Computational Fluid Dynamic Analysis of a Bifurcating Artery
Skill Report

Ashley Miller

Applied and Engineering Physics Department, Cornell University.

(Dated: February 19, 2024)

This skill project report presents a computational fluid dynamic (CFD) analysis of a bifurcating artery using Ansys Fluent, focusing on the impact of arterial obstruction on fluid flow dynamics. The simulation results indicated that arterial obstruction leads to increased wall shear stress and flow disruptions, highlighting the importance of computational biofluid dynamics in assessing health risks and informing diagnostic and treatment strategies. The report concludes with insights into future directions for refining the model and reflections on the skills learned throughout this project.

I. INTRODUCTION

Biofluid dynamics is a discipline at the intersection of physics and biomedical science that aims to apply fluid dynamics to analyze the relationship between biological fluid flows and physiological processes to better understand health and disease. Biofluid dynamics has important applications in medicine since it studies the mechanisms behind disease formation in critical areas such as the circulatory, pulmonary, and synovial systems [1]. For example, blood vessel wall shear stress is a major indicator of the formation of lethal disorders such as aneurysms and cardiovascular disease [2]. Insights into these fluid flow systems are crucial towards developing diagnostic and treatment tools that will help patients suffering from a wide range of disorders.

Meanwhile, the use of simulation software has become increasingly advanced and popular as an engineering tool to understand and model fluid flow in virtually any situation. These computational fluid dynamics (CFD) software have become invaluable in the study of biological fluid flow systems.

This report will focus specifically on utilizing CFD to simulate fluid flow in a bifurcating artery in various conditions. Arteries are part of the cardiovascular system, which is responsible for delivering oxygenated blood via the arterial system to various organs and tissues and recirculating deoxygenated blood back to the right ventricle via the venous system so it can be oxygenated by the lungs. In addition to delivering oxygen, the cardiovascular system is responsible for facilitating heat and mass transfer throughout the body as it delivers important nutrients to organs. To accomplish this, the heart acts as a pump to circulate about 5 L of blood throughout the body at an impressive rate of about $6\frac{L}{min}$. Blood, composed of mainly blood cells and plasma, is a non-Newtonian fluid and viscoelastic material with a density of $1060\frac{kg}{m^3}$. However, the non-Newtonian effects are small enough to be ignored in this analysis [1]. The simulations in this report will represent the situation where an artery becomes obstructed, impeding proper fluid flow. Common obstructions within arteries include plaques and blood clots. Artery obstructions can be extremely dangerous and cause a heart attack, stroke, or even death. CFD simulations can be very helpful in modeling the formation of these dangerous disorders and testing treatment approaches such as stents. Not only can CFD handle complex physics problems, but it provides an opportunity for personalized medical treatment with technology that can create customized 3D models of patients' anatomy [2].

II. METHODOLOGY

A. Computational Fluid Dynamics: Ansys Fluent

CFD software is a tool that takes in user inputs, solves fluid dynamics equations using mathematical models, and outputs useful color pictures and results. User inputs include geometry, meshing, and boundary conditions. The mathematical model that the computations are based on has some physical assumptions and principles that are based on the user inputs. The CFD software solves for selected variables at selected points in order to generate color pictures, plots, and animations [3].

