Journal of Positive School Psychology
2022, Vol. 6, No. 2, 3336 – 3343 http://journalppw.com

Blood Flow in Arteries with Stenoses: A Three-Dimensional Unsteady Flow

[1]H. Girija Bai, [2]M. Prasanna Jeyanthi, [3]P. Hemavathy, [*4]M. Premkumar, [5]A. Mohamed Ismail

[1]*Department of Mathematics, Sathyabama Institute of Science and Technology, Chennai-600 119, Tamil Nadu, India,* girijanameprakash@gmail.com
[2]*Department of Mathematics, Sathyabama Institute of Science and Technology, Chennai-600 119, Tamil Nadu, India,* prasannajeyanthi@gmail.com
[3]*Department of Computer Applications, Hindustan Institute of Science and Technology, Chennai- 603103, Tamil Nadu, India,* phemavathy@hindustanuniv.ac.in
[*4]*Assistant Professor Department of Mathematics, Sathyabama Institute of Science and Technology Chennai-600119, Tamilnadu, India,* mprem.maths3033@gmail.com
[5]*Associate Professor, Department of Mathematics, Sathyabama Institute of Science and Technology, Chennai-600119, Tamilnadu, India,* ismailbtl@yahoo.co.in

Abstract

Three-dimensional unsteady blood flow in arteries with two axisymmetric stenoses of dilation of 25% was mathematically investigated in this work. Hemodynamic parameters such as velocity, pressure, and streamline pattern are examined using computational fluid dynamics methods. The finite volume approach is used to address the problem. The CFD applications Fluent and Gambit are used to perform numerical simulations. In the stenoses, low pressure and rapid velocity are noted. This is a sign that blood flow has been disrupted. The root of surgical interference is the detection and quantification of numerous stenoses. This research is crucial for understanding the link between hemodynamics and rupture risk.

Keywords: Multiple Stenoses; Computational Fluid Dynamics; Hemodynamics; Numerical simulation; Finite Volume Method.

Introduction

Hemodynamics is the study of the physiological variables that control blood flow in the circulatory system. Unsteady flow phenomena dominate blood flow in arteries. Blood flow in normal physiological situations, as well as blood flow in sick situations, is an important field of research. The majority of fatalities in industrialised nations are due to cardiovascular disorders, the majority of which are linked to some sort of abnormal blood flow in arteries. The arteries are dynamic organs that alter and change in response to changes in hemodynamic circumstances. Plaques arise when cholesterol deposits and connective tissue proliferate in an arterial wall, growing inward and restricting natural blood flow.

The stenosis may worsen as a result of the blockage, which may harm the interior cells of the wall. Stenosis is a condition in which a blood artery narrows abnormally as a result of the formation of vascular plaques or other forms of aberrant tissue. This type of vascular disorder is rather common, especially in mammalian arteries. This disease has the potential to be lethal if it develops into a severe version. Although the exact mechanism for the advancement of this high blood pressure is unknown, a few investigators have stated that the formation of vascular atherosclerotic plaque and the protrusion of ligaments and spurs on the blood vessel wall are two major factors in the initiation and progression of this high blood pressure.

Due to technical interest as well as potential medicinal applications, the homodynamic behaviour of blood flow in arterial vascular disease bears certain relevant elements. Young studied the impact of stenosis on the circular pipe and the dart in 1968. J. Daly (1976) [11] performed a numerical investigation of pulsatile flow from stenotic canine femoral arteries with lumen constrictions ranging from 0% to 61%. V. O'Brien and L.W. Ehrlich [1] investigated a basic pulsatile flow in a constricted artery in 1985. H. Huang et al. [9] used a finite difference approach for steady and transient flow to examine flow in a pipe with a blockage in 1995. Somkid Amornsanmankul et al. [4] explored the pulsatile blood flow via a stenotic artery in 2006.

Using the spectral element approach, Seung E. Lee et al [7] quantitatively studied the blood flow dynamic behavior in a stenosed, specific topic carotid bifurcation in 2008. Quan Long et al. [8] conducted a numerical analysis in 2008 to determine the potential of the Circle of Wills (CoW) provide the collateral blood flow for individuals with unilateral carotid artery stenosis. The consequences of identical blood flow behavior in a small artery were investigated by V.P. Srivastava et al. [3] in 2010. mathematical model for investigating unsteady blood flow via a stenosed channel under various density variations caused by antiplatelet drugs is discussed in [14]. The analysis of blood flow due to anti-platelet drug was obtained in [15]. Because of its crucial applications in physiology, the study of magnetohydrodynamic (MHD) blood flow in arteries has sparked a lot of interest. The MHD flow through circular ducts has been analyzed in [16].

