STM JOURNALS

Journal of
Polymer & Composites

ISSN: 2321-2810 (Online)
ISSN: 2321-8525 (Print)
Volume 11, Special Issue 13, 2023
DOI (Journal): 10.37591/JoPC

http://engineeringjournals.stmjournals.in/index.php/JoPC/index

Research JoPC

Computational Analysis of Non-Newtonian Blood Flow through Bifurcated Coronary Artery: Insights into Hemodynamics and Wall Shear Stress

Abdulrajak Buradi[1,*], Md Yousuf Ahmed Khan[2], Sanjaytharan Tamilselvan[3], Kapilan N.[4]

Abstract

This abstract presents a study on the computational fluid dynamics (CFD) simulations of blood flow through a bifurcated coronary artery using non-Newtonian fluid model. The objective of this study is to investigate the hemodynamic characteristics in Bifurcated Coronary artery. The methodology involved the utilization of ANSYS SpaceClaim software for creating a geometric model of the bifurcated coronary artery. A mesh independent study was conducted to ensure the accuracy and reliability of the simulations. Fluent, a popular CFD software was employed for running the simulations, with appropriate boundary conditions set for the velocity inlet, outflow, and no-slip wall conditions. Additionally, a user-defined function file was implemented to define the velocity at the inlet. The simulations were performed using two different time step sizes: 1.15 seconds and 1.30 seconds representing two Cardiac Cycle Systole and diastole. This enabled the analysis of the temporal changes in the flow characteristics and their influence on WSS, pressure, and velocity profiles. The results obtained from the simulations were post-processed to extract the desired parameters. The analysis of the results revealed significant variations in the WSS, pressure, and velocity profiles along the bifurcated coronary artery. The pressure distribution within the artery ranged from a minimum of -406.23 Pa to a maximum of 426.55 Pa. The velocity profiles obtained during the simulations ranged from a minimum of 0 m/s to a maximum of 0.879 m/s. The wall shear stress (WSS) distribution along the arterial walls ranged from a minimum of 0 Pa to a maximum of 25 Pa. Based on the findings, it can be concluded that the temporal changes in the flow characteristics within the bifurcated coronary artery play a crucial role in determining the hemodynamic conditions. This study contributes to a better understanding of the fluid dynamics in bifurcated coronary arteries and can potentially aid in the diagnosis and treatment of cardiovascular diseases.

***Author for Correspondence**
Abdulrajak Buradi

[1]Associate Professor, Department of Mechanical Engineering, Nitte Meenakshi Institute of Technology, Bangalore, Karnataka, India
[2]UG Student, Department of Mechanical Engineering, Nitte Meenakshi Institute of Technology, Bangalore, Karnataka, India
[3]UG Student, Department of Mechanical Engineering, Nitte Meenakshi Institute of Technology, Bangalore, Karnataka, India
[4]Professor, Department of Mechanical Engineering, Nitte Meenakshi Institute of Technology, Bangalore, Karnataka, India

Received Date: November 27, 2023
Accepted Date: December 20, 2023
Published Date: February 19, 2024

Citation: Abdulrajak Buradi, Md Yousuf Ahmed Khan, Sanjaytharan Tamilselvan, Kapilan N. Computational Analysis of Non-Newtonian Blood Flow through Bifurcated Coronary Artery: Insights into Hemodynamics and Wall Shear Stress. Journal of Polymer & Composites. 2023; 11(Special Issue 13): S160–S168.

Keywords: Hemodynamics, Computational fluid dynamics (CFD), Wall Shear Stress (WSS), Bifurcated coronary artery, Cardiovascular diseases (CVD).

INTRODUCTION

Computational Fluid Dynamics (CFD) has resulted in numerous medical-related studies in the past two decades, including examination of flow in definite arteries. Hemodynamics within arteries affects cardiovascular diseases, as low WSS values have been linked to building of atherosclerosis. Despite a collective number of inquiries, CFD applications in bio medical remain inadequate, rendering the physiological relevance of hematologic data attained from simulations dubious. Accurateness of geometry prototype, complication of fluids flowing in body of human, and forced boundary conditions are regularly acknowledge restraining factors disturbing the precision of CFD simulation outcomes in the field of bio medical engineering. Utilization of several Medical imaging techniques such as ultrasound imaging, magnetic resonance imaging (MRI) & computed tomography (CT) offers tailored patient specific models for CFD Studies [1, 2]. On the other hand, Non-invasive techniques such as pressure wire, Doppler ultrasound etc. provides flow & pressure information at precise sites of arterial system, which can be used as boundary conditions in CFD studies. However, this information replicates definite data of an explicit case at a detailed time, and cannot be opted for what-if scenarios to conduct inclusive exploration. Therefore, Utilization of apt boundary conditions in simulations entails distinct consideration. Proper implementation of outlet boundary conditions in simulations of flow in a specific artery requires considering the impact of the rest of arterial network, which can be achieved by using boundary conditions that designates a parallel between exit flow and pressure [3].

