Dynamical behaviour of non newtonian spiral blood flow through arterial stenosis

Mohammad Ali¹, Md. Mahmudul Hasan¹ and Mahbub Alam Maruf¹

¹Department of Mechanical Engineering, Bangladesh University of Engineering and Technology, Dhaka-1000, Bangladesh.

E-mail: mali@me.buet.ac.bd

Abstract. The spiral component of blood flow has both beneficial and detrimental effects in human circulatory system. A numerical investigation is carried out to analyze the effects of spiral blood flow through an axisymmetric three dimensional artery having 75% stenosis at the center. Blood is assumed as a non-Newtonian fluid. Standard $k-\omega$ model is used for the simulation with the Reynolds number of 1000. A parabolic velocity profile with spiral flow is used as inlet boundary condition. The peak values of all velocity components are found just after stenosis. But total pressure gradually decreases at downstream. Spiral flow of blood has significant effects on tangential component of velocity. However, the effect is mild for radial and axial velocity components. The peak value of wall shear stress is at the stenosis zone and decreases rapidly in downstream. The effect of spiral flow is significant for turbulent kinetic energy. Detailed investigation and relevant pathological issues are delineated throughout the paper.

1. Introduction

At present, congenital and acquired cardiovascular diseases have become one of the most important causes of morbidity and mortality in the world. Among the acquired cardiovascular diseases, atherosclerosis is the most common. These diseases are often characterized by arterial stenosis. Arterial stenosis refers to the localized narrowing of arterial passages. This stenosed artery is responsible to reduce the maximum flow of blood. It happens as a result of coagulation of calcium and fatty materials such as cholesterol and triglyceride. This deposition is named as plaque. In the worst cases plaque may rupture, causing the formation of thrombus cause of death of the tissues. This is the main reason for brain strokes or heart attack and sometimes causes inadequate blood flow in the lower part of the body. In recent years, researchers are interested to simulate the blood flow by computational techniques in a model of three dimensional arteries. The hemodynamical characteristics of flow through arterial stenosis have been continually investigated numerically and experimentally for single as well as multiple restrictions. An experimental investigation of pulsating flow through a smooth constriction was conducted by Ahmed [1]. These experiments were conducted over physiologically relevant mean Reynolds number of 600 based on the tube diameter and the time-averaged value of upstream centerline velocity. A numerical investigation was carried out for laminar sinusoidal pulsating flow through a modeled arterial stenosis with a trapezoidal profile by Hasan and Das [2]. Finite element based numerical technique was used to solve the fluid flow governing equations. The effects of pulsation,
stenosis severity, Reynolds number and Womersley number on the flow behavior were studied. Investigation of physiological pulsatile flow in a model arterial stenosis using large-eddy and direct numerical simulations was done by Molla [3]. Physiological pulsatile flow in a 3D model of arterial stenosis was investigated by using large eddy simulation (LES) technique. The effect of spiral flow of blood was studied by Paul [4]. They investigated for 75% reduction in cross sectional area by standard $k-\omega$ model for Reynolds number 500 and 1000 considering blood as a Newtonian fluid. A numerical simulation of unsteady blood flow through multi-irregular arterial stenosis was conducted by Mustapha [5]. In this paper, effects of spiral blood flow through arterial stenosis are numerically investigated by CFD analysis. Standard $k-\omega$ model is used to simulate the model. Blood is considered as a Non-Newtonian fluid and Reynolds number of 1000. Artery wall is assumed as rigid.

1.1. Nomenclature

- $D$: Artery diameter (m)
- $\rho$: Density (kg/m$^3$)
- $\mu$: Dynamic viscosity (Ns/m$^2$)
- $V$: Stream wise bulk velocity (m/s)
- $v$: Instantaneous velocity (m/s)
- $\Omega$: Spiral velocity (rad/sec)
- $Re$: Reynolds number
- $R$: Radius of blood vessel (m)
- $r$: Radial location from axis of artery (m)

2. Problem formulations

A 75% axisymmetric three dimensional artery is considered here. A parabolic velocity profile with swirling flow is used as inlet boundary condition. The stenosis in the blood vessel is created by using cosine formule. Both 2D and 3D models of calculation domain are shown in figure 1..The total length of the model is taken as 540 mm (27D) where diameter $D=20$ mm. Four different spiral velocities are used for investigation. The inlet boundary condition for the stream wise velocity and spiral speed are written in C-language using the interface of User Defined Function (UDF) of Fluent and linked with the solver.

![Figure 1](image.jpg)

Figure 1. a) 2D model of stenosed artery (b) 3D model of stenosed artery.

3. Numerical method

The problem is solved numerically with simulating 2D and 3D Navier Stokes equations using proper boundary conditions.

3.1. Governing equations

Navier Stokes equation represents the conservation laws of mass and momentum.

- Conservation of mass
\[ \frac{\partial p}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0 \quad (1) \]

- Conservation of momentum
\[ \frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial (\rho uv)}{\partial y} + \frac{\partial (\rho uw)}{\partial z} = \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \quad (2) \]
\[ \frac{\partial (\rho v)}{\partial t} + \frac{\partial (\rho uv)}{\partial x} + \frac{\partial (\rho v^2)}{\partial y} + \frac{\partial (\rho vw)}{\partial z} = \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \quad (3) \]
\[ \frac{\partial (\rho w)}{\partial t} + \frac{\partial (\rho uw)}{\partial x} + \frac{\partial (\rho vw)}{\partial y} + \frac{\partial (\rho w^2)}{\partial z} = \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \quad (4) \]

3.2. Discretization of the Domain

ANSYS FLUENT is used for the present computation. This CFD tool will solve the governing integral equations for the conservation of mass and momentum. Figure 2 shows the discretization model. The domain consists of 921416 cells, 2793268 faces and 951246 nodes. The geometry is created in solid works and imported in Ansys fluent. Quadrilateral cells were used for this geometry because they can be stretched easily to account for different flow gradients in different directions. The cells near the surface have high aspect ratios. For viscous flow through the stenosis, finely spaced Grid was constructed to calculate the details of the flow near the wall. Three different mesh numbers are used for grid independent test.

