 doi:10.20944/preprints202106.0234.v1

Computational fluid dynamic (CFD) simulation of Bifurcate Artery

Yas Barzegar

[1] Mechanical Engineering Department, Sapienza University, Rome, Italy

Email: barzegar.1799951@studenti.uniroma1.it

Atrin Barzegar

[2] Computer Engineering Department, Sapienza University, Rome, Italy

Email: barzegar.1839406@studenti.uniroma1.it

Abstract:

Heart attacks and strokes are one of the leading causes of death in the world today, and heart attacks caused by clogged arteries that carry blood to the heart muscle are a significant part of these strokes. These are caused by the accumulation of fat particles in the walls of the arteries and the reduction of blood flow through it over a long process. The process of fat penetration in the underlying layers of the Artery wall has been the focus of many researchers, and various researches and Simulations have been done on it, in each of them, the effect of specific parameters has been considered. In the present study, the effect of blood flow rate on the flow pattern in a bifurcate artery with two ducts has been investigated using FLUENT software with Computation fluid dynamic Method. The effect of the angle between the two ducts of the Artery on the flow pattern has been investigated.

Keywords: flow pattern, Artery wall, heart muscle, flow rate, computational fluid dynamic

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 8 June 2021 doi:10.20944/preprints202106.0234.v1

1-Introduction:

Cardiovascular diseases are one of the leading causes of death in the world today. For example, in a report published by the American Heart Association [1], about 20 million people in the US adult population, or one in three people, have cardiovascular disease. Also, 1.1 million Americans die each year from vascular obstruction, which accounts for 80 percent of all deaths in American society. [2] According to World Health Organization estimates, cardiovascular disease is the leading cause of death in It will be presented worldwide by 2020. In Iran, official statistics from the Ministry of Health and Medical Education show that more than 40 percent of deaths are due to cardiovascular disease and more than 13 percent are due to heart attacks. As Iran has a record of the highest heart death rate in the world. In Iran, 900 people die every day in the country due to heart complications. However, the risk of cardiovascular disease after the age of 40 is 43% in men and 93% in women. According to a survey by the Ministry of Health, 49.3% of people are overweight, 16.1% have high blood pressure and 92.3% of people are not physically active. [3] This statistic indicates an increase in non-communicable diseases, especially cardiovascular diseases, in the future, while 20% of premature deaths are due to preventable heart diseases. [4] Therefore, studying intravenous flow patterns and examining the factors affecting them will lead to a reduction in mortality. The main cardiovascular disease is vasoconstriction, which generally occurs in medium and large diameter arteries. Clogged arteries sometimes present as heart attacks, half of which result in death. The main arteries exposed to these diseases are often the arteries, of which the carotid artery and its coronary arteries are the most affected. More than 100 factors are known to cause coronary heart disease. But the major proven causes for clogged arteries are smoking, high blood pressure, diabetes, stress and genetic background. The observers considered the effect of the pulsation of the current as well as the change in the curvature of the vessel and obtained an approximate value for the LDL discharge from the wall over time by using the single layer model for the wall. [5] Also, Fazli et al. [6] predicted the effect of flow pulse and areas prone to high fat concentrations in the geometry of clogged carotid arteries. The importance of this issue prompted researchers to take a closer look at specific cases such as forks [7], curvatures [8], the effect of secondary currents and the rotation region.

2-Artery wall structure:

The artery wall generally has 6 different layers. These layers are shown in the figure. Glycocalyx 1, the first layer in contact with the bloodstream, is stretched as a very thin coating (about 500 nm thick) on the endothelial layer. The endothelial layer is made up of a row of endothelial cells that separate the lumen from the vessel wall. The next layer (intima) is a quasi-homogeneous layer consisting of collagen and proteoglycan fibers and its thickness at normal pressure and without clogging is about 155 nm. This layer is in a state

Clogging of the arteries can also involve smooth muscle cells. The post-intima layer is the inner elastic layer (IEL) that acts impervious to particles. This layer has scattered holes on its sides through which particles pass and their diameter is about 91 nm. The thickness of this layer is about 1 micrometer. The fifth layer is called the media layer, which consists of smooth muscle cells and elastic connective tissues and has the greatest thickness between different layers. In this layer, on the surface, it has a thickness of 16 between different layers (about 200 micrometers). LDL particles in this layer are adsorbed on smooth muscle cells. The last layer of the vessel wall and its outer layer, the Adonticia layer. This layer is composed of fibrous tissues that are mainly made of elastic and lymphatic fibers.