Numerical simulation of blood flow is becoming increasingly popular as a means of predicting cardiovascular disease and assisting with treatment decision-making. This is because measuring blood flow in an artery in vivo is expensive, often inaccessible, and generally not accurate enough to predict pressure distribution and wall shear stress Congenital heart defects CFD models of blood flow in arteries may assist researchers in determining how flow conditions contribute to CHD, determining disease severity, and identifying preventative measures [4, 5].With the progress of CFD and the introduction of supercomputers, computer modeling has gained significance. In the 1970s, mathematical formulations were conducted using a vorticity stream function method by OBrien et al. to investigate unsteady flow through symmetric bifurcations in two dimensions. However, further research revealed that two-dimensional modeling failed to provide a comprehensive understanding of the way of flow due to lack of non-negligible subordinate movement in branches. As a result, researchers focused on three-dimensional simulations, which were initially limited to either symmetric bifurcation geometry or steady flow in the 1980s. Recently, researchers have been able to simulate pulsatile flow in realistic arterial bifurcations using three-dimensional simulations [6, 7]. The non-Newtonian models that exhibit shear-thinning are often thought to represent the real rheological behavior in multifaceted artery structure, but they do not precisely depict behavior of blood. These models counsel viscosity as a function of rate of shear, which is grounded on experimental consequences that are obtained with certain assumptions. To improve blood flow modeling, researchers have proposed more advanced blood rheology models that model red blood cells (RBCs) [8].

Various studies have tackled the issue of approximating WSS by using different methods such as MRI. These methods rely heavily on the quality of the apparatus used, and can produce dependable outcomes over time [9]. Computational modeling which involves creating a 3D model based on geometries from various sources to examine general aspects of the problem. Preceding lessons have used diverse methods to assess the problem from various perspectives [10, 11]. The molecular viscosity of blood, static pressure, velocity strain rate, and WSS have a substantial impression on the hemodynamic conditions of the cardiovascular system [12]. The part of viscosity in the advancement and evolution of atherosclerosis cannot be overlooked. Atherosclerosis tends to occur in areas where blood flow is slow or disrupted, and WSS and its gradients are low [13]. Several studies support the theory that an increase in plasma viscosity could be a linkage amongst cardiovascular peril aspects and atherosclerosis. Viscosity is non-Newtonian, meaning it varies depending on the velocity gradient and other factors such as vascular geometry and flow conditions [14, 15].

METHODOLOGY

Coronary Artery Geometry

Utilizing ANSYS 2022 R2 Space Claim and Fusion360 software, the coronary arteries geometry was constructed, taking into account anatomical details such as vessel diameter and length, as depicted in the Figure 1, to ensure a realistic representation. The CAD software provided exact control over the geometry, which guaranteed an accurate depiction of the coronary arteries anatomy and dimensions, allowing for a meticulous examination of the complicated relationship between blood flow and the arterial wall.

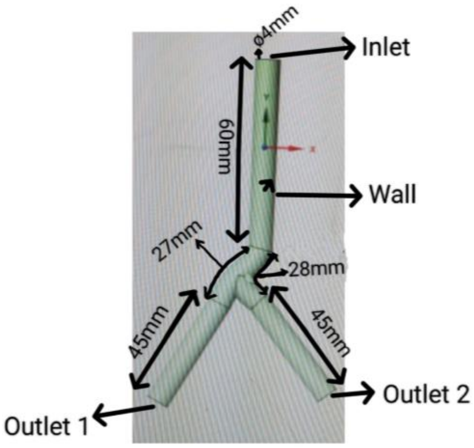

Figure 1. Geometry of the artery with dimensions.

