anastomosis

Anastomoses have been widely used in coronary artery disease and diseases of the peripheral vessels, in organ transplantation and implantation of medical devices, and in trauma. Depending on patient condition and surgical objective, the anastomosis can be created in an end-to-end, side-to-side, side-to-end, or end-to-side configuration [14, 15, 16, 17], as shown in Figure 1.3. Hemodynamic parameters, such as wall shear stress (WSS) [18, 19, 20], spatial gradient of WSS (WSSG) [21], and oscillatory shear index (OSI) [22], play an important role in vascular remodeling and graft failure at anastomotic sites. As
a result, blood flow in anastomoses has received considerable attention in recent years. Studies have focused on the hemodynamics of vascular end-to-side anastomoses, with an emphasis on arterial grafts [23, 24, 25, 26] and arteriovenous (AV) grafts [27, 28] (Figure 1.4). These studies have shown that the WSS has a significant impact on the development of intimal hyperplasia (IH), characterized by the thickening of the intima of a blood vessel [19, 20, 29, 30, 31, 32]. IH is a major cause of long-term failure of vascular grafts. Complex flow or low WSS near an anastomotic site triggers a biological response in endothelial cells [33], which leads to proliferation and migration of smooth muscle cells into the intima [34]. As a result, a stenosis forms and ultimately, vascular grafts fail due to thrombus formation.

![Diagram of anastomoses](image1)

Figure 1.3: Schematic of anastomoses: (a) end-to-end, (b) side-to-side, (c) end-to-side.

![Diagram of grafts](image2)

Figure 1.4: (a) Arterial graft, (b) AV graft [17].

Anastomotic flow is more complicated in the aorta. The flow often exhibits transition to turbulence near peak systole [17], which might lead to the increased risk of early
hemolysis and thrombosis. An example is left ventricular aortic bypass (Figures 1.1 and 1.2), which is widely used for implantation of LVADs, which assist or replace the function of a failing left ventricle. The bypass procedure creates a side-to-end anastomosis on the apex of the left ventricle by a LVAD inflow graft and an end-to-side one on the ascending aorta through an outflow graft. The flow division in the LVAD anastomosis varies widely depending on the state of the native heart, and these wide variations in flow division have a significant impact on the hemodynamics in the aorta.

1.4 Summary of Previous Work

While the fluid mechanics within LVADs has been studied extensively [35, 36, 37, 38, 39], the flow patterns in the aorta, when connected to the LVAD, have not been thoroughly investigated either numerically or experimentally. Sherwin et al.[25] used both a laminar computational model and magnetic resonance imaging (MRI) to investigate the effect of out-of-plane attachment on the flow patterns within a rigid, distal, end-to-side anastomosis model. They found that the non-planar attachment may promote flow mixing and reduce low shear regions. May-Newman et al.[40] used a laminar, steady, computational fluid dynamics (CFD) model to study how different outflow cannula anastomotic locations affect flow patterns in a rigid, adult aortic model. Their study demonstrated that distal anastomotic configurations on the aortic arch may be not appropriate for patients with low or no flow from the native heart. The effect of the great vessels on the aortic arch, however, was not considered. Litwak et al. studied the effect of LVAD outflow graft locations on the aortic flow in both an animal model [41] and a mock circulatory loop [42] using continuous and pulsatile LVADs. They found that the LVAD outflow graft location has a large impact, at least in the short term, on flow splitting in the aortic branch vessels. The detailed flow field in the aorta was not provided. Minakawa et al.[43] visualized three-dimensional (3D) flow patterns in a glass adult aortic model during extracorporeal
circulation using particle image velocimetry (PIV) with the cannulae attached to the ascending aorta. Turbulent flow was observed near the aortic cannulation sites. WSS distribution near the cannulation site and flow splitting in the great vessels, however, were not reported.

In contrast to adult LVADs, PVADs are often associated with hemolysis and thrombosis, partly due to higher peak inflow into the aorta as a result of smaller graft diameters. Therefore, it is important to study the turbulent flow in the pediatric aorta with PVAD support. Many studies have focused on understanding the consequence of pumping mode (continuous versus pulsatile) associated with PVAD assistance on vascular pulsatility. Taylor et al. [44] demonstrated that the pulsatility of cerebral flow helped to regulate the blood flow to the brain for infants during extracorporeal membrane oxygenation (ECMO). Undar et al. [45] studied the flow in a neonatal piglet and found that pulsatile perfusion improved cerebral, renal and myocardial blood flow after cardiopulmonary bypass (CPB). Pentalos et al. [46] performed an acute animal study in piglets of left ventricular dysfunction with the pCAS to study differences between pulsatile and continuous flows. They found that pulsatile flow produced pulsatility three times greater than continuous flow in cases of less than total left ventricular bypass by the pCAS. The impact of PVAD assistance on the local hemodynamics near the graft sites has received less attention. Pekkan et al. [47] simulated the detailed pulsatile hemodynamics of a patient-specific neonatal aortic model during cardiopulmonary bypass (CPB) based on a laminar CFD model. They found that the local hemodynamics depends on the cannulation configuration.

It is clear that turbulence has been largely ignored in the previous studies. The adult and pediatric VAD anastomoses can cause strong mixing of flow in the aorta and impingement jet on the aortic wall. As a result, turbulence may occur and needs to be considered in the analysis.
1.5 Turbulence Modeling

Current computational methods for turbulent flow range from the direct solution of the flow field to modeling the flow statistics. Direct solution of the flow field is by direct numerical simulation (DNS). This is the most accurate approach as it resolves all scales of flow motion and no modeling is required. This method, however, is limited to simple geometries and low Reynolds number \((Re)\) turbulent flows due to its computational expense. In contrast to DNS, Reynolds-Average Navier-Stokes (RANS) modeling is computationally inexpensive but less accurate. It solves the ensemble-averaged Navier-Stokes equations to predict the flow field statistics. Large eddy simulation (LES) \([48, 49]\) lies between DNS and RANS. It explicitly resolves the large-scale motion (grid scale (GS)) while modeling the energy transfer to smaller motions (subgrid scale (SGS)) through a set of filtered Navier-Stokes equations. As a result, LES can only be used to calculate the flow statistics determined by the large scales. Detached-Eddy Simulation (DES) is a class of models used to combine RANS modeling near surfaces with LES away from surfaces. Compared to LES, DES can significantly reduce the computational cost for wall-bounded flows. Recently, Grinstein and DeVore \([50]\) presented results from 3D, time-dependent simulations of turbulent flows obtained without explicit SGS stress models. They reasoned that the discretization errors of monotone convection algorithms provide built-in or implicit SGS models which satisfy physically required SGS properties and incorporate most effects of the SGS motion \([51, 52]\). This is called monotone integrated large eddy simulation (MILES), or implicit large eddy simulation (ILES).

Figure 1.5 shows the ranges of turbulence motions resolved by various modeling approaches on a turbulence energy spectrum. The cut-off wavenumber for RANS is 0 because no turbulence motions are resolved. The cut-off for DNS lies to the right of the dissipation range (region D) because all the scales are resolved. The cut-off wavenumbers for LES/MILES lies between RANS and DNS. DES allows a variable cutoff between RANS and LES/MILES. Region A contains the scales associated with the energy transfer
between resolved and unresolved energy production. All modeling approaches fail in this region.

![Schematic of a turbulent energy spectrum](image)

**Figure 1.5**: Schematic of a turbulent energy spectrum, showing (A) the region of very little energy production and no resolved dissipation, (B) the energy-containing region, (C) the initial subrange, (D) the dissipation region. \( E(k) \) is the kinetic energy spectral density, and \( k \) is the wavenumber.

Simulating turbulent flows in the cardiovascular system involves several numerical obstacles. First, turbulence often occurs in complex 3D geometries, and therefore, sufficient mesh resolution is required to resolve all the turbulence scales. Second, the flow often exhibits transition to turbulence and a complex relaminarization process not found in fully turbulent flows. Nevertheless, various modeling approaches have been used to study turbulent blood flows, particularly stenotic flow. Varghese and Blackburn [53] and Ryval et al. [54] used unsteady RANS models to study a stenotic flow. Reasonable agreement was found between the numerical and experimental results in term of complex flow features downstream of the stenosis. Mittal et al. [55] applied a LES model to study turbulent flow in a simplified arterial stenosis model under a sinusoidal flow waveform. Approximately a quarter of a million grid cells were used in their simulation. The turbulent
data were sampled for eight cycles after the flow stabilized. Their simulation predicted features of post-stenotic flow reported by previous experiments and provided time accurate information for a variety of scales not available with RANS. Varghese et al.\[56, 57\] performed DNS of both steady and pulsatile flows, through an axisymmetric and eccentric 75% stenosed tubes, using a high-order spectral-element method. Two hundred and fifty-six and 1024 processors were used for the axisymmetric and eccentric models, respectively. Twenty-five cycles were used for turbulent data collection. Their DNS predicted a completely laminar flow under both the steady and pulsatile flow conditions for the axisymmetric model. Transition to turbulence was found in the post stenotic region of the eccentric model for both the steady and pulsatile (near peak systole) conditions. Their results provide a valuable database for turbulence in straight pipe flows with a stenosis. Lee et al.\[58\] studied transitional flow in a stenotic carotid bifurcation using DNS. Two hundred and fifty-six processors and nearly two million grid cells were used. Complex flow features such as vortex shedding, Dean's flow, and shear-layer destabilization were observed. DNS has been used in studying flow in AV grafts. AV grafts create abnormally high flow rates with relatively lower pulsatility than that in arterial flows. As a result, in contrast to stenotic flow, flow in AV grafts may be turbulent throughout the entire flow cycle. Lee et al.\[28\] studied transitional turbulent flow in an AV graft under various flow divisions using DNS. Steady inflow conditions were used in their study. Their results indicate the importance of flow division on the development of turbulence in the graft.

Overall, although DNS has a clear advantage over other turbulence models in modeling transitional turbulent flow, a large number of processors (i.e., hundreds or thousands) are required. This is especially true for the aortic flow with VAD anastomoses, which often exhibit high flow rates at peak systole and involves complex multi-branch geometries. Therefore, very fine temporal and spatial resolutions are required to resolve the smallest (Kolmogorov) scales of the flow \[59\], which makes DNS very expensive. In this study, a MILES method based on a finite volume approach is used to resolve the
energy-containing structures of the flow while the SGS effects are incorporated implicitly through numerical dissipation. The advantage of this method over DNS is that it can predict relatively accurate macroscopic behavior of the flow using a much smaller number of computational resources.

1.6 Blood Viscoelasticity

Blood is a suspension of cells (red blood cells, white blood cells, and platelets) in plasma. The viscoelasticity of blood is largely dependent on red blood cells. At shear rates lower than $50 \text{s}^{-1}$, the viscoelasticity of blood increases exponentially due to the formation of irregular large red cell aggregates and rouleaux. At high shear rates (above $100 \text{s}^{-1}$), rouleaux are dispersed, and individual red cells are deformed and aligned with the direction of flow [60, 61, 62, 63, 64, 65]. As a result, viscoelasticity decreases. Flow in the aorta is usually assumed to be Newtonian due to the relatively high shear rate. However, non-Newtonian effects could still be critical in small geometries such as the pediatric aorta. It has been reported that in PVADs, Newtonian blood analogs produce higher values of WSS and Reynolds stresses than non-Newtonian blood analogs [66]. The viscoelasticity of blood needs to be considered.

1.7 Hemolysis

Hemolysis (lysis of red cells), which often occurs at high fluid stresses, remains a problem to the development of VADs. Figure 1.6 shows the damage curves of red blood cells and platelets. It is clear that hemolysis depends not only on stresses that cells are exposed to, but also on the exposure time. As stresses increase, the exposure time required for causing damage decreases. For example, when red cells are exposed to stresses higher than $10^4 \text{dynes/cm}^2$, an exposure time on the order of milliseconds is sufficient for hemolysis, as shown in Figure 1.6. The platelet damage curve has a similar trend to that of red
blood cells, but has a different combination of stresses and exposure times, as shown in Figure 1.6.

Figure 1.6: Damage curves of red blood cells and platelets based on exposed stresses and exposure times. The region above and to the right of each curve represents the region with significant cell lysis of each type. Lysis data are taken from [60].

Models have been developed to study hemolysis in implantable devices. One widely used empirical model proposed by Giersiepen et al. [67] can be expressed as:

$$H = \gamma \tau^{\alpha}_{vEq} t^\beta,$$  \hspace{1cm} (1.1)

where $H$ is plasma free hemoglobin fraction, $\tau_{vEq}$ is viscous shear stress equivalent, $t$ is time, and $\alpha$, $\beta$, and $\gamma$ are constants, defined as:

$$\alpha = 2.416, \beta = 0.785, \gamma = 3.62 \times 10^{-7}.$$  \hspace{1cm} (1.2)

Many researchers [47, 68, 69, 70] have used this formula to estimate hemolysis by inte-
grating it along flow streamlines. This approach has several major deficiencies. First, its accuracy is always limited by the number and location of streamlines chosen for integration. Second, any transient effect in the flow is ignored. Third, the detailed distribution of hemoglobin concentration (in the plasma) inside a device cannot be obtained. As a result, hemolysis high spots inside the device cannot be identified. To tackle these problems, Farinas et al.[71, 72] presented a linear advection-reaction partial differential equation as a tool for the design of cardiovascular prosthetic devices:

\[
\frac{\partial H_l}{\partial t} + U \cdot \nabla H_l = R_l, \tag{1.3}
\]

where \(U\) is velocity vector, \(R_l\) is the reaction term (production of hemolysis) as a function of shear stresses, and \(H_l\) is a linear hemolysis variable, defined as:

\[
H_l = H^{1/\beta} = \gamma^{1/\beta} \tau_v^{n/\beta} \tau_{vE} t. \tag{1.4}
\]

The detailed derivation of the model is presented in Section 2.5. Lacasse et al.[73, 74] modified Farinas’s hemolysis model by introducing a step function in the source term to account for the threshold value of shear stresses required to produce hemolysis.

The mechanism of hemolysis in turbulent flows is not well understood. It is generally accepted that turbulent eddies can directly interact with red cells through energy transfer only if the length scales of both are comparable [75]. An often-cited hemolysis threshold of Reynolds shear stresses is 4000 dynes/cm\(^2\) [76] for about 1 ms. This value, however, was based on one-component laser Doppler anemometry (LDA) measurement. Lu et al.[77] measured the flow under similar flow conditions using a two-component LDA. They found that the threshold principal Reynolds shear stress for hemolysis is 8000 dynes/cm\(^2\).
1.8 Platelet Activation and Thrombosis

The clinical application of VADs is also often complicated by thrombosis and shear-induced thromboembolism. It is estimated that VADs have a thromboembolic rate of up to 50% [78, 79, 80, 81]. The device-induced thrombus formation usually starts in low flow or stagnant flow regions with the rapid adsorption of plasma proteins on the device surface. The adsorption of von Willebrand factor (vWF) leads to the adhesion of platelets (both activated and resting platelets) onto the device surface through glycoprotein (Gp) Ib receptor. This adhesion further triggers platelet activation and aggregation, with the subsequent release of adenosine diphosphate (ADP) and synthesis of thromboxane A2 (TxA2) and thrombin from the activated platelets. These newly-generated chemical agonists serve as stimuli for additional platelet adhesion and aggregation, resulting in a potential positive feedback loop. As a result, thrombosis occurs [60].

Fluid mechanics plays a critical role in platelet activation and aggregation. Pathologically high local shear stresses (≥50 dynes/cm²) can directly activate platelets in the presence of vWF and platelet membrane Gp receptor functional complexes GpIb/IX/V and GpIIb-IIIa via a diacylglycerol (DAG)-independent pathway of protein kinase C (PKC) [82]. The key events include shear-induced vWF binding to GpIb and a subsequent vWF-dependent Ca²⁺ influx, which gives rise to the activation of GpIIb-IIIa and allows it to bind vWF and fibrinogen [82, 83]. This is different from chemical agonist-induced platelet activation, in which the signal pathway is DAG dependent and Ca²⁺ is released from the intracellular stores. The rate and extent of platelet aggregation also show strong dependence on the local shear rate. On the one hand, platelet aggregation can be enhanced by the increased frequency and efficiency of platelet collision, which is proportional to the local shear rate [84, 85, 86], and this shear-induced platelet aggregation may in turn be further enhanced by the release of nucleotides (ADP) due to the disruption of red cells and platelets when exposed to high shear rate for an extended time (see Figure 1.6) [60]. On the other hand, platelet aggregate growth may be limited by the
high local shear rate when the mechanical force is strong enough to break the molecular bridges that connect the platelets together. Recent work has shown that platelets are reversibly aggregated at relatively low shear stress (12 dynes/cm$^2$) via the direct binding of fibrinogen (on the surfaces) to the platelet membrane receptor GpIIb-IIIa, whereas higher shear stress (108 dynes/cm$^2$) results in irreversible aggregation, a process in which the passing platelets are first slowed down with the aid of the GpIb-vWF binding and then stabilized by the interaction between GpIIb-IIIa and fibrinogen [87, 88, 89, 90].

The presence of a thrombus can dramatically alter the local blood flow. In blood vessels, excessive clot formation may significantly increase the local flow shear stress, causing more platelet activation. The high shear stress may also embolize the thrombus, resulting in the occlusion of capillary vessels in the microcirculation.

1.9 The Current Study

1.9.1 Methodology

Here we investigate the hemodynamics of the end-to-side anastomosis of a VAD to the aorta. Both a simplified planar adult aortic model and a more sophisticated pediatric patient-specific aortic model are used for analysis. A second-order accurate Navier-Stokes solver is used to simulate the flow based on a finite volume method (FVM). MILES is used to resolve the turbulence. The flow splitting among the three great vessels on the aortic arch, the brachiocephalic (innominate) artery, the left common carotid artery, and the left subclavian artery, is modeled using a resistance boundary condition. Hemolysis is predicted using a linear advection-reaction transport equation.

1.9.2 Validation

The accuracy of temporal/spatial schemes employed here are examined using a two-dimensional (2D) decaying Taylor-Green vortex. The flow solver is validated by studying
fully developed pulsatile pipe flow and pulsatile flow in a simplified pediatric aorta. The MILES method is validated by using a fully developed turbulent channel flow at Re$_r$ = 395. The non-Newtonian effect is also examined in the simplified pediatric aortic model using PIV measurements.

1.9.3 Significance

This is the first detailed study of local hemodynamics in the aorta under both pulsatile and continuous left ventricular assistance. Hemodynamic parameters such as pulsatility, flow splitting, velocity, Reynolds stress, WSS, OSI, and hemolysis indices are calculated as a function of flow conditions. These findings will help to identify the risk of graft failure for patients with pulsatile and continuous VADs.
Chapter 2

Methodology

This chapter describes the detailed numerical methodologies that have been developed to study turbulent flow in the end-to-side anastomoses of VADs to the aorta. The experimental designs, procedures, and techniques used for validation are provided as well.

2.1 Basic Equations

The governing equation for the motion of an incompressible fluid with density $\rho$ and dynamic viscosity $\mu$, can be expressed with the continuity equation:

$$ \nabla \cdot U = 0, \quad (2.1) $$

and the Navier-Stokes equation:

$$ \rho \frac{\partial U}{\partial t} + \rho \nabla \cdot (UU) = -\nabla p + \mu \nabla^2 U, \quad (2.2) $$

\[ \text{Inertial} \quad \text{Viscous} \]
where \( U \) and \( p \) are the velocity vector and pressure, respectively. The ratio between the inertial term and viscous term is measured by the Reynolds number \( Re \), defined as:

\[
Re = \frac{\rho U_{\text{mean}} L}{\mu},
\]

(2.3)

where \( U_{\text{mean}} \) is the mean velocity magnitude, and \( L \) is the length scale of the flow.

\[ \text{2.2 Turbulence} \]

At low \( Re \), the flow is laminar. Any perturbation of the flow is damped out by the viscous dissipation. Turbulence occurs when \( Re \) reaches a certain limit. The limit depends not only on the characteristics of the base flow, but also on the amplitude and type of perturbations enacted. Turbulent flows are different from laminar flows in two aspects. First, turbulent flows are random processes in time and space. This means that they cannot be experimentally reproduced at any specific time. Second, in comparison to laminar flows, turbulent flows have a wide range of scales in time and space. It is generally believed that there is an energy cascade process inside turbulent flows, especially inside high \( Re \) turbulent flows. Large scales, which contain most of the energy, pass the energy by forming smaller scales. Eventually energy is dissipated at the smallest scales of flow [49].

\[ \text{2.3 Large Eddy Simulation Principles and Modeling} \]

\[ \text{2.3.1 Filtering} \]

LES was first proposed by Smagorinsky in 1963 [91] in a study of a quasi-2D atmospheric turbulence. Based on the theory of Kolmogorov [59], small scale eddies are homogeneous and isotropic regardless of the large scale turbulent motions, which are highly depen-
dent on flow conditions. The fundamental idea of LES is to resolve the flow-dependent large scale motions and to model the smaller scales. By doing so, the total number of computational mesh cells can be significantly reduced when compared to DNS.

The large scales of motions can be separated from the small scales by applying a filter such as:

$$\bar{f}(x, t) = \int_V f(x_1, t)G(x, x_1; \Delta)dx_1,$$  

where $f(x, t)$ is any flow variable, $V$ is the flow domain, and $\Delta$ is the filter width, a measure of the wavelength of the smallest scale retained by the filtering. $G(x, x_1; \Delta)$ is the filter function, which satisfies

$$\int_V G(x, x_1; \Delta)dx_1 = 1.$$  

Gaussian, volume-averaged box, and Fourier cutoff filters are the common filter functions used for LES [92, 93]. The Gaussian filter is defined as:

$$G(x, \Delta) = \sqrt{\frac{6}{\pi \Delta^2}} \exp\left(-\frac{6x^2}{\Delta^2}\right).$$  

The volume-averaged box filter works as a volume averaging in the computational cells, defined as:

$$G(x, \Delta) = \begin{cases} \frac{1}{\Delta} & \text{if } |x| \leq \frac{\Delta}{2} \\ 0 & \text{otherwise.} \end{cases}$$  

The Fourier cutoff filter is defined in Fourier space as:

$$\hat{G}(k, \Delta) = \begin{cases} 1 & \text{if } k \leq \frac{\pi}{\Delta} \\ 0 & \text{otherwise.} \end{cases}$$
Note quantities denoted by a overbar $\bar{\cdot}$ and a caret $\hat{\cdot}$ are the filtered quantities in real space and Fourier space, respectively.

Figure 2.1 shows the shapes of the three filters in both real and Fourier space. The shape of the Gaussian filter is a Gaussian function in both real and Fourier space. The box filter averages flow variables over computational cells. The Fourier cut-off filter is the most accurate filter, as it completely cuts off the small scales below the wavenumber threshold. In contrast, the Gaussian and box filters affect both large and small scales of the flow, as shown in Figures 2.1(b), (d) and (f).
Figure 2.1: Typical filter functions in real and Fourier space: (a) the Gaussian filter in real space; (b) the Gaussian filter in Fourier space; (c) the volume-averaged box filter in real space; (d) the volume-averaged box filter in Fourier space; (e) the Fourier cutoff filter in real space; (f) the Fourier cutoff filter in Fourier space.
2.3.2 Governing Equations

Once applying the filtering (Equation 2.4) to Equations 2.1 and 2.2, one can obtain the continuity and Navier-Stokes equations for the filtered flow field, written as

\[ \nabla \cdot \mathbf{U} = 0, \quad (2.9) \]

and

\[ \rho \left( \frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{U} \mathbf{U}) \right) = -\nabla p + \mu \nabla^2 \mathbf{U}. \quad (2.10) \]

It is clear that the large scales of flow motions are completely decoupled from the small scales in the continuity equation (Equation 2.9), as it is linear. This decoupling, however, does not extend to the Navier-Stokes equations, due to the nonlinear convective term, as shown in Equation 2.10. Hence a SGS stress term is introduced, expressed as:

\[ \tau_{\text{sgs}} = \rho (\mathbf{U} \mathbf{U} - \mathbf{U} \mathbf{U}). \quad (2.11) \]

Substituting Equation 2.11 into Equation 2.10 yields:

\[ \rho \left( \frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{U} \mathbf{U}) \right) = -\nabla p - \nabla \cdot \tau_{\text{sgs}} + \mu \nabla^2 \mathbf{U}. \quad (2.12) \]

Equations 2.9 and 2.12 are used to calculate the resolved flow motions.

By the LES definition, velocity vector \( \mathbf{U} \) can be decomposed as:

\[ \mathbf{U} = \mathbf{U} + \mathbf{U}'. \quad (2.13) \]

where \( \mathbf{U}' \) is the velocity in the SGSs. Substituting Equation 2.13 into Equation 2.11 leads to:

\[ \tau_{\text{sgs}} = \rho ((\mathbf{U} + \mathbf{U}') (\mathbf{U} + \mathbf{U}') - \mathbf{U} \mathbf{U}) = L_{\text{sgs}} + C_{\text{sgs}} + R_{\text{sgs}}, \quad (2.14) \]
where $L_{sgs}$, $C_{sgs}$, and $R_{sgs}$ are the Leonard stresses, the cross stresses, and the Reynolds-like stresses, defined as:

$$L_{sgs} = \rho(\overline{UU} - \overline{U} \overline{U}), \quad (2.15)$$

$$C_{sgs} = \rho(\overline{UU}' + \overline{U}' \overline{U}), \quad (2.16)$$

$$R_{sgs} = \rho\overline{U'U'}. \quad (2.17)$$

$L_{sgs}$, only a function of the resolved velocity, represents the contribution of the resolved scales to sub-grid turbulence. $C_{sgs}$ represents the energy transfer between the resolved scales and SGSs. $R_{sgs}$ represents the interactions between the SGSs [92]. Neither Leonard or cross stresses are Galilean invariant, but their sum is. Hence the SGS stresses are Galilean invariant [94]. Given that the Navier-Stokes equation is Galilean invariant, the SGS stresses are usually modeled as a single entity.

It should be noted that the FVM (used in the present study) automatically applies an implicit box filter in the governing equations based on the computational cells, and therefore, no explicit filtering operation is required.

### 2.3.3 Sub-grid Scale Models

SGS models have to be compatible with filters chosen [95]. Since the FVM used in this study works as an implicit box filter, only SGS models in this framework are presented here. Some of the models which are not used in the present study are only briefly discussed.
2.3.3.1 Eddy-Viscosity-Based Models

Since the SGS eddies are more homogeneous and isotropic than the large ones, most SGS models are eddy-viscosity models with a general form:

$$\tau_{sgs} - \frac{2}{3} k_{sgs} I = -2 \mu_{sgs} \overline{S},$$  \hspace{1cm} (2.18)

where $I$ is the unit tensor, $\mu_{sgs}$ is the SGS eddy viscosity, $k_{sgs}$ is the SGS kinetic energy, defined as:

$$k_{sgs} = \frac{1}{2} \sum_i (\tau_{sgs})_{ii},$$  \hspace{1cm} (2.19)

and $\overline{S}$ is the resolved-scale strain-rate tensor

$$\overline{S} = \frac{1}{2} (\nabla U + \nabla U^\text{tr}),$$  \hspace{1cm} (2.20)

where the superscript $\text{tr}$ represents the transpose of a matrix. If the small scales of motion are in equilibrium (energies received and dissipated are statistically equal), the Smagorinsky model [91] can be derived for the eddy viscosity:

$$\mu_{sgs} = \rho (C_s \triangle)^2 \left| \overline{S} \right|.$$  \hspace{1cm} (2.21)

Here, $\left| \right|$ represents the magnitude of a variable, $\triangle$ is the grid size (the filter width), and $C_s$ is the Smagorinsky constant, typically values of which are $\mathcal{O}(10^{-1})$, determined from a decaying isotropic turbulence [96].

