Computational Fluid Dynamics Model of the Cardiopulmonary System

Arianna Worthy and Grayson Hawkins (UTK)
Mentor : Dr. Kwai Wong (UTK)

Joint Institute for Computational Sciences

Abstract

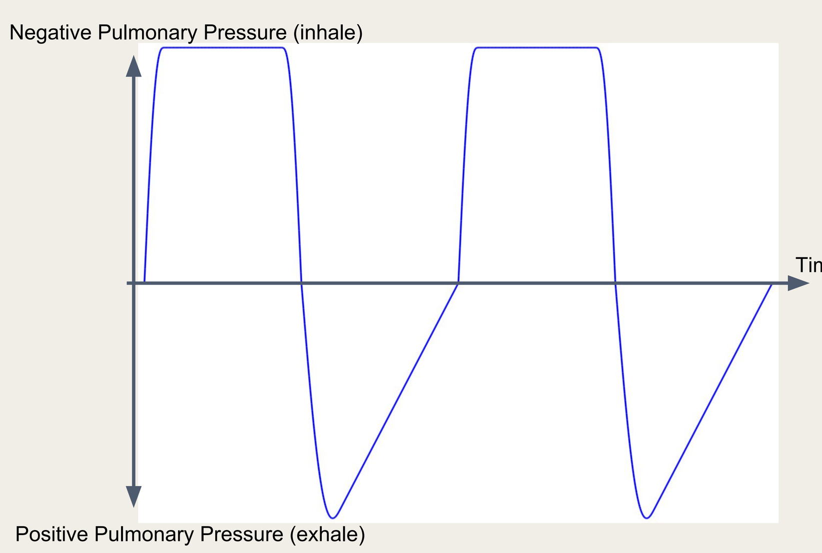
The rise of high performance computing has given scientists, researchers and physicians the ability to simulate the biological processes that drive life. In this project, we look at the geometry of the pulmonary and cardiovascular systems and conduct finite element Computational Fluid Dynamic simulations to gain a better understanding of fluid behavior and oxygen diffusion in these intricate processes. Using simplified geometries of increasing complexity for both the cardiovascular and pulmonary systems, we slowly work our way to a representative simulation of the human heart-lung system in the multiphysics software Comsol.

Methodology

Comsol is a multiphysics software that uses the Finite Element Method to apply fluid flow and transport equations to a geometry. The basics of accomplishing any complex result in a CFD simulation are beginning with a simple geometry and benchmarking/correcting key variables, namely pressure, velocity and oxygen content. For the lungs, we began with a single bifurcation before advancing to a second and third generation. For the cardiovascular system, we began with a simple pulsating flow through a smooth duct to accurately simulate a heartbeat. Finally, we are coupling the two systems together.

