Investigation of Blood Flow Behavior in an Aneurysm Sac Using Computational Fluid Dynamics

Görkem GÜÇLÜ[1], Talha GÜNAYDIN[1], Can ÜNSAL[1], Ali Bahadır OLCAY[1,*]
[1]Yeditepe University, Faculty of Engineering, Department of Mechanical Engineering, Kayisdagi Cad., 34755, Istanbul 34755, Turkey.
Corresponding author's e-mail address[*]: bahadir.olcay@yeditepe.edu.tr

Cem BİLGİN[2], Bahattin HAKYEMEZ[2]
[2]Uludag University School of Medicine, Department of Radiology, Gorukle, Bursa 16059, Turkey

***Abstract*—Blood flow behavior inside an aneurysm is a complex three-dimensional fluid mechanics problem and clarity on this subject can improve or enhance the effectiveness of the treatment. In this study, the fluid flow properties such as velocity, viscosity, wall shear, and pressure are examined by using a computational fluid dynamics (CFD) technique. The patient's angiographic images are provided by the Radiology Department of Uludag University Medical School. These images are converted into a three-dimensional solid model; then, this model is used as a fluid domain in CFD. The blood flow is defined as non-Newtonian, laminar and parabolic pulsatile with two different maximum velocities so that the difference in velocity can be evaluated. The findings are presented in charts for each fluid flow parameter (e.g., velocity, viscosity, wall shear, pressure).**

Keywords*—aneurysm sac; internal carotid artery; computational fluid dynamics

I. Introduction

Aneurysms are formed and developed by the blood pressure due to the weakness of the arterial wall. Blood pressure eventually forms a bulged structure at the artery with a heartbeat. Generally, an aneurysm may not cause any symptom and it can be fatal when an aneurysm rupture occurs. Rupture of a cerebral aneurysm causes subarachnoid hemorrhage with potentially severe neurologic complications [1].

It is hard to be completely sure about deciding to intervene in the aneurysm when it is unruptured. Even though the intervention is done successfully, there is no guarantee that the patient will respond to the treatment. The success of the treatment highly depends on clot formation [2]. As researches show, the clot formation depends on many factors among with viscosity. High blood velocity inside the aneurysm prevents blood viscosity from getting higher and does not allow fibrinogen proteins to stay inside the aneurysm by restricting the clot formation [3].

High blood velocity inside an aneurysm simultaneously causes high wall shear that can lead to blood leakage from the aneurysm. Measuring the parameters such as wall shear is difficult and expensive with experimental methods [4]. However, the CFD model can provide clarity on these parameters.

Thanks to modern modeling techniques, three-dimensional aneurysms can be obtained by using a patient's angiographic images. Thus, aneurysms can be analyzed by using computational fluid dynamics methods. ANSYS CFX was used for this work and the physical properties of the blood are accurately defined to obtain reliable results. The inspiration for this study is to be able to see the behavior of blood flow inside an aneurysm that belongs to a real patient. Since blood is a shear-thinning non-Newtonian fluid, its viscosity is variable. Therefore, a realistic model of blood viscosity is used in CFD to get accurate results [5]. The important feature of the blood flow is pulsatile behavior due to the intermittent ejection of blood from the left ventricle chamber to the system. Under normal conditions, the arterial flow is considered as laminar flow [6] since the Reynolds number in the artery stays less than 2300.

II. Methodolody

By obtaining the brain tomography scans, three-dimensional geometry is created by using the 3D Slicer and MeshMixer programs. Then, the geometry is smoothed further and lastly, the geometry is turned into a fluid domain for CFD model by using ANSYS SpaceClaim tool.

After obtaining the fluid domain, the aneurysm is cut into three pieces with identical volumes, so that the differences between viscosity, velocity, pressure and wall shear stress can be analyzed region by region. Time and mesh independencies are also performed. This process is done with half heartbeat cycle for different mesh and time step sizes until the results no longer indicate significant changes. The boundary conditions are introduced and explained with respect to numbering in Figure 1. Briefly, surface number 1 is the velocity inlet of the the CFD model. The velocity profile is introduced in here as a parabolic pulsatile profile with maximum values of 0.3m/s and 0.6m/s so that the effect of velocity on viscosity and wall shear can be demonstrated. Surface numbers 2, 3 and 4 are defined as to be pressurized openings rather than outlets because of the probability of reverse flow [4]. The opening

978-1-7281-2420-9/19/$31.00 ©2019 IEEE

condition allows reverse flows. Therefore, these surfaces are pressurized as 80 mmHg which is a mean blood pressure for the patient. The walls of the whole model are defined as rigid which means no deflections are allowed.