The Smagorinsky model is only valid for equilibrium conditions. It performs poorly in wall bounded flows, where non-equilibrium conditions commonly occur. To address this problem, one-equation eddy-viscosity models were developed. The basic idea is to solve the transport equation of a SGS quantity on which the eddy viscosity depends. One commonly used quantity is the SGS kinetic energy, $k_{sgs}$, which represents the SGS
velocity scales. By subtracting the filtered Navier-Stokes equation (Equation 2.12) from its exact counterpart (Equation 2.2) and then multiplying the results by the velocity in the SGSs ($U'$), one can derive the transport equation for $k_{sgs}$ as [97]:

$$\rho \left[ \frac{\partial k_{sgs}}{\partial t} + \nabla \cdot (k_{sgs} U) \right] = \nabla \cdot \left[ (\mu + \mu_{sgs}) \nabla k_{sgs} \right] - \epsilon - \tau_{sgs} \cdot \overline{S},$$

(2.22)

where the SGS eddy viscosity $\mu_{sgs}$ and the dissipation $\epsilon$ are defined as:

$$\mu_{sgs} = C_k k_{sgs}^{1/2} \Delta,$$

(2.23)

$$\epsilon = C_\epsilon k_{sgs}^{1/2} / \Delta,$$

(2.24)

where $C_k$ and $C_\epsilon$ are two constants. One major advantage of the one-equation eddy viscosity model over the Smagorinsky model is that it accounts for non-equilibrium effects by introducing independent SGS velocity scales for the SGS eddy viscosity with a small increase of cost [93].

### 2.3.3.2 Stress Transport Models

Although the eddy-viscosity-based models have been relatively successful, they have several major weaknesses [93]. First, it is based on the assumption of isotropy in the SGS scales. This does not hold true for low $Re$ turbulent flow or high $Re$ turbulent flow with coarse grids. Second, the correlation (Equation 2.18) between the SGS stress tensor, $\tau_{sgs}$, and the resolved strain rate tensor, $\overline{S}$, is poorly justified. Third, they are unable to produce “backscatter” (the kinetic energy cascaded to the SGS scales returns to the large scales without being dissipated). Various models have been developed to account for these discrepancies. Deardorff [98] and Fureby et al.,[99] developed a transport equation for the SGS stress tensor $\tau_{sgs}$. Fureby et al. found that at low $Re$ flows (fine grids), the stress transport model predicts “backscatter” similar to the linear combination model (explained in Section 2.3.3.3). At high $Re$ flows (coarse grids), it behaves
increasingly like an eddy-viscosity-based model, with only one-way energy transfer. The stress transport model requires solving six additional transport equations (six SGS stress tensor components) instead of one for the one-equation eddy-viscosity model. Hence, it is computationally more expensive.

2.3.3.3 Scale Similarity and Linear Combination Models

Scale similarity and linear combination models are based on the hypothesis that the interaction between the GSs and SGSs of motion takes place mainly between the smallest GSs and the largest SGSs [100]. The largest SGSs of motion can be obtained by applying an additional explicit filter on the SGS velocity. This approach gives better prediction for the energy transfer between the large and small scales than the eddy-viscosity-based models, which are purely dissipative. One drawback is that the scale similarity model hardly dissipates any energy and thus needs to be combined with the eddy-viscosity model (linear combination models).

2.3.3.4 Dynamic Models

Dynamic models, first proposed by Germano et al.[101], are a modification to the basic Smagorinsky model with the Smagorinsky constant $C_s$ as a function of both space and time. In dynamic models, $C_s$ is determined by the energy content of the smallest resolved scales. This is accomplished by applying a second filtering operation. The principal drawback for the dynamic model is the use of averaging in space or over time and additional clipping to avoid excessive fluctuations in the model coefficients.

2.3.3.5 Sub-grid Scale Model Choice

The explicit SGS model used in this study is the one-equation eddy-viscosity model, only for validation purposes. This choice is justified by the comparative study of different SGS models by Fureby et al.[102]. They found that only the SGS stress model predicted
better results than did the one-equation eddy-viscosity model. The SGS stress model, however, tends to significantly increase the computational cost, and therefore, is not included in the present study. Neither the scale similarity model nor the dynamic model is considered, as they predict very similar energy spectra and macroscopic flow features when compared to the one equation eddy viscosity model in simple flows [52].

2.3.4 Wall Treatment

The eddy-viscosity-based model may cause excessive dissipation near the walls. Therefore, wall treatments are needed to reduce the sub-grid eddy viscosity as a function of wall-normal distances. In this study, the mixing length model [103] is used:

\[ \mu_{sgs} = \rho (\kappa y_w)^2 D_a(y_w) |S|, \]  

(2.25)

where \( \kappa \) is the von Karman constant, \( y_w \) is the normal distance from the wall, \( S \) is the local strain rate, and \( D_a(y_w) \) is the “van Driest” damping function [104], defined as:

\[ D_a(y_w) = \left[ 1 - e^{-\left( \frac{\rho u \tau y_w}{\mu} \right)^{k_c}} \right]^m, \]  

(2.26)

where \( A^+, m \) and \( k_c \) are the constants, and \( u_\tau \) is the friction velocity, defined as:

\[ u_\tau = \sqrt{\frac{\tau_w}{\rho}}, \]  

(2.27)

where \( \tau_w \) is the WSS. The Van Driest damping function suppresses turbulence in the laminar sub-region \( (y^+ \lesssim 5) \) near the wall by reducing the length scale. \( y^+ \) is a non-dimensional wall distance, defined as:

\[ y^+ = \frac{\rho u z y_w}{\mu}. \]  

(2.28)
2.4 Monotone Integrated Large Eddy Simulation

Conventional LESs have several drawbacks, such as masking of the SGS terms by the numerical truncation errors, aliasing errors in high-order methods, (e.g., spectral schemes), and difficulty of design of SGS models for complex geometries [105]. This situation has motivated the investigation of unconventional LES approaches, such as implicit SGS modeling. Here in this study, we present the MILES concept proposed by Boris [51].

2.4.1 Rational

MILES is a class of practical methods for dealing with turbulent flows in very complex geometries. It is based on the fundamental hypothesis that the action of SGSs on the resolved scales is strictly equivalent to a dissipation [48]. A wide class of efficient numerical methods, generally based on monotone convective algorithms, have a built-in (implicit) SGS model, which include flux corrected transport (FCT) [106], monotone upstream-centered schemes for conservation laws (MUSCL) [107], piecewise parabolic method (PPM) [108], total variation diminishing (TVD) [109], and the second-order Godunov method [110].

Numerical simulation of the convection induces several types of errors, including numerical diffusion and dissipation, and the Gibbs phenomenon oscillations introduced by the finite resolution [111]. These errors are intrinsically intertwined. The problem is further complicated by the coupling with the continuity equation. Nonlinear, monotone convective algorithms can minimize these errors by their built-in physical properties (global conservation, monotonicity (positivity), causality, and locality [111]). These physical properties are essential for determining the suitability of any convective algorithm for MILES. Conservation means that in Eulerian formulations, the total quantity of a convected quantity in a control volume (CV) is not affected by the convection algorithm, as it literally implies. Conservation can be easily enforced using the FVM. In the FVM,
the volume integrals of the convective term of the Navier-Stokes equations are converted to surface integrals, using Gauss’s theorem. The convective quantities are then evaluated as fluxes at the surfaces of each finite volume. Because the influx of a given finite volume must be equal to the outflux (enforced by continuity equation), the FVM is conservative.

Monotonicity ensures no unphysical oscillation of the convected quantity despite the presence of the numerical dispersion effect and finite-resolution Gibbs effect. Positivity ensures any positively defined convected quantity does not turn negative. Monotone methods tend to have this property built-in.

Causality requires that any traveling fluid particle in the computational domain has to pass through all the computational cells on the flow path. This property is automatically implemented by ensuring the maximum Courant number (the ratio of a time step to a cell residence time) of the flow is less than 1.

Locality means that the derivatives (gradients, curls, and divergences) are calculated locally, which ensures that monotone algorithms adjust the amount of numerical diffusion based on local grids. As a result, the SGS modeling in the MILES is anisotropic in nature.

By enforcing all these properties, nonlinear monotone algorithms induce a local, time-dependent numerical dissipation which acts as SGS models [51, 52, 112, 113, 114, 115].

2.4.2 Numerics

There are a few general approaches for designing stable methods which mimic functional subgrid modeling [48, 111]. In this study, the MILES approach within the framework of flux-limiting FVMs is used and therefore, presented here. Fureby and Grinstein [52, 116, 105] have carried out a theoretical analysis of this approach, to examine the existing relationship between leading numerical error terms and tensorial subgrid viscosities.

Based on Gauss’s theorem, the integration of the convection in the Navier-Stokes
equation over a control cell $\Omega$ can be written as:

$$\int_\Omega \nabla \cdot (\overline{U} \overline{U}) d\Omega = \int_A (\overline{U} \cdot n) \overline{U} dA, \quad (2.29)$$

where $A$ is the surface of the CV $\Omega$, and $n$ is the normal vector on the surface. The surface flux integral can be discretized as:

$$\int_A (\overline{U} \cdot n) \overline{U} dA \approx \sum_f \phi_f (\overline{U})_f, \quad (2.30)$$

where the subscript $f$ denotes the face index, and $\phi$ is the flux as a function of $\overline{U}$. For flux-limiting methods, the numerical flux blends a high-order flux function $\phi^h_f$ which works well in smooth regions and a low-order dispersion-free flux function $\phi^l_f$ which behaves well near sharp gradients:

$$\phi_f = \phi^l_f + \Psi_f (\phi^h_f - \phi^l_f). \quad (2.31)$$

Typically, $\phi^h_f$ is obtained from second (linear) or fourth-order (cubic) accurate central schemes, while $\phi^l_f$ is obtained from a first-order (upwind) accurate scheme. The flux limiter $\Psi_f$ is designed in a way such that the inclusion of the correction term $(\phi^h_f - \phi^l_f)$ does not affect the physical properties (causality, conservation, monotonicity, locality, and positivity) of the monotone schemes.

To see the effects of this convection algorithm, Fureby and Grinstein [116, 105] analyzed the leading error term by applying a three-point backward scheme for time integration, a first-order functional reconstruction for the high-order flux function $\phi^h_f$, and an upwind differencing for the lower-order flux function $\phi^l_f$. As a result, the leading dissipative error term is:

$$-\rho \nabla \cdot (\overline{U} C_d + C_d \overline{U} + C_d C_d), \quad (2.32)$$
with

\[ C_d = \beta_d (\nabla U) D_d, \quad (2.33) \]

\[ \beta_d = \frac{1}{2} \left( 1 - \Psi_j(\bar{U}) \right) \frac{\bar{U} \cdot D_d}{|\bar{U} \cdot D_d|}, \quad (2.34) \]

where \( D_d \) is the topology vector connecting neighboring CVs. By comparing Equation 2.32 to Equation 2.12, one can find that the implicit SGS stress tensor is as follows:

\[
\tau_{sgs}^{MILES} = - \rho (U C_d + C_d \bar{U} + C_d C_d) \\
= - \rho \beta_d \left[ (\bar{U} D_d)^{tr} \bar{U} + \nabla \bar{U} (\bar{U} D_d)^{tr} \right] + \rho \beta_d^2 \left[ (\nabla \bar{U}) D_d \right] \left[ (\nabla \bar{U}) D_d \right] \quad (2.35)
\]

where the first term I represents an eddy-viscosity type of model with a tensorial viscosity \( \rho \beta_d U D_d \) (see Equation 2.18), and the second term II represents a Leonard type of tensor (see Equation 2.15). The combination of the two terms yields a mixed model which is very similar to linear combination models of LES [48] (see Section 2.3.3.3).

In the present study, TVD-based flux limiters are used to simulate turbulent flows. The details of the construction of TVD-based flux limiters are presented in Section 2.7.2.1.

### 2.5 Hemolysis Modeling

#### 2.5.1 Hyperbolic Transport Equation

To evaluate hemolysis levels in various cardiac devices, the empirical Giersiepen-Wurzinger formula (Equations 1.1 and 1.2) has been widely used [67]. It is clear from Equations 1.1 and 1.2 that the exposure time \( t \) is nonlinear. As a result, the net outflux of hemolysis \( H \) in a device is not equal to the sum of hemolysis which occurs inside the device. To overcome this problem, a linear hemolysis variable \( H_l \) is introduced and defined as in Equation 1.4 [71, 72]:

\[ H_l \]
\[ H_t = H^{1/\beta} = \gamma^{1/\beta} \tau_v^{\alpha/\beta} t. \]

Since a Couette viscometer was used to attain this empirical hemolysis formula [117], the shear stresses acting on the material volume can be assumed to be constant. Hence, the time derivative along a streamline of \( H_t \) is constant, and is given by:

\[ \frac{D}{Dt} H_t = \gamma^{1/\beta} \tau_v^{\alpha/\beta} \]

Farinas et al. [71, 72] further extended Equation 2.36 to any material volume along a streamline. Hence, a hyperbolic transport equation can be derived as shown in Equation 1.3:

\[ \frac{\partial H_t}{\partial t} + U \cdot \nabla H_t = R_t, \]

where the reaction coefficient \( R_t \) is:

\[ R_t = \gamma^{1/\beta} \tau_v^{\alpha/\beta}. \]

Equation 1.3 could significantly overestimate the hemolysis, as the reaction (hemolysis) is independent of the red cell concentration (zero-order reaction). As a result, the hemolysis continues even after all red cells in the CV are lysed. To account for the saturation of hemolysis, Equation 1.3 is modified by introducing a first-order reaction source term as [72]

\[ \frac{\partial H_t}{\partial t} + U \cdot \nabla H_t = R_t(1 - H_t). \]

For turbulent flows, the Reynolds stresses could interact with red blood cells if the smallest scales of turbulence are comparable to the size of red cells. The reaction coeffi-
cient of Equation 2.38 for turbulent flows has a form:

\[ R_t = \gamma^{1/\beta}(\tau^{n/\beta}_{vE} + \delta_s \tau^{n/\beta}_{REq}), \]  \hspace{1cm} (2.39)

where \( \tau_{REq} \) is the Reynolds stress equivalent, and the step function \( \delta_s \) is used to account for the hemolytic threshold for the Reynolds stresses [74], defined as:

\[ \delta_s = \begin{cases} 
0 & \text{if } \tau_{REq} < \tau_s \\
1 & \text{otherwise,} 
\end{cases} \]  \hspace{1cm} (2.40)

where \( \tau_s \) is the threshold limit.

### 2.5.2 Stress Equivalent

Both viscous (\( \tau_{vE} \)) and Reynolds stress (\( \tau_{REq} \)) equivalents are scalars, which can be calculated using the principal normal viscous and Reynolds stresses (\( \sigma_1, \sigma_2, \sigma_3 \)), respectively [72]. Three different equivalent forms have been commonly used:

1. von Mises form [118]:

\[ \tau_{vM} = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}. \]  \hspace{1cm} (2.41)

2. Tresca form [119]:

\[ \tau_T = \frac{1}{2} \max \{|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_3 - \sigma_1|\}. \]  \hspace{1cm} (2.42)

3. Normalized von Mises form [72]:

\[ \tau_{vM}^{\text{norm}} = \frac{\tau_{vM}}{\sqrt{3}}. \]  \hspace{1cm} (2.43)
The von Mises form is a popular choice and therefore is used in the present study. To avoid calculating the principal normal stresses, the von Mises stress equivalent can be calculated using the stress tensor components (viscous shear stress or Reynolds stress) $\tau$:

$$\tau_{vM} = \sqrt{\left(\tau_{11} - \tau_{22}\right)^2 + \left(\tau_{22} - \tau_{33}\right)^2 + \left(\tau_{33} - \tau_{11}\right)^2 + 6\left(\tau_{12}^2 + \tau_{23}^2 + \tau_{31}^2\right)}.$$  \hspace{1cm} (2.44)

### 2.5.3 Hemolysis Indices

The normalized index of hemolysis (NIHe) and the modified index of hemolysis (MIH) are two common indices to evaluate hemolysis in cardiac devices. The NIHe is a measure of the increase in the plasma-free hemoglobin, in grams per 100 liters of blood. The MIH is a measure of the increase in plasma-free hemoglobin, normalized by the total quantity of hemoglobin (in both plasma and red cells) in blood. These indices can be calculated as \cite{71, 72}:

$$\text{NIHe}(g/100\text{ l}) = 100H_t(H(\tau, t))_{\text{mean}} = 100H_t(H_t)^{\beta}_{\text{mean}},$$  \hspace{1cm} (2.45)

$$\text{MIH} = 10^6(H(\tau, t))_{\text{mean}} = 10^6(H_t)^{\beta}_{\text{mean}},$$  \hspace{1cm} (2.46)

where $H_t$ is the total blood hemoglobin concentration of the plasma and red cells. For pulsatile flows, if the hemolysis fraction at the inlet is assumed to be zero, the average linear hemolysis fraction $(H_t)_{\text{time-averaged}}$ can be calculated as

$$(H_t)_{\text{time-averaged}} = \frac{\int_0^T \int_{A_o} U \cdot nH_t dA_o}{\int_0^T \int_{A_o} U \cdot n dA_o},$$  \hspace{1cm} (2.47)

where $T$ is the period of flow cycle, and $n$ is the unit normal vector on the outlet surfaces ($A_o$). For steady flows, Equation 2.47 can be simplified to

$$(H_t)_{\text{mean}} = \frac{\int_{A_o} U \cdot nH_t dA_o}{\int_{A_o} U \cdot n dA_o}.$$  \hspace{1cm} (2.48)
2.6 Hemodynamics Parameters

2.6.1 Surplus Hemodynamic Energy

To make meaningful comparisons between the continuous mode (CM) and pulsatile mode (PM) of VADs, surplus hemodynamic energy (SHE, ergs/cm³) is used to characterize pulsatile hemodynamic performance of the flow, and is defined as:

\[
\text{SHE} = 1332(\text{EEP} - \text{TAP}),
\]

where energy equivalent pressure (EEP, mmHg) and time-averaged pressure (TAP, mmHg) are defined as:

\[
\text{EEP} = \frac{\int_0^T pq \, dt}{\int_0^T Q \, dt},
\]

and

\[
\text{TAP} = \int_0^T p \, dt,
\]

where \(Q\) is the flow rate. SHE, which represents the extra energy generated by flow pulsations, has been used extensively in the literature [46, 120, 121].

2.6.2 Wall Shear Stress and Oscillatory Shear Index

Time-averaged WSS and OSI, two important hemodynamic parameters for vascular remodeling [17], are defined as:

\[
(\tau_w)_{\text{time-averaged}} = \frac{1}{T} \int_0^T \tau_w \, dt,
\]

and

\[
\text{OSI} = \frac{1}{2} \left[ 1 - \frac{\left( \frac{1}{T} \int_0^T |\tau_w| \, dt \right)}{\frac{1}{T} \int_0^T |\tau_w| \, dt} \right],
\]
where WSS $\tau_w$ is defined as:

$$
\tau_w = \mu n_w \cdot [\nabla U + (\nabla U)^{tr}],
$$

(2.54)

where $n_w$ is the unit normal vector on the wall. OSI represents the temporal oscillation of the WSS vector.

### 2.6.3 Principal Reynolds Stress

Reynolds stresses have been widely used to assess the turbulence level in cardiac devices. Reynolds stresses vary for different coordinate systems as they are not invariant to coordinate transformations. Hence, the principal Reynolds stresses are used in this study because they represent the maximum Reynolds stresses, and therefore are potentially the most destructive. The three principal normal Reynolds stresses $\sigma_R$ can be calculated as the eigenvalues of the Reynolds stress tensor $\tau_R$ as below:

$$
\begin{vmatrix}
(\sigma_R)_{1} & -(\tau_R)_{12} & -(\tau_R)_{13} \\
-(\tau_R)_{21} & (\sigma_R)_{2} - (\tau_R)_{22} & -(\tau_R)_{23} \\
-(\tau_R)_{31} & -(\tau_R)_{32} & (\sigma_R)_{3} - (\tau_R)_{33}
\end{vmatrix} = 0.
$$

(2.55)

Equation 2.55 can be transformed into a cubic equation as:

$$
(\sigma_R)^3 - I_1(\sigma_R)^2 + I_2(\sigma_R) - I_3 = 0,
$$

(2.56)

where

$$
I_1 = (\tau_R)_{11} + (\tau_R)_{22} + (\tau_R)_{33},
$$

(2.57)

and
\[ I_2 = (\tau_R)_{11}(\tau_R)_{22} + (\tau_R)_{11}(\tau_R)_{33} + (\tau_R)_{22}(\tau_R)_{33} - (\tau_R)_{31}(\tau_R)_{13} - (\tau_R)_{32}(\tau_R)_{23} - (\tau_R)_{21}(\tau_R)_{12}, \]  

(2.58)

and

\[ I_3 = (\tau_R)_{11}(\tau_R)_{22}(\tau_R)_{33} + (\tau_R)_{12}(\tau_R)_{23}(\tau_R)_{31} + (\tau_R)_{13}(\tau_R)_{21}(\tau_R)_{32} - (\tau_R)_{31}(\tau_R)_{22}(\tau_R)_{13} - (\tau_R)_{32}(\tau_R)_{23}(\tau_R)_{11} - (\tau_R)_{33}(\tau_R)_{21}(\tau_R)_{12}. \]

(2.59)

The roots of Equation 2.55 are the principal normal Reynolds stresses \((\sigma_R)_1, (\sigma_R)_2,\) and \((\sigma_R)_3\). The maximum principal Reynolds shear stress can be determined by

\[
[(\tau^p_R)_{ij}]_{\text{max}} = \left[ \frac{1}{2} |(\sigma_R)_i - (\sigma_R)_j| \right]_{\text{max}} (i \neq j),
\]

(2.60)

where \(\tau^p_R\) is the principal Reynolds shear stress.

### 2.7 Finite Volume Discretization

The most widely employed numerical methods for CFD are finite element, finite difference, finite volume, and spectral methods. The FVM offers the best compromise between flexibility and accuracy and therefore is used in this study [93]. This section presents the discretization of the FVM, which has several properties [122]:

1. Because the governing equations are discretized in integral form over each cell CV, the basic quantities, such as mass and momentum, are conserved at the cell level. This is critical in fluid dynamics.

2. The governing equations are solved in a fixed Eulerian system.

3. The cell CVs can be any arbitrary shape. All dependent variables are saved in the centroid of each CV (the colocated or non-staggered variable arrangement) [123, 124].
4. The systems of partial differential equations are solved separately, with the inter-
equation coupling treated explicitly. Non-linear convective terms are linearized
before the discretization.

2.7.1 Domain Discretization

The discretization of solution domain consists of spatial discretization and temporal
discretization. The spatial discretization divides the computational domain into com-
putational cells, or CVs, on which the governing equations are discretized to algebraic
forms. Figure 2.2 shows typical polyhedral CVs, formed by a set of convex faces. The
CVs do not overlap each other and share boundary faces with adjacent CVs. The vector
d, connects the centroids (C and N) of two adjacent CVs, while the vector nf is the unit
normal vector on the common face Af. Note that nf passes through the center of the
face Af. All main dependent variables are defined at the cell centroid (C), and some
derived properties may be defined at the cell surface Af or vertices.
The time-domain parabolic equation [125] can be solved by dividing the time domain into a finite number of time intervals and obtaining the solution at each time step based on an initial condition. The time interval can be constant, or vary depending on solution parameters.

2.7.2 Spatial Discretization

The spatial integral form of the transport equation for a scalar property \( \psi \) is:

\[
\frac{\partial}{\partial t} \int_{\Omega} \psi d\Omega + \int_{\Omega} \nabla \cdot (U\psi) d\Omega - \int_{\Omega} \nabla \cdot (\Gamma_{\psi} \nabla \psi) d\Omega = \int_{\Omega} S_{\psi}(\psi) d\Omega
\]

\[
\text{Temporal derivative} \quad \text{Convection} \quad \text{Diffusion} \quad \text{Source}
\]

(2.61)
where $\Gamma_\psi$ is the diffusivity of $\psi$. This is a second-order partial differential equation, as the diffusion term is a second-order derivative. To obtain good accuracy, the spatial and temporal discretization must be equal to or higher than the order of the equation [122].

For a second-order accurate method, $\psi$ must be linear around the centroid $C$ and time $t$, mathematically expressed as:

$$
\psi(x_C + \Delta x_C) = \psi(x_C) + \Delta x_C \cdot (\nabla \psi)_C,
$$

(2.62)

$$
\psi(t + \Delta t) = \psi(t) + \Delta t \cdot \left( \frac{\partial \psi}{\partial t} \right)_t,
$$

(2.63)

$$
\int_\Omega \psi d\Omega = \psi_C \Omega,
$$

(2.64)

where $\Delta x_C$ is the mesh related length scale around the cell centroid $C$, $\Delta t$ is the time step interval, and $\psi_C$ is the variable $\psi$ at the cell centroid $C$ of the CV, $\Omega$.

Gauss’ theorem is applied throughout to linearize the volume integral of a divergence as follows:

$$
\int_\Omega \nabla \cdot \psi d\Omega = \int_A dA \cdot \psi = \sum_f A_f \cdot \psi_f.
$$

(2.65)

The face value $\psi_f$ of $\psi$ can be obtained by numerical interpolation.

### 2.7.2.1 Convection

The convection term can be discretized as follows:

$$
\int_\Omega \nabla \cdot (U \psi) d\Omega = \int_A dA \cdot (U \psi) = \sum_f (A_f \cdot U_f) \psi_f = \sum_f \phi_f \psi_f.
$$

(2.66)
In the FVM, the mass flux $\phi_f$ satisfies the continuity equation (mass conservation) in every CV, mathematically expressed as

$$\sum_f \phi_f = 0. \quad (2.67)$$

The face value $\psi_f$ of the variable $\psi$ can be obtained from the values in the cell centers by using the convection differencing schemes.

**Upwind Differencing**

Upwind differencing (UD) is a bounded first-order accurate scheme, in which $\psi_f$ is determined by the direction of the flow:

$$\psi_f = \begin{cases} 
\psi_f = \psi_C & \text{for } \phi_f \geq 0 \\
\psi_f = \psi_N & \text{for } \phi_f < 0 
\end{cases} \quad (2.68)$$

The boundedness of UD implicitly induces a numerical diffusion term [122], which could affect the accuracy.

**Central Differencing**

Central differencing (CD) is an unbounded second-order accurate scheme. It is based on the assumption that the variation of $\psi$ between $C$ and $N$ is linear (Figure 2.3), mathematically expressed as:

$$\psi_f = \varphi \psi_C + (1 - \varphi) \psi_N, \quad (2.69)$$

with

$$\varphi = \frac{|FN|}{|CF|}, \quad (2.70)$$
where $|FN|$ is the distance between $F$ and $N$, and $CF$ is the distance between $C$ and $F$. Note $F$ is on the face $A_f$.