Meshing of Artery Model

Accurately capturing the complex features of the coronary artery geometry required high-quality mesh generation [16], which was performed using ANSYS 2022 R2 Meshing Application. Depending on the complexity of the geometry and resolution requirements, an unstructured mesh was generated as shown in Figure 2. To ensure sufficient mesh resolution near the arterial walls, advanced meshing techniques such as boundary layer meshing and tetrahedral meshing were employed [17]. The ratio of transition in the walls of the blood vessels is 0.272, and the mesh comprises approximately 635662 elements. The mesh was carefully designed to provide adequate resolution of the boundary layer close to the arterial walls, where significant flow gradients occur.

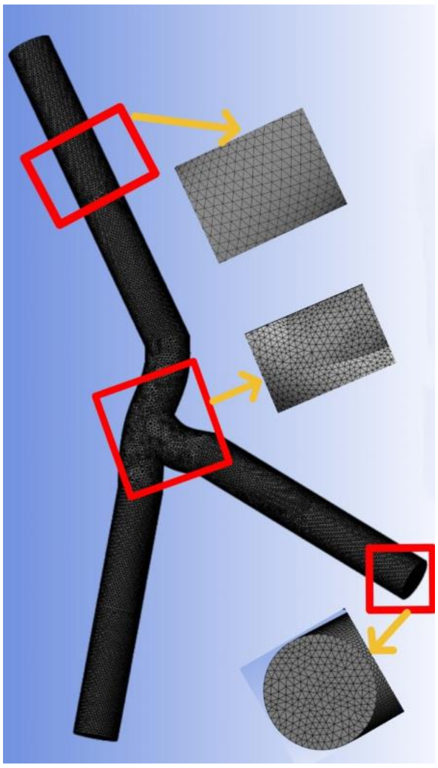

Figure 2. Representation of meshing of model.

Table 1. Mesh Independence Analysis

Mesh Type	Number of Elements	Max Velocity (m/s)	Max Pressure(pa)	Max WSS (pa)
Coarse	318065	1.79094	405.332	69.5036
Medium	454578	1.81201m/s	417.845	81.7405
Fine	686734	1.81387m/s	420.654	85.8556

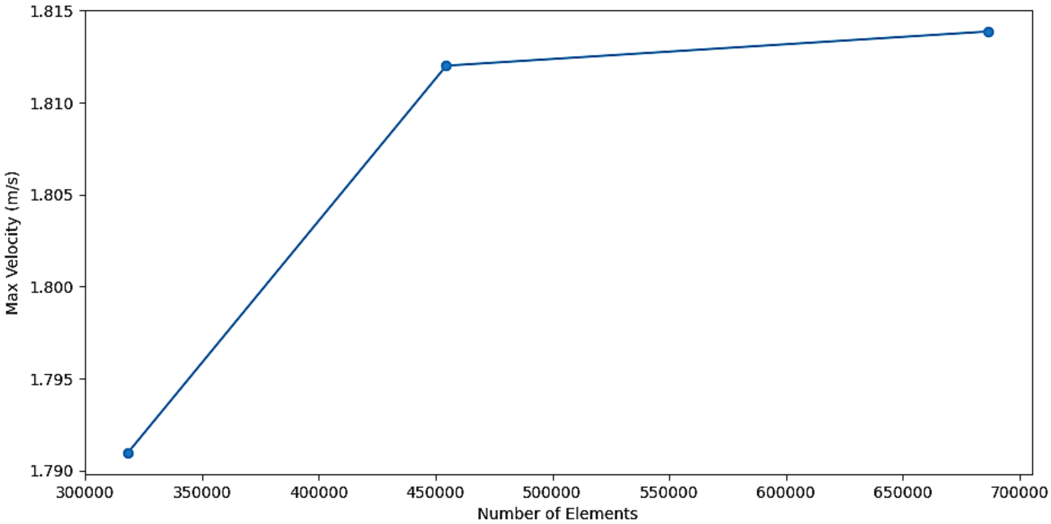

Figure 3. Max Velocity vs Number of Elements.

A study was steered to judge the compassion of results to mesh resolution. Multiple meshes with varying levels of refinement were created and simulations were performed on each mesh. Key hemodynamic parameters such as pressure distribution, velocity profiles, and WSS were compared among different mesh configurations to determine the minimum level of mesh refinement required for reliable and accurate results. Convergence criteria were considered when assessing mesh independence. The optimal mesh for subsequent simulations was selected based on consistent results with minimal variation. Three tests were run for 1,000 iterations each on three meshes with elements of3,18,065, 4,54,578, and 6,86,764 as represented in Table 1.