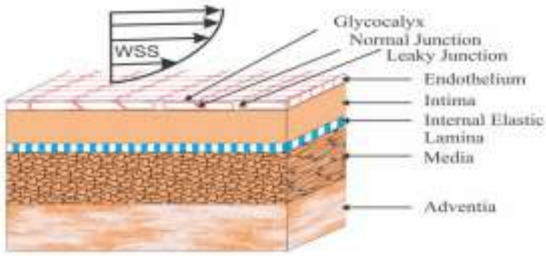

Figure 1: Artery wall structure

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 8 June 2021 doi:10.20944/preprints202106.0234.v1

3-Computational Fluid Dynamic:

Computational Fluid Dynamic is a branch of science to prediction fluid flow, heat transfer, mass transfer (eg in decomposition), phase change (in freezing and boiling), chemical reactions (such as combustion), mechanical displacement (such as fan rotation) Stress or deformation in the structure of solids (such as bending of beams due to wind), and other phenomena, and this prediction is done by solving the mathematical equations governing the processes mentioned above and by using numerical algorithms in the computer.

4-Procedure and Methodology:

Finite Difference methods for finding the approximate solution of incompressible Navier-Stokes equations differ significantly in terms of accuracy and efficiency. In finite difference methods, although the central differences are second order, most of these methods suffer from numerical instability. Upwind methods are stable and have first-order accuracy and are artificially affected. Second-order methods are not better for high Reynolds numbers than first-order methods. High-order methods are stable and have high numerical accuracy. The solution method used in this project is Upwind second order.

4-1 Geometry of Model:

The following figure shows the dimensions and geometry of the simulated model. As shown in the figure, the angle between the two branches of the vessel is 140 degrees and the diameter of the vessels is 3.23 and 2.41, respectively.

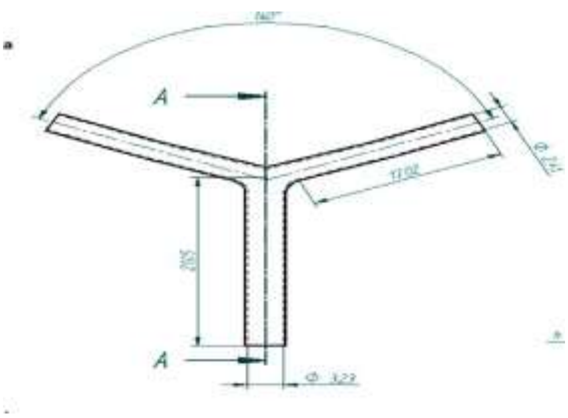

Figure 2: Geometry of Model

As shown in the figure below, 3D modeling was performed in catia software version 21.

Figure 3:3D Modeling in Catia Software

4-2 Boundary conditions:

The boundary conditions governing the simulation are given in the table 1.

Table1 :Boundry Conditions

cm/s38.8:Inlet velocity	**Boundary conditions**
Pascal 0: Outlet Pressure	
kgm/s :Blood viscosity 0.003	
laminar:Type of Flow	
No slip condition	

4-3 Simulation steps:

1. Read the Gambit model
2. Check the model
3. Apply mechanical properties to the model
4. Apply boundary conditions to the model
5. Select the type of solvent

4-4 Meshing the Model:

The following figure 5 shows the meshing of the bifurcated vessel model in Gambit software. In the meshing environment, we must specify the type of element and the distances between the elements. The elements used for meshing the plates are of the triangular type. The T-grid element is used to meshes the volume of the

 doi:10.20944/preprints202106.0234.v1

shape. The distances used between the elements are 0.001.