Figure 2.3: Face interpolation.

**Total Variation Diminishing Differencing**

Neither UD or CD is bounded and accurate at the same time. One way to tackle this problem is to use TVD schemes [109], which provide at least second-order accuracy in smooth regions of the flow and produce numerical solutions free from spurious oscillations associated with the classical CD scheme. The total variation (TV) of the solution variable $\psi$ at time step $n$ is defined as

$$TV(\psi^n) = \sum_f |\psi^n_C - \psi^n_N|.$$  \hfill (2.71)
where \( C \) and \( N \) are the points around the face \( A_f \). The TVD schemes satisfies [126]:

\[
TV(\psi^n) \leq TV(\psi^{n+1}). \tag{2.72}
\]

For flux-limiting methods, the variable \( \psi_f \) on the surface \( A_f \) is decomposed as the weighted sum of a high-order variable \( \psi_f^h \) and a first-order variable \( \psi_f^l \) as follows:

\[
\psi_f = \psi_f^l + \Psi_f (\psi_f^h - \psi_f^l). \tag{2.73}
\]

If the limiter \( \Psi_f \) is assumed to be a function of the local ratio of consecutive gradients of \( \psi, \xi \), Sweby [126] showed that the flux-limiting schemes satisfy the TVD condition only if:

\[
0 \leq \left( \frac{\Psi(\xi)}{\xi}, \Psi(\xi) \right) \leq 2. \tag{2.74}
\]

It is worth noting that all TVD-based flux-limiting schemes are monotonicity preserving.

### 2.7.2.2 Diffusion

The diffusion term can be linearized as follows:

\[
\int_{\Omega} \nabla \cdot (\Gamma_\psi \nabla \psi) d\Omega = \int_A dA \cdot (\Gamma_\psi \nabla \psi) = \sum_f A_f \cdot (\Gamma_\psi \nabla \psi)_f. \tag{2.75}
\]

If the mesh is orthogonal (the centroid vector \( d_c \) is parallel to the unit normal face vector \( n_f \)), the face gradient discretization is implicit and expressed as follows:

\[
A_f \cdot (\Gamma_\psi \nabla \psi)_f = |A_f| \frac{(\Gamma_\psi \psi)_N - (\Gamma_\psi \psi)_C}{|d_c|}. \tag{2.76}
\]
If the mesh is not orthogonal, an additional explicit term is introduced as follows:

\[
A_f \cdot (\Gamma_\psi \nabla \psi)_f = \left( (A_f)_{dc} \right) \frac{(\Gamma_\psi \psi)_N - (\Gamma_\psi \psi)_C}{|d_c|} + \underbrace{(A_f)_\Delta \cdot \left( \tilde{\nabla}_\psi \right)_f}_{\text{Non-orthogonal}}, \quad (2.77)
\]

and

\[
A_f = (A_f)_{dc} + (A_f)_\Delta, \quad (2.78)
\]

where the vector \((A_f)_{dc}\) is parallel to the centroid vector \(d_c\), and the vector \((A_f)_\Delta\) is the remainder of \(A_f\). The gradient of \(\psi\) on the surface, \(\left( \tilde{\nabla}_\psi \right)_f\), is evaluated by interpolating the cell centered gradients:

\[
\left( \tilde{\nabla}_\psi \right)_f = \varphi (\nabla \psi)_C + (1 - \varphi) (\nabla \psi)_N, \quad (2.79)
\]

where the cell-centered gradient is evaluated as:

\[
(\nabla \phi)_C = \frac{1}{\Omega} \sum_f A_f \cdot \phi_f. \quad (2.80)
\]

### 2.7.2.3 Source Term

The source term accounts for any sources and sinks that either create or destroy the variable \(\psi\). If the reaction is assumed to be first order, the source term \(S_\psi(\psi)\) can be expressed as:

\[
S_\psi(\psi) = k_{\text{zero}} + k_{\text{first}} \psi, \quad (2.81)
\]

where \(k_{\text{zero}}\) and \(k_{\text{first}}\) are the zero-order and first-order reaction coefficients, respectively. According to Equation 2.64, the volume integral of the source term can be discretized implicitly as:

\[
\int_\Omega S_\psi(\psi) \, d\Omega = k_{\text{zero}} \Omega + k_{\text{first}} \psi_C \Omega. \quad (2.82)
\]
2.7.3 Temporal Integration

For transient problems, the spatial integral of the transport equation (Equation 2.61) has to be integrated along time $t$ as:

$$
\int_t^{t+\Delta t} \left[ \frac{\partial}{\partial t} \int_{\Omega} \psi \, d\Omega + \int_{\Omega} \nabla \cdot (U \psi) \, d\Omega - \int_{\Omega} \nabla \cdot (\Gamma \psi \nabla \psi) \, d\Omega \right] \, dt
= \int_t^{t+\Delta t} \left[ \int_{\Omega} S_{\psi}(\psi) \, d\Omega \right] \, dt. \tag{2.83}
$$

Substituting Equations 2.66, 2.75 and 2.82 into Equation 2.83 gives:

$$
\int_t^{t+\Delta t} \left[ \left( \frac{\partial \psi}{\partial t} \right)_C \Omega + \sum_f \phi_f \psi_f \psi_f - \sum_f A_f \cdot (\Gamma \psi \nabla \psi)_f \right] \, dt
= \int_t^{t+\Delta t} \left[ k_{\text{zero}} \Omega + k_{\text{first}} \psi C \Omega \right] \, dt. \tag{2.84}
$$

Equation 2.84 is usually called the “semi-discretized” form of the transport equation [127].

Second-order Backward Differencing

The backward differencing scheme (BD) [122] uses the values of $\psi$ at three different time steps ($n-2$, $n-1$, and $n$),

$$
\psi^{n-2} = \psi^{t-\Delta t}, \tag{2.85}
$$

$$
\psi^{n-1} = \psi^{t}, \tag{2.86}
$$

and

$$
\psi^{n} = \psi^{t+\Delta t} \tag{2.87}
$$

to calculate the temporal derivative. $\psi^{n-2}$ and $\psi^{n-1}$ can be approximated using Taylor expansions around the time $n$ as:

$$
\psi^{n-2} = \psi^{n} - 2 \left( \frac{\partial \psi}{\partial t} \right)^n \Delta t + 2 \left( \frac{\partial^2 \psi}{\partial t^2} \right)^n \Delta t^2 + O(\Delta t^3), \tag{2.88}
$$
\[ \psi^{n-1} = \psi^n - \left( \frac{\partial \psi}{\partial t} \right)^n \Delta t + \frac{1}{2} \left( \frac{\partial^2 \psi}{\partial t^2} \right)^n \Delta t^2 + \mathcal{O} \left( \Delta t^3 \right). \quad (2.89) \]

If Equation 2.89 is multiplied by four and then subtracted from Equation 2.88, one can obtain a second-order accurate approximation of the temporal derivative at the time step \( n \):

\[ \left( \frac{\partial \psi}{\partial t} \right)^n = \frac{3}{2} \psi^n - 2 \psi^{n-1} + \frac{1}{2} \psi^{n-2} \frac{1}{\Delta t}. \quad (2.90) \]

Substituting Equation 2.90 into 2.84 and neglecting the temporal variation in the face values leads to:

\[ \frac{3}{2} \psi^n_C - 2 \psi^{n-1}_C + \frac{1}{2} \psi^{n-2}_C \Omega + \sum_f \phi_f^n \psi_f^n - \sum_f A_f \cdot (\Gamma \nabla \psi)^n_f = k_{\text{zero}} \Omega + k_{\text{first}} \psi^n_C \Omega. \quad (2.91) \]

With proper boundary conditions implemented, Equation 2.91 produces a system of algebraic equations of \( \psi^n_C \) as:

\[ [A] [\psi_C] = [b]. \quad (2.92) \]

where \([A]\) is a sparse matrix, \([\psi_C]\) is a vector of \( \psi \) at the cell centroids for all CVs, and \([b]\) is a constant source vector determined by the previous time steps.

**2.7.4 Linear Solvers**

The set of Equation 2.92 are solved using iterative linear solvers. Two different solvers, preconditioned bi-conjugate gradient (PBICG) solvers and generalized geometric/algebraic multi-grid (GAMG), are used in this study.

**2.7.4.1 PBICG Solver**

The PBICG solver solves a preconditioned linear system of

\[ [M]^{-1} [A] [\psi_C] = [M]^{-1} [b], \quad (2.93) \]
where $[M]$ is a simplified diagonal-based incomplete lower upper (DILU) preconditioner [128]. The PBICG solver is typical of asymmetric Krylov subspace methods, which use matrix-vector multiplications to form the Krylov subspace so that matrix-matrix operations are avoided [129].

### 2.7.4.2 GAMG Solver

The basic principle behind the GAMG solver is to use a series of coarse grid levels to dampen out high frequency errors and to generate an approximated solution for the fine grid [127]. The geometric coarsening of the grid is done by faceAreaPair method [128]. The solver operates a V-shape multigrid cycle with pre- and post-smoothing on each of the coarse grids using Gauss-Seidel method. The matrix of the coarsest grid level is solved directly using incomplete Cholesky preconditioned conjugate gradient (ICCG) and incomplete Cholesky preconditioned biconjugate gradient (BICCG) for symmetric and asymmetric matrices, respectively [130].

### 2.7.5 Navier-Stokes Discretization

The integral form of the Navier-Stokes equation (Equation 2.2) can be expressed as:

$$
\int_{t}^{t+\Delta t} \left[ \frac{\partial}{\partial t} \int_{\Omega} \rho U d\Omega + \int_{\Omega} \rho \nabla \cdot (UU) d\Omega - \int_{\Omega} \nabla \cdot (\mu \nabla U) d\Omega \right] dt \\
= \int_{t}^{t+\Delta t} \left[ -\int_{\Omega} \nabla p d\Omega \right] dt.
$$

(2.94)

This is different from the integral of the general transport equation (Equation 2.83) in two aspects: First, the Navier-Stokes equation is nonlinear, which cannot be treated in the general fashion; Second, the pressure-velocity coupling introduces a new unknown variable pressure, $p$, into the system.
2.7.5.1 Nonlinear Convection Term

The convection term in Equation 2.2 ($\nabla \cdot (UU)$) is nonlinear. As a result, the discretized form of this term leads to a nonlinear system of algebraic equations. Given the complexity of nonlinear equation solvers and the computational effort required, the convection term is first linearized by using the flux of the previous time step $\phi^{n-1}$ to calculate the velocity $U^n$ at time step $n$ as follows [122]:

$$
\int_{\Omega} \nabla \cdot \rho(UU) d\Omega = \int_A \rho dA \cdot (UU)
$$

$$
= \sum_f \rho A_f^n \cdot (U_f U_f)^n
$$

$$
= \sum_f \rho \phi_f^n U_f^n
$$

$$
\approx \sum_f \rho \phi_f^{n-1} U_f^n.
$$

(2.95)

2.7.5.2 Pressure Equation

To derive the pressure equation, a semi-discretized form of the Navier-Stokes equation can be derived by the discretization of Equation 2.94 using the procedures described previously [122]:

$$
a_C U_C = M - \nabla p,
$$

(2.96)

where the term $M$ is defined as:

$$
M = - \sum_N a_N U_N + C_t.
$$

(2.97)

The coefficient $a_C$ includes the contribution from the temporal derivative, convection and diffusion terms of the Navier-Stokes equation corresponding to $U_C$, while the coefficient $a_N$ includes the contributions from the neighbor cells corresponding to $U_N$. The coefficient $C_t$ represents the contribution from the previous time steps due to the temporal integration.
Dividing both sides of Equation 2.96 by $a_C$ gives:

$$U_C = \frac{M}{a_C} - \frac{\nabla p}{a_C}. \quad (2.98)$$

Velocity on the cell face can be obtained by the face interpolation of Equation 2.98 on both sides:

$$U_f = \left( \frac{M}{a_C} \right)_f - \left( \frac{\nabla p}{a_C} \right)_f. \quad (2.99)$$

Based on the continuity equation in the discrete form (Equation 2.67), one obtains:

$$\sum_f A_f \cdot \left( \frac{M}{a_C} \right)_f = \sum_f A_f \cdot \left( \frac{\nabla p}{a_C} \right)_f. \quad (2.100)$$

The flux on the face $A_f$ can be calculated according to Equation 2.99:

$$\phi = A_f \cdot \left[ \left( \frac{M}{a_C} \right)_f - \left( \frac{\nabla p}{a_C} \right)_f \right]. \quad (2.101)$$

With Equation 2.100 satisfied, the face flux is guaranteed conservative. The pressure gradient in Equations 2.96, 2.98, 2.99, 2.100, and 2.101 can be discretized in a similar manner to the diffusion term as previously described.

### 2.7.5.3 Pressure-Velocity Coupling

The pressure implicit with splitting of operators (PISO) method [131] is used to deal with the pressure-velocity coupling in a segregated manner. The detailed procedure is shown in Figure 2.4, with each step described below [93, 122]:

1. Assemble and solve the momentum predictor equation (Equation 2.96) using pressure $p$ and flux $\phi$ of the last time step. The resulting velocity generally does not satisfy the continuity equation. This step is called the *momentum predictor*.

2. Update the coefficients of $M$ (off-diagonal matrix components) using the newly
calculated velocity field.

3. The pressure equation (Equation 2.100) is solved to obtain a new pressure field. This step is called the pressure solution. The pressure equation is solved iteratively due to the explicit nature of the non-orthogonal component of the face interpolation of pressure. In most cases, one or two corrector steps are sufficient to get convergence of the non-orthogonal component.

4. The new conservative flux field is updated using the new pressure field based on Equation 2.101. The velocity field is updated explicitly using Equation 2.98. This is the explicit velocity correction.

5. Steps 2-4 are now repeated iteratively until a predetermined error tolerance is reached.

6. Go to the next time step.

It should be noted that a number of non-orthogonal pressure correctors are repeated in the algorithm without updating the matrix coefficients of $M$. This is based on the assumption that for low Courant numbers, the pressure-velocity coupling is much stronger than the nonlinear coupling [122].
Figure 2.4: PISO procedure.
2.7.6 Large Eddy Simulation Model Implementation

For LES models, the SGS stress has to be included in the Navier-Stokes equation, as shown in Equation 2.12. When the eddy-viscosity-based SGS models (Equation 2.18) are used, the \(-2\mu_{sgs}S\) term is combined with the viscous shear stress, while the \(\frac{2}{3}k_{sgs}I\) term is sub-summed into the pressure gradient for numerical reasons, similar to that employed in RANS modeling. As a result, the new diffusion term of the filtered Navier-Stokes equation in integral form becomes:

$$\int_\Omega \nabla \cdot Bd\Omega = \int_\Omega (\nabla U + \nabla U^{tr}) d\Omega,$$

where \(B\) is the combined stress, and the effective dynamic viscosity is defined as:

$$\mu_{eff} = \mu + \mu_{sgs}. \quad \text{(2.103)}$$

In this study, the gradient and transposed gradient terms are discretized separately, with the gradient treated as a diffusion and the transposed gradient as a source. If both terms are treated as a diffusion, it will form a linked system of linear equations with the value of one velocity component dependent on the other two, which requires significantly more computational resources to solve [93]. As a result, the discretized form of Equation 2.102 is:

$$\int_\Omega \nabla \cdot Bd\Omega = \sum_f (\mu_{eff})_f A_f \cdot (\nabla U)_f + \nabla \cdot \left[ \mu_{eff} \left( \nabla U^{tr} \right)^{n-1} \right]_C \Omega,$$

where the transposed gradient \(\nabla U^{tr}\) is calculated using the velocity from the last time step \(n - 1\). The cell face and cell-centered velocity gradients are approximated using Equations 2.77 and 2.80, respectively.
One-equation Eddy Viscosity Model

The integral form of the one-equation eddy-viscosity model (Equation 2.22) can be expressed as:

\[ \int_{t}^{t+\Delta t} \left[ \frac{\partial}{\partial t} \int_{\Omega} \rho k_{sgs} d\Omega \ + \ \int_{\Omega} \rho \nabla \cdot (k_{sgs} \vec{U}) d\Omega \ - \ \int_{\Omega} \nabla \cdot ((\mu + \mu_{sgs}) \nabla k_{sgs}) d\Omega \right] dt \]

\[ = \int_{t}^{t+\Delta t} \left[ \int_{\Omega} (-\epsilon - \tau_{sgs} \cdot \vec{S}) d\Omega \right] dt. \]

(2.105)

In Equation 2.105, each term can be discretized in a manner similar to the discretization schemes described previously for the general transport Equation 2.61. One thing worth noting is that the SGS kinetic energy \( k_{sgs} \) is always positive. Since the CD scheme is unbounded and could lead to negative \( k_{sgs} \), it is not suitable for the face interpolation of \( k_{sgs} \). In this study, a TVD scheme is used instead.

2.7.7 Discretization Errors

The discretization errors come from two sources. One is the numerical diffusion as a result of the discretization schemes of temporal integration and convection differencing, which are not fully second-order accurate in order to preserve the boundedness of the solution. Higher-order errors are not discussed. Another is a result of the discretization of the solution domain, which depends on the computational mesh quality [122].

54
2.7.7.1 Numerical Diffusion from Convection Differencing Schemes

If we assume that TVD is a blender of the first-order UD and the second-order CD schemes and the face flux, \( \phi_f \geq 0 \), the TVD-based face interpolation of the transport variable \( \psi_f \) can be expressed based on Equations 2.68, 2.69 and 2.72:

\[
(\psi_f)_{\text{TVD}} = \psi_C + \Psi_f [(\varphi \psi_C + (1 - \varphi) \psi_N) - \psi_C].
\]  

(2.106)

If Equation 2.69 is subtracted from 2.106, one obtains the difference \( \delta \) of the face value \( \psi_f \) between the CD and TVD schemes:

\[
\delta (\psi_f) = (\psi_f)_{\text{CD}} - (\psi_f)_{\text{TVD}}
\]

\[
= [\varphi \psi_C + (1 - \varphi) \psi_N]
\]

\[
- [\psi_C + \Psi_f ((\varphi \psi_C + (1 - \varphi) \psi_N) - \psi_C)].
\]  

(2.107)

Equation 2.107 can be rearranged as the following form:

\[
\delta (\psi_f) = [\Psi_f (1 - \varphi) - (1 - \varphi)] \psi_C + [(1 - \varphi) - \Psi_f (1 - \varphi)] \psi_N
\]

\[
= (1 - \varphi) (1 - \Psi_f) (\psi_N - \psi_C).
\]  

(2.108)

The difference for the whole convection term can be expressed as [122]:

\[
E_{\text{TVD}} = \sum_f \rho A_f \cdot U_f \delta (\psi_f) = \sum_f A_f \cdot U_f [(1 - \varphi) (1 - \Psi_f) (\psi_N - \psi_C)]
\]

\[
= \sum_f A_f \cdot U_f \left[(1 - \varphi) (1 - \Psi_f) d_c \cdot (\nabla \psi)_f \right]
\]

\[
= \sum_f A_f \cdot [(1 - \varphi) (1 - \Psi_f) U_f d_c] \cdot (\nabla \psi)_f
\]

\[
= \sum_f A_f \cdot (\Gamma_{\text{num}} \cdot \nabla \psi)_f,
\]  

(2.109)

where \( \Gamma_{\text{num}} \), the numerical diffusivity tensor, is defined as:

\[
\Gamma_{\text{num}} = (1 - \varphi) (1 - \Psi_f) U_f d_c.
\]  

(2.110)
It is clear from Equation 2.109 that TVD schemes indeed introduce a diffusion-like numerical error, depending on both velocity field and the gradient of $\psi$. If the first-order UD scheme is used for convection differencing, $\Psi_f$ is equal to zero. As a result, the diffusivity tensor, $\Gamma_{\text{num}}$, becomes:

$$\Gamma_{\text{num}} = (1 - \varphi) U_f d_c.$$ (2.111)

2.7.7.2 Numerical Diffusion from Temporal Integration

The numerical error of the BD scheme for temporal integration, Equation 2.91,

$$\frac{3}{2} \dot{\psi}^n_C - 2\psi^{n-1}_C + \frac{1}{2} \psi^{n-2}_C \Omega + \sum_f \phi^n_f \psi^n_f - \sum_f A_f \cdot (\Gamma_\psi \nabla \psi)^n_f = k_{\text{zero}} \Omega + k_{\text{first}} \psi^n_C \Omega,$$

is obtained by comparing to the fully second-order accurate Crank-Nicholson discretization of Equation 2.84 [122],

$$\frac{\psi^n_C - \psi^{n-1}_C}{\Delta t} \Omega + \frac{1}{2} \sum_f \phi^n_f \psi^n_f + \frac{1}{2} \sum_f \phi^{n-1}_f \psi^{n-1}_f - \frac{1}{2} \sum_f A_f \cdot (\Gamma_\psi \nabla \psi)^n_f - \frac{1}{2} \sum_f A_f \cdot (\Gamma_\psi \nabla \psi)^{n-1}_f = k_{\text{zero}} \Omega + \frac{1}{2} k_{\text{first}} \psi^n_C \Omega + \frac{1}{2} k_{\text{first}} \psi^{n-1}_C \Omega.$$ (2.112)

The resulting difference between the two schemes can be expressed as:

$$E_{BD} = \left[ \frac{1}{2} \psi^n_C - \psi^{n-1}_C + \frac{1}{2} \psi^{n-2}_C \Omega \right] - \frac{1}{2} \left[ \sum_f \phi^n_f \psi^n_f - \sum_f \phi^{n-1}_f \psi^{n-1}_f \right]$$

Temporal

$$+ \frac{1}{2} \left[ \sum_f A_f \cdot (\Gamma_\psi \nabla \psi)^n_f - \sum_f A_f \cdot (\Gamma_\psi \nabla \psi)^{n-1}_f \right]$$

Convection

$$Diffusion$$

$$= \left[ k_{\text{first}} \psi^n_C \Omega - k_{\text{first}} \psi^{n-1}_C \Omega \right].$$ Source (2.113)
It is clear that the numerical error comes from all four terms. Jasak [122] showed that the leading term of the truncation error of the BD scheme is four times larger than for the Crank-Nicholson scheme.

2.7.7.3 Mesh Induced Errors

Mesh induced errors are mainly from three sources: insufficient mesh resolution, non-orthogonality and skewness [93, 122]. For a second-order spatial discretization, the mesh resolution error is mainly due to the assumption of the linear variation of the solution in the control cells. Insufficient mesh resolution could significantly affect the accuracy of the numerical solution, especially in the regions where the flow changes rapidly.

Mesh non-orthogonality, if excessive, could lead to unboundedness, causing the failure of the solution. The diffusion term of the general transport equation, for example, exhibits bounded behavior. Its discrete form (Equation 2.75) is bounded only on orthogonal meshes. For meshes with a large number of non-orthogonal cells, the non-orthogonal correction (Equation 2.77) could create unboundedness. As a result, numerical simulation could become unstable and unphysical oscillations could appear. One way to control this is to set a limiter on the non-orthogonal correction such that its contribution does not exceed a specified fraction of the orthogonal contribution. This, however, could violate the order of accuracy of the discretization.

Skewness causes numerical diffusion [122]. Skewness error occurs when the vector connecting the two adjacent cell centroids, \( d_C \), does not pass through the center of the shared face \( A_f \), as shown in Figure 2.2. Hence, the face center value can not be obtained by the linear interpolation of the adjacent cell centroid values.
2.8 Particle Image Velocimetry Principles

2.8.1 Basic Setup

PIV is a non-invasive technique for acquiring planar velocity field measurements in a fluid. A typical PIV system consists of several sub-systems, as shown in Figure 2.5. The flow is seeded with small tracer particles. These particles are illuminated in a plane (light sheet) of the flow by a laser at least twice within a very short time delay, which is dependent on the mean velocity field and the magnification at imaging. The light scattered by the particles is recorded on two separate frames on a special cross correlation charge-coupled device (CCD) sensor. The PIV recordings are then digitized by a scanner and the output is saved in real time on a computer. The velocity field in the plane is determined by the particle displacements during the time delay. If the particle displacements with the delay time are assumed to be linear, the flow velocity can be calculated as:

\[
    u = \frac{\Delta x_p}{(\Delta t)_{pl}},
\]

and

\[
    v = \frac{\Delta y_p}{(\Delta t)_{pl}},
\]

where \( u \) and \( v \) are the velocities in the \( x \) and \( y \) directions, and \( \Delta x_p \) and \( \Delta y_p \) are the particle displacement in the \( x \) and \( y \) directions, and \((\Delta t)_{pl}\) is the time pulse delay.
2.8.2 Classification

Depending on image density, PIV recordings can be classified as three types [132], as shown in Figure 2.6. With sufficient low image density, the motion of individual particles can be tracked, as shown in Figure 2.6(a). This is called particle tracking velocimetry, or PTV. For medium image density (Figure 2.6(b)), although the image of individual particles can be detected, it is no longer possible to track the motions of individual particles. As a result, a statistical approach is required to track the displacement of a group of particles within an indicated small volume, within the light sheet, often referred to as interrogation volume, as shown in Figure 2.7. Since statistical PIV is the most popular among the three, it is usually called standard PIV. In the case of high image density, it is not possible to detect individual particles as their images overlap to form speckles. This is called laser speckle velocimetry, or LSV (Figure 2.6(c)). In the present study, standard PIV is used.
2.8.3 Tracer Particles

Particle image velocimetry measurement is based on the assumption that the particles follow the local flow faithfully and preserve the original flow dynamics. As a result, the particle density has to be sufficiently low, so that the particle-particle interaction can be neglected. The level of the particle-fluid coupling can be estimated by the Stokes number, the ratio of the particle relaxation time $\lambda$ to the bulk flow local time scale $t_b$ (the laser pulse delay $(\Delta t)_{pl}$) [133]:

$$St = \frac{\lambda}{t_b}$$

(2.116)

with the particle relaxation time $\lambda$ given by:

$$\lambda = \frac{m_p}{6\pi r_p \mu},$$

(2.117)

where $m_p$ and $r_p$ are the mass and equivalent radius of the particle, respectively. For $St \leq 1$, the particle will follow the bulk flow closely, while for $St > 1$, the particle will move independently of the bulk flow. To keep the Stokes number $St$ low, one can either decrease the particle size or increase the laser pulse delay $(\Delta t)_{pl}$. The particle size, however, cannot be too small, because small particles scatter little light and are hard to
visualize. Similarly, \((\Delta t)_{pl}\) cannot be too long. The trajectory of the particle motions during long \((\Delta t)_{pl}\) could become nonlinear. As a result, Equations 2.114 and 2.115 are no longer valid. In some cases, the particles could move out of the interrogation volume during long \((\Delta t)_{pl}\), which could significantly affect the accuracy of the results.

2.8.4 Cross-Correlation

Typically, PIV recordings are subdivided into interrogation regions during evaluation, as shown in Figure 2.7. Their geometrical back-projection into the light sheet is often referred to as the interrogation volume. PIV recordings are most often evaluated by locally cross-correlating two frames of single exposures of the tracer ensemble. Cross-correlation, which is based on the assumption that all particles in an interrogation area move a similar distance and direction during \((\Delta t)_{pl}\), is pattern recognition of the particle displacement within the interrogation area.