The Figure 3 shown above illustrates the maximum velocity observed in the velocity profile for varying numbers of elements. The horizontal x-axis denotes the number of elements, while the vertical y-axis embodies the maximum velocity. The graph indicates that as the quantity of elements increases, the max. velocity also increases. This observation suggests that a more comprehensive discretization of the model offers a more detailed representation of the velocity profile. Subdividing the model into smaller segments by swelling the quantity of elements captures the velocity variations more precisely, resulting in higher maximum velocity values. This improved resolution enables the detection of localized features or fluctuations in the velocity profile.

Figure 4 exposes that an increase in the quantity of elements primes to a rise in the maximum pressure. This suggests that a finer mesh or higher resolution model achieved by adding more elements can improve pressure estimation. A finer mesh can capture smaller-scale variations and intricate details of the flow field, leading to higher pressure estimates at localized regions or near flow obstructions.

The Figure 5 shows a similar trend of increasing maximum WSS with a higher number of elements. A finer mesh with more elements can better resolve small-scale flow features and variations [18], making localized regions with higher WSS visible [19]. Additionally, the complexity of the flow field and geometry of the artery can contribute to this behavior. The Mesh Independent Study we conducted

Journal of Polymer & Composites
Volume 11, Special Issue 13
ISSN: 2321-2810 (Online), ISSN: 2321-8525 (Print)

guaranteed that the simulations accurately portrayed the crucial flow characteristics in the carotid artery, without placing an unreasonable computational burden. In our investigation, we opted for a Medium Mesh, which strikes a balance between computational efficiency and the requirement to resolve the flow features of interest.

Boundary Conditions

The optimal pick of boundary conditions is imperative in procurement of field of flow interest. Realistic blood flow conditions were emulated by applying physiologically relevant boundary conditions. The inlet conditions were set with a blood density of 1060 Kg/m^3 and the viscosity model carreau was utilized. Atottering velocity UDF was demarcated and incorporated as an inlet condition. The walls of artery were set to no slip conditions, while both the outlets were set to outflow condition. Figure 6 displays the inflow velocity conditions for one cardiac cycles, each of which has a duration of 0.75 seconds. A time step size of 1.15 seconds and 1.30 seconds were chosen for computational efficiency and temporal resolution balance.

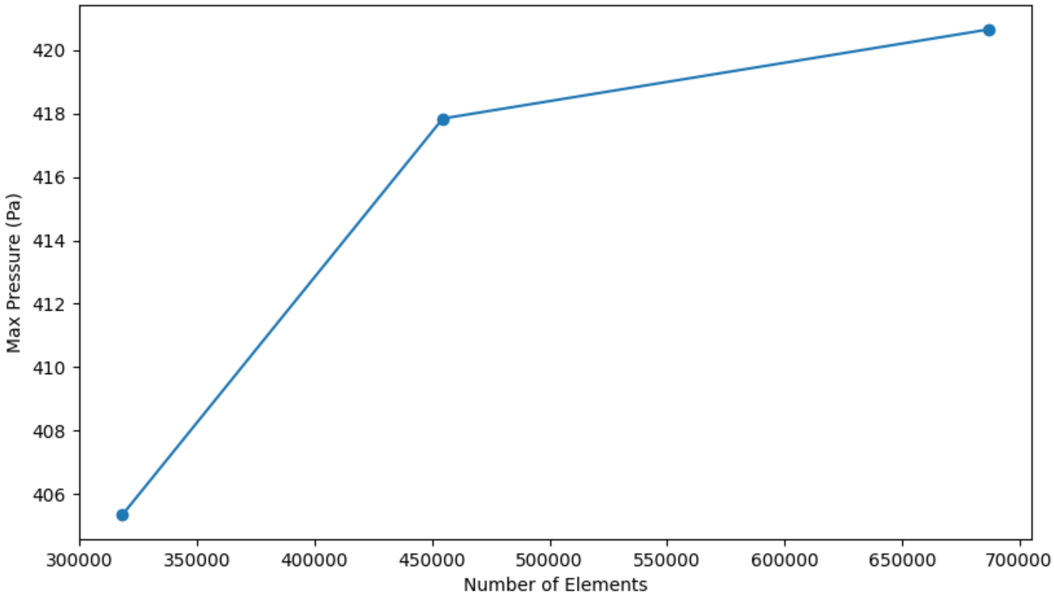

Figure 4. Max pressure vs number of elements.