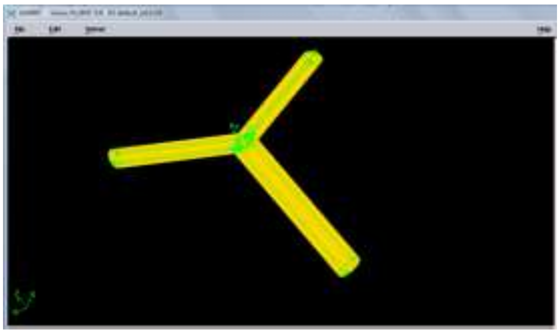

Figure 4:Meshing Biffurcate vessel

5-Evaluate research results

In this section we want to evaluate pressure contour and velocity contour and effect of flow rate on Pressure and velocity in Bifurcate vessel**.**

5-1 Pressure Contour

The figure 6 shows the static pressure contour applied to the bifurcate artery. In this case, the flow rate is the same in both ducts. As can be seen, the static pressure at the bifurcation site as well as at the inlet site is maximal, reaching 1.35 Pascal. At the end of the two ducts it reaches the value of zero Pascal.

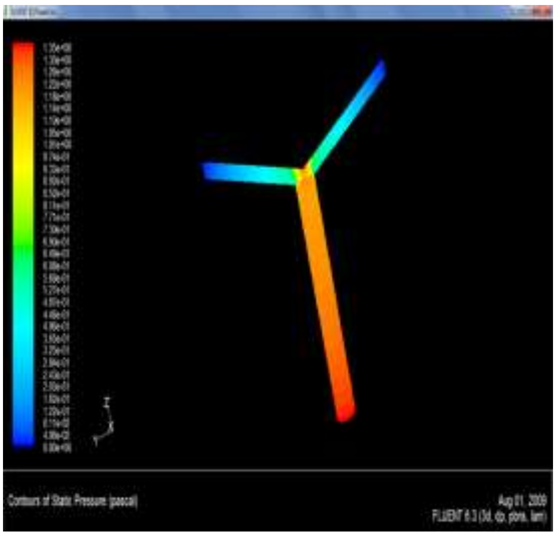

Figure 5:Pressure Contour

5-2 Velocity Contour

The figure 7 shows the velocity contour on bifurcate artery. In this case, the flow rate is the same in both ducts. As can be seen, the maximum speed at the internal cross-section of the two ducts has reached a maximum of 1.04 m / s.

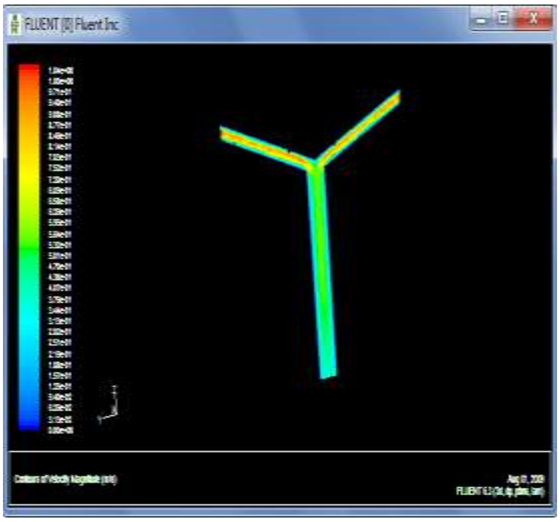

Figure 6:Velocity contour

5-3 Effect of flow rate on pressure contour:

The following figure 8 shows the pressure contour in the two ducts. The flow rate in the two ducts is 0.2 (in the left duct) and 0.8 (in the right duct). As can be seen, the pressure in the left ventricle has increased.