To understand what really happens under the hood in a CFD solver requires a basic understanding of fluid dynamics. The governing equations of fluid dynamics are based on three fundamental conservation laws: energy, mass, and momentum. These governing equations have complementary differential and integral forms. One can imagine the differential forms as applying laws to infinitesimal fluid particles and the integral forms as applying laws to a finite volume in a flow domain [4]. Unfortunately, these governing equations are coupled and nonlinear, so they can only be solved manually in a number of special cases. Fortunately, CFD is able to solve the equations for complex systems approximately by converting the governing equations to a large set of algebraic equations with numerical methods. I used Ansys Fluent, a leading general-purpose CFD solver used in industry, for this project. Fluent employs a numerical method known as the finite volume method. Finite volume analysis consists of dividing the fluid domain into infinitesimal control volumes and applying the conservation laws to each volume chunk using the integral form of the governing equations [5]. These control volumes are defined by
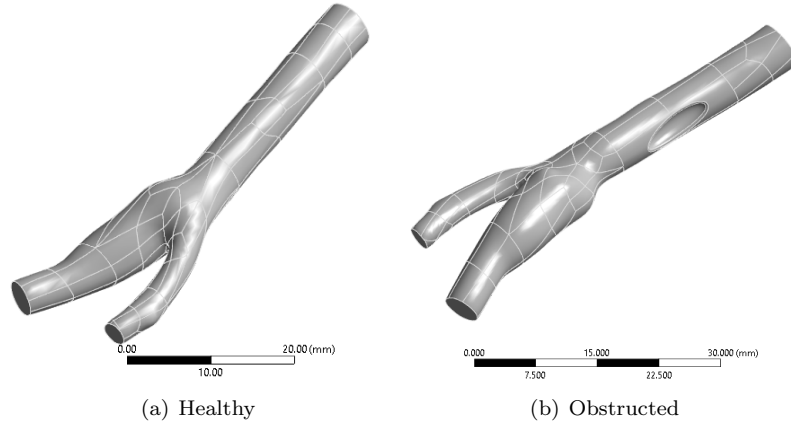
(a) Healthy   (b) Obstructed

FIG. 1. 3D CAD model made from a luminal casting of a bifurcating carotid artery. Assigned these CAD models as 'geometry' inputs in Fluent setup.
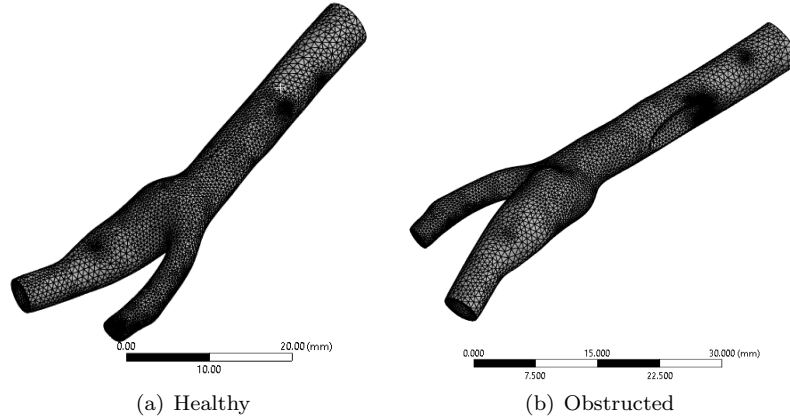

(a) Healthy   (b) Obstructed

FIG. 2. Mesh generated by using the 3D CAD model as an input to Ansys Fluent Meshing

the meshing of the geometry, which is defined by the user during pre-processing. Ansys includes both pre-processing and post-processing tools to use along with the Fluent CFD solver. The pre-processing will be described in the "Setup" sections while the post-processing will be described in the "Results" sections.

B. 3D Bifurcating Artery Setup

The setup for a CFD simulation consists of pre-processing steps such as creating a model of the geometry using CAD (computer-aided design) software, generating the mesh, and providing sufficient problem specifications to the CFD solver to inform the mathematical model of initial conditions, boundary conditions, assumptions, and other necessary information. I performed two simulations of a 3D bifurcating artery: one without a blockage and one with a blockage. The only difference between the setups of these two simulations was the CAD geometry, as shown in Figure 1. The original, healthy bifurcating artery geometry is a computer model based off of a lumi-

nal casting of a real carotid artery bifurcation, which I retrieved from [6]. To model a blockage within the artery, I created an ellipsoidal-shaped indentation on the wall of the artery using DesignModeler, the CAD tool supported by Ansys.