![Figure 2.7: Schematic representation of geometric imaging [132].](image)

The cross-correlation function for two interrogation areas of image frames A and B
can be written as [132]:

\[ R_{II}(s, \Theta, \triangle D) = \langle I_A(X, \Theta)I_B(X + s, \Theta, \triangle D) \rangle, \]  \hspace{1cm} (2.118)

where \( R_{II} \) is the cross-correlation function, \( s \) is the shifting vector in the correlation plane, \( \Theta \) is the distribution of particle locations in the interrogation volume, \( \triangle D \) is the displacement vector for each particle, \( X \) is the coordinates in the image plane, \( I_A \) is the intensity distribution of image frame A, \( I_B \) is the intensity distribution of image frame B, and \( \langle \rangle \) is the spatial average. For a given distribution of particles inside the flow, the peak correlation value of \( R_{II} \) occurs when \( s = \langle \triangle D \rangle \) [132]. This suggests that the location of the peak value represents the average in-plane displacement. As a result, the velocity vectors can be calculated using Equations 2.114 and 2.115 based on the cross correlation peak shift. Figure 2.8 shows an example of an interrogation area cross correlation function of frames A and B.

Figure 2.8: Cross correlation of interrogation areas of frames A and B: (a) the interrogation area of frame A, (b) the interrogation area of frame B, (c) the cross correlation function of the interrogation areas.
2.8.5 Error Analysis

The overall error of PIV relies on various factors from the image recording to the post-processing. The absolute measurement error in the estimation of a single displacement vector, $\delta_{tot}$, can be decomposed into a bias error, $\delta_{bias}$, and a measurement uncertainty error, $\delta_{rms}$:

$$\delta_{tot} = \delta_{bias} + \delta_{rms},$$

(2.119)

where $\delta_{rms}$ is a measure of data repeatability, and $\delta_{bias}$ is associated with the methods for velocity evaluation. Here various contributing factors for $\delta_{bias}$ are briefly discussed [132].

2.8.5.1 Particle Size

The optimum particle image diameter for digital PIV evaluation using the cross-correlation is slightly over two pixels. Images with larger particle diameter can lead to wide cross-correlation peaks, and therefore, reduce the accuracy of the peak finding algorithm. In addition, larger interrogation areas are required for larger particles to contain an appropriate number of particles, causing a decrease in spatial resolution [134]. In contrast, images with particle diameter less than two pixels could introduce a bias error called peak locking. As a consequence, the displacement peaks tend to be biased toward integer values [135]. This error can be overcome by defocusing the particle image, increasing the sampling rate, or preconditioning the images using filters to optimize the particle image diameters with respect to the peak estimator [132].

2.8.5.2 Particle Density

The effective particle image pair density within an interrogation area can be determined by [136]:

$$(N_p)_{eff} = (N_p)_A F_l F_o,$$

(2.120)
where \((N_p)_{\text{eff}}\) is the effective particle image pair density, \((N_p)_A\) is the total number of particles in frame A, \(F_i\) is the factor of in-plane particle loss, and \(F_o\) is the factor of out-of-plane particle loss. The particle density affects the accuracy of the PIV measurement in two ways. First, the probability of a valid displacement increases with increasing \((N_p)_{\text{eff}}\) in an interrogation area. For single exposure/double frames PIV, \((N_p)_{\text{eff}}\) has to be at least greater than five to achieve a 95% valid detection probability. Second, high \((N_p)_{\text{eff}}\) can reduce the measurement uncertainty substantially [132].

2.8.5.3 Background Noise

Minor background noise has little effect on the cross correlation and measurement uncertainty [132, 134]. Minor background noise could slightly decrease the peak to base ratio of the cross-correlation function and widen the peak.

2.8.5.4 Displacement Gradients

High displacement gradients across an interrogation area tend to underestimate the mean particle displacement since the particles with slower motions are likely to stay longer in the interrogation volume than those with higher motions. This bias error can be minimized by reducing the size of the interrogation volume or the laser pulse delay.

2.8.5.5 Out-of-plane Motion

In the case of highly 3D flows, out-of-plane particle loss \(F_o\) (Equation 2.120) could be significantly high such that the correlation peak signal strength diminishes. This out-of-plane motion can be compensated for by three methods [132]. First, the out-of-plane motion can be reduced by reducing the laser pulse delay. Second, the out-of-plane motion can be accommodated by thickening the light sheet. This method, however, could reduce the energy density in the light sheet. Third, the mean out-of-plane flow component can be accommodated with an increase in the distance between the two light sheets. In
general, the three methods need to be combined in order to achieve the best results.

2.9 Principle of Fluid Viscoelasticity Measurement

2.9.1 Governing Equation

Measurements of the viscoelasticity are based on the principle of oscillatory flow within a cylindrical tube. A general form of the Navier-Stokes equation for fully-developed sinusoidal flow of an incompressible viscoelastic fluid in a straight pipe is [137]:

\[ \rho \frac{\partial u^\dagger}{\partial t} = -\frac{\partial p^\dagger}{\partial x} + \eta^\dagger \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u^\dagger}{\partial r} \right) \right], \tag{2.121} \]

where the superscript \( \dagger \) represents the complex form of a variable, \( x \) is the coordinate in the axial direction, \( r \) is the coordinate in the radial direction, \( u^\dagger \) is the axial component of complex velocity vector \( U^\dagger \), and \( \eta^\dagger \) is the complex viscoelasticity, defined as:

\[ \eta^\dagger = \eta' - i\eta'' = \eta e^{-i\theta}, \tag{2.122} \]

where the viscosity, \( \eta' \), and the elasticity, \( \eta'' \), represent the loss and storage moduli of energy, and \( \theta \) is the viscosity angle, which is related to the phase shift between energy dissipation and recovery. The complex pressure, \( p^\dagger \), is independent of \( r \) and has a form of

\[ -\frac{\partial p^\dagger}{\partial x} = \Upsilon e^{i\omega t} \tag{2.123} \]

for sinusoidal flows, where \( \Upsilon \) is the amplitude, and \( \omega \) is the radian frequency of flow oscillation. If a no slip boundary condition is applied to the wall, the complex velocity can be derived as

\[ u^\dagger = \left[ \frac{i\Upsilon}{\rho \omega} \right] \left[ \frac{J_0^\dagger(Kr)}{J_0^\dagger(KR)} - 1 \right] e^{i\omega t}, \tag{2.124} \]
where \( R \) is the radius of the pipe, \( J^0_0 \) is the complex zero-order Bessel function of the first kind, and \( K \) is defined as

\[
K^2 = -\frac{i\omega \rho}{\eta^i}.
\] (2.125)

### 2.9.2 Impedance

The impedance for oscillatory flows is given by

\[
Z^\dagger = \frac{\Delta p^\dagger}{Q^\dagger},
\] (2.126)

where \( \Delta p^\dagger \) and \( Q^\dagger \) are the complex pressure drop along the pipe and the complex volumetric flow rate, respectively. \( \Delta p^\dagger \) is equal to

\[
\Delta p^\dagger = \Upsilon e^{i\omega t} L,
\] (2.127)

where \( L \) is the length of the pipe. \( Q^\dagger \) can be calculated by integrating Equation 2.124 along the cross section of the pipe as:

\[
Q^\dagger = 2\pi \int_0^R ru^\dagger dr
\]

\[
= \left[ \frac{\Upsilon \pi R^2}{i\omega \rho} \right] \left[ 1 - \frac{2}{KR J^1_0(KR)} \right] e^{i\omega t}.
\] (2.128)

where \( J^1_1 \) is the complex first-order Bessel function of the first kind. Substituting Equations 2.127 and 2.128 into Equation 2.126 gives:

\[
Z^\dagger = \left[ \frac{i\rho \omega L}{\pi R^2} \right] \left[ 1 - \frac{2}{KR J^1_0(KR)} \right]^{-1}.
\] (2.129)
For small pipe flows at low frequencies, the velocity profile can be assumed to be parabolic, in which case Equation 2.129 can be simplified as [138]:

\[
Z^\dagger = \frac{8\eta' L}{\pi R^4} + i \left( \frac{4\rho \omega L}{3\pi R^2} - \frac{8\eta'' L}{\pi R^4} \right) .
\] (2.130)

The magnitude of the resulting complex shear rate at the pipe wall can be expressed as:

\[
|\varsigma^\dagger| = \left|\frac{4Q^\dagger}{\pi R^3} \left[ 1 + i\frac{\rho \omega R^2}{24\eta^\dagger} \right] \right| .
\] (2.131)

The complex WSS can be calculated as

\[
(\tau_w)^\dagger = \eta^\dagger |\varsigma^\dagger| = |\varsigma^\dagger| \eta' - i |\varsigma^\dagger| \eta'' .
\] (2.132)

By measuring \(Q^\dagger\) and \(\Delta p^\dagger\), the viscosity and elasticity can be calculated using Equation 2.130. The shear rate and WSS can be calculated using Equations 2.131 and 2.132, respectively.

### 2.9.3 Maxwell Relaxation Time

For sinusoidal oscillatory flow, viscous energy dissipation is in a different phase from elastic energy storage. If the Maxwell model is used, the viscosity and elasticity can be expressed as [138]:

\[
\eta' = \frac{\eta_{\text{dash}}}{1 + (\omega \lambda)^2} ,
\] (2.133)

and

\[
\eta'' = \frac{\eta_{\text{dash}} \omega \lambda}{1 + (\omega \lambda)^2} .
\] (2.134)

The relaxation time \(\lambda\) is defined as

\[
\lambda = \frac{\eta_{\text{dash}}}{k_{\text{spring}}} = \frac{\eta''}{\omega \eta'} .
\] (2.135)
where $\eta_{\text{dash}}$ and $k_{\text{spring}}$ are the dash constant (energy dissipation) and spring constant (energy storage) of the Maxwell model, respectively. It is clear from Equations 2.133 and 2.134 that the Maxwell relaxation time can be easily obtained from the viscosity $\eta'$ and elasticity $\eta''$ measurements. The relaxation time, a measure of the rate at which blood microstructures change in response to the change in flow, is a key parameter to blood viscoelastic properties. It plays a direct role in the balance of energy storage versus energy dissipation [138]. The relative importance of the relaxation time can be characterized by the Deborah number ($De$), which is the ratio of the relaxation time ($\lambda$) to the bulk flow local time scale ($t_b$), expressed as [139]:

$$De = \frac{\lambda}{t_b}.$$  \hspace{1cm} (2.136)

For $De \ll 1$, viscosity dominates; for $De \approx 1$, the material is viscoelastic; for $De \gg 1$, elasticity dominates. Elastic yield stress is the maximum attainable energy stored per unit volume per unit strain, which is related to the point which blood loses its ability for energy storage [140].

It is worth noting that the relaxation time $\lambda$ of the Maxwell model varies with shear rate, as shown in Equation 2.135. This suggests that the Maxwell model is inadequate for modeling of blood.

### 2.10 Experimental Setup

An *in vitro* pediatric mock circulatory loop is designed and built to measure the flow characteristics in a simplified acrylic pediatric aortic model using PIV as a part of the flow solver validation. Here the details of the experimental setup are presented.
2.10.1 Pediatric Mock Circulatory Loop

The pediatric mock circulation (consisting of a heart simulator, pediatric aorta, great vessels on the aortic arch, compliance, resistance, and reservoir) is used to simulate the flow in the pediatric aorta. The loop is based on the adult loop first designed by Rosenberg [141], with the flow volume reduced and the overall resistance increased to mimic the pediatric circulation. Snapshots and schematic of the experimental setup are shown in Figures 2.9 and 2.10, respectively.

Figure 2.9: Pictures of the in vitro mock circulatory loop composed of (1) heart simulator, (2) flow probe, (3) acrylic pediatric aortic model, (4) compliance chambers, (5) pressure transducer, (6) clamp, (7) resistance plate, (8) reservoir, and (9) air line. (a) The front view, (b) the back view, and (c) the enlarged view of the connection between heart simulator and pediatric aortic model.
Figure 2.10: Schematic of the in vitro mock circulatory loop. Note that the red arrows represent the flow direction.

### 2.10.1.1 Heart Simulator

An acrylic model of the 12cc Penn State PVAD is used in this study to simulate the heart, with the flow driven by a pneumatic pump, as shown in Figure 2.10. The acrylic is made from two separate pieces for easy cleaning and replacement of the diaphragm. Pediatric-sized CarboMedics bileaflet valves are used for both the PVAD inlet and outlet, with a diameter of 16 mm and an effective orifice area of 1.1 cm$^2$. Both the inlet and outlet CarboMedics valves are oriented such that the B-datum lines are perpendicular to the diaphragm. The pediatric aortic model is connected to the PVAD outlet.
2.10.1.2 Pediatric Aortic Model

An acrylic model is used to mimic the natural pediatric aorta. The pediatric aortic model, as shown in Figure 2.11 (a), is a simplified version of a 42-month female pediatric aorta (taken from Cleveland Clinic). The model is scaled down to represent a healthy pediatric aorta of a one-year old. The great vessels on the aortic arch, the brachiocephalic artery, the left common carotid artery, and the left subclavian artery, are included in the model to account for the effect of flow splitting on the aortic flow. Excessive curvatures on the aortic arch and the great vessels are removed for the convenience of PIV measurement. The diameter of the aortic model is 11.6 cm, and all the major dimensions of the model are in good agreement with morphological measurements by Machii and Becker [142]. For ease in machining, the acrylic, shown in Figure 2.11 (b), is fabricated in six separate pieces and then glued together, as shown in Figure 2.11 (c).

Figure 2.11: The pediatric aorta: (a) the pediatric aortic model, (b) the acrylic model, (c) the split-up of the acrylic model.
2.10.1.3 Resistance

The overall resistance in the loop is dependent on the size and length of the tubings, the resistance plate and the clamp, as shown in Figure 2.10. The tubing used in this study is flexible Tygon® laboratory tubing R-3603 (Saint-Gobain Performance Plastics, Aurora, OH) of varying diameters. The resistance plate, which consists of two 10" × 6" plates, is used to represent the systematic resistance. By placing a section of the tubing between the two plates, the resistance can be adjusted by either tightening or loosening the plates. The flow ratio between the aortic outlet and the great vessels on the aortic arch is adjusted using the clamp, as shown in Figure 2.10.

2.10.1.4 Compliance

The compliance of the system is controlled by the two compliance chambers, with one on the arterial side and another on the venous side, as shown in Figure 2.10. Each compliance chamber has a piston resisted by a flexible spring bar. As a result, the flow volumes of the compliance chambers vary along the pressure pulse, providing the compliance for the arterial and venous sides.

2.10.1.5 Fluid

Both Newtonian and 40% hematocrit (HCT) non-Newtonian blood analogs are used for flow visualization. The analogs are made of Xanthan gum, glycerin, sodium iodide, and water [65]. The glycerin provides the increased viscosity when compared to water, while the Xanthan gum provides the elasticity and shear-thinning properties for the 40% HCT analog. The sodium iodide is added to match the refractive index (RI) of the analogs to that (1.49) of the acrylic model. The weight ratio of each chemical for the two blood analogs are summarized in Table 2.1:
Table 2.1: Chemical compositions of the blood analogs.

<table>
<thead>
<tr>
<th></th>
<th>Water</th>
<th>Glycerin</th>
<th>Sodium Iodide</th>
<th>Xanthan Gum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Newtonian</td>
<td>30%</td>
<td>20%</td>
<td>50%</td>
<td>0%</td>
</tr>
<tr>
<td>40% HCT</td>
<td>34.47%</td>
<td>15.5%</td>
<td>50%</td>
<td>0.03%</td>
</tr>
</tbody>
</table>

A Vilastic-3 Viscoelastic Analyzer (Vilastic Scientific Inc., Austin, Texas, USA) is used to measure the viscoelasticity of the blood analogs. The Vilastic-3 measures oscillatory pressure across and flow through a vertically-oriented tube. A frequency of 2 Hz is used, as it is near to the pulse rate. All measurements are performed at 22 °C. The kinematic viscosity and the RI of the Newtonian blood analog are $3.77 \times 10^{-6}$ m²/s ($3.77 \times 10^{-2}$ Stokes) and 1.483, respectively, while the kinematic viscosity and elasticity of the 40% HCT blood analog show good agreement with those of the 40% HCT pediatric blood, as shown in Figure 2.12. The RI of the 40% HCT blood analog is 1.482. Both fluids are seeded with glass spheres with a diameter of 10 microns and a density of 2.5 g/cm³ (Potters Industries Inc., Valley Forge, PA) that act as tracers for the PIV measurements. The resulting Stokes number based on Equation 2.116 is much less than 1. Hence, the particles will follow the flow faithfully.
Figure 2.12: Viscosity and elasticity of the pediatric blood and blood analog at 40% HCT.

2.10.2 Flow Measurement

Flow rates at the aortic inlet (the PVAD outlet), the aortic outlet, and the great vessels on the aortic arch are measured using an ultrasonic flow probe system (Transonic System Inc., Millis, MA). The aortic pressure is measured at the compliance chamber on the arterial side using Maxxim Medical pressure transducers (Maxxim Medical, Athens, TX). All the pressure and flow rate signals are converted and displayed on a computer using a WaveBook acquisition system and WaveView software (IOtech, Inc., Cleveland, OH).

A 2D Gemini PIV 15 system (New Wave Research, Inc., Fremont, CA) is used to measure the flow. Dual Nd:YAG lasers are used to produce two overlapping 6 mm diameter beams as the light source to illuminate particles seeded in the blood analogs. Each beam is formed into a 200 μm thick light sheet by using a −25 mm cylindrical lens coupled with a 25 mm diameter high-energy mirror and a 500 mm spherical lens. A one megapixel CCD camera (TSI, Inc., Shoreview, MN) with a Micro-Nikkor 60mm f/2.8D
lens (Nikon Corporation, Tokyo, Japan) is used to take images with an array of 1000 x 1016 pixels. A cross correlation method employing a “frame straddling” technique is used with a LaserPulse Synchronizer (TSI, Inc., Shoreview, MN) and a personal computer (Dell, Inc., Round Rock, TX). Insight™ 6 software (TSI, Inc., Shoreview, MN) is used to control the image acquisition. The images are captured in time using a trigger signal from the starting inflow of the system. Four hundred image pairs are collected at every 10 ms, as this has been shown to be sufficient for mean flow statistics calculation.

The images are first masked using an in-house Matlab (The MathWorks, Inc., Natick, MA) code by setting the image intensity of the out-of-fluid region to zero. The image walls are found by identifying the locations of the maximum absolute intensity gradients across the boundary. Insight™ 3G software (TSI, Inc., Shoreview, MN) is then used to calculate the displacement and velocity vector. To improve image quality, a pair of background images are generated for each time step using approximately 100 image pairs based on a minimum filter. The masked images are then subtracted by the background images to remove the image background and the light reflection from stationary objects. This is called *pre-processing*. The newly-generated images are then processed using a Hart correlation algorithm [143] with a deformation grid used with an original interrogation region of $32 \times 32$ pixels and a final region of $16 \times 16$ pixels. Figure 2.13 shows an example of how a PIV image is processed.
Figure 2.13: An example of PIV images at different stages of processing: (a) raw image, (b) masked image, (c) background image, (d) pre-processed image after background subtraction.
Validation

Validation is one of the primary methods for building confidence in computational simulations. Unfortunately, validation has many different meanings in the various branches of engineering and computer simulations. Here, we use the definition from Oberkampf and Trucano [144] and Roy [145] where validation is the assessment of the accuracy of the numerical solution by reproducing experimental (or analytical) data. In this chapter, several cases are presented to validate the flow solver and numerical schemes.

3.1 Two-dimensional Decaying Taylor-Green Vortex

3.1.1 Analytical Solution

The 2D Taylor-Green Vortex is an analytical solution to the Navier-Stokes equation, which has long been used for validation of temporal and spatial accuracy of numerical schemes. In the domain $-\frac{\pi}{2} \leq x, y \leq \frac{\pi}{2}$, the solution is given by

$$
\begin{align*}
    u &= \sin x \cos y e^{-2vt} \\
    v &= -\cos x \sin y e^{-2vt}.
\end{align*}
$$

(3.1)
where $\nu$ is the kinematic viscosity of the fluid. The pressure field can be obtained by substituting Equation 3.1 into the Navier-Stokes equation (Equation 2.2), given by:

$$p = \frac{\rho}{2} (\cos 2x + \sin 2y) \left( e^{-2\nu t} \right)^2.$$  

(3.2)

Figure 3.1 shows the analytical velocity magnitude contour, velocity vector, and pressure contour at $t = 1\text{s}$. The kinematic viscosity of the fluid $\nu$ is assumed to be $1\text{ m}^2/\text{s}$. The velocity field has odd symmetry, and the pressure field is periodic in both the $x$ and $y$ direction.

Figure 3.1: Analytical solution of the 2D Taylor-Green Vortex at time $t = 1\text{s}$: (a) velocity magnitude contour, (b) velocity vector, (c) pressure contour.
3.1.2 Numerical Simulation

3.1.2.1 Schemes and Procedure

The transient Navier-Stokes flow solver described in Section 2.7.5 is used to simulate the flow with four different square mesh resolutions (8 × 8, 16 × 16, 32 × 32 and 64 × 64). Three different convection differencing schemes (Section 2.7.2.1), UD, CD, and TVD, are used for the convection face interpolation. The second-order BD (Section 2.7.3) is used for the temporal integration. The time step size \( \Delta t \) is adjusted such that the maximum Courant number for every mesh resolution at each time step is 0.01. Symmetric boundary conditions are applied for all the boundaries except for the empty faces in the \( z \)-direction, as indicated from Figure 3.1. Equations 3.1 and 3.2 are used to prescribe the initial conditions of the simulation at \( t = 0 \).

3.1.2.2 Results

Figure 3.2 shows the contour of numerical velocity magnitude for all four meshes at \( t = 1 \) s. Only the results based on the TVD scheme are presented here. As the mesh is refined, the results move closer to the analytical solution (Figure 3.1).
Figure 3.2: Numerical velocity contours of the 2D Taylor-Green Vortex at time $t = 1\text{s}$ using the TVD scheme: (a) $8 \times 8$ mesh, (b) $16 \times 16$ mesh, (c) $32 \times 32$ mesh, (d) $64 \times 64$ mesh.

To quantify the numerical simulation error, the mean velocity errors over the computational domain normalized by the maximum velocity magnitude are compared among the three convection differencing schemes, as shown in Figure 3.3. This is plotted against the relative grid resolution, defined as $\Delta x / \Delta x_{\text{finest}}$. As a result, $\Delta x / \Delta x_{\text{finest}} = 1, 2, 4, 8$ represents the finest ($64 \times 64$), fine ($32 \times 32$), medium ($16 \times 16$), and coarse ($8 \times 8$) meshes, respectively. It is clear from Figure 3.3 that the first-order UD has a lower rate of grid convergence, while the CD (second-order) and TVD share a higher rate of convergence. This suggests that the TVD scheme is second-order accurate in space.
Figure 3.3: Normalized average velocity error over the computational domain versus normalized grid resolution for different numerical schemes at $t = 1s$. $|U_S - U_A|_{\text{mean}}$ is the mean velocity error between the simulation results ($U_S$) and analytical results ($U_A$) over the computational domain, $U_{\text{max}}$ is the maximum velocity magnitude, $\Delta x$ is the grid spacing, and $\Delta x_{\text{finest}}$ is the finest grid spacing.

3.2 Fully-Developed Turbulent Channel Flow

To validate the accuracy of the FVM based on the MILES method using the TVD-based flux-limiter (Section 2.4), a fully-developed turbulent channel flow at $Re_\tau = 395$ ($Re_\tau = h u_\tau / \nu$, where $u_\tau$ is the friction velocity and $h$ is the channel half height) is studied using both the LES and the MILES methods. The results are compared to the DNS data of Moser et al.[146].
3.2.1 Computational Setup

3.2.1.1 Geometry

The computational domain is shown in Figure 3.4, which represents a subdomain of two infinite parallel plates. The size of the domain is chosen as $2\pi h \times 2h \times \pi h$ in the streamwise, cross-stream, and span-wise directions. This is believed to be adequate to contain the largest eddy structures with the computational cost minimized. The grid consists of $80 \times 100 \times 60$ hexahedral cells with uniform spacing in the streamwise and spanwise directions. A constant cell expansion ratio is applied in the cross-stream direction such that $y^+$ value (defined in Equation 2.28) at the wall-adjacent cells is less than 1.

![Figure 3.4: Schematic of a fully-developed turbulent channel flow geometry. $x$ is the streamwise direction, $y$ is the cross-stream direction, and $z$ is the span-wise direction.](image)

3.2.1.2 Numerical Schemes

Both the LES and the MILES models are used to study the flow. Because the FVM applies an implicit filter to the Navier-Stokes equation based on the computational grids
(Section 2.3.1), no explicit filtering is applied. The SGS turbulence is modeled using two SGS models; one-equation SGS eddy-viscosity model [97] with Van Driest damping function [104] (LES) (Section 2.3.3.1) and the TVD-based flux limiting scheme (MILES) (Section 2.4). The transport equation of the SGS kinetic energy $k_{sgs}$ for the one-equation eddy-viscosity model is discretized according to Section 2.7.6. The second-order accurate CD and TVD schemes are applied for the discretization of the convection term of the Navier-Stokes equation for the LES and MILES models, respectively. The convective term of the transport equation of $k_{sgs}$ is discretized using a second-order accurate TVD scheme. The second-order accurate BD scheme is applied for the time integration of both models.

### 3.2.1.3 Fluid Properties

Both LES and MILES data are compared to the DNS data by Moser et al.[146] at $Re_\tau = 395$. In the DNS simulation, the kinematic viscosity of the fluid $\nu$ is taken as $2 \times 10^{-5} \, \text{m}^2/\text{s}$, and the streamwise bulk velocity $u_b$ is chosen as 0.1335 m/s. The same values are used in the present study to ensure consistency [93].