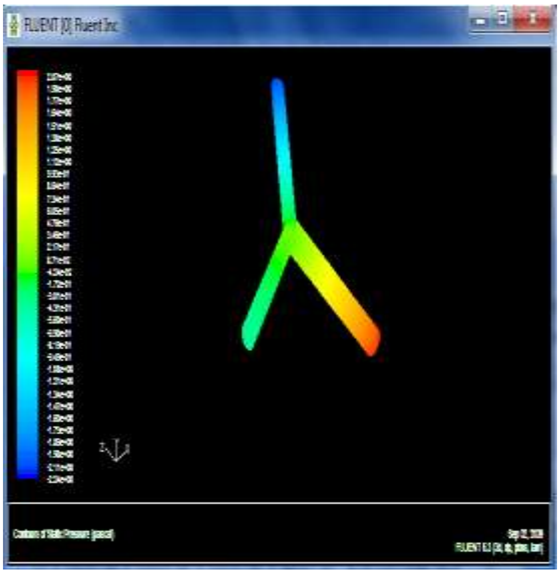

Figure 7: Effect of flow rate on pressure contour

The following figure 9 shows the pressure contour in the two ducts. The flow rate in the two ducts is 0.6 (in the left duct) and 0.4 (in the right duct). As can be seen, the pressure in the right duct has increased.

 doi:10.20944/preprints202106.0234.v1

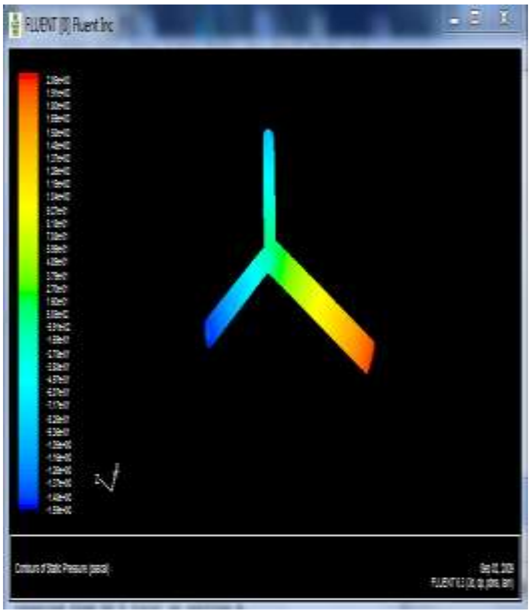

Figure 8:Pressure contour

5-4 Effect of flow rate on Velocity contour:

The figure 10 shows the velocity contour in the two ducts. The flow rate in the two ducts is 0.8 (in the left duct) and 0.2 (in the right duct). As can be seen, the velocity in the left duct has increased to 0.83 m / s.

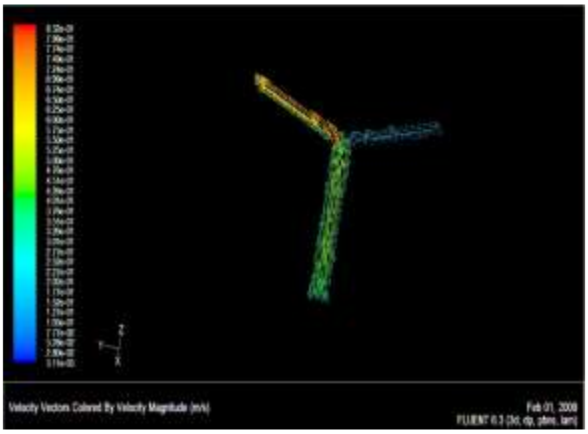

Figure 9:Velocity contour

5-5 Pressure contour in new model with different angle between two Branches

The pressure contour in the other model is shown with distances between the branches of the duct of 70 degrees.

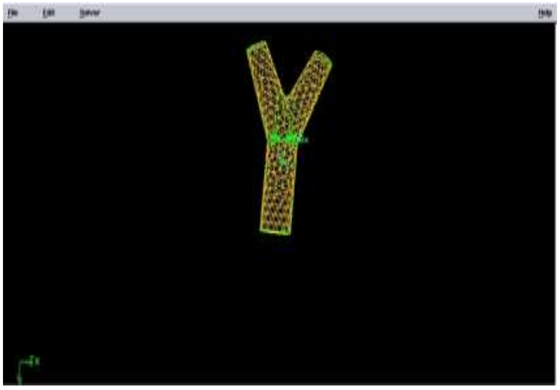

Figure 10:Meshing new Model

In the figure 11 shows the Pressure contour in new Model.