Next, I performed identical meshing procedures on both geometries, as seen in . The mesh tells the CFD solver which finite volumes to use and at which points to solve for selected variables. In pathline or streamline simulations, Fluent simulates each particle pathline to begin from the nodes of the mesh at the inlet. While smaller mesh element sizes would be a more accurate mathematical model, there is a tradeoff between accuracy and computation time. Further, there is a point of diminishing returns when refining the mesh where smaller element sizes no longer change the results; the best practice is to test simulations with smaller element sizes until the results no longer show noticeable changes - at that point the model is ideal. I went through that process when refining the mesh here. I used an element size of 1 mm applied to the whole geometry and subsequently used sphere of influence, inflation, and surface meshing tools to further refine the mesh. It is generally best to

refine the mesh (make the element sizes smaller) around corners or areas where relatively more force is expected.

After I completed the geometry and meshing setups, I imported them into Fluent to begin the physics setup. I made the assumption that the walls of the artery are stationary and no-slip, which refers to the boundary condition that the tangential component of the velocity vector equals the speed of the wall (which is zero here for the stationary case), while the normal component has value zero [7]. The only external force I added was gravity. For boundary conditions, I followed guidelines from [6]. First, I defined the inlet velocity to be $0.315\frac{m}{s}$ in accordance with a Reynolds number of 600. Next, I defined the outlet pressures to be 13332 Pa, which I got by taking the average of the systolic and diastolic pressures of a healthy human. Additionally, I defined the fluid domain to be containing blood and assigned the material properties of blood to be 1060 kg/m3 and 0.0035 m/s [6]. Since the inlet velocity corresponds to a Reynolds number of less than 2000 and I was taking this simulation to be steady-state, I selected the laminar flow regime. Laminar flow refers to highly organized flow along streamlines [1]. In Fluent, each streamline initiates from the nodes of the mesh at the inlet. Finally, I set the calculation to perform 2000 iterations and ran the simulation.

III. RESULTS

Once the CFD solver completes the simulation and successfully runs through the specified number of iterations, results can be generated in post-processing software. Ansys Fluent has a built in post-processing software that generates the familiar color pictures and contours. I used this post-processing software to generate pathline animations, velocity vector fields, and pressure, velocity, and wall shear contours. Two of these results are especially meaningful in the scope of blood vessel health and analysis: wall shear and velocity vector fields [2].

Wall shear stress results are shown in Figure 3. The blockage increased the maximum shear stress, which is seen at the bifurcation, by 1.36 Pa (about a 7% increase), putting the artery at increased risk for developing disorders such as aneurysms. Additionally, Figure 3(b) shows increased stress around the blockage indentation.

Further, in Figure 4, normalized velocity fields are shown for particular sections within the healthy and obstructed arteries. I generated these vector fields by taking identical cut sections of the arteries and using the post-processing capabilities to solve for the velocity fields within those planes. First, in Figure 4(a), the bulge of the larger outlet has a significant amount of swirl and reverse flow. Since the outlet geometries are identical on both the healthy and obstructed models, this swirl is present on both models. No other regions in the healthy model exhibited observable swirl or reverse flow within the velocity field. However, in Figure 4(b), an additional region of slight swirl and back flow is seen at the boundaries of
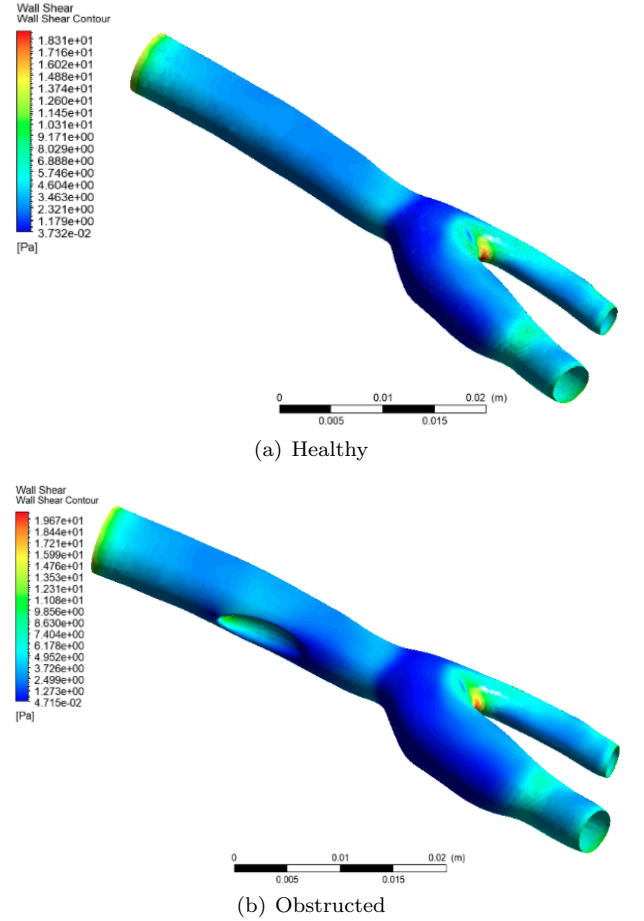