The simulation of complex dynamic processes that appear in nature or in industrial applications poses a lot of challenging mathematical problems, opening a long road from the basic problem, to the mathematical modelling, the numerical simulation, and finally to the interpretation of results.

The aim of the present study is to find the unsteady flow pattern for the multiple stenoses using CFD analysis which has not been thoroughly investigated so far. Axisymmetric pulsatile flow of a viscous fluid in a constricted vessel is considered. Limitations on the amount of the constriction are ignored. Since arterial wall is gently elastic, we neglect the wall dispensability. Change in diameter in arteries is on the order of 10% [12];

METHODS

A. Formulation of the problem

The geometry of the stenosed vessel in Fig.1. is given by

$$y(x) = \begin{cases} a\left(l - \frac{\delta}{2a}\left(l + \cos\left(\frac{\pi x}{x_0}\right)\right)\right), & |x| \le x_0 \\ a, & |x| > x_0 \end{cases}$$

(1)

B. Boundary Conditions:

u=0, v=0 ,w = 0, 0n stenosed vessel

u=u(t), v=0, w=0,on inflow segment

fx=0, fy=0, fz =0, on outflow segment

fx and fy are the components of an arbitrary vector valued

function f defined by

$\overline{f} = \overline{n} \cdot \overline{\tau} = -p\,\overline{n} + \mu\,[\overline{n}\cdot(\nabla\overline{u}) + \nabla\cdot(\overline{n}\cdot\overline{u})]$ [2]

C. Governing Differntial Equaions

Equations of momentum and mass conservation for incompressible fluid can be written as:

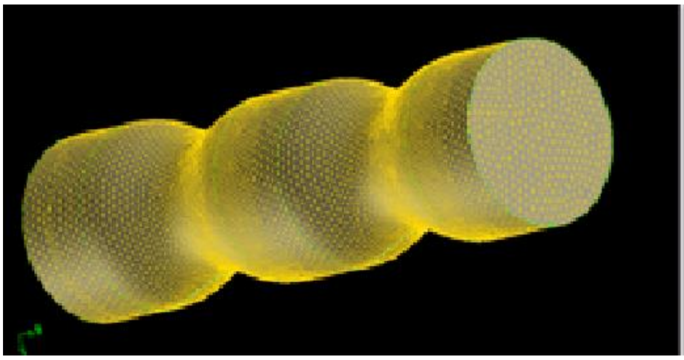

Fig.1. *An arterial vessel model with two asymmetric stenoses.*

$$\nabla \cdot \overline{v} = 0$$

(2)

$$\rho\left(\frac{\partial \bar{v}}{\partial t} + \bar{v}.\nabla\bar{v}\right) = -\nabla p + \mu\nabla^2\bar{v} \qquad (3)$$

where: ρ - density of blood, (v)¯-velocity field, p - pressure, μ = co-efficient of viscosity.

The Navier-Stokes System of Equations: Incompressible Viscous Flow case:

The vector form of unsteady three-dimensional motion of a viscous, incompressible and isothermal flow:

$$\frac{\partial \vec{Q}}{\partial t} + \frac{\partial \overrightarrow{E_i}}{\partial x} + \frac{\partial \overrightarrow{F_i}}{\partial y} + \frac{\partial \overrightarrow{G_i}}{\partial z} = \frac{\partial \overrightarrow{E_v}}{\partial x} + \frac{\partial \overrightarrow{F_v}}{\partial y} + \frac{\partial \overrightarrow{G_v}}{\partial z} \quad [13]\ (4)$$

Where Q ⃗ is the vector containing the primitive variables and Ei, Fi , Gi are the vectors containing the inviscid fluxes in the x, y and z directions is given by

$$\vec{Q} = \begin{bmatrix} 0 \\ u \\ v \\ w \end{bmatrix}, E_i = \begin{pmatrix} u \\ u^2 + p \\ uv \\ uw \end{pmatrix}, F_i = \begin{pmatrix} v \\ vu \\ v^2 + p \\ vw \end{pmatrix},$$

$$G_i = \begin{pmatrix} w \\ wu \\ wv \\ w^2 + p \end{pmatrix}$$

The viscous fluxes in the x, y and z directions are defined as follows

$$E_v = \begin{pmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \end{pmatrix} \quad F_v = \begin{pmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ \tau_{yz} \end{pmatrix}, \quad G_v = \begin{pmatrix} 0 \\ \tau_{zx} \\ \tau_{zy} \\ \tau_{zz} \end{pmatrix},$$