### 3.2.1.4 Boundary Conditions and Flow Initialization

Two pairs of periodic boundary conditions are applied in the streamwise and spanwise directions and no slip boundary conditions are applied at the top and bottom walls. The flow is initialized using a parabolic laminar profile with superimposed streamwise streaks [93]. Once a statistically steady state is reached, sampling is performed for about 150 flow-through times for both models, followed by a combined streamwise and spanwise averaging over the entire channel.
3.2.2 Model Comparison

3.2.2.1 Velocity

Figure 3.5 shows the normalized mean streamwise velocity and normalized root mean square (rms) fluctuating velocity components predicted using the LES and MILES models along with the DNS data of Moser et al.[146]. Only minor differences between the MILES and LES models are found in the mean streamwise velocity profile, as shown in Figure 3.5(a). Both models, however, slightly over-predict the mean velocity in the log region \(y^+ > 30\) when compared to the DNS data. All the predictions for the rms component \((u_{rms}, v_{rms}, w_{rms})\) in the core region are similar and slightly below the DNS data, as shown in Figure 3.5(b). In the near-wall region, the LES model gives better agreement with the DNS data in terms of \(u_{rms}\) than does the MILES model, but significantly over-predicts \(v_{rms}\) near the wall when compared to the DNS and MILES data, due to its isotropy assumption in the unresolved scales (SGS scales).
3.2.2 Three-dimensional Vortex Structures

One of the major characteristics of a high $Re$ turbulent flow is its highly random, intense vorticity field. In comparison to laminar flows, the vorticity field of high $Re$ turbulent...
flows contains a wide range of scales with large variations in both time and space. In this work, vortex cores are identified using the second largest eigenvalue \((\lambda_e)_2\) method proposed by Jeong and Hussain [147], which is derived based on the assumption that vortex cores contain a local pressure minimum. According to their method, a vortex core can be identified as a connected region where the second largest eigenvalue \((\lambda_e)_2\) is less than zero [93]. The local strain rate of resolved scales, \(\overline{S}\), is defined in Equation 2.20, with anti-symmetric part of the resolved velocity gradient tensor, \(\overline{W}\), given by:

\[
\overline{W} = \frac{1}{2} \left( \nabla \overline{U} - (\nabla \overline{U})^{\text{tr}} \right).
\]

Figure 3.6 shows the comparison of instantaneous vortex structures predicted by the LES and MILES methods. The isosurface contours are colored using normalized helicity \((He)\), defined as [148, 149]:

\[
He = \frac{\overline{U} \cdot \overline{\omega}_{\text{vor}}}{|\overline{U}| |\overline{\omega}_{\text{vor}}|},
\]

where \(\overline{\omega}_{\text{vor}}\) is the vorticity of the resolved velocity, \(\overline{U}\). The normalized helicity physically represents the cosine of the angle between the velocity vector and vorticity vector. \(He = \pm 1\) means that the velocity and vorticity vectors are parallel to each other, while \(He = 0\) means that the vorticity vector is normal to the velocity vector, and therefore, the flow is 2D. It is clear from Figure 3.6 that the LES and MILES predict very similar vortex structures, which are dominated by stream-wise vortices. Helicity is close to zero near the walls, indicating that the flow becomes 2D in these regions.
Figure 3.6: Instantaneous vortex structures in the fully-developed turbulent channel flow visualized through the isosurface contours of the second largest eigenvalue \(((\lambda_e)_2 < 0)\) colored by normalized helicity: (a) LES, (b) MILES.

### 3.3 Laminar Pulsatile Pipe Flow

To validate the capability of the flow solver in simulating pulsatile flows, we study fully developed pulsatile flow in a straight pipe with a diameter, \(D\), of 10 mm and a length, \(L\), of 10 mm. The geometry of the computational domain is shown in Figure 3.7.
3.3.1 Analytical Solution

A fully-developed pulsatile flow in a rigid straight pipe in which the driving pressure gradient varies in time is governed by:

$$\rho \frac{\partial u(t,r)}{\partial t} = -\frac{\partial p(x,t)}{\partial x} + \mu \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u(t,r)}{\partial r} \right) \right],$$  \hspace{1cm} (3.5)

where $x$ is the coordinate in the axial direction, $r$ is the coordinate in the radial direction, and $u$ is the axial component of velocity $U$. Equation 3.5 is similar to Equation 2.121 in Section 2.9 for viscoelastic fluids.

The pulsatile pressure gradient can be expressed by the Fourier series:

$$\frac{\partial p(x,t)}{\partial x} = \sum_0^\infty C_n \cos \left( \frac{2n\pi t}{T} \right) + \sum_1^\infty D_n \sin \left( \frac{2n\pi t}{T} \right),$$  \hspace{1cm} (3.6)

where $C_n$ and $D_n$ are the Fourier constants given by:

$$C_0 = \frac{1}{T} \int_0^T \frac{\partial p(x,t)}{\partial x} dt, \hspace{1cm} (3.7)$$
\[ C_n = \frac{2}{T} \int_0^T \frac{\partial p(x,t)}{\partial x} \cos \left( \frac{2n\pi t}{T} \right) dt \quad (n > 0), \quad (3.8) \]

\[ D_n = \frac{2}{T} \int_0^T \frac{\partial p(x,t)}{\partial x} \sin \left( \frac{2n\pi t}{T} \right) dt \quad (n > 0), \quad (3.9) \]

where \( T \) is the period of the flow cycle.

The analytical solution of Equation 3.5 based on the pressure gradient in Equation 3.6 can be expressed as follows [150]:

\[ u(r,t) = \sum_{0}^{\infty} A_n(r) \cos \left( \frac{2n\pi t}{T} \right) + \sum_{1}^{\infty} B_n(r) \sin \left( \frac{2n\pi t}{T} \right), \quad (3.10) \]

with \( A_n(r) \) and \( B_n(r) \) defined as:

\[ A_0(r) = \frac{C_0 R^2}{4\mu} \left( 1 - \left( \frac{r}{R} \right)^2 \right), \quad (3.11) \]

\[ A_n(r) = \frac{C_n R^2}{\mu} \Re \left\{ -\frac{1}{\Lambda^2} \left( 1 - \frac{J_0(\xi)}{J_0(\Lambda)} \right) \right\} + \frac{D_n R^2}{\mu} \Im \left\{ -\frac{1}{\Lambda^2} \left( 1 - \frac{J_0(\xi)}{J_0(\Lambda)} \right) \right\} \quad (n > 0), \quad (3.12) \]

\[ B_n(r) = -\frac{C_n R^2}{\mu} \Im \left\{ -\frac{1}{\Lambda^2} \left( 1 - \frac{J_0(\xi)}{J_0(\Lambda)} \right) \right\} + \frac{D_n R^2}{\mu} \Re \left\{ -\frac{1}{\Lambda^2} \left( 1 - \frac{J_0(\xi)}{J_0(\Lambda)} \right) \right\} \quad (n > 0), \quad (3.13) \]

where \( \Re \) and \( \Im \) are real and imaginary parts of a function, respectively, \( J_0 \) is the Bessel function of the first kind of order zero, \( R \) is the radius of the pipe, and \( \Lambda \) is a function of \( r \) defined by

\[ \xi = \Lambda \frac{r}{R}, \quad (3.14) \]
where $\Lambda$ is defined by

$$\Lambda = \left( \frac{i - 1}{\sqrt{2}} \right) W_o,$$

(3.15)

with the Womersley number $W_o$ given by

$$W_o = R \sqrt{\frac{\rho \omega}{\mu}},$$

(3.16)

and the angular frequency $\omega$ is defined as

$$\omega = \frac{2\pi}{T}.$$

(3.17)

### 3.3.2 Numerical Solution

#### 3.3.2.1 Computational Setup

The transient Navier-Stokes flow solver described in Section 2.7.5 is used to simulate the flow. The second-order accurate TVD convection differencing scheme is used for the convection face interpolation. The second-order BD scheme (Section 2.7.3) is used for the temporal integration. A composite pressure waveform with a period of 0.5 s is applied at the pipe inlet with zero pressure applied at the outlet. Thirty harmonics are used to represent the composite waveform with reasonable accuracy when compared to the experimental waveform, as shown in Figure 3.8. A zero normal gradient is applied for the velocity field at both the pipe inlet and the outlet. A no-slip boundary condition is applied at the wall. A total of approximately 70,000 hexahedral cells are generated in the geometry with fine boundary cells near the wall. The radial cross-section of the mesh is shown in Figure 3.9. A constant time step interval $\Delta t$ of $2 \times 10^{-4}$ s is used in the simulation and the maximum Courant number through the cycle is 0.5. Ten cycles are required to achieve convergence for the transient analysis.
Figure 3.8: Pipe inlet pressure and flow waveforms.

Figure 3.9: Radial cross-section of the pipe mesh.
3.3.2.2 Numerical Results

The numerical results are compared to the analytical solution in terms of the flow rate and velocity field. Figure 3.8 shows the difference between the numerical and analytical flow waveforms based on the same pressure waveform. Less than 5% relative error is found at every time step. To compare the velocity field, the composite Fourier coefficients \( A_n(r) \) and \( B_n(r) \) (Equations 3.10, 3.11, 3.12 and 3.13) are used. The advantage of comparing the composite Fourier coefficients rather than the velocity field is that the composite Fourier coefficients are independent of time and are measures of velocity accuracy throughout the flow cycle. By definition, the coefficients \( A_n(r) \) and \( B_n(r) \) can be expressed as follows:

\[
A_0(r) = \frac{1}{T} \int_0^T u(r, t) \, dt, \tag{3.18}
\]

\[
A_n(r) = \frac{2}{T} \int_0^T u(r, t) \cos \left( \frac{2n\pi t}{T} \right) \, dt \quad (n > 0), \tag{3.19}
\]

\[
B_n(r) = \frac{2}{T} \int_0^T u(r, t) \sin \left( \frac{2n\pi t}{T} \right) \, dt \quad (n > 0). \tag{3.20}
\]

Therefore, the coefficients can be determined by numerical integration of Equations 3.18, 3.19, and 3.20 using the numerical velocity data through the trapezoidal rule. Figure 3.10 shows the comparison between the numerical and analytical results in terms of the first three of the composite Fourier coefficients of velocity field. Higher order coefficients are much smaller and therefore, not presented here. It is clear from Figure 3.10 that the numerical and analytical data are very close to each other for every Fourier coefficient component. This suggests that the flow solver predicts the flow field accurately. The \( Wo \) in this simulation is 9.5, and the \( Re \) based on the peak velocity is around 577.
3.4 Pediatric Aortic Flow

The first three validation cases are all based on simple geometries. The performance of the flow solver in complex geometries remains unknown. For this reason, special effort has been made in this work to study the flow in a simplified pediatric aorta both experimentally and numerically.

3.4.1 Experiment

3.4.1.1 Flow Conditions

The experimental setup is well described in Section 2.10, and therefore is not presented here. Both Newtonian and 40% HCT blood analogs are used to examine the effect

Figure 3.10: Comparison of the analytical and numerical solutions in terms of the velocity composite Fourier coefficients.
of viscoelasticity on the flow. The mock circulatory loop is adjusted to ensure that both fluids are compared under the same flow conditions. Figures 3.11 and 3.12 show the experimental flow and pressure waveforms for the Newtonian and 40% HCT blood analogs. The heart rate (HR) used in this study is 100 beat/min. It is clear from Figure 3.11 that the Newtonian and 40% HCT blood analogs give similar flow waveforms at each boundary. The pressure waveforms are shown in Figure 3.12, with the relative cycle averaged difference of the aortic pressure between the Newtonian and 40% HCT blood analogs being less than 5%. The aortic pressure is measured in the compliance chamber on the arterial side, as shown in Figure 2.9. The systolic/diastolic pressure are in the range of one-year old infants as reported by Kent et al.[151].

![Flow Rate Waveforms](image)

**Figure 3.11:** Experimental flow waveforms on each boundary for the Newtonian and 40% HCT blood analogs.
Figure 3.12: Experimental aortic pressure waveforms for the Newtonian and 40% HCT blood analogs.

The flow splitting in the great vessels is summarized in Table 3.1. Both fluids show similar flow splitting in the great vessels. The total flow splitting in the great vessels are 45.3% and 47.3% for the Newtonian and 40% HCT blood analogs, respectively. According to Fogel et al.[152], approximately 40% to 50% of the CO is diverted to the great vessels in the pediatric aortic system. This suggests that the flow splitting used in this study is physiological.
Table 3.1: Flow splitting in the aorta. Flow percentage is the ratio between the flow rate and the CO.

<table>
<thead>
<tr>
<th>Fluid</th>
<th>CO [L/min]</th>
<th>Brachiocephalic Artery (%)</th>
<th>Left Common Carotid Artery (%)</th>
<th>Left Subclavian Artery (%)</th>
<th>Total Flow Division in the Great Vessels (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Newtonian</td>
<td>1.11</td>
<td>23.3</td>
<td>13.2</td>
<td>8.8</td>
<td>45.3</td>
</tr>
<tr>
<td>40% HCT</td>
<td>1.09</td>
<td>24.0</td>
<td>13.7</td>
<td>9.5</td>
<td>47.3</td>
</tr>
</tbody>
</table>

3.4.1.2 The Effect of Viscoelasticity

Figure 3.13 shows the experimental velocity contours (measured by PIV) in planes 12 mm, 10 mm and 12.8 mm as a function of time using both the Newtonian and 40% HCT blood analogs. The locations of planes 12mm, 10 mm and 12.8 mm are shown in Figure 3.14. The Newtonian and 40% HCT blood analogs give similar velocity contours at every time step. The flow is skewed toward the inner wall of the ascending aorta and the proximal walls of the great vessels at early systole, as shown in Figure 3.13 (1). Flow separation develops first near the proximal walls of the great vessels in the middle of flow acceleration (Figure 3.13 (3)), and stays until late systole when retrograde flows occur (Figure 3.13 (7)). The maximum velocity occurs at peak systole (Figure 3.13 (5)), with the flow in the great vessels skewed toward the distal walls. Overall, the viscoelasticity has little impact on the flow field. Note that the blank regions inside the flow contours are where acrylic pieces are glued together (see Figure 2.11). Laser beams are significantly scattered in these regions, and therefore, no data can be obtained.
(a) Newtonian, 12 mm Plane  
(b) 40% HCT, 12 mm Plane  
(c) Newtonian, 10 mm Plane  
(d) 40% HCT, 10 mm Plane  
(e) Newtonian, 12.8 mm Plane  
(f) 40% HCT, 12.8 mm Plane

(1) 40 ms
(2) 60 ms
(3) 80 ms
(a) Newtonian, 12 mm Plane
(b) 40% HCT, 12 mm Plane
(c) Newtonian, 10 mm Plane
(d) 40% HCT, 10 mm Plane
(e) Newtonian, 12.8 mm Plane
(f) 40% HCT, 12.8 mm Plane

(4) 100 ms
(5) 160 ms
(6) 220 ms
(7) 240 ms
(8) 270 ms
(10) 350 ms
(11) 450 ms
Figure 3.13: Experimental velocity contours in planes 12 mm, 10 mm and 12.8 mm at various time steps.
3.4.2 Numerical Simulation

Numerical simulation is performed for the simplified pediatric aortic model under the same flow conditions as the experiment. Only the Newtonian fluid is used for the simulation, since the effect of viscoelasticity is negligible, as determined experimentally in Section 3.4.1.2.

3.4.2.1 Numerical Setup

High-quality unstructured mesh cells (mainly tetrahedra) are generated for the pediatric aortic model using GAMBIT 2.3.16 (Fluent Inc., Lebanon, NH, USA) with fine boundary cells applied near the walls. The total cell number is approximately 500,000. The flow is solved using the time-dependent Navier-Stokes solver described in Section 2.7.5. The
second-order BD scheme is used for temporal discretization, and the second-order TVD scheme is used for the convection differencing. The pressure-velocity is coupled using the PISO.

The experimental aortic inlet and outlet flow waveforms are applied at the aortic inlet and outlet, respectively. Fifty harmonics are used to make the composite flow waveforms match the experimental ones. At every time step, the velocity profiles at the aortic inlet and outlet are assumed to be parabolic. A zero normal gradient is applied for pressure at the aortic inlet and outlet. At the outlets of the great vessels, zero normal gradient is applied for the velocity with a resistance boundary condition applied for pressure, defined as [153, 154]:

\[ p = \chi Q + p_0 \]  \hspace{1cm} (3.21)

where \( \chi \) is the resistance constant, and \( p_0 \) is a base pressure used to maintain a physiological pressure level in the system. Approximate resistance values are applied to the great vessels such that the flow splitting is consistent with the experiment. A no-slip boundary condition is applied at the wall.

A Newtonian fluid is used for the simulation, with a kinematic viscosity, \( \nu \), of \( 3.77 \times 10^{-6} \text{ m}^2/\text{s} \) (3.77 \( \times 10^{-2} \text{ Stokes} \)), consistent with one used in the experiment. The time step is chosen such that the maximum Courant number in the flow domain is around 0.5. The simulation takes three flow cycles to converge.

3.4.2.2 Comparison of Results

Figure 3.15 shows the experimental and numerical flow waveforms at each boundary. Only slight differences are found in the flow waveforms of the great vessels, especially in late systole. This suggests that the resistance boundary condition provides a good approximation of real-time flow splitting throughout the cycle.

Figure 3.16 shows the comparison of the experimental and numerical flow fields at
planes 12 mm and 10 mm. The experimental and numerical contours share a similar pattern in planes 12 mm and 10 mm throughout the flow cycle, albeit with slightly different magnitude. The difference is mainly due to two reasons. First, there is a slight difference between the experimental and numerical flow waveforms in the great vessels, as shown in Figure 3.15. This contributes to the deviation of the numerical flow magnitude from the experimental one in the great vessels, especially during late systole and early diastole. Second, the assumption of the parabolic flow profile at the aortic inlet is not entirely true, as shown in Figure 3.13. It is interesting that the numerical results at peak systole and early diastole show a flow separation near the inner wall of the aortic arch at plane 12 mm, as shown in Figures 3.16 (5) and (6). This flow separation region is partly overlapped with the “blind spots” of the PIV measurements. The experimental contours in these regions remain unknown.
Figure 3.15: Comparison of the experimental and numerical flow waveform at each boundary based on the Newtonian blood analog.
(1) 40 ms
(2) 60 ms
(3) 80 ms
(a) Experimental

(b) Numerical

(4) 100 ms
(5) 160 ms
(a) Experimental

(b) Numerical

(6) 220 ms
(a) Experimental

(b) Numerical

(7) 240 ms
(8) 270 ms
(a) Experimental

(b) Numerical

(10) 350 ms
Figure 3.16: Comparison of the numerical and experimental flow field at different time steps.

3.5 Conclusion

We validated the flow solver using four different cases. The 2D decaying Taylor-Green vortex was used to examine the temporal/spatial accuracy of the numerical schemes. The TVD scheme is found to be second-order accurate in space. The MILES method based on the TVD scheme was validated by comparing to documented results for a
fully-developed turbulent channel flow at $Re_x = 395$. The MILES is found to predict reasonable turbulence statistics without excessive numerical dissipation. One advantage of this method over the one-equation eddy-viscosity LES model is the anisotropic nature of its SGS modeling, which is based on the local grid. The performance of the flow solver in pulsatile flows was validated using a fully-developed pulsatile pipe flow and a simplified pediatric aortic flow. Reasonable agreements were found between the numerical and analytical/experimental results. It is worth mentioning that the flow solver can be used for both laminar and turbulent flows. In laminar flows, the TVD scheme provides necessary boundedness to the flow without significantly violating the accuracy of the solution. In turbulent flows, the TVD-based flux-limiting scheme implicitly acts as a SGS model and stabilizes the flow. The accuracy of the results, however, is entirely dependent on the grid.
Chapter 4

Blood Flow in the Adult Left Ventricular Assist Device Anastomoses

In this chapter, we study the turbulent flow in the end-to-side anastomosis of a LVAD to the adult aorta using the previously validated transient flow solver [155]. Two different anastomotic flows (proximal and distal) are simulated for 50% and 100% LVAD support, and the results are compared with a healthy adult aortic flow. All the analyses are based on a planar aortic model under steady inflow conditions for simplification. Our results reveal that the outflow cannulae induce high exit jet flows in the aorta, resulting in turbulent flow. The distal configuration causes more turbulence in the aorta than does the proximal configuration. The turbulence, however, may not cause any hemolysis due to low Reynolds stresses and relatively large Kolmogorov length scales compared to blood red cells. The LVAD support causes an acute increase of flow splitting in the great vessels for both anastomotic configurations, although its long-term effect on the flow splitting remains unknown. A large increase of WSS is found near the cannulation sites during the LVAD support. This work builds a foundation for more physiologically realistic simulations under pulsatile flow conditions.
4.1 Methods

4.1.1 Engineering Model Geometry

The aorta is modeled in six segments (Figure 4.1 (a)): ascending, arch, descending, brachiocephalic (innominate) artery, left common carotid artery, and left subclavian artery. The ascending and descending aorta are assumed to be straight pipes with internal diameters of 2.3 cm and lengths of 1.61 cm and 20.94 cm, respectively. The aortic arch is assumed to be curved in a single plane with a radius of 5.07 cm. The three great vessels are modeled as in-plane straight pipes with inner diameters of 1.65 cm, 1.05 cm, and 1.25 cm, respectively. The branch lengths are chosen such that the flow patterns at the inlet of the branches are independent of the branch outlet velocity boundary conditions (zero normal gradient) [156]. An outflow cannula, modeled as a curved pipe with an internal diameter of 1.8 cm, is sutured onto the ascending aorta as a proximal anastomotic configuration (Figure 4.1 (b)) or onto the aortic arch as a distal anastomotic configuration (Figure 4.1 (c)) [40].

Figure 4.1: (a) Healthy aorta, (b) proximal configuration with the cannula attached on the ascending aorta, (c) distal configuration with the cannula attached on the aortic arch.
4.1.2 Governing Equations and Non-dimensionalization

The flow in the aorta is governed by the time-dependent Navier-Stokes equations. To simplify the numerical simulation, the momentum and continuity equations are first non-dimensionalized by the diameter of the aorta $D$ and the mean velocity at the aortic inlet for the normal CO (5 L/min) $U_{\text{mean}}$. The corresponding dimensionless inner radius $R_i^*$ and outer radius $R_o^*$ of the aortic curvature are equal to 1.6 and 2.6, respectively, where superscript $\ast$ refers to a dimensionless form. The coordinates $x$, $y$, and $z$ are defined in Figure 4.1 (a), and all are referenced to the center of the aortic curvature. For the convenience of numerical data presentation, the coordinates $x$ and $y$ can be transformed into the cylindrical coordinates $r$ and $\theta$, as shown in Figure 4.1 (a). This coordinate system is applied to both the proximal and distal configurations.

4.1.3 Grid Generation

High-quality mesh cells (mainly hexahedra and tetrahedra) are generated for the anastomotic models using GAMBIT 2.3.16 (Fluent Inc., Lebanon, NH, USA). Three different grids (coarse grid = $\sim 400,000$ cells; medium grid = $\sim 2,000,000$ cells; fine grid = $\sim 7,000,000$ cells) are constructed for the grid study. Fine boundary cells are used near the walls for all three to resolve the viscous sublayer of the turbulent wall flows. The dimensionless heights of the near-wall spacing are 0.01, 0.005, and 0.0025 in the coarse, medium, and fine grids, respectively. The interior volumes of the geometries are filled with a combination of hexahedral and tetrahedral cells with nearly isotropic dimensionless sizes of 0.06, 0.03, and 0.02 for the coarse, medium and fine grids, respectively. The maximum non-orthogonality and skewness of each mesh are shown in Table 4.1.
Table 4.1: Mesh non-orthogonality and skewness.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Max Non-orthogonality</th>
<th>Max Skewness</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>78.69</td>
<td>1.16</td>
</tr>
<tr>
<td>Medium</td>
<td>78.41</td>
<td>1.89</td>
</tr>
<tr>
<td>Fine</td>
<td>66.16</td>
<td>0.85</td>
</tr>
</tbody>
</table>

4.1.4 Flow Solvers and Numerical Algorithms

Numerical simulations are performed using the previously validated time-accurate Navier-Stokes solver (see Chapter 3) for the anastomotic models. The set of non-dimensionalized governing equations is discretized into algebraic equations using the FVM described in Section 2.7.5. The time derivative term is discretized using the second-order BD scheme. The convective term in the momentum equation is discretized using the second-order TVD scheme, which implicitly satisfies the physically required SGS properties and stabilizes the flow (MILES). This method has been carefully validated in Section 3.2. The linear solvers used in the present work are the GAMG solver for the pressure and the PBICG solver for the velocity. The pressure and velocity terms are coupled using the PISO scheme. The initial conditions for all the field variables are attained using a steady Navier-Stokes solver [128]. The semi-implicit pressure linked equations (SIMPLE) algorithm is applied for the pressure-velocity coupling in the steady flow solver.

4.1.5 Fluid Properties and Computational Parameters

Blood is modeled as an incompressible Newtonian fluid with a density, $\rho$, of $1.057 \times 10^{-3}$ kg/m$^3$ and a dynamic viscosity, $\mu$, of $3.7 \times 10^{-3}$ kg/(m·s). The transient simulations start and continue until a statistically stationary state is reached. Flow data are then sampled at every time step for roughly one flow-through time, and flow statistics are
collected. The flow-through time is the time taken by a fluid particle to traverse from the aortic inlet along the aortic arch to the aortic outlet with a velocity, defined by the CO (5 L/min) divided by the cross-sectional area of the aorta. The corresponding dimensionless flow-through time in this study is 16. The residuals of all the variables are converged to $10^{-6}$ at all time steps. The maximum Courant numbers for all three grids are around 0.7. Simulations are performed in parallel using a PC cluster within the Penn State High Performance Computing Group. Sun SunFire v20z 1U Rackmount Boxes are used as the compute nodes, each with Dual 2.4 GHz AMD Opteron Processors and 8 GB of ECC RAM, and are interconnected using high-speed Silverstorm Infiniband network. The cluster runs the GNU/Linux operating system. Four, sixteen, and thirty-two processors are used for the coarse grid, medium grid, and fine grid, respectively. Computations take approximately 16, 60, and 110 hours to simulate one flow-through time for the respective grids.

4.1.6 Boundary Conditions

For each LVAD anastomotic configuration, three different operating conditions are considered (Table 4.2). Case 1 represents a healthy aortic flow condition with a CO of 5.0 L/min and serves as a no LVAD support baseline. For case 2, all the blood flow goes through the LVAD, short-circuiting the left heart. For case 3, the left heart works in parallel with the LVAD.
Table 4.2: Summary of operating flow conditions applied for both anastomotic configurations.

<table>
<thead>
<tr>
<th>Condition</th>
<th>Aortic Inflow (L/min)</th>
<th>Re at the Aortic Inlet</th>
<th>LVAD Outflow (L/min)</th>
<th>Re at the Outflow Cannula Inlet</th>
<th>Model</th>
<th>Inlet Boundary Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>5</td>
<td>1318</td>
<td>0</td>
<td>0</td>
<td>Laminar</td>
<td>Steady</td>
</tr>
<tr>
<td>II</td>
<td>0</td>
<td>0</td>
<td>5</td>
<td>1647</td>
<td>MILES</td>
<td>Steady</td>
</tr>
<tr>
<td>III</td>
<td>2.5</td>
<td>659</td>
<td>2.5</td>
<td>823</td>
<td>MILES</td>
<td>Steady</td>
</tr>
</tbody>
</table>

A plug flow velocity profile is applied at both the inlet of the aorta and the outlet of the LVAD outflow cannula. The velocity boundary conditions at the outlet of the descending aorta and the great vessels are zero normal velocity gradients. All the outlets of the great vessels are assigned an outflow resistance boundary condition (Equation 3.21). $p_0$ is assumed to be 100 mmHg in this study. The resistance of each great vessel is determined with reference to steady flow in the healthy aorta (Figure 4.1 (a)) for the CO of 5 L/min. According to Middleman [157], approximately 5% of the CO is diverted to each great vessel in the adult human aortic system. Numerical experiments are performed to find the physiologically correct pressure at each outlet of the great vessels needed to maintain this 5% output. The resistance of each great vessel is then calculated based on Equation 3.21 and maintained at this value for the cannulated aorta study. The aorta and LVAD outlet cannula are assumed to be rigid, and a no-slip boundary condition is applied at the walls.
4.2 Results

4.2.1 Grid Study

Solution verification is essential to the MILES method because the effect of SGS motions is modeled by numerical dissipation. In this study, the numerical simulations for the anastomotic models are performed using three different grids. The results are compared in terms of mean velocity magnitude and resolved temporal energy spectra. Figures 4.2 (a) and (b) shows the effect of grid size on the mean velocity magnitude along the line of \( r^* = \frac{R^*_o + R^*_i}{2} \) and \( \theta = 45^\circ \) and the line of \( r^* = \frac{R^*_o + R^*_i}{2} \) and \( \theta = 15^\circ \) for the distal configuration under operating condition III, respectively. Both line locations are close to the cannulation site. It is clear from Figure 4.2 that the medium grid gives reasonable results and sufficient spatial resolution when compared to the fine grid.
Figure 4.2: The effect of grid size on mean velocity magnitude near the distal cannulation site under operating condition III: (a) Position I: \( r^* = \frac{R_0^* + R_1^*}{2}, \theta = 45^\circ \); (b) Position II: \( r^* = \frac{R_0^* + R_1^*}{2}, \theta = 15^\circ \).

