Modeling and Numerical Blood Flow Analysis of Tibial Artery using CFD

S.Manimaran
Department of Biomedical Engineering
Rajalakshmi Engineering College, Chennai, Tamil Nadu, India

C.Muralidharan M.E
Assistant Professor
Department of Biomedical Engineering
Rajalakshmi Engineering College, Chennai, Tamil Nadu, India

Surendra Bogadi M.E,(Ph.D)
Assistant Professor
Department of Aeronautical Engineering
Rajalakshmi Engineering College, Chennai, Tamil Nadu, India

Abstract-The aim of this research is to estimate the effect of blood flow through the tibial artery on leg portion. Blood flow in tibial artery is an unsteady and pulsatile flow. A three dimensional numerical analysis is carried out to understand the effects of blood flow presents on tibial artery. For CFD simulations the initial step is the construction of geometrical model of the tibial artery. To achieve this, 3D realistic and idealized models of tibial artery were constructed from CREO software. Blood is used as the working fluid and enters the micro tube at 300k.After completion of geometry creation; Grid generation on modeled tibial artery is done by using ANSYS ICEM CFD. Finally analysis is solved using ANSYS FLUENT-14.0. The outcome of results gives better accuracy of biomechanical characteristics on tibial artery's blood flow for comparing and analysis with various patients model of tibial artery from other software like MIMICS, 3D DOCTOR, SurgiCase CMF, etc.

Keywords—Blood flow, CFD, Tibial artery

I. INTRODUCTION

Vascular diseases are the results of traumatic and non-traumatic injury, especially to the lower limb portion starts from the knee joint. Vascular disease patients are susceptible to these types of injury because of their aging and disease condition. Thus, patients that receive this injury will experience swelling, pain, abrupt blood flow changes due to the clots, dissection, or aneurysm of the arteries involved. The blood flow in tibial artery may be affected by serious vascular injury to the patient. . Numerical simulation is one method that can be applied to simulate the blood flow pattern through the tibial artery starts from knee joint. In this arterial research, a geometrical linear with irregular shape of artery was used for performing the blood flow analysis of the artery. The numerical study of blood flow in arteries was undertaken by 3D models of geometrical tibial artery with different diameters is considered. The numerical study analyzes the fluid dynamic variable like velocity profile in the artery. A rigid tube of constant circular cross section, which is bent into a linear and irregular shape, is considered for the steady state fluid flow. Patients are vulnerable to vascular injuries of the lower limb, especially at the knee joint. The popliteal artery divides into the posterior and anterior tibial arteries. It is clear that many vascular injuries affect the main arteries on human vasculature structure, such as tibial artery for diabetic patients. In this artery is very important in supplying blood around the leg portion. The cause of a tibial artery injury for a diabetic patients and vascular disease patients is hypertension, dislocation, and ligament rupture. Thus, such an injury causes interruption of flow in the tibial artery and patients may complain of pain, parenthesis, or loss of sensation to the leg. In the lower limb anatomy, the main artery from the knee joint is a tibial artery.

A numerical study of the blood flow in vessels with a clots using Open FOAM code reported by Pinto, S. I. S., et al. The CFD package was used to perform the numerical study because it is an open software package that can be freely modified. Continuous flow of a blood was considered and implemented in a numerical code. Velocity profiles were obtained a tibial artery, considering constant inlet velocity. The profiles obtained through ANSYS FLUENT are in agreement with those obtained through the analytical solution. Velocity profiles were also obtained for a healthy model; however, as far as we know Modeling and blood flow analysis of tibial artery for healthy model has not yet been literature. For this reason indefinite collaborations between CREO and ANSYS FLUENT may be an important methodology to understand the biomechanical characteristics of the development and progression of disease and for establishing and creating treatment modalities in the vascular defects. In the present work, results are obtained, firstly, for a modeling of tibial artery from CREO and, then CFD for blood flow analysis with affordable biomechanical characteristics.

