International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 01, January-2018

CFD Analysis of Haemodynamics in Carotid Artery Bifurcation Model of Healthy and Atherosclerotic Blood Vessel

Keerthana.S, Visalakshi.Cho
Sri Ramakrishna Engineering College,
Department of Biomedical Engineering,
Coimbatore

***Abstract*– The paper reviews the computational fluid dynamics (CFD) analysis on three dimensional model of carotid artery bifurcation constructed for finite element analysis to scrutinize the velocity ,pressure gradient and stress analysis. The simulations are performed for 3 cases – the first, with a healthy blood vessel with no calcification or blockage; the second, through a moderately calcified blood vessel and the third, through a severely calcified or occluded blood vessel. The results will show the differences in pressure, velocity profile, and the outlet flow through the 2 branches. The wall displacement and stress analysis use geometrically non- linear shell theory where incrementally linearly elastic wall behavior is assumed. The flow analysis applies the time-dependent, three-dimensional, incompressible Navier-Stokes equations for non-Newtonian inelastic fluids. In an iteratively coupled approach the equations of the fluid motion and the transient shell equations are numerically solved using the finite element method.Appropriate boundary conditions were applied to yield the correct geometry in the unloaded state, and physiological levels of pressure and axial stretching were applied. The model took into account the varying thickness of the arterial wall along the bifurcation.**

I.INTRODUCTION

Vascular diseases causes illness and sometimes may lead to death. Atherosclerosis is one such dreadful disease. Atherosclerosis is a narrowing of the arteries caused by a build up of plaque. It's also called arteriosclerosis or hardening of the arteries, which affects the elastic property of the heart.

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of mathematical modeling (partial differential equations) numerical methods ,software tools (solvers, pre- and postprocessing utilities).

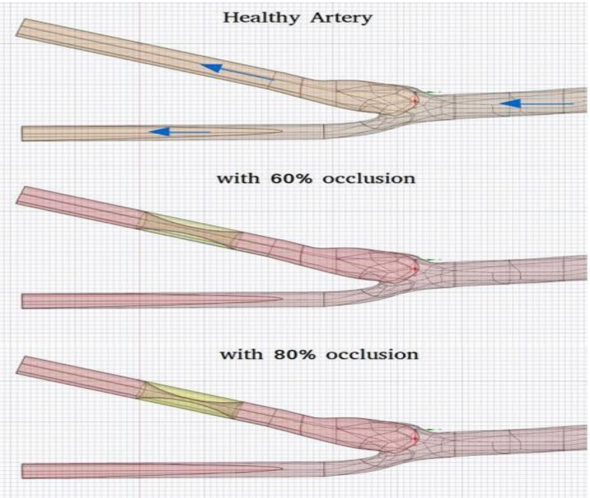


Figure (1): The simulations to be performed for 3 cases – the first, with a healthy blood vessel with no calcification or blockage; the second, through a moderately calcified blood vessel and the third, through a severely calcified or occluded blood vessel.

Figure(2): requirements for a good analysis result.

The flow analysis applies Navier-Stokes equations for non-Newtonian inelastic fluids.

The Navier-Stokes equations govern the motion of fluids and can be seen as Newton's second law of motion for fluids. In the case of a compressible Newtonian fluid, this yields

$$\underbrace{\rho\left(\frac{\partial \mathbf{u}}{\partial t}+\mathbf{u}\cdot\nabla\mathbf{u}\right)}_{1}=\underbrace{-\nabla p}_{2}+\underbrace{\nabla\cdot\left(\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)-\frac{2}{3}\mu(\nabla\cdot\mathbf{u})\mathbf{I}\right)}_{3}+\underbrace{\mathbf{F}}_{4}$$

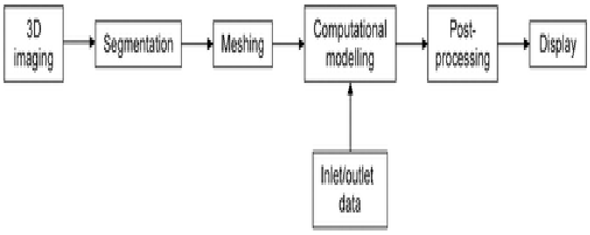
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 01, January-2018

where **u** is the fluid velocity, ***p*** is the fluid pressure, ***ρ*** is the fluid density, and ***μ*** is the fluid dynamic viscosity. The different terms correspond to the inertial forces (1), pressure forces (2), viscous forces (3), and the external forces applied to the fluid (4).

The Reynolds number, **Re=*ρUL/μ***, corresponds to the ratio of inertial forces (1) to viscous forces (3). It measures how turbulent the flow is. Low Reynolds number flows are laminar, while higher Reynolds number flows are turbulent.