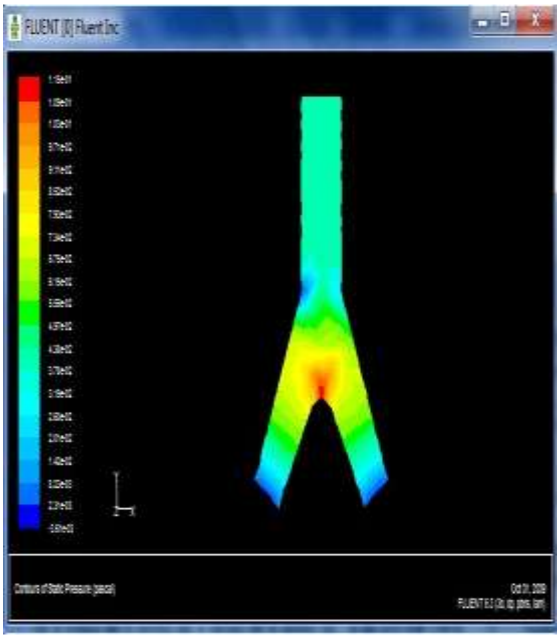

Figure 11:Pressure Contour

6-Results and discussion:

As mentioned before, the static pressure at the bifurcation site as well as at the inlet site is maximal, reaching 1.35 Pascal. At the end of the two ducts it reaches the value of zero Pascal. the maximum speed at the internal cross-section of the two ducts has reached a maximum of 1.04 m / s. when the flow rate increased, the Pressure Decreased

7- Conclusion:

With increasing flow in the duct and also decreasing the angle between the two ducts from 140 degrees to 70 degrees, the speed of blood flow has increased. So that when the

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 8 June 2021 doi:10.20944/preprints202106.0234.v1

angle between the two ducts decreases from 140 degrees to 70 degrees, the maximum speed from 1.04 meters per second to 1.44 Meters per second increased.

Refrences:

[1] Kevin Mc Namara, Hamzah Alzubaidi, and John Keith Jackson, "Cardiovascular disease as a leading cause of death: how are pharmacists getting involved?",2019

[2] Salim S. Virani, Alvaro Alonso , Hugo J. aparicio, Heart Disease and Stroke Statistics, Vol. 143, No. 8,2021

[3] MehrNewsAgency. http://www.mehrnews.com ,2014

[4] http://www.tebyan.net/newindex.aspx?pid,2010

[5]M. K. Kolandavel, E.-T. Fruend, S. Ringgaard, and P. G. Walker, "The effects of time varying curvature on species transport in coronary arteries," Annals of biomedical engineering, vol. 34, pp. 1820-1832, 2006.

[6] S. E. Fazli S., Sadeghi M.R., "Numerical simulation of LDL mass transfer in a common carotid artery under pulsatile flows," Journal of Biomechanics, vol. 44, pp. 68-76, 2011.

[7]M. Motomiya, Karino, T" ,.Flow pattern in the human carotid bifurcation," Stroke, vol. 15, pp. 50-56, 1984.

[8] S. Wada, Karino, T., "Theoretical prediction of low-density lipoproteins concentration at the luminal surface of an artery with multiple bend," Annals of Biomedical Engineering, vol. 30,pp. 778-791, 2002.

Authors:

[1] Yas Barzegar, Mechanical Engineering Department, Sapienza University, Rome, Italy

Email:barzegar.1799951@studenti.uniroma1.it

[2] Atrin Barzegar, Computer Engineering Department, Sapienza University, Rome, Italy

Email:barzegar.1839406@studenti.uniroma1.it