Practical Patient-Specific Cardiac Blood Flow Simulations Using SPH

Scott Kulp\textsuperscript{1}, Mingchen Gao\textsuperscript{1}, Shaoting Zhang\textsuperscript{1}, Zhen Qian\textsuperscript{2}, Szilard Voros\textsuperscript{2}, Dimitris Metaxas\textsuperscript{2}, and Leon Axel\textsuperscript{2}

\textsuperscript{1}Rutgers University, \textsuperscript{2}Piedmont Heart Institute, \textsuperscript{3}New York University

Introduction

- After a heart attack, the movement of the heart walls changes, affecting the motion of blood. This could potentially lead to thrombus and stroke.
- Recent developments in heart wall models used in blood flow simulations have become increasingly realistic and complex, but the computation time for simulations are very high.
- Smoothed Particle Hydrodynamics (SPH) is a widely-used method for fluid simulations that is very fast and easily parallelizable. However, dealing with complex boundary conditions is an unsolved problem.
- We present a method to accurately and quickly manage the boundary conditions in a cardiac blood flow SPH simulation, cutting the total computation processing time by orders of magnitude.

Smoothed Particle Hydrodynamics

- In SPH, the fluid is represented by particles, each with their own mass and velocity.
- Density at a point can be computed as the weighted sum of the masses of the neighboring particles.
- The pressure gradient is computed similarly, by taking the gradient of the weighting function. The forces due to the pressure gradient are applied to update the velocity of each particle.

\[ \rho_i = \sum_{j=1}^{N} m_j W(r_i - r_j, h) \]
\[ f_{\text{pressure}}^{i} = -\sum_{j=1}^{N} \frac{m_j P_i + P_j}{\rho_j} \nabla W(r_i - r_j, h) \]

Boundary Management

- Solid wall boundary condition enforcement drives the flow.
- Before simulation begins, we generate a level set around the surface, to determine the distance from the boundary at every point in the domain.
- Boundary particles are generated uniformly inside the level set region. The walls must have a thickness of at least 2h (twice the sphere of influence), so that fluid particles on either side cannot interact.
- Boundary particles move with the walls each time step, and compute their own densities/pressures. Fluid particles near the walls use the boundary particles in the pressure gradient step as normal.

Collision Detection

- We seek a fast way to perform collision detection between boundary (red) and fluid (black) particles, so that there is no penetration.
  (a) Initial state: Sphere of radius h surrounds each particle
  (b) Boundary moves distance d, new location too close to fluid
  (c) Pass 1: Boundary box collision to detect “danger” pairs
  (d) Pass 2: For all danger pairs, determine if line segment d intersects the fluid sphere. If so, push particles forward in same direction as boundary
  (e) New positions (boundary no longer too close to fluid)

Results

- The accuracy of the simulation can be improved by adjusting the lengths of the time steps. We used $dt=0.001s$ (SPH1), 0.0005s (SPH2), and 0.00025s (SPH3).
- Visualizations of the flow fields can be seen in the figure below. Blue regions represent low velocity, red regions represent high velocity. We compare the SPH1 and SPH3 experiments to an FDM simulation. Velocities are higher than expected near valves in SPH due to thickened walls, and thus smaller openings.
- The running times and ejection fraction for these experiments are shown in the table below. The computation time is far lower with similar levels of accuracy, compared to FDM.

<table>
<thead>
<tr>
<th>Speed of Sound (c)</th>
<th>FDM</th>
<th>SPH1</th>
<th>SPH2</th>
<th>SPH3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>N/A</td>
<td>10</td>
<td>20</td>
<td>30</td>
</tr>
<tr>
<td>Speed of Sound (c)</td>
<td></td>
<td>Adaptive</td>
<td>.005</td>
<td>.0025</td>
</tr>
<tr>
<td>Simulation Time</td>
<td>4 days</td>
<td>30 min</td>
<td>62 min</td>
<td>126 min</td>
</tr>
<tr>
<td>Ejection Fraction</td>
<td></td>
<td>-45</td>
<td>-42</td>
<td>-48</td>
</tr>
</tbody>
</table>