Pulmonary Simulations

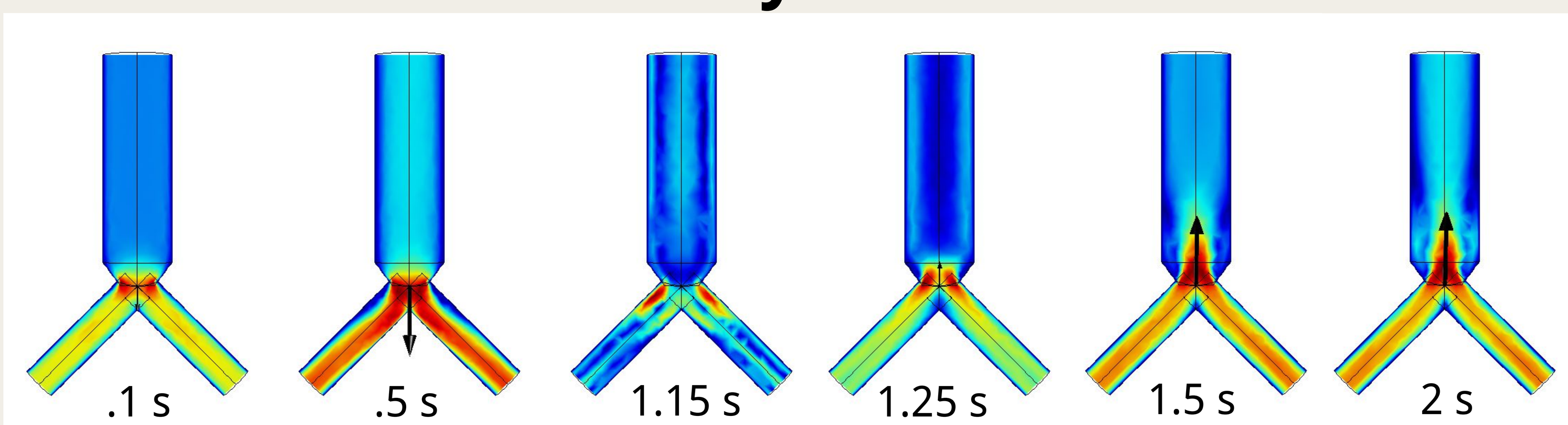
Inhaling and exhaling, when represented on a graph of volumetric flow vs time, are very different. We manipulated the inlet pressure conditions of the lung with the following equations...



- These are two separate waveforms, with a third (square) waveform serving as a "switch" between them
- The waveforms are then multiplied by separate factors such that...

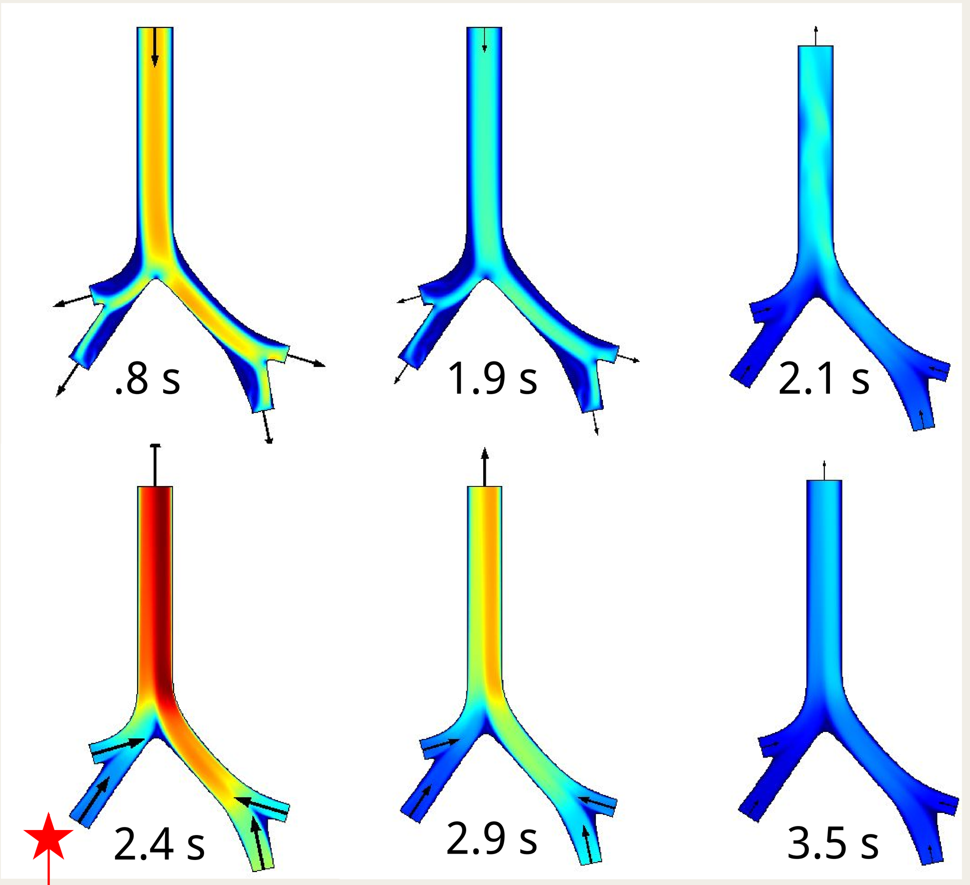
$\int Q_{in}\,dt = -\int Q_{out}\,dt = V_{Tidal} = 0.5L$