Since we made the assumptions of an incompressible flow, appropriate nondimensional terms and expressions for shear stresses must be used; these expressions are given as follows

$$\tau_{xx} = \frac{2}{Re_L}\frac{\partial u}{\partial x},$$

$$\tau_{yy} = \frac{2}{Re_L}\frac{\partial v}{\partial y},$$

$$\tau_{zz} = \frac{2}{Re_L}\frac{\partial w}{\partial z},$$

$$\tau_{xy} = \frac{1}{Re_L}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) = \tau_{yx} \ ,$$

$$\tau_{xz} = \frac{1}{Re_L}\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right) = \tau_{zx,}$$

$$\tau_{yz} = \frac{1}{Re_L}\left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right) = \tau_{zy}$$

Transformation of the governing equations from cartesian coordinates (x,y,z,t) to computational space with generalised curvilinear coordinates $(\xi, \eta, \varsigma, t)$

$$\frac{\partial \hat{Q}}{\partial t} + \frac{\partial \widehat{E_i}}{\partial \xi} + \frac{\partial \widehat{F_i}}{\partial \eta} + \frac{\partial \widehat{G_i}}{\partial \varsigma} = \frac{\partial \widehat{E_v}}{\partial \xi} + \frac{\partial \widehat{F_v}}{\partial \eta} + \frac{\partial \widehat{G_v}}{\partial \varsigma} \quad (5)$$

Where

$$\left.\begin{aligned}
\hat{Q} &= \frac{\vec{Q}}{J_x} \\
\widehat{E_i} &= \frac{1}{J_x}\left(\xi_x\overrightarrow{E_i} + \xi_y\overrightarrow{F_i} + \xi_z\overrightarrow{G_i}\right) \\
\widehat{F_i} &= \frac{1}{J_x}\left(\eta_x\overrightarrow{E_i} + \eta_y\overrightarrow{F_i} + \eta_z\overrightarrow{G_i}\right) \\
\widehat{G_i} &= \frac{1}{J_x}\left(\varsigma_x\overrightarrow{E_i} + \varsigma_y\overrightarrow{F_i} + \varsigma_z\overrightarrow{G_i}\right) \\
\widehat{E_v} &= \frac{1}{J_x}\left(\xi_x\overrightarrow{E_v} + \xi_y\overrightarrow{F_v} + \xi_z\overrightarrow{G_v}\right) \\
\widehat{F_v} &= \frac{1}{J_x}\left(\eta_x\overrightarrow{E_v} + \eta_y\overrightarrow{F_v} + \eta_z\overrightarrow{G_v}\right) \\
\widehat{G_v} &= \frac{1}{J_x}\left(\varsigma_x\overrightarrow{E_v} + \varsigma_y\overrightarrow{F_v} + \varsigma_z\overrightarrow{G_v}\right)
\end{aligned}\right\} \quad [6]$$

$\widehat{(Q)}$ is the vector containing the primitive variables and $\widehat{E_i}$, $\widehat{F_i}$, and $\widehat{G_i}$ are the vectors containing the inviscid fluxes in the ξ, η and ζ directions respectively, and are given by

$$\left.\begin{aligned}
&\hat{Q} = \frac{1}{J_x}\begin{pmatrix} 0 \\ u \\ v \\ w \end{pmatrix} \\
&\widehat{E_i} = \frac{1}{J_x}\begin{bmatrix} U \\ uU + p\xi_x \\ vU + p\xi_y \\ wU + p\xi_z \end{bmatrix}; \quad \widehat{F_i} = \frac{1}{J_x}\begin{bmatrix} V \\ uV + p\eta_x \\ vV + p\eta_y \\ wV + p\eta_z \end{bmatrix}, \\
&\widehat{G_i} = \frac{1}{J_x}\begin{bmatrix} W \\ uW + p\varsigma_x \\ vW + p\varsigma_y \\ wW + p\varsigma_z \end{bmatrix} [13]
\end{aligned}\right\} \quad \{7\}$$

Where U, V and W are the contravariant velocities. The shear stresses in the computational space C are obtained and substituting in viscous flux vectors $\widehat{E_v}$, $\widehat{F_v}$ and $\widehat{G_v}$ in the ξ, η and ζ directions respectively.

$$\left.\begin{aligned}
&\widehat{E_v} = \\
&\frac{1}{J_x Re_L}\begin{bmatrix} 0 \\ (\nabla\xi.\nabla\xi)u_\xi + (\nabla\xi.\nabla\eta)u_\eta + (\nabla\xi.\nabla\zeta)u_\zeta \\ (\nabla\xi.\nabla\xi)v_\xi + (\nabla\xi.\nabla\eta)v_\eta + (\nabla\xi.\nabla\zeta)v_\zeta \\ (\nabla\xi.\nabla\xi)w_\xi + (\nabla\xi.\nabla\eta)w_\eta + (\nabla\xi.\nabla\zeta)w_\zeta \end{bmatrix}
\end{aligned}\right\} \quad \{8\}$$