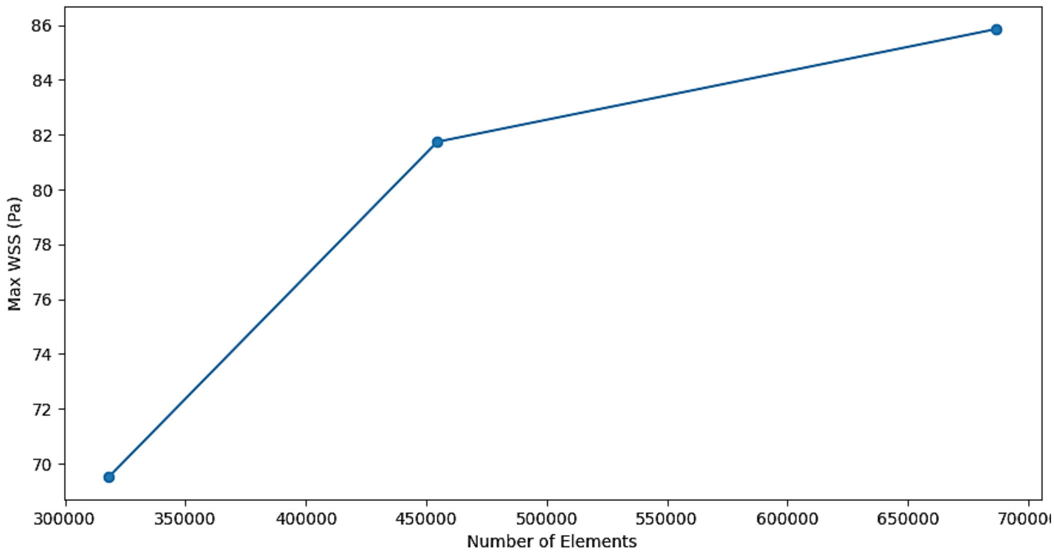

Figure 5. Max. WSS vs number of element.

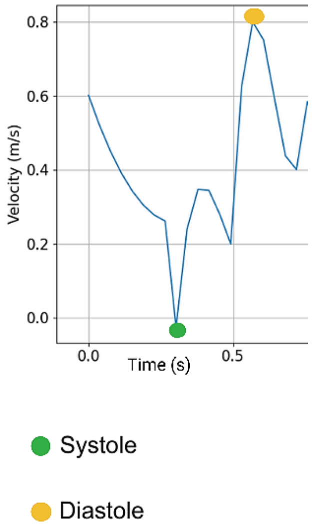

Figure 6. Velocity profile of the blood during a cardiac cycle.

Mathematical Formulation

The cruise of blood through a division in an artery can be viewed as a 3-D incompressible and time-sensitive flow. To accurately model this issue, it is necessary to fulfill the conservation principles of both momentum & mass simultaneously.

The continuity and momentum equations fundamental structures for blood with a consistent density of ρ and a fluctuating viscosity of μeff[20], the basic forms are:

$$\nabla \cdot \mu = 0 \tag{1}$$

$$\&\rho\ \delta\mu \div \delta T\ +\ \rho\nabla \cdot (\mu\ \otimes\ \mu) = -\nabla\rho + \nabla \cdot (\mu eff \nabla\mu) \tag{2}$$

where μ is the viscosity and ρ is density

The non-Newtonian model, power law relation between shear rate γ&shear stress τcan be derived as:

$$\tau\ =\ -M|\gamma|^{n} \tag{3}$$

where m and n are constants for a given hematocrit. Hence the equation mentioned above can be written as [21]:

$$\mu eff = m\ |\gamma|^{n-1} \tag{4}$$

RESULTS AND DISCUSSION

3-D incompressible transient flow in CA is simulated for above mentioned models. Two different time locations are considered at cardiac cycle namely, t_1=1.15 Sec (diastolic flow rate) and t_2=1.3 Sec (systolic flow rate). The Figure 7 illustrates the contour plots of the pressure on the bifurcated coronary artery based on a Carreau model. The pressure in the entire coronary artery ranged from -406.77 to 426.080 [22]. The site of the max. and the min. WSS are clearly marked. The maximum pressure occurs at an inlet point of the CA whereas the minimum pressure occurs at a point right outlet of the CA the pressure at time steps at 1.15s and 1.3s is maximum at 426pa and minimum at -406.77pa [18].