Figure 4.3 shows the effect of grid size on the resolved energy spectra at probe 1 for the distal configuration under operating condition II. The probe is located near the
distal cannulation site (Figure 4.4), where most turbulence occurs. The cut-off frequency is chosen as 100 Hz, determined by

\[ f_{\text{cutoff}} = \frac{q}{\Delta x}, \quad (4.1) \]

where \( \Delta x \) is the grid size, and \( q \) is defined by

\[ q = \sqrt{\frac{1}{3} (u'^2 + v'^2 + w'^2)}, \]

where \( u', v' \) and \( w' \) are the three components of fluctuating velocity. It is clear from Figure 4.3 that the resolved energy spectra are close to each other for all three grids in the low frequency region. In the high frequency region, however, the spectra are roughly an order of magnitude higher for the medium and fine grids when compared to the coarse grid. The higher spectra indicate a higher proportion of resolved scales for higher grid resolutions. Overall, the medium grid captures the turbulent eddies without excessive numerical dissipation. Therefore, the results presented from here are based on the medium grid.
Figure 4.3: The effect of grid size on the temporal energy spectra at probe 1 for the distal configuration under operating condition II.

Figure 4.4: Summary of the plane and probe locations used in this study.
4.2.2 Mean Primary Flow Field

The flow field in the aorta is presented in the planes $z^* = 0$, $z^* = -0.3$, and $z^* = 0.3$, as shown in Figure 4.4. Plane $z^* = 0$ is the mid-plane of the aorta. Plane $z^* = 0.3$ is the plane close to the outflow cannula side, while plane $z^* = -0.3$ is near the posterior wall of the aorta. The flow fields in the mid-plane of the cannula (plane $x^* = -2$ for the proximal anastomotic configuration; plane $x^* = 1.6$ for the distal anastomotic configuration) are also presented.

The primary flow fields are presented in Figure 4.5. Figure 4.5 (a) shows the flow field for the healthy aortic flow (condition I). The flow at the entrance to the aorta is skewed toward the inner aortic wall by the curved entrance. As the flow moves downstream, it skews toward the outer wall of the aorta, due to the centrifugal force. Within the secondary branches, the flow is highly skewed toward the distal walls, as shown in plane $z^* = 0$ in Figure 4.5 (a). The separation areas near the proximal walls of the branch vessels are due partially to the unnaturally sharp edges in the branch junction areas.

An impingement jet is found on the aortic arch near the outflow anastomosis for condition II. This impingement directs a high-velocity jet into the brachiocephalic artery for the proximal anastomotic configuration, as shown in the planes of $z^* = 0$ and $z^* = -0.3$ in Figure 4.5 (b). A strong impingement jet, however, is not found for condition III, as shown in Figure 4.5 (c). The flow starts to reorganize downstream in the descending aorta for all three conditions. The flow in the cannula is found to be skewed toward the outer wall due to the effect of curvature, as shown in plane $x^* = -2$ in Figures 4.5 (b) and (c).

The primary flow fields for the distal anastomotic configuration under flow conditions II and III are presented in Figures 4.5 (d) and (e), respectively. The flow in the aorta is highly disturbed with an impingement jet on the aortic arch for condition II, as shown in plane $z^* = -0.3$ in Figure 4.5 (d). A large stagnant flow region ($z^* = 0$ in Figure 4.5 (d)) is found near the aortic root in flow condition II. The flow within the great vessels
for condition II is skewed toward the proximal walls, as shown in plane $z^* = 0$ in Figure 4.5 (d). This is clearly different from the flow in the proximal configuration under the same condition, as shown in Figure 4.5 (b).

In contrast to condition II, the inflow from the cannula has less effect on the upstream flow field proximal to the cannula insertion site in operating condition III, as shown in Figure 4.5 (e). The flow in the branch vessels is skewed toward the distal walls of the branches, similar to the flow in operating condition I. The flow downstream, however, is highly disturbed. A region of flow separation is found in the planes of $z^* = 0$ and $z^* = 0.3$ distal to the cannula insertion site, as shown in Figure 4.5 (e).
4.2.3 Mean Secondary Flow Field

The curvature of the aortic arch and the strong mixing of the flows, combined with the presence of the great vessels, induce significant secondary flow within the aorta. Figures 4.6 and 4.7 illustrate the mean secondary flow fields under flow condition III for both proximal and distal anastomoses in thirteen different planes. The cross-flow vectors represent the direction of the tangential velocities on each plane. The vectors are all of
uniform length, so that they only indicate flow direction, not magnitude. The velocity contours in Figures 4.6 and 4.7 provide the magnitude of the mean three-component velocities.

Figure 4.6 shows the secondary flows for the proximal configuration. The cannula inflow disrupts the secondary flow patterns in the aorta, as shown in plane P2. As the flow moves downstream, a rotation cell is found near the anterior aortic wall in plane P3. This cell continues revolving, as shown from planes P4 to P7. A pair of Dean cells start developing from plane 4 and persist through plane P7. The secondary flows in the branch vessels near the main aorta are dominated by the flow impingement on the branch walls. As the flow moves further downstream into the branch vessels, it reorganizes and the secondary flow decays. The flow in the branches is found to be skewed toward the posterior branch walls. This is more pronounced in the brachiocephalic artery and left common carotid artery (planes P8 and P10), as they are closer to the cannulation site than is the left subclavian artery. This skewness of the flow in each secondary branch decays further downstream.

Figure 4.7 shows the secondary flow motions for the distal configuration. As the flow enters the aortic arch, the secondary flow is displaced from the outer wall to the inner wall, as shown in P1. A pair of Dean cells sets up immediately downstream of the aortic entrance, due to the imbalance between centrifugal force and radial pressure gradient [158], as shown in P2. As the flow progresses downstream, a pair of separation cells are formed near the inner wall, as shown in P3. This is a result of a secondary boundary-layer separation near the inner wall [159]. The separation cells disappear and a second symmetric pair of counter-rotating vortices emerges in plane P4, in which the secondary flow has a four-vortex pattern. This type of four-vortex mode is usually found for fully-developed curved-pipe flow when the Dean number is larger than 957 [160]. In this study, the Dean number based on the aortic arch curvature is 909. The appearance of the four-vortex pattern in the slightly lower Dean number is possible because the flow
is not fully-developed. The Dean cells are also found in plane P5, while the second pair of cells disappears. The secondary flow motions are highly disturbed in the downstream region distal to the cannula anastomosis, as seen in P6 and P7. A clear flow separation region is found near the anterior aortic wall on the cannula side, as shown in P6 and P7. It should be pointed out that the secondary flow in the branch vessels for the distal configuration is different from that for the proximal configuration. For the distal configuration, the secondary flow in the branch vessels is symmetrical to the mid-plane $z^* = 0$, because the distal anastomosis does not strongly affect the upstream aortic flow. This is not the case for the proximal configuration.
Figure 4.6: Mean secondary flow fields for the proximal configuration under flow condition III.
Figure 4.7: Mean secondary flow fields for the distal configuration under flow condition III.
4.2.4 Turbulence

Figure 4.8 shows the resolved energy spectra of fluctuating velocity at probe 2 under flow conditions II and III for the distal configuration. The location of probe 2 is near the cannulation site, as shown in Figure 4.4. The cutoff frequency is set as 100 Hz. The results from Figure 4.8 clearly show that the exit jet flow from the outflow cannula can cause turbulent flow in the aorta and that condition II tends to cause more turbulence than condition III.

![Figure 4.8: Power spectra of the fluctuating velocities at probe 2 for the distal configuration.](image)

We further compare the principal normal Reynolds stress field for the proximal and distal configurations. The three principal normal stresses are calculated as the eigenvalues of the Reynolds stress tensor, as described in Section 2.6.3. Figure 4.9 shows the maximum principal normal Reynolds stress component at planes $z^* = 0$, $z^* = -0.3$, and $z^* = 0.3$ for the proximal and distal configurations under condition II. The principal normal Reynolds stresses for condition III are much smaller, and therefore are not
presented. It is found from Figure 4.9 that the stress field for the distal configuration is much higher than the stress field for the proximal configuration. The maximum normal stress is approximately 410 dynes/cm², as shown in Figure 4.9 (b).

Figure 4.9: The maximum principal normal Reynolds stress at planes \( z^* = 0 \), \( z^* = -0.3 \), and \( z^* = 0.3 \) for flow condition II: (a) proximal configuration, (b) distal configuration.

4.2.5 Mean Wall Shear Stress

Figure 4.10 shows the distribution of the magnitude of the mean WSS. For the healthy aortic flow (condition I), the WSS maxima (31 dynes/cm²) occur on the outer wall of the aortic arch between the branch entrances and the distal walls of the secondary vessels, while the minimum WSS occur along the inner wall of the aortic arch, the proximal walls of the branch vessels, and the wall of the descending aorta. With the LVAD support
(flow conditions II and III), the cannula attachment areas show very high WSS, with the proximal configuration causing a higher WSS distribution near the cannulation site than does the distal configuration. The maximum WSS for the proximal and distal configurations are 80 dynes/cm² and 67 dynes/cm², respectively. For the distal configuration with operating condition II, the high WSS regions in the branch vessels occur on the proximal walls instead of the distal walls, consistent with the observations from the velocity contours.

Figure 4.10: Mean WSS magnitude contour: (a) condition I; (b) condition II, proximal configuration; (c) condition III, proximal configuration; (d) condition II, distal configuration; (e) condition III, distal configuration.

4.2.6 Flow Splitting in the Great Vessels

Table 4.3 shows the effects of the anastomotic configurations on the flow rates in the great vessels. For condition I, the flow rate at each branch vessel is 5% of the CO. With the LVAD support, the flow rate at each great vessel increases for both configurations,
except for the flow rate at the left subclavian artery under condition III with the proximal configuration. The maximum increase (roughly 66.6%) of total flow division in the secondary vessels occurs in the distal configuration under flow condition II.

Table 4.3: Flow splitting in the great vessels. Flow percentage is the ratio between the flow rate and the CO.

<table>
<thead>
<tr>
<th>Anatomostic configuration</th>
<th>Flow Conditions</th>
<th>Brachiocephalic Artery (%)</th>
<th>Left Common Carotid Artery (%)</th>
<th>Left Subclavian Artery (%)</th>
<th>Total Flow Division in the Great Vessels (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>No LVAD Support I</td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>15</td>
<td></td>
</tr>
<tr>
<td>Proximal II</td>
<td>6.63</td>
<td>5.36</td>
<td>5.00</td>
<td>16.99</td>
<td></td>
</tr>
<tr>
<td>Proximal III</td>
<td>5.28</td>
<td>5.3</td>
<td>4.84</td>
<td>15.42</td>
<td></td>
</tr>
<tr>
<td>Distal II</td>
<td>7.62</td>
<td>8.62</td>
<td>8.75</td>
<td>24.99</td>
<td></td>
</tr>
<tr>
<td>Distal III</td>
<td>6.57</td>
<td>7.05</td>
<td>7.42</td>
<td>21.04</td>
<td></td>
</tr>
</tbody>
</table>

4.3 Discussion

4.3.1 Anastomotic Flow

Our numerical results suggest that LVAD anastomoses have a significant effect on the aortic flow (Figures 4.5, 4.6 and 4.7). The greatest disruption of the aortic flow occurs during the 100% LVAD support (condition II). The proximal configuration causes a higher exit jet flow in the aorta than does the distal configuration, as the exit jet directs flow both upstream and downstream in the distal configuration, while in the proximal configuration the exit jet directs flow mainly downstream. The high-velocity jet in the proximal configuration is diverted into the brachiocephalic artery, sparing the left common carotid and left subclavian arteries, which might explain the perioperative right-sided stroke rate because most emboli caused by the aortic graft flow into the brachiocephalic artery [161, 162]. The distal configuration exhibits a greater region of flow stagnation near the
aortic valve than does the proximal configuration during the 100% support, suggesting that the distal configuration may provide a site for thrombosis near the aortic valve, consistent with the findings by May-Newman et al.[40]. Therefore, this type of surgical configuration is not recommended for patients with poor cardiac function.

Turbulence is found near the cannulation sites during the LVAD support. The distal configuration causes more turbulence than does the proximal configuration (Figure 4.9), as a consequence of its increased large scale unsteadiness of the flow. For the same configuration, condition II (100% support) causes more turbulence than does condition III (50% support) (Figure 4.8), due to the higher exit jet flow into the aorta under condition II. To evaluate possible hemolysis in the turbulent flow, the principal Reynolds stresses are calculated, and the maximum Reynolds stress is approximately 410 dynes/cm², well below the hemolysis threshold value (4000 dynes/cm²) reported in the literature [76].

The Kolmogorov length scale of the flow, $\eta_K$, can be estimated by [49]

$$\frac{D}{\eta_K} = \left( \frac{\rho \mu q}{D} \right)^{3/4},$$  \hspace{1cm} (4.2)

and is approximately 150 $\mu$m, which is about twenty times larger than the size of red cells. Because the Reynolds stresses are low and the smallest eddies of the flow are not comparable to the size of red cells, the turbulent flow may not cause any hemolysis. The nature of the mean flow field, however, is influenced.

The maximum WSS for the healthy aortic flow is 31 dynes/cm², which is located in the vicinity of the aortic arch bifurcations (Figure 4.10). This is roughly half of the maximum WSS at peak systole (64 dynes/cm²) reported by Friedman et al.[163] in their aortic bifurcation model, which occurred near the flow divider of the aortic bifurcation. The WSS distribution on the aortic wall is highly altered during LVAD support. High WSS regions (up to 80 dynes/cm²) are found on the aortic arch near the cannulation sites (Figure 4.10). The proximal configuration causes a higher WSS distribution near
the cannulation site than does the distal configuration, as a result of the higher exit jet flow in the proximal configuration. For each configuration, condition II, in which all the flow goes through the LVAD, causes the highest wall shear stresses near the cannulation sites.

A large increase in flow rate in the great vessels is found for both proximal and distal configurations due to the increase of the pressure in the branches (Table 4.3). In general, the distal configuration causes higher flow rates at the great vessels than does the proximal configuration. As the percentage of the LVAD support increases, the flow rates in the great vessels tend to increase. These results are qualitatively consistent with the flow splitting data reported by Litwak et al.[41] in their acute animal study. Their data showed that the flow in the descending aorta is decreased at any level of continuous or pulsatile ventricular assist device (VAD) support when the LVAD is connected to the descending aorta. We remind the reader that both our results and the data by Litwak et al. only represent the short-term effect of the LVAD support on the flow splitting. The downstream resistances in the secondary vessels are strongly auto-regulated and may adapt to maintain blood supply at a physiological level in the long run. Unfortunately, there is a lack of long-term data dealing with the flow splitting in patients with LVAD support. Systematic in vivo studies would be useful.

4.3.2 Limitations

First, there are significant differences between the simplified planar aortic model used in this study and the native adult aorta. Both the tapering and 3D convolution of the aorta [164] are ignored. However, good qualitative agreement is found between our results for condition I (healthy aortic flow) and the data reported by Shahcheraghi et al.[156] using a more realistic aortic model during peak systole. Both studies show that the flow is skewed toward the inner aortic wall near the entrance, the outer wall along the aortic arch, and that the flow in the secondary branches is highly skewed toward the distal
walls. Moreover, both studies indicate that the maximum WSS occurs near the aortic bifurcations. This suggests that our results will provide a good basis for studying the anastomotic flow using more complicated aortic models.

Second, pulsatility of both the native heart and LVAD will also need to be incorporated. Pulsatile flows can provide better end-organ perfusion than can continuous flows [165]. Furthermore, pulsatile flows from the LVAD are expected to induce a much higher jet flow into the aorta, causing more turbulence. The effects of pulsatility and three-dimensionality of the native aorta will be studied in the next two chapters.

4.4 Conclusion

We studied the turbulent flow in a simplified planar aortic model with proximal and distal LVAD anastomotic configurations using a previously validated FVM based MILES. A systematic grid study was performed to verify the numerical results. Our results demonstrate that the LVAD anastomosis induces high-velocity jets into the aorta. The proximal configuration directs the jet mainly into the brachiocephalic artery, which might explain the dominance of right hemispheric perioperative strokes in adult patients. As the LVAD support increases, turbulence may occur in the aorta. The distal configuration causes more turbulence than does the proximal configuration. The turbulence may not cause hemolysis due to its relatively low Reynolds stresses and large size relative to the red cells. The distal configuration is not recommended for patients with low or no flow from the heart due to the lack of blood washout near the aortic valve. The LVAD anastomosis leads to large increases in flow splitting to the great vessels, acutely. Relatively high WSS is found near the cannulation sites during the LVAD support when compared to the healthy aortic flow.
Chapter 5

Blood Flow in the Pulsatile Pediatric Ventricular Assist Device Anastomosis

Numerical simulations are performed to investigate the flow within the end-to-side proximal anastomosis of a pulsatile PVAD to an aorta [166]. The anastomotic model is constructed from a patient-specific pediatric aorta. The three great vessels originating from the aortic arch, the brachiocephalic (innominate) artery, the left common carotid artery, and the left subclavian artery, are included. A MILES method based upon a FVM is used to study the resulting turbulent flow. A resistance boundary condition is applied at each branch outlet to study flow splitting. The PVAD anastomosis is found to alter the aortic flow dramatically. More flow is diverted into the great vessels with the PVAD support. Turbulence is found in the jet impingement area at peak systole for 100% bypass and a maximum principal normal Reynolds stress of 7081 dynes/cm$^2$ is estimated based on ten flow cycles. This may be high enough to cause hemolysis and platelet activation. Regions prone to IH are identified by combining the time-averaged wall shear stress and oscillatory shear index. These regions are found to vary with the percentage of flow bypass.
5.1 Methods

5.1.1 Geometrical Model

The natural aorta is three-dimensionally convoluted [164]. Hemodynamic parameters, such as WSS, are largely dependent on the local anatomy of the aorta. Therefore, an accurate study of a pediatric aortic graft flow requires patient-specific pediatric aorta morphology. In this study, a patient-specific healthy aorta (eight years old) geometry (Figure 5.1 (a)) reconstructed using MRI (NIH-Georgia Tech Fontan Anotomy Database ID: CHOP007) is scaled down to represent a healthy pediatric aorta of a one year old with an aortic inlet diameter of 11.6 mm. The primary dimensions of the new aortic model are well in the range of morphological measurements reported by Machii and Becker [142]. The length of each branch vessel on the aortic arch is extended about ten times its internal diameter such that the flow pattern at the branch inlet is independent of the branch outlet boundary condition. A curved pipe with an internal diameter of 6 mm is attached to the ascending aorta as a PVAD outflow graft, as shown in Figure 5.1 (b). The nomenclature of the end-to-side anastomosis is presented in Figure 5.1 (c). The 6 mm outflow graft fits pediatric patients from 0-12 months, weighing 2-8 kg [13].
Graft flow in large vessels such as the aorta is often complicated with turbulence, as indicated in Chapter 4. In contrast to adult LVADs, PVADs are often associated with hemolysis and thrombosis, partly due to higher peak inflow into the aorta as a result of smaller graft diameters. This problem may be worse for pulsatile pumps, as $Re$ at peak systole is much higher than $Re$ during continuous support. In this study, the previously validated MILES method based on the FVM (Section 3.2) is used to resolve the large energy-containing structures of the flow, while the SGS effects are incorporated implicitly through numerical dissipation. The second-order TVD scheme (validated in Section 3.1), for discretization of the convective terms in the momentum equations, is used to implicitly satisfy the physically required SGS properties and to stabilize the flow [112].
5.1.3 Boundary Conditions

Three different flow conditions are studied in this work (Table 5.1). In case 1, aortic flow is studied based on the healthy pediatric aortic model with a CO of 1.0 L/min and a HR of 120 beats per minute (BPM) as a no-support baseline. In case 2, 50% of CO goes through the PVAD as an intermediate VAD support. Case 3 is the condition of 100% bypass flow, and the entire pulsatility in the aorta is due to the ejection of the PVAD. For simplicity, only synchronized flow is studied in case 2. The beat rates of the left heart and PVAD in cases 2 and 3 are assumed to be 120 BPM.

Table 5.1: Summary of flow conditions.

<table>
<thead>
<tr>
<th>Case</th>
<th>Beat Rate of PVAD (BPM)</th>
<th>Beat Rate of HR (BPM)</th>
<th>Total Flow (L/min)</th>
<th>Percentage Support (%)</th>
<th>Peak Re at Aortic Inlet</th>
<th>Peak Re at Outflow Cannula</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>N/A</td>
<td>120</td>
<td>1.0</td>
<td>0</td>
<td>2400</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>120</td>
<td>120</td>
<td>1.0</td>
<td>50</td>
<td>1200</td>
<td>2134</td>
</tr>
<tr>
<td>3</td>
<td>120</td>
<td>0</td>
<td>1.0</td>
<td>100</td>
<td>0</td>
<td>4269</td>
</tr>
</tbody>
</table>

At the inlet of the aorta and outflow cannula, plug flow velocity profiles are applied together with scaled waveforms taken from an *in vivo* animal study by Pantolos et al.[46]. A Fourier transform with 15 harmonics is applied to formulate matching flow conditions for the computational simulations. The velocity boundary conditions at the outlet of the descending aorta and the three primary branch vessels on the aortic arch are zero normal velocity gradients. Each outlet is assigned an outflow resistance boundary condition for pressure using Equation 3.21:

\[ p = \chi Q + p_0. \]

The pressure at the aortic outlet, \( p_0 \), is assumed to be 60 mmHg [151] in this study to provide a physiological pressure level in the system. The resistance of each outlet is determined with reference to pulsatile flow in the healthy aortic model. Approximately
40% to 50% of the CO is diverted to the three branch vessels in the pediatric aortic system [167]. In this study, the flow splits among the great vessels, the brachiocephalic (innominate) artery, the left common carotid artery, and the left subclavian artery, are specified proportional to the cross-sectional area of each vessel as 21.41%, 10.58% and 14.76% such that the total split of the branch vessels is 46.75%. Numerical experiments are performed to find the resistance at each outlet needed to maintain the physiologically correct aortic pressure range and flow splits. The same resistances are then used for cases 2 and 3. The aorta and outflow graft are assumed to be rigid, and a no-slip boundary condition is applied at the walls.

5.1.4 Grid Generation

High-quality unstructured grids are generated for the computational geometries using GAMBIT 2.3.16 (Fluent Inc., Lebanon, NH, USA). Three different grids (coarse = ~ 180,000 cells; medium = ~ 580,000 cells; fine = ~ 1,600,000 cells) are constructed for a grid study. Fine cells are used near the walls for all three grid generations to resolve the viscous sub-layer of the turbulent wall flows. The heights of the near-wall spacing are 0.5 mm, 0.1 mm, and 0.05 mm in the coarse, medium, and fine grids, respectively. The interior volumes of the geometries are filled with a combination of tetrahedral and hexahedral cells with nearly isotropic sizes of 1.0 mm, 0.7 mm, and 0.5 mm for the coarse, medium and fine grids, respectively.

5.1.5 Flow Solver and Numerical Algorithm

Numerical simulations are performed using the time-accurate Navier-Stokes solver validated in Chapter 3. The set of the governing equations is discretized into algebraic equations using the FVM described in Section 2.7.5. The time derivative term is discretized using the second-order BD scheme (Section 2.7.3). The convective term in the momentum equation is discretized using the second-order TVD scheme (Section 2.7.2.1).
The accuracy of the TVD scheme has been validated previously in Section 3.1. The linear solvers used in the present work are the GAMG solver for the pressure and the PBICG solver for the velocity. The pressure and velocity terms are coupled using the PISO scheme described in Section 2.7.5.3.

5.1.6 Fluid Properties and Computational Parameters

Blood is modeled as an incompressible Newtonian fluid with a density, \( \rho \), of \( 1.057 \times 10^{-3} \) kg/m\(^3\) and a dynamic viscosity, \( \mu \), of \( 3.7 \times 10^{-3} \) kg/(m \cdot s). The time steps are chosen such that the Courant number through the flow cycle is below 1. The residuals of all the variables are converged to \( 10^{-6} \) at all time steps. Simulations are run in parallel using a PC cluster within the Penn State High Performance Computing Group. Dell PowerEdge 1950 1U Rackmount Servers are used as the compute nodes, each with Dual 3.0 GHz Intel Xeon E5450 (Harpertown) Quad-Core Processors and 32 GB of ECC RAM, and are interconnected with high-speed Silverstorm Infiniband network. Four, sixteen, and sixteen processors are used for the coarse, medium, and fine grids, respectively. The computational times of one cycle for the coarse, medium, and fine grids are roughly 16, 26, and 82 hours, respectively. Convergence is achieved after three cycles for the healthy aortic model and 50% bypass. The maximum time averaged relative error of WSS of the fifth cycle to the fourth cycle over the computational domain is less than 1%. This suggests that the flows for the healthy aortic model and 50% bypass are laminar. Transitional turbulent flow occurs near peak systole for 100% bypass, as indicated by large variance in WSS (over 100%) at peak systole from cycle to cycle. The simulation for 100% bypass starts and runs for three flow cycles to eliminate the initial transient effect. The flow is then sampled for the following ten flow cycles (4-13) to collect turbulent statistics.
5.2 Results

5.2.1 Solution Verification

A systematic grid study is performed using the three different grids to verify the numerical solutions. Figure 5.2 shows the effect of grid on the volume flow rate and pressure at the inlet and outlets of the healthy aortic model. No significant difference is found among the three grids. The maximum time averaged relative error of volumetric flow rate and pressure through the flow cycle between the coarse grid and the fine grid is less than 1%. A large pressure pulse is found at early diastole at the aortic inlet due to a lack of compliance in the aortic model. The systolic and diastolic pressures in the aorta are 103/60 mmHg, measured from the aortic inlet, which are well in the pressure range of one-year-old pediatric patients [151]. Minor back flow is observed in each branch vessel during late systole. It is clear that the coarse grid gives reasonable integral results when compared to the medium and fine grids.