Similarly to calculate $\widehat{F_v}$ and $\widehat{G_v}$

Equations (5), (6), (7) and (8) are the governing equations of an incompressible viscous flow in strong conservation form in computational space. These equations has to be solved using difference method.[13]

D. Methodology

Computational fluid dynamics is used in many areas, such as engineering and medical field. This new field provides very detailed information about fluid characteristics. Medical science lent this new technology to study hemodynamic within the body. CFD uses Finite Volume Method (FVM) as like FEM, FDM. In FVM ,two interpolation structures can be used: (i) Piecewise constant interpolation (ii) Piecewise linear (or bilinear) interpolation. Ist is denoted by cell-centered method and the second is denoted by the cell-vertex method. In both methods, the cells and group of cells around a node are used as volumes.

A CFD solution involves the following basic steps:

- Creation of the geometry
- Choice of the models
- Apply of the boundary conditions
- Flow field computation
- Post processing

CFD helps us to understand their formation, growth, and rupture of stenosis. The purpose of our study is to show the possibility of development of computational analyses of velocity, pressure, and patterns of streamlines.

Discretization is conducted in Gambit. Governing equations were solved in FLUENT which use finite volume method. The file obtained in Gambit is imported in software program FLUENT v.6.3.26 (ANSYS. Inc.) and processed in 3 stages, namely preprocessing, processing and post processing.

Although blood has actually non-Newtonian behavior, in the simulation it is considered Newtonian because there were no significant differences in the distribution of wall shear stress [6].

Reynolds number and Womersley number are the two physical parameters necessary to solve an incompressible fluid flow problem.

Womersley number (α) depends on: flow rate, model geometry and fluid viscosity and varies with vessel diameter.

Womersley number is: $\alpha = r\sqrt{\frac{2\pi\upsilon\rho}{\mu}}$

(6)

where: **r** [m] - entry of the vessel radius, **υ** - flow rate, **ρ** [kg/m^3] - blood density, $\boldsymbol{\mu}$ [kg / ms] - blood viscosity.

The Womersley parameter α can be interpreted as the ratio of the unsteady forces to the viscous forces. When the Womersley parameter is low, viscous forces dominate, velocity profiles are parabolic in shape, and the centerline velocity oscillates in phase with the driving pressure gradient (Womersley1955, McDonald1974). For Womersley parameters above 10, the unsteady inertial forces dominate, and the flow is essentially one of piston-like motion with a flat velocity profile.[10]

The dimensionless Reynolds number (Re) gives the flow command. This number varies with the diameter of the vessel for each case.

$$Re = \frac{\boldsymbol{inertia\ force}}{\boldsymbol{friction\ force}}$$

i.e. $Re = \frac{\boldsymbol{\rho v d}}{\boldsymbol{\mu}}$ **(7)**

where: **ρ** [kg/m^3] - blood density, v [m/s] - maximum speed of blood flow at the entrance, **d** [m] - diameter at the entrance of the vessel, $\boldsymbol{\mu}$ [Kg/ms]-blood viscosity. [5]

Depending on this number, blood flow values in the model can be:

- Laminar when Re ≤ 2300,
- Transient when 2300 <Re <10000,
- Turbulent when Re> 10000.

If flow is laminar, wall shear stress is defined as the velocity gradient at the wall, through the relation: $\tau_{\omega} = \mu\frac{\partial v}{\partial n}$

(8)

where τ_{ω} [Pa] - tension tangential to the wall, **μ**[kg / ms] - blood viscosity, v [m / s] - the speed of blood flow in the vessel, **n** - normal direction to the vessel wall. [5]

RESULTS

Governing equations were solved in FLUENT which use finite volume method for discretization conducted in Gambit. Detailed study of velocity, pressure and streamlines are made. By considering the Domain

TABLE 1. *DOMAIN OF THE GEOMETRY*

	MIN.(m)	**MAX(m)**
X	-2	5
Y	-1	1
Z	-1	1

Geometry of the vessel has been created using MATLAB. Imported the values in Gambit by considering blood density ρ = 1060 [kg/m3] and dynamic viscosity η = 0.003 [kg/ms] (Poiseuille) and by giving boundary condition with velocity 0.6(m/s), velocity inlet, axisymmetric flow, as Interval size 0.1 grid has been generated.