The Figure 8 displays velocity streamline of the bifurcated CA. The velocity streamline in the entire bifurcation region ranged from 0 to 0.879 ms^{-1} [21]. The site of the max. and the min. velocity streamline are clearly marked. The max. velocity streamline occurs at a point before the bifurcation of the CA The min. velocity streamline occurs at a point just after the bifurcation of the CA the velocity streamline at time steps at 1.15s and 1.3s is maximum at 0.879 ms^{-1} and minimum at 0 ms^{-1} [22].

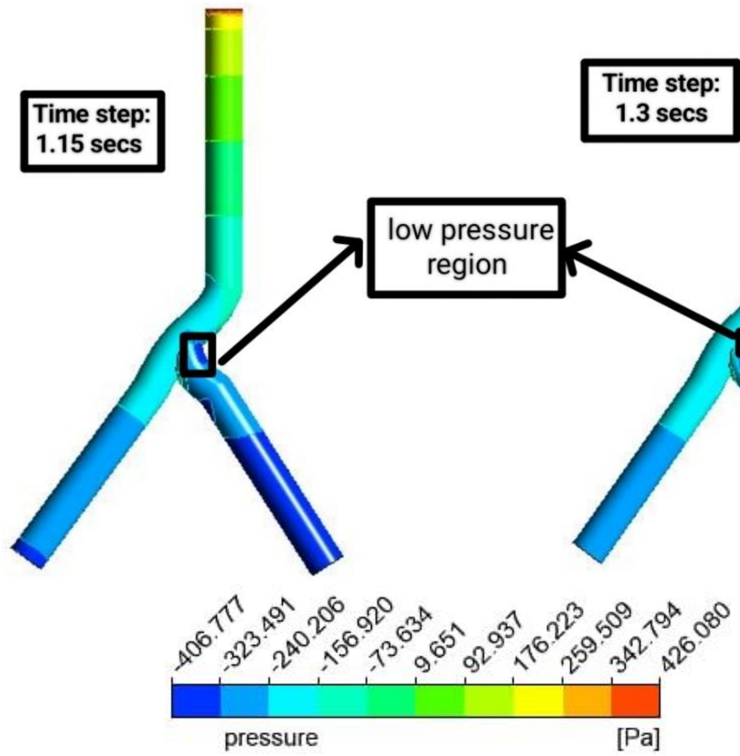

Figure 7. Pressure contour.

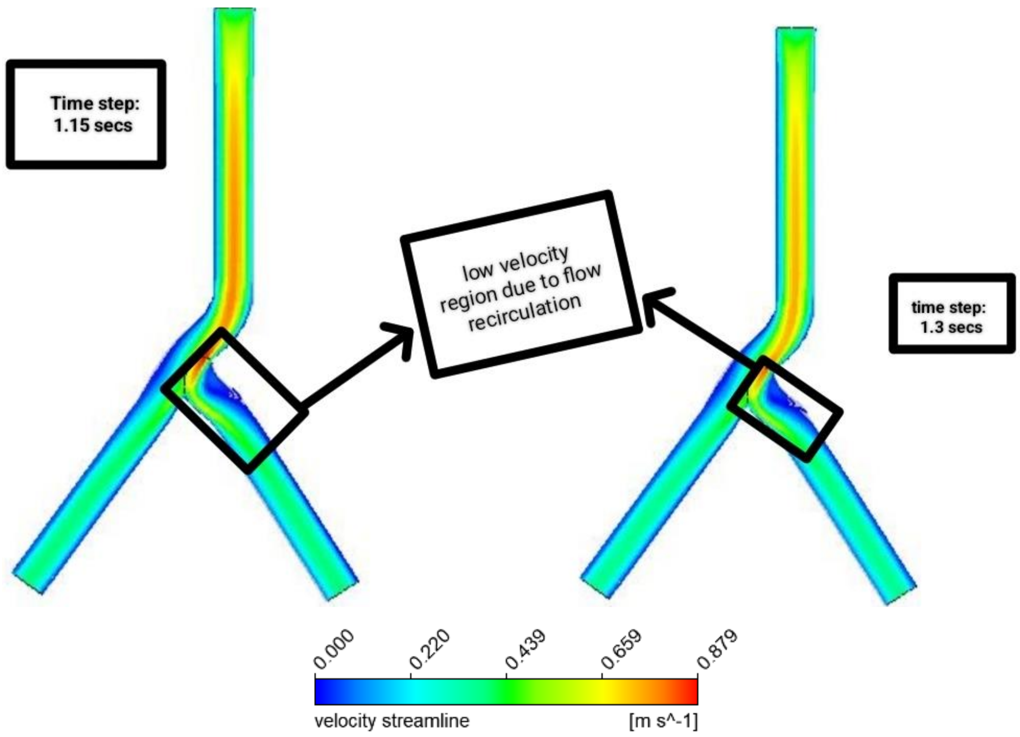

Figure 8. Velocity stream line.