3D Pulmonary Simulation



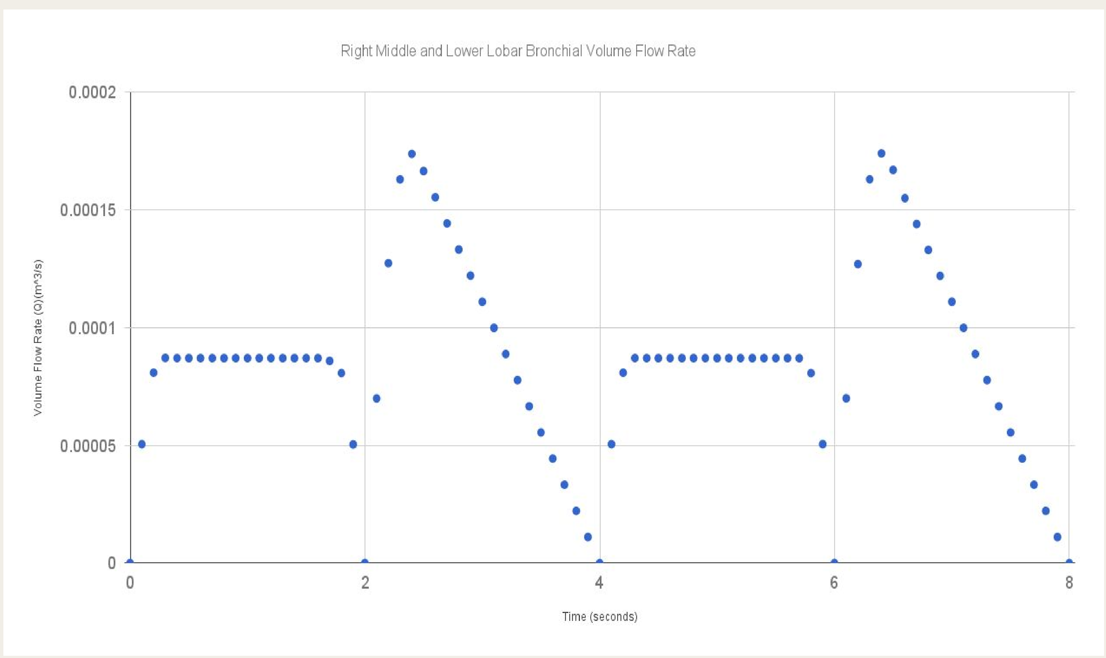
This shows the velocity gradient of the final iteration of our 3D model, a volumetrically accurate breathing cycle. The mean pressure at this first bifurcation required to match with the 0.5L tidal volume was ∓0.1766 Pa.

2D Pulmonary Simulation



The following data is taken from this outlet, representing the right middle/lower lobes of the lung.

- We used the 2D solver in Comsol to hasten our research and achieve a more representative geometry.
- Steady state sims. were used to estimate the pressure values at the outlets.
- This shows the velocity gradient of the latest iteration of our 2D simulation, showing inspiration from t = (0,2) and expiration from t = (2,4).