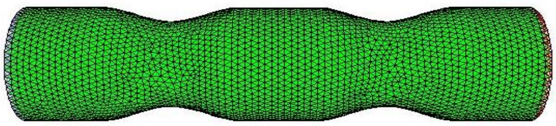

Fig.2. *Grid display for an unsteady flow.*

TABLE 2 *By considering Interval size = 0.1*

Nodes	**25,581**
Mixed wall faces	**10,040**
Mixed outflow faces	**752**
Mixed velocity– inlet faces	**752**
Mixed interior faces	**2,65,276**
Tetrahedral cells	**1,35,524**

For unsteady flow for 100 time steps, 10 seconds for 1000 flow time, iterations per time step 20 and reporting interval as 10 has been calculated.

TABLE 3 *VOLUME STATISTICS:*

MIN.(m^3)	**MAX(m^3)**
2.381751-005	3.956449e-004

Total volume = 1.90732e001
Face area statistics:
Min. face area (m^2) = 1.243686e-003
Max. face area (m^2) =1.257079e-002
The velocity magnitude for unsteady flow with inlet velocity= 0.6(m/s)

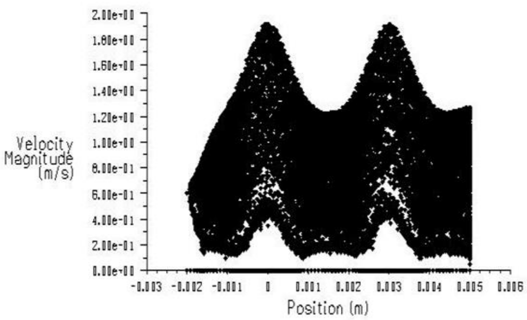

Fig.3 (a). *Velocity magnitude for an unsteady flow*

By considering different planes ,we analyse the velocity variation prominently. At the region of stenoses velocity is high compared to other region fig 3(b)

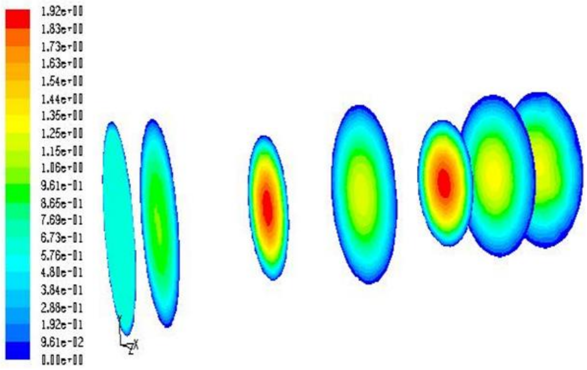

Fig.3 (b). *Velocity magnitude at planes x= -0.0015, 0, 0.0015, 0.003, 0.004, 0.005*

By considering the velocity contours for the plane section along xy- axis, setting z = 0, contours of the flow pattern has been analyzed. This contours shows that there is a circulation of flow in the region of stenoses and also the flow is getting disturbed due to contriction fig 3(c).

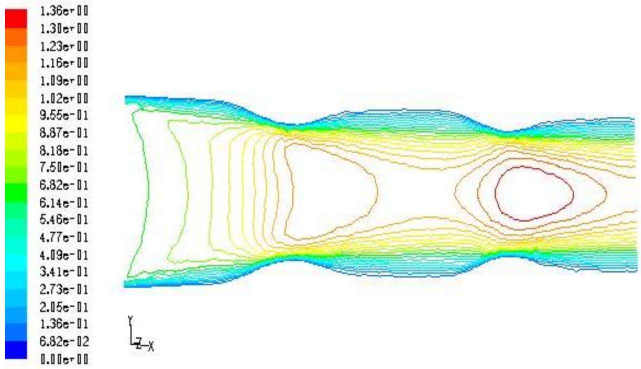

Fig.3 (c). *Velocity magnitude for a plane section x= -.002 to 0.005 and y = -0.001 to 0.001 and z=0*

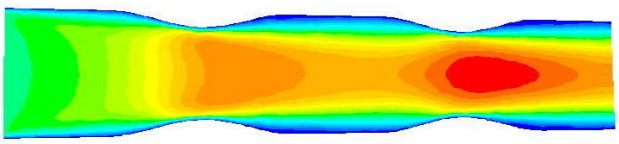

Fig.3 (d). *Velocity magnitude for a plane section x= -.002 to 0.005 and y = -0.001 to 0.001 and z=0, filled*

Velocity drop can be analyse through the xy plot.