The Figure 9 expresses contours WSS on the bifurcated CA. The WSS in the entire bifurcation region ranged from 0 to 25pa [19]. The place max. and the min. WSS are clearly marked the max. WSS occurs at a point just before the bifurcation of the CA whereas the min WSS occurs at a point just after the bifurcation of CA the WSS at time steps at 1.15s and 1.3s is max. at 25pa and min at 0 - 4.167pa [21].

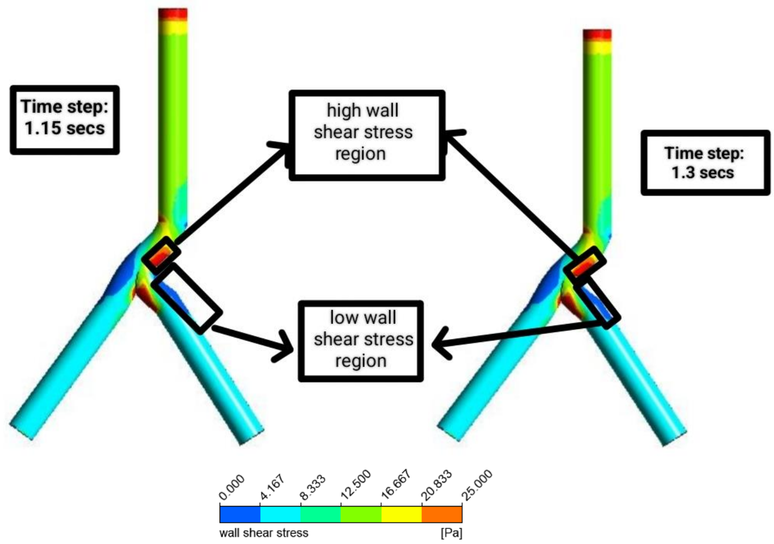

Figure 9. Wall shear stress.

CONCLUSION

This paper presents a numerical analysis (CFD) of blood flow through simplified 3-D models of a coronary artery bifurcation. The Carreau model was utilized to account for blood viscosity, and both steady state and transient conditions were examined. Non-Newtonian models have demonstrated that the distribution of WSS within bifurcated CA is non-uniform. Specific areas near the walls of bifurcations have been identified as hemodynamically vulnerable, with low WSS regions that are prone to plaque formation. The study successfully analyzed the fluid dynamics of blood flow in coronary arteries, considering the pulsatile nature of blood. However, the continuous flow field underestimates parameters like pressure, velocity, and WSS. Further studies that closely resemble the physical scenario are necessary to investigate the impact of these flow parameters on stenosis development.

Acknowledgments

We would like to express our deepest gratitude to our respected and learned guide, Dr. Abdulrajak Buradi, Associate Professor, NMIT, Bangalore, for his invaluable help and guidance throughout this project. We are thankful for his encouragement and support in completing this project. We are also grateful to Dr. Kapilan N, Head of the Department of Mechanical Engineering, NMIT, Bangalore for allowing us to use all the necessary facilities. We also extend our appreciation to the officials and staff members of the Department of Mechanical Engineering at NMIT, Bangalore who provided us with valuable knowledge and support during our project work. Finally, we would like to express our deep appreciation to our classmates and extend our indebtedness to our parents for their moral support and encouragement.

REFERENCES

1. Apostolidis, Alex J., Adam P. Moyer, and Antony N. Beris. "Non-Newtonian effects in simulations of coronary arterial blood flow." Journal of Non-Newtonian Fluid Mechanics 233 (2016): 155-165
2. Hallad, Nitesh Basavaraj, et al. "Computational Analysis of Blood Flow in a Curved Bifurcated Coronary Artery." Conference on Fluid Mechanics and Fluid Power. Singapore: Springer Nature Singapore, 2021.
3. Jhunjhunwala, Pooja, Pramod Padole, and S. Thombre. "CFD analysis of pulsatile flow and non-Newtonian behavior of blood in arteries." MCB: Molecular & Cellular Biomechanics 12.1 (2015): 37-47
4. Mahalingam, Arun, et al. "Numerical analysis of the effect of turbulence transition on the hemodynamic parameters in human coronary arteries." Cardiovascular diagnosis and therapy 6.3 (2016): 208.