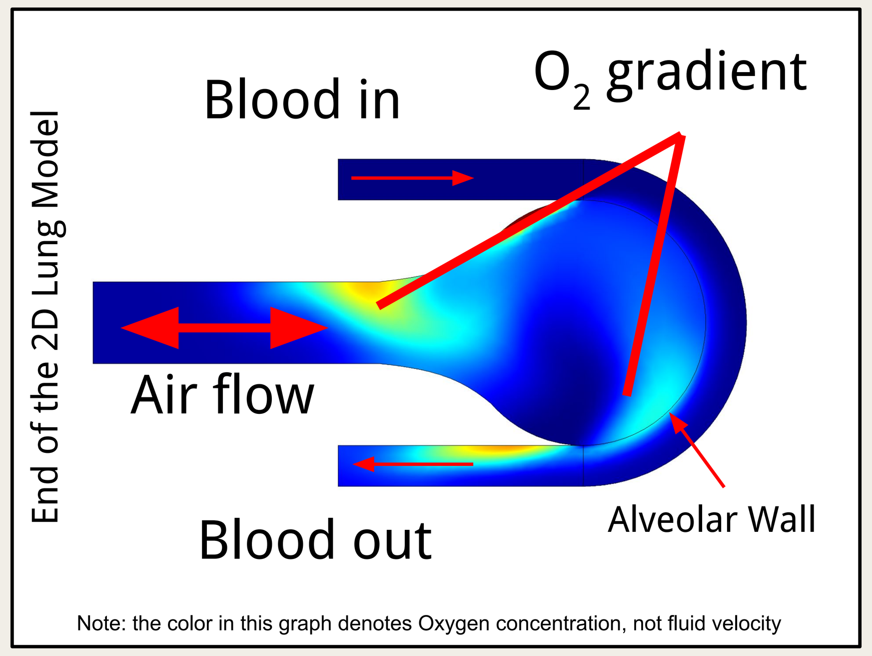


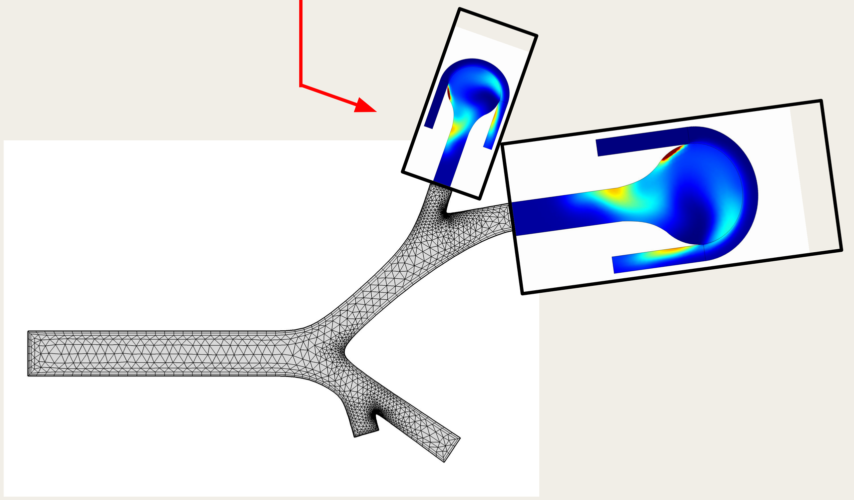
This graph shows the absolute value of the flow rate in and out out of the RM/RL lobar bronchiole.

Continuing our Work...