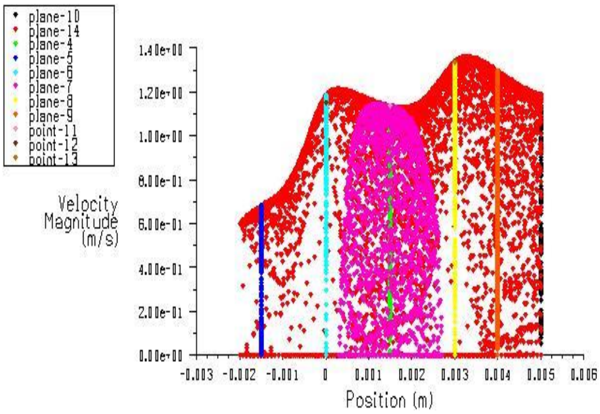

Fig.3(e). *Velocity magnitude graph for an unsteady flow at different planes of x*

From fig.3(b), 3(c), 3(d), 3(e), we observe that there is high velocity at both the region of constriction. The velocity profile is not same throughout the domain. There is a variation of velocities throughout the vessel especially due to constriction the flow is much affected and there is a restriction of flow.

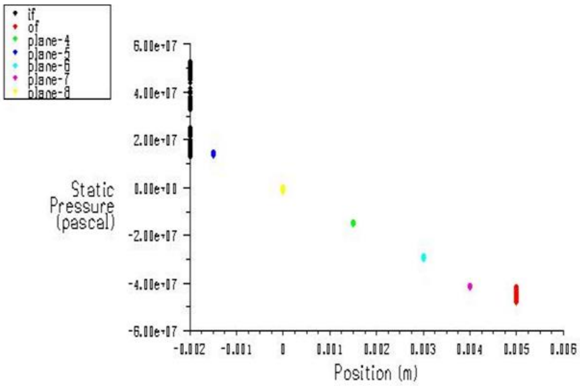

Fig. 4(a). *Pressure magnitude for an unsteady flow*

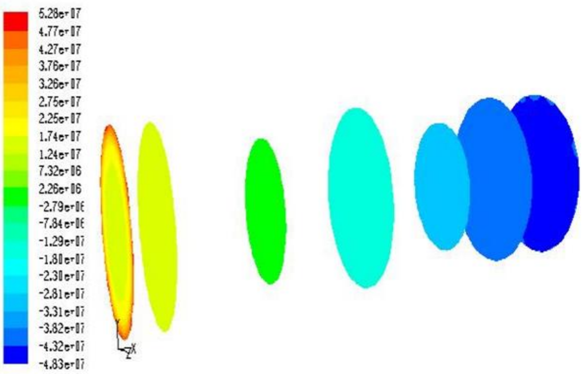

Fig. 4(b). *Pressure magnitude for a plane section x= -.002 to 0.005 and y = -0.001 to 0.001 and z=0*

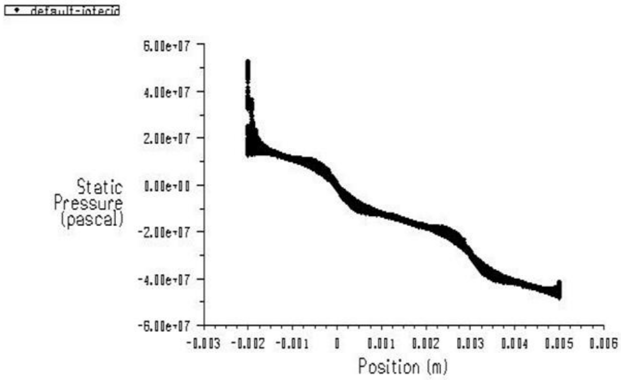

Fig. 4(c). *Pressure difference for an unsteady flow*

It is clear that there is a considerable pressure variation in the direction normal to the X-axis in the stenosed region. A knowledge of the axial pressure loss across the stenoses relative to the overall pressure loss across the entire stenosed vessel is of great physiological interest. The pressure gradient from the inflow segment to the peak of stenoses is much higher than the pressure gradient across the whole of the stenoses and it increases with δ.