After these steps, the blood is defined as the CFD model with a non-Newtonian Bird-Carreau model. The density of blood is 1047 kg/m^3 and high and low shear viscosity values are selected as 0.0035 Pa.s and 0.056 Pa.s, respectively [7]. Finally, CFD analyses were performed for two different maximum velocity values and the results are obtained.

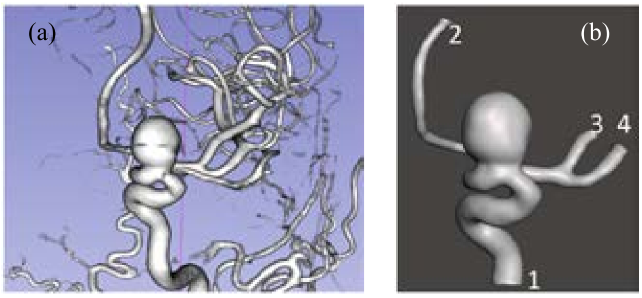

Fig 1. 3D Slicer (a) and MeshMixer (b) appearance of the geometry

III. Results

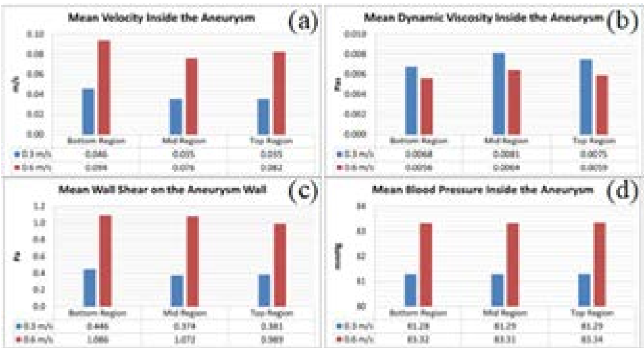

Figure 2 shows the data obtained from the CFD analysis. The fluid flow property values are shown to compare for each region and velocities. The results show that when the inlet velocity of the artery is high, the mean velocity in the aneurysm is large as expected as given in Figure 2(a). Besides, the dynamic viscosity of the blood is increased where the velocity is getting declined (Figure 2(b)). Mean wall shear stress was analyzed only at the walls of the aneurysm and it was indicated that mean wall shear stress is high at the walls for high-velocity simulations as shown in Figure 2(c). Moreover, mean blood pressure was inclined when the inlet velocity is high for the simulations (Figure 2(d)). On the other hand, important differences are observed when different velocities are considered. All fluid flow parameters, which can lead to a leakage or rupture, increase when the maximum velocity becomes large. Furthermore, mean high blood pressure and mean wall shear stress causes an extra load in the radial, axial and tangential directions on the aneurysm.

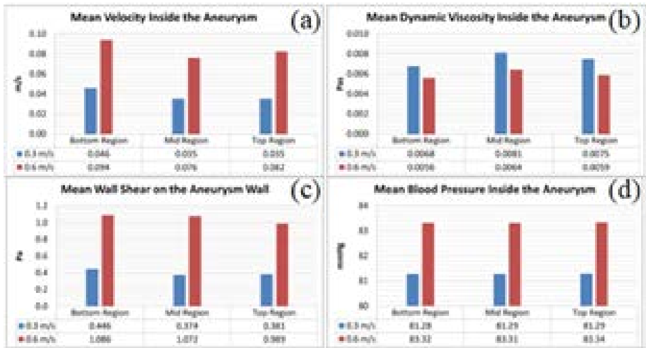

Fig 2. CFD Post Data inside the Aneurysm and on the Aneurysm Wall. Mean Velocity (a), Mean Dynamic Viscosity (b), Mean Wall Shear Stress (c), Mean Blood Pressure (d)

In Figure 2, it is noted that mid-region viscosity values appear to be larger than both the bottom and top region. This actually implies that blood flow in the aneurysm sac occurs near the aneurysm wall and blood flow is slower at the mid-region than other regions. This is also supported by the mean velocity values at the mid-region since the mean velocity values are smaller than other regions as well. When mean wall shear values given in Figure 2 are considered, the increase in inlet velocity from 0.3 to 0.6 m/s causes more than two-fold rise in the mean wall shear values on the aneurysm wall. Lastly, no significant differences are observed among bottom, mid and top regions for mean blood pressure in the aneurysm sac. On the other hand, velocity contour plots are presented in Figure 3 to compare the effect of inlet velocity of 0.3 and 0.6 m/s. It is noted that 0.6 m/s inlet velocity causes higher fluid velocities inside the aneurysm sac.