Coupling Systems

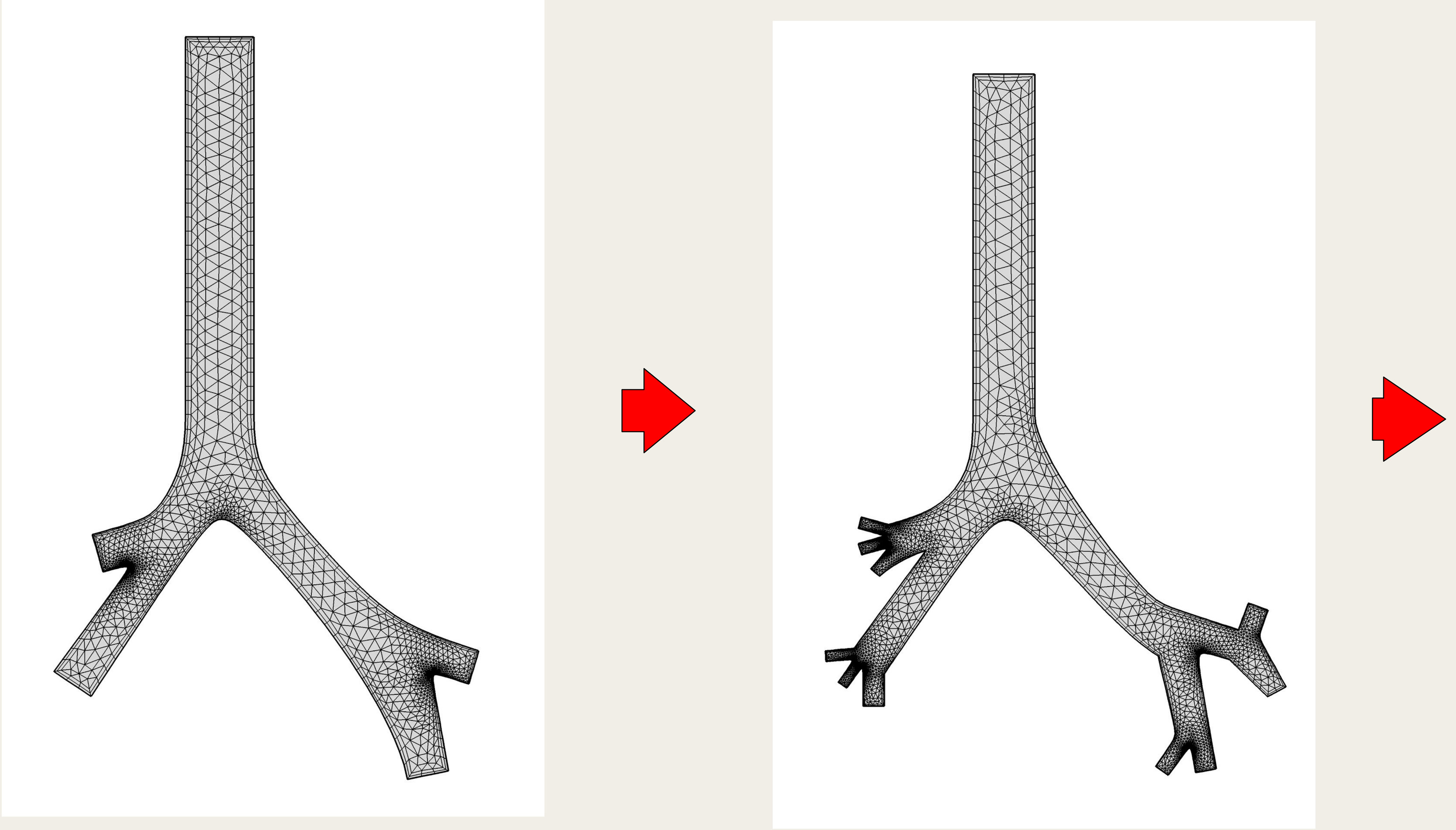
- Find representative diffusion coefficients and pressure gradients
- Match computational results to the bulk characteristics of blood, reiterate the simulation.
- Place "alveolar sacs" (see photo) to the end of the 2D pulmonary model...





- Using Transport of Concentrated Species, oxygen will be modeled to diffuse through the alveolar wall into the blood stream, with an accurate oxygen consuption rate.

Third and Fourth Bifurcations



More complex geomeries,as pictured above, are the next steps in the following weeks. The more realistic the geometries designed in Comsol, the easier "real world" numbers can be computed.

Acknowledgements

University of Tennessee, Knoxville. Computational Resources from XSEDE.. Mentorship of Dr. Kwai Wong, UTK. The project is part of the National Science Foundation's Research Experiences for Undergraduates program.

Major References

Hlastala and Berger, *Physiology of Respiration*

Des Jardins, *Cardiopulmonary Anatomy and Physiology*

The University of Tennessee Knoxville

Oak Ridge National Laboratory