(a) Healthy

(b) Obstructed

FIG. 3. Wall shear contours generated by Ansys Fluent

the obstruction. Intuitively, swirl and reverse flow will be detrimental in artery performance and cause less efficient circulation, forcing the heart and body to work harder to compensate and successfully circulate blood throughout the body. The simulations showed that the obstruction causes an additional region of swirl.

IV. DISCUSSION

A. Verification and Validation of Results

Arguably, the most important step of a CFD simulation is the verification and validation of the results. This step involves ensuring the simulation results are valid and meaningful by confirming the assumptions and conditions that the problem is based on. Without a basic understanding of how Fluent works and what the expected results of the simulation should be, I would not have an intuitive idea of whether the calculation was successful. I went through an identical verification process for both the healthy and obstructed geometries, and followed guidelines from [6].

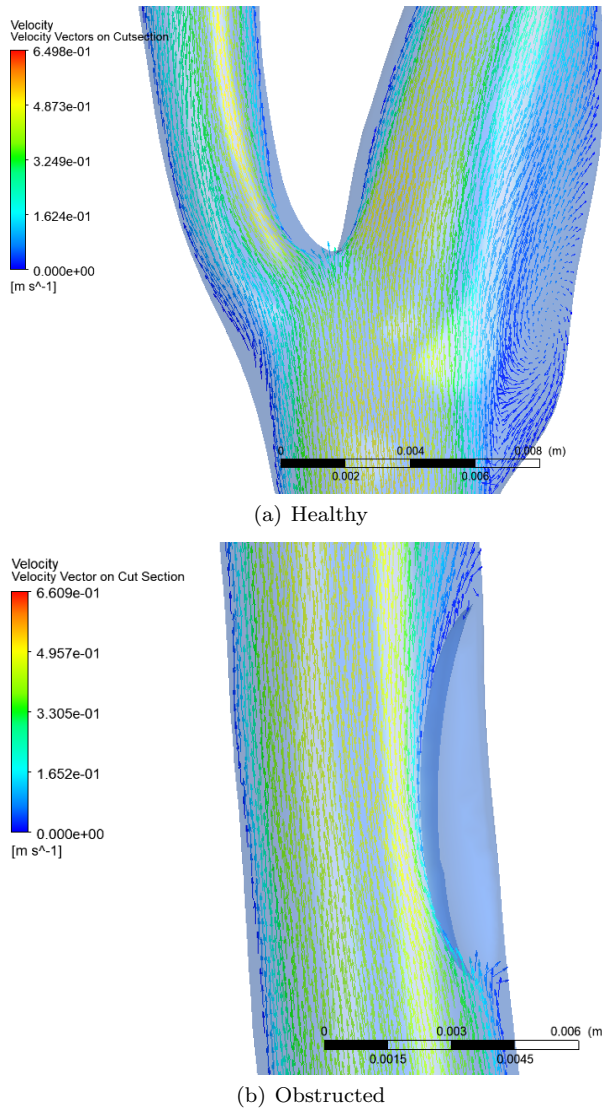The first step of the verification process is checking

(a) Healthy

(b) Obstructed

FIG. 4. Normalized velocity vector field generated by Ansys Fluent

condition was satisfied by checking the velocity profile at the inlet during the calculation.