Streamlines are a family of curves that are instantaneously tangent to the velocity vector of the flow. This show the direction of fluid element which travels at any point at any time. These stream lines are representative of the speed in a given time

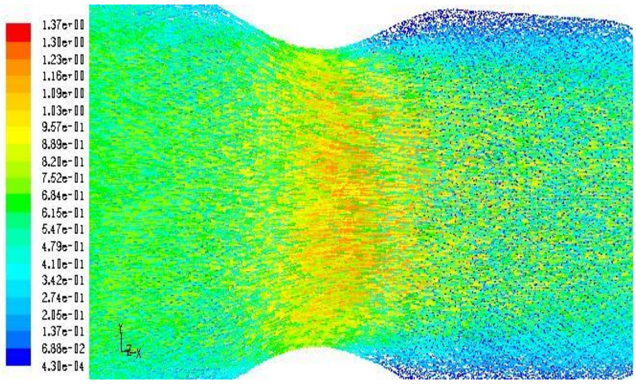

Fig. 5. *Streamlines for the fluid particles*

Wall fluxes, wall shear stress on different positions

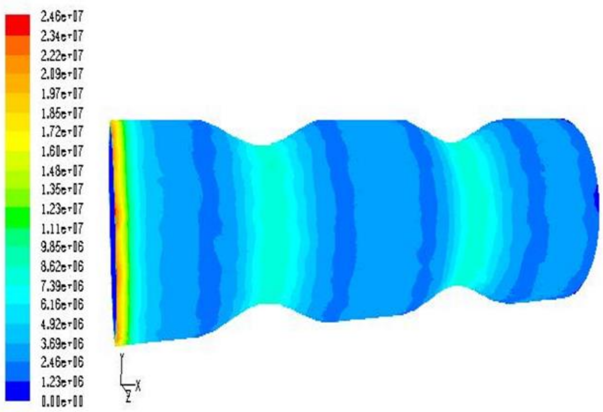

Fig. 6(a). *Wall shear stress for an unsteady flow*

"Flux Report"
Mass Flow Rate (kg/s)
------------ --------------------
default-interior 0.0040111574
if 0.0019948452
of -0.0019948441
wall 0
---------------- --------------------
Net 1.1641532e-09

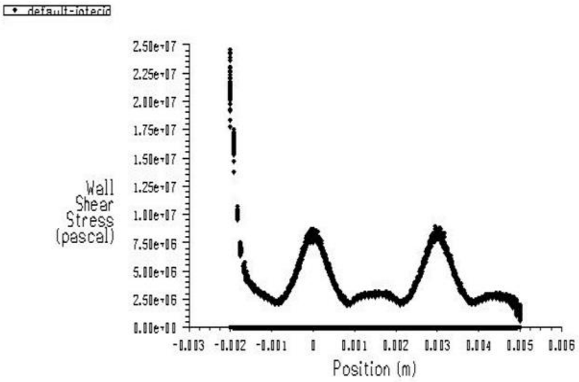

Fig. 6(b). *Wall shear stress plot on the domain*

Pathlines of fluid particles at different position of x is shown in fig. 7

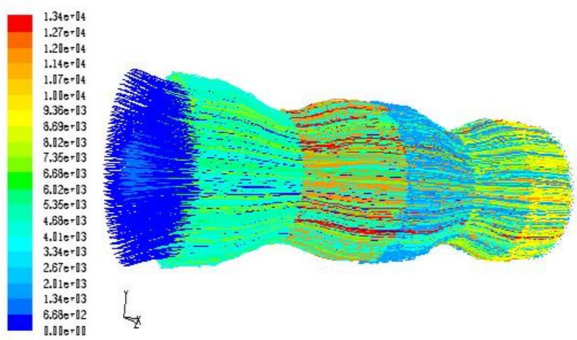

Fig. 7. *Pathlines for an unsteady flow*

Iterations were calculated and solution converges correct to the decimals with equation of continuity (black), and momemtum equation x - velocity (red), y – velocity (green), z-velocity (blue)

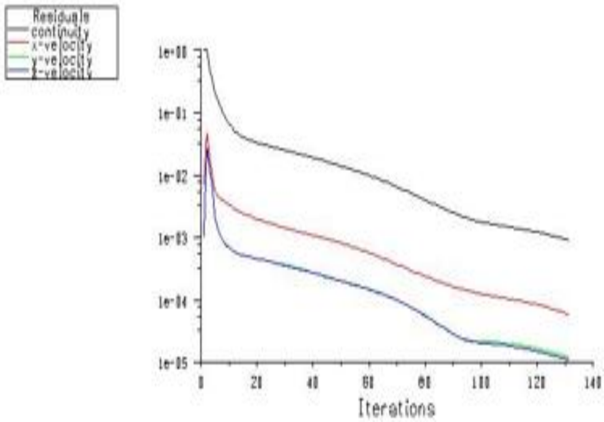

Fig. 8. *Convergence of iterations for an unsteady flow*

In case of non-convergence, some parameters have to be tuned adequately. For explicit solvers the 'cfl' no. and for implicit solvers the under relaxation factors can be changed. The solution converges at 128th iterations where the continuity , x, y, z velocity values are -9.9286e-04, 6.5562e-05, 1.3315e-05, 1.1970e-05 respectively. The study of unsteady flow reveals several interesting new features. It appears that there is a correlation between regions of recirculation, which is a prominent feature of the unsteady flow, and the location of lesions.[9]