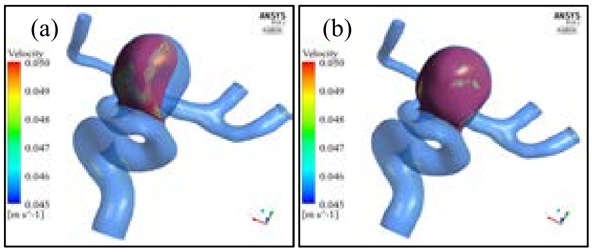

Fig 3. CFD post velocity values for 0.3m/s (a) and 0.6m/s (b) inlet velocities

Figure 4 indicates that how magnitudes of velocity vectors vary inside the aneurysm sac according to inlet velocity values. When the pulsatile flow is at its maximum value, the aneurysm allows more and high-speed blood inside it. Also, when high speed occurs inside the aneurysm, the vortex that is shown Figure 4 is getting larger in the mid-region. It prevents the clot to form. So, aneurysm is getting larger by the time.

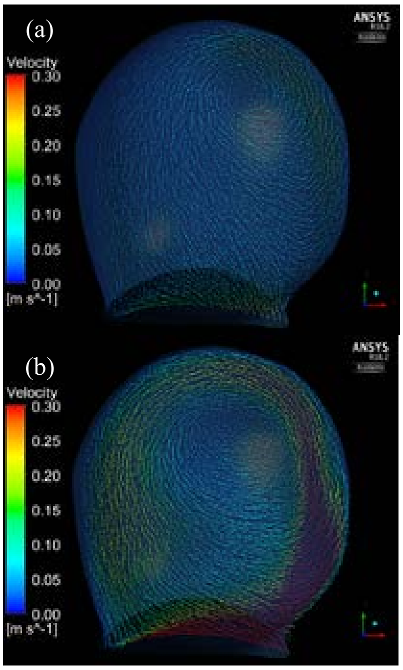

Fig 4. CFD post velocity vector plots for 0.3m/s (a) and 0.6m/s (b) inlet velocities

IV. Conclusion

In the present study, the fluid flow behavior of blood in the aneurysm sac is evaluated at three different regions of the sac. While blood velocity was obtained to be smaller in the mid-region compared to bottom and top planes, high velocity was observed near the aneurysm sac wall. It is noted that small velocity values in the mid-region can cause large dynamic viscosity at that location. The mean dynamic viscosity values also support these findings. On the other hand, while treating the aneurysm, the main objective is to slow down the blood flow in the aneurysm sac. This builds a stagnation region in the aneurysm, so the aneurysm can no longer get any blood flow from the main artery. Besides, stagnation region implies that velocity of blood flow to be very small and this indicates that the dynamic viscosity at those locations can reach very high values. In other words, when the velocity of blood flow decreases, the dynamic viscosity of the blood increases and it leads to a stagnation region inside the aneurysm sac. As previous studies discussed, the stagnation regions will lead to a clot formation later [1][7].

Acknowledgment

This work has been supported by TUBITAK (The Scientific and Technological Research Council of Turkey) under 1001 Program, Project #: 117M491.

References

[1] A. Valencia, H. Morales, R. Rivera, E. Bravo, and M. Galvez, "Blood flow dynamics in patient-specific cerebral aneurysm models: The relationship between wall shear stress and aneurysm area index," *Med. Eng. Phys.*, vol. 30, no. 3, pp. 329–340, 2008.

[2] O. Article, "Computational modelling of clot development in patient-specific cerebral aneurysm cases," no. November 2015, pp. 262–272, 2016.

[3] S. S. Lazarus, "Studies on clot density*," no. March, pp. 99–106, 2018.

[4] M. Jahangiri, M. Saghafian, and M. R. Sadeghi, "Effects of Non-Newtonian Behavior of Blood on Wall Shear Stress in an Elastic Vessel with Simple and Consecutive Stenosis," *Biomed. Pharmacol. J.*, vol. 8, no. 1, pp. 123–131, 2015.

[5] J. E. Hippelheuser, A. Lauric, A. D. Cohen, and A. M. Malek, "Realistic non-Newtonian viscosity modelling highlights hemodynamic differences between intracranial aneurysms with and without surface blebs," *J. Biomech.*, vol. 47, no. 15, pp. 3695–3703, 2014.

[6] B. Thomas and K. S. Sumam, "Blood Flow in Human Arterial System-A Review," *Procedia Technol.*, vol. 24, pp. 339–346, 2016.

[7] O. Mutlu, A. B. Olcay, C. Bilgin, and B. Hakyemez, "Evaluating the effectiveness of two different flow diverter stents based on the stagnation region formation in an aneurysm sac using lagrangian coherent structure," *World Neurosurg.*, 2019.