The Mach number, **M=*U/c***, corresponds to the ratio of the fluid velocity, ***U***, to the speed of sound in that fluid, ***c***. The Mach number measures the flow compressibility.

II.CFD ANALYSIS OF CAROTID ARTERY BIFURCATION



Figure(3):Steps involved in CFD analysis

Using a combination of three-dimensional (3D) imaging and computational modeling. For blood flow, 3D arterial lumen information is obtained from 3D imaging. Computational fluid dynamics is then used to estimate the 3D velocity field within the lumen, from which wall shear stress may be calculated. For arterial mechanics, the 3D arterial wall geometry is integrated with solid modeling to estimate of the strain field and stress field within the artery wall. For intraplaque stresses, this has been achieved through the use of detailed two- dimensional (2D) intraplaque geometry from MRI.

MESHING

The mesh used here is hex-dominant parametric. This meshing algorithm allows you to create high-quality hex- dominant meshes for arbitrary geometries with optionally refined boundary layers using the "Snappy Hex Mesh" tool. For complex flows and geometries, a "finite element" method must be used. The arterial geometry is broken down into much smaller volume elements, such as prisms or tetrahedrons, creating a 3D mesh.

Segmenting the geometry into these discrete "finite elements" allows the equations of flow to be solved over a much smaller volume and hence increases the accuracy of the simulation.

Figure (5.1)

By combining the results for each element, a universal flow field can be created for the whole geometry. Although creating smaller and more numerous mesh elements will lead to more accurate solutions, especially when modeling turbulent flows, it is also more computationally demanding.

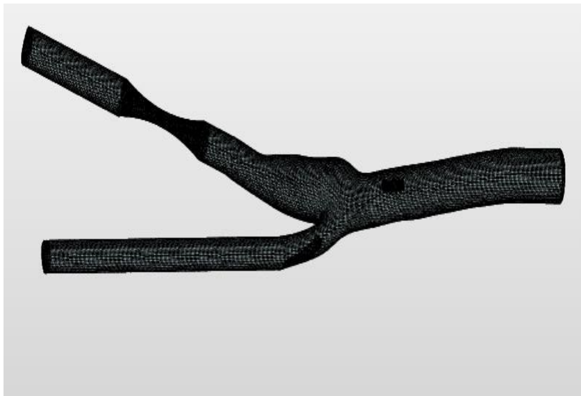


Figure (5.2)

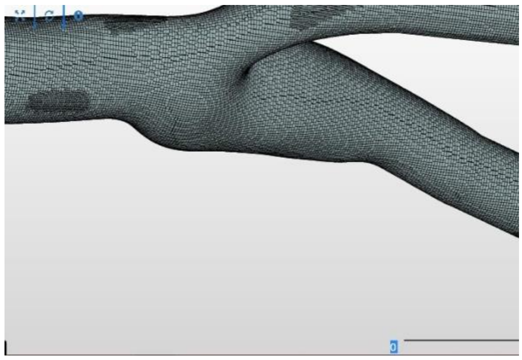


Figure (5.3)

Figure 5.1, 5.2, 5.3, – indicate the Figure 5.1, 5.2, 5.3, – indicate the meshing result of
meshing result of carotid artery of three different CAD models of 80% occlusion,60% and normal artery

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 01, January-2018

INLET DATA

Inlet boundary condition data are needed for CFD. It is common to measure the flow or velocity inlet waveform from MRI or ultrasound and apply this at the inlet assuming axial flow. Typically, the geometry includes a small extension, of a few diameters length, before the inlet to allow the flow to become fully developed by the time the "true" inlet is reached. Fully developed flow is axial, symmetric and with no secondary flow components (*i.e.* no spiraling). It is known that flow in realistic geometries may be asymmetric, and there may be spiraling. However, there is little work on the effect of different inlet velocity components on the flow field estimated from CFD. On a study compared different inlet flow conditions in a bifurcated geometry, including fully developed flow and two cases in which spiraling was imposed. It was found that there was little difference in the flow field or WSS within the bifurcation, and it was recommended that effort be given to more accurate reconstruction of the geometry rather than true characterization of the velocity inlet data.