CONCLUSION

An unstable 3D flow is not the same for all stenoses, according to flow analysis. The morphology of the arteries and the varying constriction diameters of stenoses have a big impact on flow characteristics. In the area of stenoses, low pressure velocity are achieved. These computer flow study approaches are critical for understanding the link between hemodynamics and rupture risk. The most horrible biological response is the creation of a stenoses. As a result, a thorough knowledge of the link between pressure, blood flow, and symptoms in cardiovascular stenoses is still a major challenge. New stenotic artery repair devices are currently being developed. The science of arterial blood flow will contribute to the modeling of individual hemodynamic flows in any patient in the future, as well as the creation of diagnostic tools for quantifying illness and the fabrication of devices that imitate or change blood flow. Fluid mechanics challenges involving three-dimensional, pulsatile flows abound in this field.

Reference

[1] V.O'Brien and L.W. Ehrich, (1985) Simple Pulsatile flow in an artery with a constriction, J. Biomechanics, vol. 18, No. 2, pp 117-127.

[2] B.V. Rathish Kumar and K.B. Naidu(1996), A transient UVP finite element analysis of a Nonlinear pulsatile flow in a stenosed vessel, IJCFD,vol 0, pp 1-6.

[3] V.P. Srivastava and Shailesh Mishra, Rati Rastogi, (2010), Non-Newtonian Arterial Blood Flow through an Overlapping Stenosis, Applications and Applied Mathematics, vol.5, No.1, pp. 225-238.

[4] Somkid Amornsamankul, Benchawan Wiwatanapataphee, Yong Hong Wu, Yongwimon Lenbury (2006), Effect of Non-Newtonian Behaviour of Blood on Pulsatile Flows in Stenotic Arteries, International Journal of Biomedical Sciences 1:1, pp. 42-46.

[5] Computational hemodynamics in a patient - specific cerebral aneurysms models S. Gaivas¹,P. Cârlescu², Ion Poeată³

[6] Malek A.M. and Izumo S. Mechanism of endothelial cell shape change and cytoskeletal remodeling in response to fluid shear stress. Journal of Cell Science 109, 713-726 (1996)

[7] Seung E. Lee, Sang-Wook Lee, Paul F. Fischer, Hisham S. Bassiouny, Francis Loth,(2008), Direct numerical simulation of transitional flow in a stenosed carotid bifurcation,Journal of Biomechanics 41, pp.2551-2561.

[8] Quan Long, Luca Luppi, Carola S. Konig, Vittoria Rinaldo, Saroj K. Das,(2008) Study of the collateral capacity of the circle of Willis of patients with severe carotid artery stenosis by 3D Computational Modeling, Journal of Biomechanics41, pp. 2735-2742.

[9] H. Huang, V. J. Modi and B.R. Saymour (1995), Fluid Mechanics of Stenosed Arteries,Int. J. Engng. Sci., vol.33 , No. 6, pp. 815-828.

[10] Blood flow in arteries, David N. Ku , Annu.Rev. Fluid Mech.1997.29:399–434, Copyright c ! 1997 by Annual Reviews Inc. All rights reserved

[11] Bart J. Daly,(1976), A Numerical study of pulsatile flow through stenosed canine femoral arteries,J. Biomechanics, vol.9, pp.465-475.

[12] Mcdonald,D.A. "Blood Flow in Arteries" Camelot, Baldwin Park, CA, 1974

[13] Joel Guerrero, Numerical Simulation of the unsteady Aerodynamics of flapping flight, Thesis, Chapter 3, pp. 34-51.

[14] Bai, H. G. (2015). CFD simulation of density variation caused by anti-platelet drug in arteries affected by stenosis. Indian Journal of Science and Technology, 8(6), 562.

[15] Bai, H. G., Naidu, K. B., & Kumar, G. V. (2015). Computational analysis of blood flow in abdominal aortic artery affected by acute stenosis. Journal of Pure and Applied Microbiology, 9(3), 2597-2611.

[16] Jeyanthi, M. P., & Ganesh, S. (2020, February). Numerical solution of Magnetohydrodynamic flow through Circular ducts. In 2020 International Conference on Emerging Trends in Information Technology and Engineering (ic-ETITE) (pp. 1-5). IEEE.