![Figure 2. Discretization of the Domain using Fluent meshing tool.](image)

3.3. Boundary Conditions

Inlet-flow: For the geometry, fully developed inlet-flow condition has been assumed as parabolic. The axial velocity distribution is given by,
\[ v(r) = 2V[1 - \left(\frac{r}{R}\right)^2] \quad (5) \]
Where, \( V \) = Streamwise mean velocity

Outlet-flow: In case of outlet boundary condition, the gauge pressure is assumed to be zero Pascal.
Wall: At all solid boundaries in the flow geometry, no-slip condition has been used. It can be mathematically expressed as,
\[ u_{wall} = 0, \text{ and } v_{wall} = 0 \quad (6) \]

3.4. Fluid Property
Fluid Material is blood, which is the working fluid in this problem. For this analysis blood is assumed as non-Newtonian fluid. Here, typical values of density, \( \rho = 1060 \, \text{kg/m}^3 \) and viscosity, \( \mu = 0.00371 \, \text{N.s/m}^2 \) are used to calculate velocity from the equation \( \text{Re} = \frac{\rho V D}{\mu} \). There are several models to solve Non-Newtonian fluid problem in CFD. The Carreau fluid model is used here to solve the problem.

3.5. Grid-Independence Test

Since numerical results are greatly dependent on the mesh generation, a grid-independence test is performed to find out the optimum number of elements to discretize the computational domain. Wall static pressure at figure 3 and axial velocity at figure 4 are used for grid independence test. Grid 1 consists of 578835 cells, Grid 2 consists of 921416 cells and Grid 3 consists of 1435082 cells. Grid 2 is giving better results than Grid 1. On the other hand Grid 3 has 55 percent more cells than Grid 2. But, there are no significant changes in results. That’s why Grid 2 is good enough for numerical solution.

![Figure 3. Static pressure for different grids.](image)

![Figure 4. Axial velocity for different grids.](image)

3.6. Validation of results

Before going to the detailed discussion, the computational results have been validated to verify the results. The validation has been done by comparing computational results with the experimental results obtained by Ahmed and Giddens [1]. Single stenosis with 75% severity model is considered for validation. Validation was done for steady inlet velocity condition. Velocity profile, in the post stenotic regions, is compared at \( Z = 2.5D \). Where \( Z \) is the normalized distance from the centre of the stenosis. Figure 5 shows that computational results are over predicted at near the artery wall and under predicted near the center of the stenosis than experimental results. But percentage of deviation is too small. However, at middle of radius computational and experimental results are in good agreement. So this model can be used to solve the problem.
Figure 5. Velocity profile comparison of the computational results and experimental data at $Z=2.5D$.

4. Results and Discussion

CFD simulation is carried out to observe the effects of spiral flow of blood on different hemodynamic parameter. Figure 6 shows that Spiral flow has a great influence on tangential velocity only, but for axial and radial velocity, its effects are negligible. All velocity components get their maximum value just after stenosis, then gradually decrease to their initial value at downstream.

The centerline total pressure decreases drastically along the length of the stenosed artery which is shown in figure 7. The spiral velocity has little effect on centerline total pressure before stenosis and just after stenosis. But at far downstream spiral flow has no effect on centerline pressure. On the other hand, figure 8 shows that turbulent kinetic energy (TKE) decreases with the increase of spiral speed. TKE is the energy content of eddies of turbulent flow. The spiral flow blood shows the characteristics of turbulent
flow. Larger the spiral velocity, higher the energy content of eddies. These eddies extract energy from mean flow. That is why TKE decreases with increase of spiral speed. From figure 9 it is seen that wall shear stress (WSS) tends to rise at the centre of the stenosis. The reason, for this rise of shear stress at the centre of stenosis, is the interaction of vortex with artery wall.

5. Conclusion
The rise of the turbulent kinetic energy in the post-stenotic region is responsible to cause damage to the blood-cell materials. This activates platelets in the blood, and subsequently they create many pathological diseases. However, the results show that the spiral effect reduces the turbulent kinetic energy, which is a beneficial effect. At the same time spiral flow produces oscillating wall shear stress in the post stenosis, which is a detrimental effect. The oscillating shear stress usually influences to cause potential damage to the inner side of a post-stenotic blood vessel which is known as endothelium. In addition, the strong circulation in the post-stenotic region due to the spiral effect is harmful. It may cause the blood to be clotted in the post-stenosis, which is a potential source of stroke. As the spiral flow has potential clinical significance, therefore, the results have some significant impacts on the understanding of blood flow dynamics and its relevance in arterial diseases.

Acknowledgments
This work has been done using different facilities of the Department of Mechanical Engineering, Bangladesh University of Engineering and Technology (BUET).

References
[1] Ahmed, S.A., and Giddens, D.P, 1983, *Journal of Biomechanical Engineering*, 16, 505-516.
[2] Hasan, A.B.M.T., and Das, D.K, 2008, *Journal of Applied Fluid Mechanics*, 1(2), 25-35.
[3] Paul, M.C., Molla, M.M., and Roditi, G.2009, *Medical Engineering & Physics*, 31, 153-159.
[4] Paul, M.C. and Larman, A. 2009, *Medical Engineering and Physics*, 31 (9), 1195-1203.
[5] Mustapha, N., Mandal, P.K., Johnston, P.R., and Amin, N. 2010, *Applied Mathematical Modelling*, 34(6), 1559-1573.