The next step, as mentioned in Section II.B, is checking the mesh by refining it and confirming that the results do not significantly change. I confirmed this result to verify my solution.

After verifying that the simulation runs properly, the results can be validated based on hand calculations or experimental data. Unfortunately, I did not have access to experimental data on this carotid artery bifurcation. It would have been interesting to compare the results to clinical data obtained with diagnostic equipment.

B. Future Directions and Next Steps

The next steps to this bifurcating artery experiment would be to modify the model to more accurately resemble the problem. Of course, this would involve a more complicated simulation setup as well as increased computing time. There are several ways that I could eliminate assumptions and simplifications in this problem to create a more rigorous model. The first modification would be to perform an unsteady simulation, meaning the steady-state flow would no longer be assumed, allowing for turbulent flow. This would be important in this case to include the pulsatile nature of circulatory flow. Further, a more advanced model would include the non-Newtonian properties of blood. Additionally, the elastic features of the walls should be included to make a more realistic model (meaning a modification of the stationary no-slip condition) [2].

V. CONCLUSION

In conclusion, I simulated a healthy and obstructed model of a bifurcating carotid artery and found that the obstruction causes flow disruptions such as increased swirl/reverse flow and increased wall shear stress. This demonstrated the importance of computational biofluid dynamics because of the health risks posed by these conditions and the advanced capabilities of software such as Ansys to inform diagnostic and treatment approaches.

Throughout this skill project, I became proficient in many areas of CFD analysis. I learned the basics of fluid dynamics as well as how to use DesignModeler, Fluent Meshing, and Fluent CFD within Ansys. Further, I learned how to use post-processing to generate meaningful results.

that the Fluent calculation was valid by confirming the governing conservation laws and my boundary conditions. One way to do this to confirm conservation of mass and the inlet boundary condition [2]. By checking the Fluent solution report for fluxes, I viewed the mass flow rate for the inlets and two outlets. For mass conservation, the mass fluxes should sum up to zero net flow rate. Here, mass was conserved as the net flow rate was approximately zero. Next, I verified the inlet boundary

[1] Biofluid dynamics, Wikipedia: `https://en.wikipedia.org/wiki/Biofluid_dynamics` (2023).

[2] 3d bifurcating artery, ANSYS: `https://courses.ansys.com/index.php/courses/fluent-3d-bifurcating-artery/`.

[3] R. Bhaskaran, What's under the cfd blackbox?, Retrieved from Cornell University Confluence Space: `https://confluence.cornell.edu/display/`

SIMULATION/The+Blackbox (2019).

[4] R. Bhaskaran, Big ideas: Fluid dynamics, Retrieved from Cornell University Confluence Space:https://confluence.cornell.edu/display/SIMULATION/Big+Ideas\%3A+Fluid+Dynamics (2019).

[5] R. Bhaskaran, Computational fluid dynamics, Retrieved from Cornell University Confluence Space:https://confluence.cornell.edu/display/SIMULATION/Computational+Fluid+Dynamics (2019).

[6] K. Works, Fluent - 3d bifurcating artery (steady), Retrieved from Cornell University Confluence Space: https://confluence.cornell.edu/pages/viewpage.action?pageId=378080320 (2019).

[7] Álvaro L. De Bortoli, G. S. Andreis, and F. N. Pereira, in *Modeling and Simulation of Reactive Flows*, edited by Álvaro L. De Bortoli, G. S. Andreis, and F. N. Pereira (Elsevier, 2015) Chap. 6, pp. 123–169.