Figure 5.3 shows the effect of grid on three components of the time-varying WSS in six different locations of the healthy aortic model. WSS is defined in Equation 2.54. It is clear that WSS is more sensitive to the grid than integral variables such as volumetric flow rate, as expected. As shown in Figure 5.3, the coarse grid predicts quite different results in locations 3 and 4 when compared to the medium and fine grids. Good agreement is found between the medium and fine grids in all locations except in location 4, as shown in Figure 5.3. The maximum time-averaged relative error of WSS through the flow cycle between the medium and fine grids is 5%, except in location 4, where the maximum time-averaged relative error is 25% for the z-component of WSS. Because of reasonable agreement and due to limited computational resources, we use the medium grid for the simulations.
Figure 5.2: Effect of grid on the pressure and flow waveforms at each boundary. Note that the three different grids are differentiated by line type.
Figure 5.3: Effect of grid on WSS in six locations. Note that the three different grids are differentiated by line type.
5.2.2 Flow Splitting

Figure 5.4 represents the volumetric flow rate and pressure at the boundaries for 50% bypass and 100% bypass. The results presented from here on for 100% bypass are averaged for ten flow cycles. Less retrograde flow is found at early diastole in the three branch vessels on the aortic arch for both cases when compared to the healthy pediatric aortic flow, as shown in Figure 5.2. Table 5.2 shows a summary of flow splitting in the three branch vessels for all three cases. For 50% bypass (case 2), the flow rate in the brachiocephalic artery is increased by roughly 1% of CO while the flow rates for the left common carotid artery and the left subclavian artery drop slightly. Overall, a slight increase is found in the total flow division in the branch vessels. With complete PVAD support (100% bypass), the total flow division in the branch vessels is increased by approximately 6%, as shown in Table 5.2. The Womersley number, \( Wo \), based on the radius of the aortic inlet for all three cases is 11.
Figure 5.4: Pressure and flow waveforms at each boundary for 50% and 100% bypass. Note that the pressure and flow waveforms are differentiated by line type.
Table 5.2: Flow splitting in the great vessels. Flow percentage is the ratio between the flow rate and the CO.

<table>
<thead>
<tr>
<th>Flow Conditions</th>
<th>Brachiocephalic Artery (%)</th>
<th>Left Common Carotid Artery (%)</th>
<th>Left Subclavian Artery (%)</th>
<th>Total Flow Division in the Great Vessels (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>21.41</td>
<td>10.58</td>
<td>14.76</td>
<td>46.75</td>
</tr>
<tr>
<td>2</td>
<td>22.48</td>
<td>10.33</td>
<td>14.14</td>
<td>46.95</td>
</tr>
<tr>
<td>3</td>
<td>25.02</td>
<td>12.09</td>
<td>15.61</td>
<td>52.72</td>
</tr>
</tbody>
</table>

5.2.3 Velocity Contours

The velocity contours are presented in three different orientations for multiple locations, as shown in Figure 5.5. Figure 5.6 shows the velocity magnitude contour for the healthy pediatric aortic model at three time points of the cardiac cycle. At peak systole, the flow profile is skewed toward the inner wall in the ascending aorta and toward the outer wall downstream. The maximum velocity occurs in the descending aorta, due to the tapering of the aorta, as shown in Figure 5.6 (a). During mid-deceleration, the flow starts to swirl and to separate from the inner wall of the aortic arch and the proximal wall of the three branch vessels, as seen in Figure 5.6 (b). In late systole, a slight retrograde flow is found in each branch vessel, and the flow is skewed toward the proximal wall of each branch vessel, as presented in Figure 5.6 (c).
Figure 5.5: Planes used in this study for velocity contours.
Figure 5.6: Velocity magnitude contour (unit: m/s) for the healthy pediatric aorta.
With graft inflow, the flow in the aorta is highly disturbed. Figure 5.7 shows the flow for 50% bypass. A jet is introduced into the aorta with a maximum velocity of 2.13 m/s at peak systole, as shown in Figure 5.7 (a). A region of flow separation is found downstream of the toe during systole, as presented in Figures 5.7 (a) and (b). The flow upstream of the site is uniform and not disturbed. Figure 5.8 shows the averaged flow contours for ten cycles for 100% bypass. A much higher jet flow is found at peak systole, with a maximum velocity of 3.6 m/s, when compared to 50% bypass. This jet impinges on the aortic floor (posterior wall), as shown in Figure 5.7 (a). The jet is then diverted into the brachiocephalic artery, sparing the left common carotid and left subclavian arteries. A larger region of flow separation is found downstream of the toe when compared to 50% bypass. This flow separation region remains through systole. As expected, a region of stagnant flow is found upstream of the heel during the entire flow cycle.
Figure 5.7: Velocity magnitude contour (unit: m/s) for 50% bypass.
5.2.4 Principal Reynolds Stress

Turbulence level for 100% bypass is examined by calculating the principal Reynolds stresses. The three-component principal normal Reynolds stresses are calculated as the eigenvalues of the Reynolds stress tensor using Equation 2.56. The maximum shear stresses are calculated using Equation 2.60. Figure 5.9 shows the maximum principal
Reynolds stresses at peak systole. High Reynolds stresses are found exclusively on the floor near the jet impingement region, with a maximum normal stress of 7081 dynes/cm\(^2\) (95% confidence interval of ± 4566 dynes/cm\(^2\), sample size = 10) and a maximum shear stress of 3439 dynes/cm\(^2\) (95% confidence interval of ± 2257 dynes/cm\(^2\), sample size = 10). The Reynolds stress in the graft is negligible at peak systole, as the flow in the graft stays laminar throughout the flow cycle. The peak Reynolds number based on the graft diameter is 4269.

![Figure 5.9](image)

Figure 5.9: Magnitude contour of Reynolds stress (unit: dynes/cm\(^2\)) at peak systole for 100 % bypass: (a) maximum principal normal stress; (b) maximum shear stress. Note that the flow waveform shown above is the cannula outflow.

5.2.5 Wall Shear Stress Distributions

Wall shear stress distributions in the aorta vary dramatically depending on the percentage of the graft inflow. The distribution of WSS at peak systole is presented in Figure
5.10. The WSS for 100% bypass represents the average of ten cycles. For the healthy aorta (Figure 5.10 (a)), the WSS maxima occur on the outer curvature of the aortic arch between the branch entrances, the distal wall of the branch vessels, and the aortic arch distal to the left subclavian artery. The maximum WSS for the healthy aorta at peak systole is 650 dynes/cm². With 50% PVAD support (Figure 5.10 (b)), the WSS distribution is increased on the aortic wall downstream of the graft site when compared to that of the healthy aorta. High WSS regions are found at the outer wall of the aortic arch between the brachiocephalic artery and left common carotid artery and near the suture line of the graft, with WSS of 950 dynes/cm² and 1800 dynes/cm², respectively. The WSS on the ascending aorta upstream of the graft site, however, is decreased due to the reduced flow from the aortic root. At 100% PVAD support (Figure 5.10 (c)), a significant increase in the WSS distribution is observed on the posterior wall of the aorta and the distal walls of the branch vessels when compared to the healthy aorta and 50% bypass, mainly due to the jet impingement. The maximum WSS occurs on the outer wall of the aortic arch between the brachiocephalic artery and left common carotid artery and near the suture line of the graft with a value of 3000 dynes/cm². Low WSS is found on the aortic floor upstream of the graft site, which suggests a stagnant flow region near the aortic root. Low WSS near the graft hood indicates flow separation at peak systole. Two flow stagnation points are found in the jet impingement region of the aortic floor by identifying the lowest WSS areas, as indicated in Figure 5.10 (c). These two stagnation points are further confirmed by identifying the highest pressure areas on the aortic floor (posterior wall), as shown in Figure 5.11.
Figure 5.10: WSS magnitude (unit: dynes/cm²) contour at peak systole: (a) healthy pediatric aorta, (b) 50% bypass, (c) 100% bypass.
Figure 5.11: Pressure contour (unit: mmHg) at peak systole for 100% bypass. Note that the flow waveform shown above is the cannula outflow.

Figure 5.12 shows the contour of the time-averaged WSS for all three cases. The time-averaged WSS is calculated using Equation 2.52. High time-averaged WSS regions are found to coincide with high WSS regions at peak systole, as shown in Figures 5.10 and 5.12. Specifically, for the healthy aorta, high time-averaged WSS regions are located at the outer wall of the aortic arch between the branch entrances, the distal wall of the branch vessels and the aortic arch distal to the left subclavian artery. For 50% bypass, higher time-averaged WSS distributions are found on the aortic wall downstream of the graft site while lower distributions are found on the wall upstream of the site when compared to the healthy aorta. The maximum time-averaged WSS occurs on the suture line with a value of 438 dynes/cm². For 100% bypass, high time-averaged WSS regions are found on the suture-line region, the posterior wall downstream of the graft site, the outer wall of the aortic arch between the branch entrances, and the distal walls of the branch vessels. The maximum time-averaged WSS (591 dynes/cm²) is on the suture line. Low time-averaged WSS regions are found on the aortic wall upstream of the site, the
anterior aortic wall downstream of the toe, the graft hood, and the inner wall of the graft near the suture line. The two flow stagnation points are also found in the time-averaged WSS contour, as indicated in Figure 5.12 (c).

Figure 5.12: Contour of time-averaged WSS (unit: dynes/cm²): (a) healthy pediatric aorta, (b) 50% bypass, (c) 100% bypass.
Figure 5.13 compares the OSI distribution for the three different flow conditions. OSI, a measure of directional changes in WSS over the cardiac cycle, is defined in Equation 2.53. High OSI regions are found on the outer wall of the ascending aorta, the proximal walls of the branch vessels on the aortic arch, and the inner wall of the aortic arch for the healthy aorta, indicating flow separation and retrograde flow in these regions. The maximum OSI of 0.47 occurs on the proximal walls of the branch vessels near the entrances, as shown in Figure 5.13 (a). For 50% bypass, high OSI is located at the aortic floor near the graft site, the anterior wall downstream of the graft toe, and the outer wall upstream of the site. High OSI is also located at the graft hood and the inner wall of the graft close to the suture line. The highest OSI regions are located at the aortic floor near the graft site and the inner wall of the graft close to the suture line, with a maximum OSI of 0.49. The OSI distribution of 100% bypass is different from that of 50% bypass in three major locations. First, low OSI is found on the aortic floor opposite the suture line for 100% bypass, except at the two flow stagnation points, as shown in the posterior view of Figure 5.13 (c). This suggests there is no significant flow reversal near the jet impingement region through the flow cycle except at the stagnation points. The high OSI at the stagnation points indicates that these areas are moving through the cycle. The second region is the aortic wall upstream of the graft site. In 100% bypass, high OSI is found on the inner wall while low OSI is found on the outer wall. This is exactly the opposite of the OSI distribution in 50% bypass. The third region is the hood of the graft. OSI is higher at the hood of the graft for 100% bypass when compared to 50% bypass, as a result of increased flow separation. The highest OSI regions are located at the anterior wall downstream of the graft toe, with a maximum value of 0.47.
Figure 5.13: Contour of OSI: (a) healthy pediatric aorta, (b) 50% bypass, (c) 100% bypass.
5.3 Discussion

Hemodynamics within the end-to-side proximal anastomosis of a PVAD to a patient-specific pediatric aortic model was simulated using a temporally and spatially second-order accurate flow solver. A systematic grid study was performed to verify the numerical solutions. The resulting flow field, WSS, and OSI were obtained and compared under three different flow conditions (healthy aorta, 50% support, and 100% support).

5.3.1 Flow Field

The numerical results for the healthy pediatric aorta have revealed a flow field structurally similar to aortic profiles previously reported in a number of experimental [167, 164] and numerical studies [47, 156]. During peak systole and mid-deceleration, the flow skewness in the ascending aorta is along the inner aortic wall while the skewness in the descending aorta is along the outer wall. The velocity profiles within the branches throughout systole are skewed toward the distal walls during positive inflow (Figures 5.6 (a) and (b)) whereas the proximal walls during retrograde inflow (Figure 5.6 (c)).

PVAD support produces a jet into the aorta and causes a jet impingement on the aortic floor near the graft site. The jet is then mainly directed into the brachiocephalic artery. This might increase the perioperative right-sided stroke rate because most emboli caused by the aortic graft flow into the brachiocephalic artery [161, 162]. This is qualitatively consistent with the findings reported by Yang et al.[155] in a proximal anastomotic model based on a planar adult aortic model under steady inflow conditions (Section 4.2.2). Turbulence occurs during 100% PVAD support at peak systole near the jet impingement on the aortic floor. The maximum Reynolds stress is 7081 dynes/cm² (95% confidence interval of ±4566 dynes/cm², sample size n = 10). These stress levels may be high enough to cause hemolysis and platelet activation [76, 168]. As a result, early graft failure may occur as a result of thrombosis.
A proximal PVAD anastomosis on the ascending aorta is found to cause an increase of total flow splitting in the great vessels on the aortic arch. As the percentage of the support increases, the total splitting increases accordingly, as a result of increased pressure at each branch outlet (Figure 5.4). Graft locations have a significant effect on the flow splitting, as demonstrated by Litwak et al. [41] and Yang et al. [155] (Section 4.2.6) in their acute in vivo animal study and numerical simulation, respectively. They both found that a distal anastomosis led to more flow splitting in the branch vessels than did a proximal anastomosis during both continuous and pulsatile VAD support. It should be noted that all these results only describe the acute effect of VAD support. Chronically, vessel downstream resistances may adjust to maintain homeostasis. The long-term effect of VAD support on the flow splitting will have to be examined carefully.

5.3.2 Wall Shear Stress and Oscillatory Shear Index

The WSS and OSI distributions for the healthy pediatric aorta are qualitatively similar to the results seen by previous aortic studies [47, 156, 169]. The low time-averaged WSS and high OSI on the proximal walls of the branch vessels and the inner wall of the aortic arch suggest flow separation and retrograde flow in these regions, consistent with the findings by Shahcheraghi et al. [156] and Huo et al. [169]. The maximum WSS at peak systole is 650 dynes/cm², which is less than half of the value (1480 dynes/cm²) reported by Pekkan et al. [47] in their neonatal aortic model. However, the peak $Re$ for Pekkan et al.’s model is nearly three times greater than that for the present model. The maximum OSI (0.47) occurs on the proximal walls of the brachiocephalic artery and the left common carotid artery, which is close to the maximum OSI (∼0.45) reported by Huo et al. in their mouse model [169].

Complex WSS and OSI occur near the graft site during PVAD support. Key observations include the presence of flow separation in the anterior aorta downstream of the toe and graft hood, and the presence of two stagnation points along the aortic floor.
The two separation zones become larger as the support increases. The two stagnation points, characterized by low WSS and high pressure, are found on the aortic floor only during 100% PVAD support. High OSI values are found around the stagnation points, suggesting that they oscillate throughout the cardiac cycle. Similar flow patterns have been observed in arterial graft flows [17, 169, 170].

5.3.3 Potential Sites for Intimal Hyperplasia

IH is one of the major causes for long-term graft failure. PVADs are currently implanted only as a temporary bridge to recovery or transplant, and not as a life-long therapy. Within the anticipated time of use, it is unlikely that IH is going to manifest in any significant way. However, VAD experiences in pediatric patients are far from mature and IH may become an issue for possible long-term pediatric VAD patients in the future. Therefore, a special effort is made to discuss potential implications for IH in the pediatric aortic graft flow. WSS is an important factor for the development of IH. Several studies [17, 19, 171] have suggested that low time averaged WSS and high OSI may be good combined predictors for IH by elevating Endothelin-1 (a smooth muscle cell (SMC) proliferator) mRNA release and inhibiting ecNOS (a SMC inhibitor) mRNA release [172]. In this study, areas of time-averaged WSS lower than 5 dynes/cm² [173, 174] and an OSI higher than 0.2 [175] are considered ‘undesirable’ and are shown in black in Figure 5.14. For the healthy aorta, IH prone regions are mainly located at the proximal walls of the branch vessels near the entrances and the inner wall of the aortic arch, as shown in Figure 5.14 (a). For 50% bypass, IH prone regions are located at the aortic floor near the graft site, the anterior wall downstream of the graft toe and upstream of the heel, the inner wall of the graft near the suture line, the proximal walls of the branch vessels near the entrance, and the inner wall of the aortic arch. IH-prone regions for 100% bypass are different from those for 50% bypass mainly in four key locations. First, a larger IH prone region is found on the anterior wall downstream of the graft toe when compared
to that for 50% bypass. Second, a large IH prone region is found on the inner wall of the ascending aorta near the aortic root for 100% bypass, due to stagnant flow. Third, no significant IH prone regions are found on the branch vessels for 100% bypass. Last, an IH prone area on the graft is located on the graft hood for 100% bypass, while it is located on the inner wall of the graft near the suture line for 50% bypass. Interestingly, the two stagnation points are not identified as IH prone regions because the time-averaged WSS in these regions is much higher than 5 dynes/cm². The suture-line region is also always associated with IH formation, caused by surgical injury, material mismatch [30, 176], or an abnormally high WSSG [21]. These “hot spots” are generally consistent with the findings in end-to-side arterial graft flows [17].
5.3.4 Limitations

This is the first study of transitional turbulent flow under pulsatile conditions in a patient-specific pediatric aortic graft model, with all primary branches, based on detailed anatom-
ical data. The primary anatomical dimensions of the pediatric aortic model are validated with the existing morphological measurements by Machii and Becker [142]. According to Machii and Becker, the external diameters and lengths of different segments of the pediatric aorta vary significantly from subject to subject. Furthermore, the reconstruction of the pediatric aorta based on a limited number of MRI slices may preclude an accurate assessment of the pediatric aortic geometry, especially near some out-of-plane curvatures. Despite these uncertainties, our results (velocity, WSS, OSI and IH-prone regions) agree with the major findings from previous studies [17, 47, 156, 169, 170]. This suggests that our data could provide general insight into the flow in the pediatric aorta with or without PVAD support. A significant advantage of numerical simulations over experimental work is that after the numerical approach is clinically justified, it can be easily applied to different subject-specific simulations. It should be kept in mind that a normal pediatric aortic anatomy is used in the present study. This is not the case for many children requiring PVAD support. The effect of the abnormality of the patient aorta on the flow needs to be examined.

One major limitation of the present study is that the MILES solver is only validated in the fully-developed turbulent channel flow under the steady inflow condition (Section 3.2). A more complete validation is to compare it to existing turbulent data under pulsatile flow conditions, as pulsatile turbulent flows are often associated with complex relaminarization. Furthermore, only ten cycles are used to calculate turbulence statistics due to limited computational resources, which are far from sufficient to predict accurate turbulent statistics. However, this work represents a significant step towards the investigation of transitional turbulent nature of the aortic graft flow both numerically and experimentally.
5.4 Conclusions

The results of this study show that a pediatric aortic graft inflow alters the flow in the aorta significantly. More flow is diverted into the great vessels with PVAD support. Turbulence is observed near the jet impingement region for total PVAD support at peak systole. The maximum Reynolds stress is over 7000 dynes/cm$^2$. This value may be sufficient to cause hemolysis and platelet activation. As a consequence, early graft failure may occur. Regions most prone to IH are identified using time-averaged WSS and OSI as two predictors. Flow separation, retrograde flow and flow stagnation points vary greatly depending on the percentage of PVAD support. In general, IH prone regions are located at the aortic floor opposite to the suture line, the anterior aortic wall downstream of the graft toe, the entire aortic wall upstream of the graft heel, the graft hood and the inner wall of the graft near the suture line. The suture-line region is also vulnerable to IH formation due to high WSSG.
Chapter 6

The Effect of Pumping Mode

Chapters 4 and 5 mainly focus on blood flow in the end-to-side anastomoses of an adult LVAD and a pulsatile PVAD, respectively. The impact of pumping mode (continuous versus pulsatile) associated with ventricular assist devices (VADs) on the aortic hemodynamics is not compared. Hence, we compare not only the hemodynamic parameters indicative of pulsatility but also the local flow fields in the aorta and the great vessels originating from the aortic arch in this chapter [177].

6.1 Methods

6.1.1 Geometrical Model

A physiologically correct aortic model is important for the study of an aortic graft flow. In this study, the pediatric patient-specific anastomotic model described in Section 5.1.1 is used for simulation, as shown in Figure 5.1. Only the proximal configuration is considered, with a 6 mm PVAD outflow graft sutured on the ascending aorta. The pediatric anastomotic model is used instead of an adult model because, in general, pediatric anastomoses cause more turbulence, and therefore, more blood damage.
6.1.2 Mathematical Model

Flow is modeled using the time-dependent Navier-Stokes equation (Equations 2.1 and 2.2). The MILES method is used to resolve the large scales of the resulting turbulent flow, while the effect of the SGS motion is incorporated through the numerical dissipation of the previously validated second-order TVD scheme (Section 3.1). This method has been validated in Section 3.2. To model hemolysis in the flow, we introduce an additional linear advection-reaction transport equation (Equation 2.38) as:

\[
\frac{\partial H_l}{\partial t} + U \cdot \nabla H_l = R_l(1 - H_l).
\]

The detailed derivation of the transport equation has been presented in Section 2.5 and therefore, is not repeated here.

6.1.3 Boundary Conditions

Five different flow conditions are studied in this work (Table 6.1). Case 1 represents a natural aortic flow condition with a CO of 1.0 L/min and a heart rate of 120 BPM. It serves as a reference flow without PVAD support. Cases 2 and 3 are intermediate VAD support conditions with 50% of CO going through the PVAD. Case 2 represents pulsatile support, while case 3 represents continuous support. Cases 4 and 5 represent 100% PVAD support in the PM and CM, respectively. For simplicity, only synchronized flow is studied. The beat rates of the left heart and PVAD are taken to be 120 BPM for all pulsatile conditions.
Table 6.1: Summary of flow conditions.

<table>
<thead>
<tr>
<th>Case</th>
<th>Percentage Support [%]</th>
<th>PVAD Mode</th>
<th>Beat Rate of PVAD (BPM)</th>
<th>HB (BPM)</th>
<th>Total Flow (L/min)</th>
<th>Peak Re at Aortic Inlet</th>
<th>Peak Re at Outflow Cannula</th>
<th>Wo based on the Aortic Inlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>N/A</td>
<td>N/A</td>
<td>120</td>
<td>1.0</td>
<td>2400</td>
<td>0</td>
<td>11</td>
</tr>
<tr>
<td>2</td>
<td>50</td>
<td>Pulsatile</td>
<td>120</td>
<td>120</td>
<td>1.0</td>
<td>1200</td>
<td>2134</td>
<td>11</td>
</tr>
<tr>
<td>3</td>
<td>50</td>
<td>Continuous</td>
<td>N/A</td>
<td>120</td>
<td>1.0</td>
<td>600</td>
<td>506</td>
<td>11</td>
</tr>
<tr>
<td>4</td>
<td>100</td>
<td>Pulsatile</td>
<td>120</td>
<td>0</td>
<td>1.0</td>
<td>0</td>
<td>4209</td>
<td>11</td>
</tr>
<tr>
<td>5</td>
<td>100</td>
<td>Continuous</td>
<td>N/A</td>
<td>0</td>
<td>1.0</td>
<td>0</td>
<td>1011</td>
<td>0</td>
</tr>
</tbody>
</table>

The flow waveforms at the inlets of the aorta and outflow graft for cases 1, 2 and 4 are scaled from an *in vivo* animal study by Pantolos et al.[46]. For case 3 (50% continuous support), a scaled waveform based on the measurement by Pantolos et al. is applied to the aortic inlet with a steady plug inflow applied at the cannula inlet. For case 5 (100% continuous support), a steady plug inflow profile is applied at the cannula inlet, while zero velocity is applied at the aortic inlet. A zero normal gradient is applied at each branch exit as a velocity boundary condition, with an outflow resistance boundary condition applied for pressure (Equation 3.21). In this study, \( p_0 \) is taken as 60 mmHg to maintain a physiological pressure level in the system [151]. The flow splits in the great vessels are set the same as those used in Chapter 5 (Section 5.1.3). We assume that the resistance of each vessel maintains its value throughout PVAD support. The aorta and outflow graft are assumed to be rigid, and a no-slip boundary condition is applied at the walls. The boundary conditions for the hemolysis fraction \( H_i \) are zero at the inlets and walls, with zero normal gradients at the outlets for all five cases.
6.1.4 Grid Generation

The results of a systematic grid study were detailed in Chapter 5 on the same pediatric aorta model used in this chapter. The medium grid (approximately 580,000 cells), which provides sufficient resolution (Section 5.1.4), is used here. The grid is generated using GAMBIT 2.3.16 (Fluent Inc., Lebanon, NH, USA). A set of fine cells is used near the walls with a height of the near-wall spacing of 0.05 mm. A hybrid of tetrahedral, hexahedral, and prism cells is used in the interior volumes of the geometries with a nearly isotropic size of 0.5 mm.

6.1.5 Flow Solver and Numerical Algorithms

The Navier-Stokes and hemolysis transport equations are discretized into algebraic ones using the FVM (Section 2.7) and solved separately. The momentum and continuity equations are solved first. The hemolysis transport equation is solved based on the resolved flow field. The numerical algorithms used for the momentum equation are the same as those described in Section 5.1.5 and therefore, are not presented here. The second-order accurate BD scheme is used to discretize the time derivative term for the hemolysis transport equations. The convective term of the hemolysis transport equation is discretized using the first-order UD scheme. Only the first-order scheme is used because of a numerical instability for higher-order schemes. The PBICG solver is used for the discretized hemolysis transport equation.

6.1.6 Fluid Properties and Computational Parameters

Blood is assumed to be an incompressible Newtonian fluid with a density, \( \rho \), of \( 1.057 \times 10^{-3} \) kg/m\(^3\) and a dynamic viscosity, \( \mu \), of \( 3.7 \times 10^{-3} \) kg/(m \( \cdot \) s). The time steps are chosen for each case such that the Courant number through the flow cycle is less than 1. The residuals of all the variables are converged to \( 10^{-6} \) at all time steps. Simulations are
run in parallel with sixteen processors using the PC cluster described in Section 5.1.6. Three cycles are required to achieve convergence of the flow field for cases 1 (healthy aortic flow), 2 (50% pulsatile support), and 3 (50% continuous support), which is based on the maximum time-averaged relative error of WSS of the fifth cycle to the fourth cycle being less than 1%. This indicates that the flows for cases 1, 2 and 3 are laminar. Turbulence occurs for cases 4 and 5. For case 4 (100% pulsatile support), the simulation first runs three cycles to eliminate initial effect, and the flow data are then sampled for ten flow cycles to collect turbulent statistics. The computation takes roughly a day per flow cycle. For case 5 (100% continuous support), the flow data are sampled at every time step for roughly five flow-through times to collect flow statistics after the flow is statistically stable. The flow-through time, 0.2 s, is the time taken by a fluid particle to traverse from the aortic inlet along the aortic arch to the aortic outlet with mean graft inlet velocity. The flow results presented from here on for cases 4 and 5 are averaged for ten flow cycles and five flow-through times, respectively. The hemolysis fractions $H_l$ for cases 1, 2, and 3 are calculated based on the flow fields of the fourth flow cycle, while $H_l$ for cases 4 and 5 are calculated based on the mean flow fields.

6.2 Results

6.2.1 Pulsatile Hemodynamic Performance

Figures 6.1 and 6.2 give the volumetric flow rate and pressure waveforms at the boundaries for all five cases. A slight retrograde flow is found at early diastole in the great vessels for the healthy aortic flow (case 1), which is reduced with the PVAD support (cases 2-5), as shown in Figure 6.1. The large pressure pulse found at early diastole at the aortic inlet for case 1, as shown in Figure 6.2, is a result of the lack of systematic compliance.
Figure 6.1: Flow waveforms at the boundaries for all five cases.
Figure 6.2: Pressure waveforms at the boundaries for all five cases.