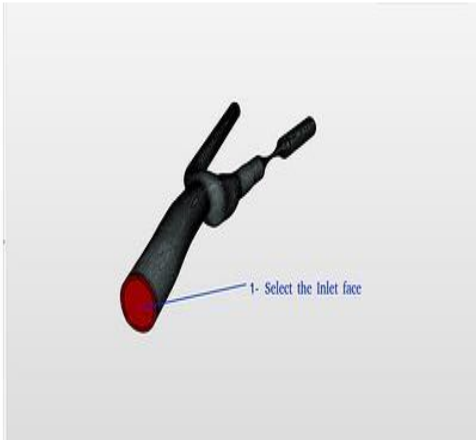


Figure (6) : the inlet face of carotid artery

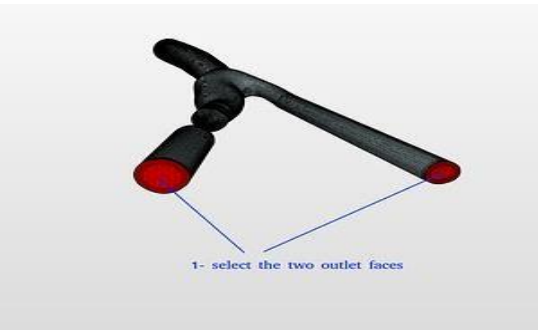


Figure (7): the outlet faces of carotid artery

SIMULATION RUN

The analysis type is Fluid dynamics and the Incompressible flow. When the Mach number is very low, it is to assume that the flow is incompressible. This is often a good approximation for liquids, which are much less compressible than gases. In that case, the density is assumed to be constant and the continuity equation reduces to $\nabla \cdot \mathbf{u} = 0$.

POST PROCESSING

Post-Processing involves extracting the desired flow properties from the computed flow field .T he computed flow properties are then compared to results. Arteries are multilayer structures in which the mechanical properties are different for each layer. There is non-linear behavior, anisotropy (different behavior in different directions), and viscous behavior. Disease is associated with 3D changes in tissue structure and hence 3D changes in mechanical behavior.

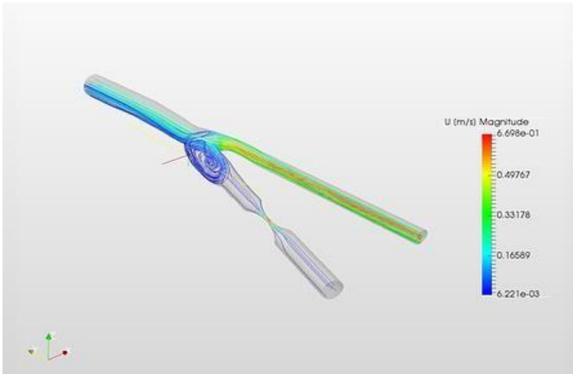


Figure (8): Streamline tracer filter used in post processing

III.CONCLUSION

Comparison of the results demonstrates the quantitative influence of the vessel wall motion. Generally there is a reduction in the magnitude of wall shear stress, with its degree depending on location and phase of the cardiac cycle. The region of slow or reversed flow was greater, in both spatial and temporal terms in the compliant model.

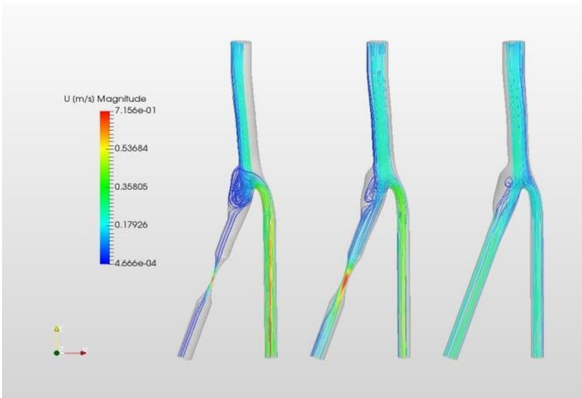


Figure (9): the hemodynamic of the three arteries, the first, with a healthy blood vessel with no calcificationor blockage; the second, through a moderately calcified blood vessel and the third, through a severely calcified or occluded blood vessel.

By using computation fluid dynamics (CFD) it's easier to analyse the hemodynamic which helps the surgical procedure easier and CFD seems to be a useful for scrutinizing the flow by non invasive technique.

IV. REFERENCES

[1] WardlawJM .Carotid imaging for secondary stroke prevention in routine practice.*Int J Stroke*2008;3:20–32. CrossRef

[2] Friedman MH ,Ehrlich LW . Numerical simulation of aortic bifurcation flows: the effect of flow divider curvature.*J Biomech*1984;17:881–8. CrossRef, Medline

[3] PerktoldK ,ReschM . Numerical flow studies in human carotid-artery bifurcations – basic discussion of the geometric factor in atherogenesis.*J Biomed Eng*1990;12:111–23. CrossRef, Medline

[4] Long Q ,XuXY , Collins MW , Griffith TM , Bourne M . The combination of magnetic resonance angiography and computational fluid dynamics: a critical review.*Crit Rev Biomed Eng*1998;26:227–74. CrossRef, Medline https://www.comsol.co.in/multiphysics/navier-stokes-equations

[5] Milner JS ,Moore JA , RuttBK , Steinman DA . Hemodynamics of human carotid artery bifurcations: computational studies with models reconstructed from magnetic resonance imaging of normal subjects.*J VascSurg*1998;28:143–56. CrossRef, Medline