5. Theodorakakos, A., et al. "Simulation of cardiac motion on non-Newtonian, pulsating flow development in the human left anterior descending coronary artery." Physics in medicine & biology 53.18 (2008): 4875
6. Buradi, Abdulrajak, and Arun Mahalingam. "Numerical analysis of wall shear stress parameters of Newtonian pulsatile blood flow through coronary artery and correlation to atherosclerosis." Advances in Mechanical Engineering: Select Proceedings of ICRIDME 2018. Springer Singapore, 2020.
7. Arzani, Amirhossein. "Accounting for residence-time in blood rheology models: do we really need non-Newtonian blood flow modelling in large arteries?." Journal of The Royal Society Interface 15.146 (2018): 20180486
8. Buradi, A., and A. Mahalingam. "Effect of stenosis severity on wall shear stress based hemodynamic descriptors using multiphase mixture theory." Journal of Applied Fluid Mechanics 11.6 (2018): 1497-1509.
9. Abugattas, C., et al. "Numerical study of bifurcation blood flows using three different non-Newtonian constitutive models." Applied Mathematical Modelling 88 (2020): 529-549.
10. Hallad, Nitesh Basavaraj, et al. "Computational Analysis of Blood Flow in a Curved Bifurcated Coronary Artery." Conference on Fluid Mechanics and Fluid Power. Singapore: Springer Nature Singapore, 2021.
11. Soulis, Johannes V., et al. "Non-Newtonian models for molecular viscosity and wall shear stress in a 3D reconstructed human left coronary artery." Medical engineering & physics 30.1 (2008): 9-19
12. Buradi, Abdulrajak, Sumant Morab, and Arun Mahalingam. "Effect of stenosis severity on shear-induced diffusion of red blood cells in coronary arteries." Journal of Mechanics in Medicine and Biology 19.05 (2019): 1950034.
13. Rezazadeh, M., & Ostadi, R. (2022). Numerical simulation of the wall shear stress distribution in a carotid artery bifurcation. Journal of Mechanical Science and Technology, 36(10), 5035-5046.
14. Dhungana, Abishek, et al. "Impact of Bifurcation and Bifurcation Angle on the Hemodynamics of Coronary Arteries." Conference on Fluid Mechanics and Fluid Power. Singapore: Springer Nature Singapore, 2021.
15. Pal, S. C., Kumar, M., & Dayal, R. (2022). A comparative study of patient-specific bifurcated carotid artery with different viscosity models. In Application of Soft Computing Techniques in Mechanical Engineering (pp. 215-230). CRC Press.
16. Buradi, Abdulrajak, and Arun Mahalingam. "Impact of coronary tortuosity on the artery hemodynamics." Biocybernetics and Biomedical Engineering 40.1 (2020): 126-147.
17. Perktold, K., Peter, R. O., Resch, M., & Langs, G. (1991). Pulsatile non-Newtonian blood flow in three-dimensional carotid bifurcation models: a numerical study of flow phenomena under different bifurcation angles. Journal of biomedical engineering, 13(6), 507-515.
18. Xu, X. Y., and M. W. Collins. "Studies of blood flow in arterial bifurcations using computational fluid dynamics." Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine 208.3 (1994): 163-175.
19. Deshpande, Prahlad V., et al. "Numerical Simulation of Blood Flow Study in an Idealized Bifurcated Coronary Artery." Conference on Fluid Mechanics and Fluid Power. Singapore: Springer Nature Singapore, 2021.
20. Nagaharish, G. N., et al. "Blood Flow Analysis Through Bifurcated and Stenosed Coronary Artery." Conference on Fluid Mechanics and Fluid Power. Singapore: Springer Nature Singapore, 2021.
21. Liu, B., and D. Tang. "Non-Newtonian effects on the wall shear stress of the blood flow in stenotic right coronary arteries." International conference on computational & experimental engineering and sciences. Vol. 17. No. 2. 2011.
22. Dahal, Prabin, et al. "Influence of Blockage on Hemodynamics of Coronary Arteries: A Numerical Investigation." Biennial International Conference on Future Learning Aspects of Mechanical Engineering. Singapore: Springer Nature Singapore, 2022.