The calculated values of EEP, TAP, and SHE (defined in Section 2.6.1), as shown in Table 6.2, are functions of flow condition. These parameters are defined in Equations
For the healthy pediatric aorta (case 1), 104%, 103%, 93% and 20% of the inlet SHE are delivered to the brachiocephalic artery, left common carotid artery, left subclavian artery, and aortic outlet, respectively. In comparison, vascular pulsatility is dampened during the left ventricular assistance (50% support and 100% support), as suggested by the SHE values in Table 6.2.

Table 6.2: Comparison of pulsatile hemodynamic performance parameters between the pulsatile and continuous PVAD support.

<table>
<thead>
<tr>
<th>Case</th>
<th>Aorta Inlet</th>
<th>Outflow Graft</th>
<th>Brachiocephalic Artery</th>
<th>Left Common Carotid Artery</th>
<th>Left Subclavian Artery</th>
<th>Aortic Outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>84.23</td>
<td>N/A</td>
<td>83.85</td>
<td>83.71</td>
<td>81.44</td>
<td>65.22</td>
</tr>
<tr>
<td>2</td>
<td>81.40</td>
<td>N/A</td>
<td>83.12</td>
<td>80.07</td>
<td>78.93</td>
<td>77.35</td>
</tr>
<tr>
<td>3</td>
<td>72.26</td>
<td>65.71</td>
<td>68.79</td>
<td>68.62</td>
<td>67.99</td>
<td>62.51</td>
</tr>
<tr>
<td>4</td>
<td>N/A</td>
<td>90.79</td>
<td>81.53</td>
<td>80.38</td>
<td>77.63</td>
<td>64.07</td>
</tr>
<tr>
<td>5</td>
<td>N/A</td>
<td>66.40</td>
<td>64.58</td>
<td>64.37</td>
<td>64.24</td>
<td>61.82</td>
</tr>
<tr>
<td>1</td>
<td>66.17</td>
<td>N/A</td>
<td>65.07</td>
<td>65.01</td>
<td>64.65</td>
<td>61.68</td>
</tr>
<tr>
<td>2</td>
<td>66.08</td>
<td>67.26</td>
<td>65.31</td>
<td>64.89</td>
<td>64.46</td>
<td>61.66</td>
</tr>
<tr>
<td>3</td>
<td>65.05</td>
<td>64.71</td>
<td>64.53</td>
<td>64.38</td>
<td>64.16</td>
<td>61.84</td>
</tr>
<tr>
<td>4</td>
<td>65.71</td>
<td>69.16</td>
<td>65.89</td>
<td>65.64</td>
<td>64.89</td>
<td>61.47</td>
</tr>
<tr>
<td>5</td>
<td>64.58</td>
<td>66.40</td>
<td>64.58</td>
<td>64.37</td>
<td>64.24</td>
<td>61.82</td>
</tr>
<tr>
<td>1</td>
<td>2.41 × 10^4</td>
<td>N/A</td>
<td>2.50 × 10^4</td>
<td>2.49 × 10^4</td>
<td>2.24 × 10^4</td>
<td>4.72 × 10^3</td>
</tr>
<tr>
<td>2</td>
<td>2.04 × 10^4</td>
<td>2.11 × 10^4</td>
<td>1.97 × 10^4</td>
<td>1.87 × 10^4</td>
<td>1.72 × 10^4</td>
<td>3.69 × 10^3</td>
</tr>
<tr>
<td>3</td>
<td>9.61 × 10^3</td>
<td>0</td>
<td>5.68 × 10^3</td>
<td>5.65 × 10^3</td>
<td>5.10 × 10^3</td>
<td>8.96 × 10^2</td>
</tr>
<tr>
<td>4</td>
<td>N/A</td>
<td>2.88 × 10^4</td>
<td>2.08 × 10^4</td>
<td>1.96 × 10^4</td>
<td>1.70 × 10^4</td>
<td>3.47 × 10^3</td>
</tr>
<tr>
<td>5</td>
<td>N/A</td>
<td>0</td>
<td>0.07</td>
<td>0.25</td>
<td>0.22</td>
<td>0.01</td>
</tr>
</tbody>
</table>

A comparison of pulsatility between the PVAD operating in the CM and PM, as
indicated by percentage change in SHE from case 1 for the great vessels and the aortic outlet, is given in Figure 6.3. Striking differences are found in pulsatility between the pumping modes. In the PM, roughly 20% reduction in pulsatility is found at each outlet for both the 50% and 100% support. In the CM, however, a roughly 80% reduction in pulsatility is found at each outlet for the 50% support, while for the 100% support no pulsatility is found in the flow, as expected.

Figure 6.3: Comparison of percentage change in SHE from the healthy aortic flow (case 1) for the 50% and 100% support in the PM and CM.

6.2.2 Flow Splitting

The effect of the PVAD support on flow splitting in the great vessels is summarized in Table 6.3. It is found that the pulsatile PVAD support (cases 2 and 4) increases the flow in the great vessels when compared to the unassisted condition (case 1), while the
continuous support (cases 3 and 5) decreases the flow. Specifically, as the percentage of the PVAD support in the PM increases, the flow in the great vessels increases significantly, with a maximum increase of 12.77% from the healthy aorta flow for 100% support (case 4). In contrast, the increase of the PVAD support has only slight impact on the flow splitting when the PVAD operates in the CM. The 50% (case 3) and 100% support (case 5) in the CM give a total flow reduction in the three branch vessels of 11.11% and 9.8%, respectively.

Table 6.3: Flow splitting in the great vessels. Flow percentage is the ratio between the flow rate and the CO.

<table>
<thead>
<tr>
<th>Flow Conditions</th>
<th>Brachiocephalic Artery (%)</th>
<th>Left Common Carotid Artery (%)</th>
<th>Left Subclavian Artery (%)</th>
<th>Total Flow Division in the Great Vessels (%)</th>
<th>Total Flow Percentage Change in the Great Vessels from Case I (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>21.41</td>
<td>10.58</td>
<td>14.76</td>
<td>46.75</td>
<td>0</td>
</tr>
<tr>
<td>II</td>
<td>22.48</td>
<td>10.33</td>
<td>14.14</td>
<td>46.95</td>
<td>0.43</td>
</tr>
<tr>
<td>III</td>
<td>19.13</td>
<td>9.25</td>
<td>13.20</td>
<td>41.58</td>
<td>-11.11</td>
</tr>
<tr>
<td>IV</td>
<td>25.02</td>
<td>12.09</td>
<td>15.61</td>
<td>52.72</td>
<td>12.77</td>
</tr>
<tr>
<td>V</td>
<td>29.42</td>
<td>9.26</td>
<td>13.49</td>
<td>42.17</td>
<td>-9.80</td>
</tr>
</tbody>
</table>

6.2.3 Velocity Contours

Figures 6.4 and 6.5 presents the velocity contours for cases 3 and 5, respectively. The velocity contours for cases 1, 2, and 4 have been shown in Figure 5.6, 5.7, and 5.8 and therefore, are not presented here. Note the velocity for cases 3 and 5 are averaged for ten flow cycles and five flow-through times, respectively. For the healthy aortic flow (case 1), the flow is skewed first towards the inner wall at the aortic arch entrance and then towards the outer wall as the flow moves downstream. Interestingly, the flow reaches its maximum velocity in the descending aorta, as a result of the aortic tapering, as shown
in Figure 5.6. The flow is highly altered during the PVAD support. The 50% PVAD support in the PM (case 2) leads to a jet into the aorta, with a maximum velocity of 2.13 m/s, as shown in Figure 5.7. The flow for the 50% support in the CM (case 3) is relatively uniform in the aorta, with a maximum velocity of 1.05 m/s occurring in the descending aorta at peak systole, as indicated in Figure 6.4 (a). At late systole (early diastole), a jet appears in the aorta and stays throughout the entire diastole due to the constant flow from the PVAD, as shown in Figure 6.4 (c). The 100% support causes a high-velocity jet to impinge on the aortic floor (posterior wall) for both the PM (case 4) and CM (case 5), as shown in Figures 5.8 and 6.5, respectively. The jet is then diverted for the most part into the brachiocephalic artery. The maximum velocities for the pulsatile and continuous support are 3.6 m/s and 0.95 m/s, respectively. A large flow separation is found downstream of the graft toe for cases 4 and 5, as indicated in Figures 5.8 and 6.5.
Figure 6.4: Velocity magnitude contour (unit: m/s) for case 3.
6.2.4 Principal Reynolds Stress

To compare the turbulence level between the PM (case 4) and CM (case 5) during 100% PVAD support, the principal values of normal Reynolds stresses are acquired as the eigenvalues of Reynolds stress tensor (Section 2.6.3). The principal normal Reynolds stresses for case 4, calculated at peak systole, are based on ten flow cycles, while the stresses for case 5 are based on five flow-through times. The peak Reynolds number based on the graft diameter for the PM (case 4) is 4269, while the Reynolds number for the CM (case 5) is 1011. Figures 6.6 (a) and (b) show the maximum principal normal Reynolds stress fields for cases 4 and 5 in the plane shown in Figure 5.5 (c). The high-velocity jets from the cannula cause high Reynolds stresses exclusively near the jet impingement on the posterior wall, for both cases. The Reynolds stresses for case 4 are much higher than for case 5 due to its higher Reynolds number. The maximum principal normal Reynolds stresses for cases 4 and 5 are 7081 dynes/cm² (95% confidence interval of ± 4566 dynes/cm², sample size = 10) and 249 dynes/cm² (95% confidence interval of ± 2 dynes/cm², sample size = 100,000), respectively. The large confidence interval for case 4 is a result of its relative small sample size.
6.2.5 Hemolysis

Table 6.4 presents the MIH for all five cases. MIH is defined in Equation 2.46. The MIH for the healthy aortic flow (case 1) is 17.75. The PVAD support in the PM causes an increase in hemolysis, with MIH values of 1.18 and 22.4 times of the physiological value (MIH for case 1) for the 50% and 100% support, respectively. Continuous support, however, causes less hemolysis, with a decrease of MIH by 72% and 61% for the 50% and 100% support, respectively. This is due to lower velocities and corresponding shears for the continuous support when compared to that for the healthy aortic flow, as shown in Figures 5.6, 6.4 and 6.5.

Figures 6.7 (a) and (b) show the distribution of hemolysis fraction $H_i$ at peak systole.
for the 100% support in the pulsatile (case 4) and continuous (case 5) modes, respectively, along a plane shown in Figure 5.5 (c). For case 4, high $H_l$ is found exclusively on the aortic floor near the jet impingement region, coinciding with the high Reynolds stress region shown in Figure 6.6 (a). $H_l$ distribution for case 5, however, is much lower when compared to that for case 4, as indicated in Figure 6.7 (b). High $H_l$ for case 5 is found closely associated with the high shear layers, whereas low $H_l$ is found in the core of the jet.

<table>
<thead>
<tr>
<th>Case</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>MIH</td>
<td>17.75</td>
<td>20.91</td>
<td>4.91</td>
<td>397.57</td>
<td>6.91</td>
</tr>
<tr>
<td>MIH Ratio</td>
<td>1</td>
<td>1.18</td>
<td>0.28</td>
<td>22.40</td>
<td>0.39</td>
</tr>
</tbody>
</table>

Figure 6.7: Linear hemolysis fraction $H_l$ contour for 100% PVAD support: (a) case 4 (PM) at peak systole, (b) case 5 (CM).

6.2.6 Wall Shear Stress Distribution

Figure 6.8 shows the distribution of time-averaged WSS for all five cases. The time-averaged WSS is defined in Equation 2.52. High WSS occurs along the outer wall of the aortic arch between the branch entrances and the distal wall of the branch vessels (cases 1-5), on the suture line (case 2-5), and on the aortic floor across from the suture
line (cases 4-5). The 50% support in the PM (case 2) causes an increase of WSS at the aortic wall downstream of the graft site but a decrease upstream when compared to the healthy aortic flow (case 1). The continuous 50% support (case 3), however, leads to an overall decrease of WSS in the aorta, as a result of the lower velocities, as shown in Figure 6.4. The maximum time-averaged WSS occurs on the suture line with values of 420 dynes/cm² and 258 dynes/cm² for cases 2 and 3, respectively. The 100% support (cases 4 and 5) causes a higher WSS distribution in the aorta when compared to the 50% support except in the aortic entrance upstream of the graft site. A large increase of WSS is found on the aortic floor across from the suture line. Two stagnation points, identified by combining lowest WSS and highest pressure, are found on the aortic floor for both cases 4 (pulsatile) and 5 (continuous) (Figures 6.8 and 6.9). The maximum WSS regions are located on the suture line with values of 575 dynes/cm² and 371 dynes/cm² for cases 4 and 5, respectively.

Figure 6.8: Time-averaged WSS magnitude (unit: dynes/cm²) contour: (a) anterior view, (b) posterior view.
Figure 6.9: Posterior view of pressure contour (unit: mmHg) for 100% bypass: (a) case 4 (PM) at peak systole, (b) case 5 (CM).

Figure 6.10 compares the OSI distribution for all five cases. High OSI, indicating flow separation and retrograde flow in these regions, are located on the proximal walls of the branch vessels near the entrances (cases 1-4), the aortic floor across the suture line (case 2-3), the anterior wall downstream of the graft toe (cases 2-5), the ascending aortic wall upstream of the graft site (case 4-5), the stagnation points (cases 4-5), and the graft hood and inner wall of the graft close to the suture line (cases 2 and 4). The maximum OSI (0.47) for the healthy aortic flow (case 1) occurs on the proximal wall of the brachiocephalic artery, as shown in Figure 6.10. The OSI distribution during the 50% pulsatile support (case 2) is similar to that during the 50% continuous support (case 3) except in two major locations. First, for case 2, relatively high OSI is also found on the graft hood (0.19) and on the inner wall of the graft close to the suture line (0.49). The OSI in the graft for case 3 is nearly zero, which suggests that the flow in the graft is steady through the flow cycle. Second, the OSI in the descending aorta for case 2 is much higher than that for case 3, as shown in Figure 6.10. During the 100% support, high OSI distributions are found on the ascending aorta upstream of the graft junction.
for both the pulsatile (case 4) and continuous support (case 5). Low OSI is found on
the aorta floor opposite the suture line for cases 4 and 5, except at the flow stagnation
points. In contrast to case 4, only one of the two stagnation points shows relatively high
OSI for case 5. Striking differences in the OSI distribution are found between cases 4 and
5 on the aortic wall downstream of the graft, the proximal walls of the branch vessels,
and the graft wall. As in the 50% continuous support (case 3), the OSI in the graft for
the 100% continuous support (case 5) is close to zero. The maximum OSIs occur on the
anterior wall downstream of the graft toe (0.47) and the inner aortic wall upstream of the
graft junction (0.44) for cases 4 and 5, respectively. Note that the time-averaged WSS
and OSI for case 4 are integrated based on the ten flow cycles averaged WSS, and the
time-averaged WSS and OSI for case 5 are integrated over the same period $T$ (0.5 s) as
for cases 1-4.

Figure 6.10: OSI contour: (a) anterior view, (b) posterior view.
6.3 Discussion

6.3.1 Pulsatility

Surplus hemodynamic energy at the aortic inlet for the healthy aortic model (case 1) is $2.41 \times 10^4$ ergs/cm$^3$, which is close to the data $((1.60 \pm 0.56) \times 10^4$ ergs/cm$^3$) reported by Pantalos et al.[46] in their in vivo neonatal animal study. It is interesting that SHE values in the brachiocephalic artery (104%) and the left common carotid artery (103%) are slightly higher than SHE at the aortic inlet for case 1. This is mainly due to the rigid walls. Greater pulsatility is found at the outlets of the brachiocephalic artery, the left common carotid artery, the left subclavian artery, and the aorta when the PVAD operates in the PM than when it is used in the CM, although both cause significant reduction in pulsatility when compared to the healthy aortic flow. During intermediate support (50%), the PM gives over three times greater pulsatility at each outlet than does the CM. This is very close to the data reported by Travis et al.[121] and Pantolos et al.[46] in their adult clinical patients (five times) and an neonatal animal model (three times), respectively. Our SHE data (nearly zero) for maximum VAD support (100%) in the CM, however, is significantly less than the experimental data $(1842 \pm 1243$ ergs/cm$^3$) by Pantolos et al., simply because, in their in vivo animal experiment, the aortic valve was not completely closed, and the natural heart contributed some pulsatility to the systemic circulation during continuous support. It is interesting that an increase of PVAD support in the PM has no significant impact on pulsatility in the great vessels.

6.3.2 Flow Splitting

The flow splitting in the great vessels increases during the pulsatile PVAD support and decreases during the continuous support when compared to the healthy aortic flow. This is a result of higher pressures at the branch outlets for the pulsatile support and lower pressures for the continuous support, as indicated by TAP values in Table 6.2. The total
flow split to the branch vessels increases with the increasing level of pulsatile PVAD support, as a result of the increase of TAP at each outlet, as shown in Table 6.2. The maximum difference in the total splitting between the pulsatile and continuous support is 10.55% of CO, which occurs during 100% support. It should be noted that our flow splitting results only provide insight into the initial responses of the aorta caused by PVAD support. Significant change in vascular structure to maintain homeostasis may take place with time.

6.3.3 Flow Field

The flow field in the aorta varies widely with the increasing PVAD assistance in both the PM and CM. Significant differences are found between the PM and CM for 50% assistance. The 50% pulsatile support causes a high-velocity jet into the aorta at peak systole while the flow for the 50% continuous support is relatively uniform due to the smaller constant graft inflow. At late systole (early diastole), a jet appears for the 50% continuous support and stays throughout diastole due to the constant PVAD inflow. In contrast, the flow fields for the 100% pulsatile (at peak systole) and continuous support look similar in shape but are significantly different in velocity magnitude. The maximum velocities for the pulsatile and continuous supports are 3.6 m/s and 0.95 m/s, respectively. Turbulence is observed exclusively near the posterior wall of the aorta for the 100% assistance, as a result of the jet and its wall impingement. The maximum principal normal Reynolds stress for the pulsatile support at peak systole is nearly thirty times greater than that for the continuous support. We remind the readers that the Reynolds stress results for case 4 are based on only ten flow cycles, so that the confidence band is very large.
6.3.4 Hemolysis

The hemolysis levels between the pulsatile and continuous support were compared in the aorta. The MIH value for the healthy aortic flow is 17.75, which is 23% of the MIH value reported by Pekkan et al.[47] for their neonatal aortic model. Pekkan et al. calculated the blood damage index by integrating the Giersiepen-Wurzinger correlation [67] along the particle path lines based on the flow field at peak systole. This approach requires calculation of streamline and residence time, and assumes that blood damage is time-independent. The MIH data by Pekkan et al., therefore, represents the blood damage at peak systole. The methodology adopted in this study, in comparison, is a volume integration of blood damage over the computational domain, and can be used to calculate time-dependent blood damage under pulsatile flows. Our MIH data, as shown in Equation 2.46, represents a cycle average. This explains why our calculated MIH value is much lower, as the hemolysis is dependent on shear stresses and maximum shear stresses occur at peak systole.

The hemolysis level increases during the pulsatile support, whereas it decreases during the continuous support when compared to the healthy aortic flow. This can be explained by the higher velocity (higher shear stress) for the pulsatile support and the lower velocity (lower shear stress) for the continuous support at peak systole when compared to that of the healthy aortic flow, as shown in Figures 5.6, 5.7, 5.8, 6.4, and 6.5, as the most hemolysis occurs during peak systole. The highest hemolysis level occurs during the 100% pulsatile support, with the MIH value of 397.57. This is higher than the MIH (299) reported by Pekkan et al.[47] for their cardiopulmonary bypass model at peak systole, partly because the contribution of Reynolds stresses was ignored in their calculation.

One advantage of the present methodology over the streamline approach is its capability of identifying local time-dependent hemolysis sites. As shown in Figure 6.7 (a), for the 100% pulsatile support, most blood damage occurs near the jet impingement, where the highest turbulence occurs. We remind the reader that the reaction coefficient $R_l$ in
Equation 2.38 is derived based on the Giersiepen-Wurzinger correlation [72], which is only valid for laminar flows. A simple integration of the Reynolds stresses using the empirical formula could significantly overestimate the blood damage index [70]. Therefore, a step function is introduced in this study to account for the threshold limit for hemolysis in turbulent flow.

6.3.5 Wall Shear Stress and Oscillatory Shear Index

The time-averaged WSS is found to increase with the increasing PVAD support, except in the ascending aorta upstream of the graft site. For the same percentage of PVAD support, the PM leads to higher WSS and OSI distributions than does the CM, due to the higher velocity and pulsatility for the PM. Two stagnation points are found on the aortic floor during the 100% assistance in both the PM and CM, which is common in arterial bypass flows [17, 169, 170]. For the pulsatile support, two stagnation points oscillate through the cycle, as indicated by their high OSI values (Figure 6.10). For the continuous support, however, the stagnation point close the outer aortic wall is nearly stationary throughout the cycle, as indicated by its low OSI value (Figure 6.10).

6.4 Conclusions

We find that the continuous PVAD support largely reduces the pulsatility in the great vessels originating from the aortic arch and also in the descending aorta. In the case of the intermediate PVAD support, the pulsatile pump yields more than three times greater pulsatility in the branch vessels than does the continuous support. The pulsatile PVAD support increases the flow percentage in the great vessels, with a maximum total increase of 12.77% from the healthy aorta for the 100% PVAD support. In comparison, the continuous support decreases the flow rate in the great vessels by 11.11% and 9.8% for the 50% support and 100% support, respectively. The flow in the aorta is highly
disturbed with the graft inflow, depending on the mode of PVAD operation. Transitional turbulence is observed during the 100% PVAD support in both the PM and CM due to the jet and its impingement on the aortic posterior wall. The correspondingly maximum principal normal Reynolds stresses are 7081 dynes/cm$^2$ and 249 dynes/cm$^2$, respectively. The pulsatile support increases hemolysis levels in the aorta while the continuous support decreases it. The 100% pulsatile support causes a significant increase of hemolysis in the aorta relative to the healthy aortic flow. Finally, the PM gives higher time-averaged WSS and OSI distribution when compared to the CM during both the intermediate and complete PVAD support. Overall, the pulsatile support provides better pulsatility, while the continuous support causes less blood damage.
Chapter 7

Summary and Conclusions

7.1 Summary

Blood flow in end-to-side anastomoses has received great interest. It is generally believed that flow disturbances induced by the anastomoses play an important role in vascular remodeling and graft failure [17]. A lot of studies have focused on the blood flow in arterial bypass grafts and the AV grafts. The anastomotic flow in the aorta, however, has received less attention. This flow is more complicated due to the multi-branch anatomy of the aorta and the possible transitional turbulence near peak systole. In this research, blood flow in end-to-side anastomoses of VADs has been thoroughly investigated.

Both adult and pediatric anastomotic models were used to study the flow. The great vessels on the aortic arch were included for all the models to study the flow splitting. The flow was simulated by a temporally and spatially second-order accurate transient flow solver based on a FVM. The turbulence was captured using a TVD-based MILES. Hemolysis was predicted by solving a time-dependent transport equation. To validate the flow solver, four cases, a two-dimensional Taylor-Green vortex, i.e., a fully-developed turbulent channel flow at $Re_r = 395$, a fully-developed pulsatile laminar pipe flow, and the flow in a simplified pediatric aorta, were studied. Good consensus was found between
the numerical results and the analytical/experimental results for all four validation cases. The effect of blood viscoelasticity was investigated by comparing the flow fields of Newtonian and 40% HCT blood analogs in the simplified pediatric aortic model measured by PIV. No significant difference was found.

The anastomotic flows were compared in different anastomotic configurations under various flow conditions in terms of flow and pressure waveforms, pulsatility, velocity, Reynolds stress, hemolysis, WSS and OSI.

7.2 Conclusions

VAD anastomoses have a significant impact on the aortic flow. Depending on the percentage of VAD support, the flow in the aorta can vary significantly. A low percentage of VAD support causes strong flow mixing in the aorta, while a high percentage of VAD support leads to jet impingement on the aortic floor. As a result, turbulence may occur. The VAD anastomosis causes wide variations of flow splitting in the great vessels. WSS and OSI vary greatly along the increase of VAD support.

7.2.1 The Impact of Anastomotic Configurations

Proximal and distal anastomotic configurations lead to very different flow fields. The proximal configuration causes a higher jet flow in the aorta than does the distal configuration. This jet is directed mainly into the brachiocephalic artery in the proximal configuration, which explains the dominance of right hemispheric perioperative strokes in adult patients. In comparison, the distal configuration causes a large region of flow stagnation near the aortic valve. Hence, this configuration is not recommended for patients with poor cardiac function due to thrombosis. More turbulence is found in the distal configuration than in the proximal configuration during continuous VAD support. More flow is diverted in the great vessels for the distal configuration than for proximal
configuration, at least in the short term.

7.2.2 The Impact of VAD Pumping Mode

VAD pumping affects not only the pulsatility, but the local hemodynamics in the aorta. The major findings are: 1) pulsatile support provides a greater degree of vascular pulsatility when compared to continuous support, which, however, is still 20% less than pulsatility in the healthy aorta; 2) pulsatile support increases the flow in the great vessels, while continuous support decreases it; 3) complete VAD support results in turbulence in the aorta, and pulsatile support causes significantly higher turbulence than does continuous support; 4) pulsatile support increases the hemolysis level in the aorta while continuous support decreases it; 5) pulsatile support causes both higher time-averaged WSS and OSI in the aorta than does continuous support.

7.3 Recommendations for Future Work

First, the MILES method is only validated in a steady inflow condition. The performance of the MILES in pulsatile flow conditions needs to be examined. Second, ten flow cycles are insufficient to predict accurate turbulence statistics. More flow cycles are required to attain good statistics. Third, normal healthy aortic models are used in this study, which is not true for pediatric patients with heart failure. The impact of abnormal aortic anatomy on the flow needs to be studied as well. Last, the effect of viscoelasticity is only examined in a simplified laminar aortic flow. Whether or not it has a big impact upon turbulent flows remains unknown and requires a detailed investigation.
Bibliography


[94] C. G. Speziale, “Galilean invariance of subgrid-scale stress models in the large-eddy


223


Vita

Ning Yang

Ning Yang was born on February 24, 1980 in Linxia, China. He graduated in July 1998 from Changqing Fifth High School in Ningxia Province, China. He then attended Zhejiang University, and received a Bachelor of Science degree in Mechanical Engineering in June, 2002. After graduation, he moved to Shenzhen, China, and worked as a product development engineer in Foxconn Enterprise Group for one year. He entered graduate school at University of California, Riverside in September, 2003, and obtained a Master of Science degree in Mechanical Engineering in August, 2005 under the guidance of Dr. Kambiz Vafai. He then moved to the Pennsylvania State University and obtained his Ph.D. in bioengineering in December, 2009. He is a member of Biomedical Engineering Society (BMES) and American Society for Artificial Internal Organs (ASAIO), and has authored several papers in biofluid mechanics and biotransport.