II. MATERIALS AND METHODS

A. CFD Introduction

Mathematical methods and physical conditions that are applied during the experimental set up of this work. Grid generation, fluid properties, governing equations boundary conditions, solver set-up and the computational method for solution is the general algorithm of applying CFD. Fluid dynamics is the science of the fluid motion. Fluid flow is studied in one of the three ways (1) Theoretical fluid dynamics is used to understanding the concepts and equations. (2) Experimental fluid dynamics is costly and a complex way to get exact results. Because it is not usually possible to develop the original situations due to limit of the repetitions or due to the human errors. (3) computational fluid dynamics (CFD) is provides a qualitative prediction of fluid flow with applying any conditions. With the use of computers, it has potential to provide great amount of data at a fraction of the cost of experiments. Fluid flows are governed by equations, which represents the Conservation laws for mass, momentum, and energy. CFD is the art of replacing such equations by a set of algebraic equations and then can be solved with the help of computers. CFD provides numerical approximation to the equations that govern the Fluid flow.

B. GEOMETRY CREATION

The computational domain of the cylindrical tube (L = 34.3 cm and R = 0.13 cm) can be taken as two dimensional rectangle due to axi-symmetry. Blood is used as the working fluid and enters the cylindrical tube at 300 K for the creation of above geometry in CREO workbench on the toolbox was selected. Then analysis type is changed from 3D to 2D by selecting property at the Geometry level. In the DESIGN MODLER (DM) screen 'cm' as the unit was chosen. Surfaces were created from the sketching with proper naming like inlet, outlet and boundary or surface. After generation of surfaces body type were defined i.e. inner tube as FLUID and the outer tube as SOLID. In initial diameter of tube is considered as 0.13 cm, then center of tube diameter is considered as 0.0075 cm, finally outer diameter is considered as 0.13 cm.

C. GRID GENERATION

The STL file of patient's 3D model is imported on ICEM CFD 14.5 for grid generation.

1. Import STL data into ANSYS ICEM CFD.
2. Set up global and part parameters for meshing.
3. Generate the mesh using the Octree approach.
4. Create material parts as velocity inlet, velocity outlet and surface on imported geometry.
5. Mesh with geometry is created. Then it is imported on ANSYS Fluent for flow analysis.

D. SETUP AND FLOW SPECIFICATION

The given meshed geometry exported to the fluent setup where physical and flow properties specified. In the problem setup pressure based solver and absolute velocity as 0.5 m/s was maintained. While all other models remain as it is. In the material section Blood was chosen as the working fluid whose thermo-physical properties mentioned below:

Zero shear rate limit (μ_0)	0.056 pa-s
Infinite shear rate limit ($\mu\infty$)	0.0035 pa-s
Relaxation time constant (λ)	3.313 S
Power law index in carreau model (n)	0.3568
Blood temp (T)	310.15 K
Blood specific heat capacity	3594 $JKg^{-1}K^{-1}$
Density (Kg/m^3)	1060

Dynamic viscosity (µ)	0.0035 (Ns/m^2)
Blood flow type	Laminar

Table 1: Thermo-physical properties of blood

E. SOLUTION

'Implicit integration' solution method was used for temporal discretization. 'Green-Gauss cell based' theorem was used to compute scalars at the cell centers. 'SIMPLE' algorithm was used for pressure velocity coupling, 'Second order upwind scheme' used for discretization of momentum and energy equations. Pressure interpolation was done using 'standard' scheme. Standard initialization was used for solution initialization and computed from the inlet. Solution control parameters i.e. under relaxation factors were kept as default values listed below:

Pressure	**Density**	**Body forces**	**Momentum**	**Energy**
0.3	1	1	0.7	1

Table 2: Value of under relaxation factors

Under run calculation, time stepping method was maintained as fixed. Maximum iteration per time step was set up to local variables is converged. Reporting interval and profile update interval were kept as the default value i.e. 1. Time step size was calculated from the frequency of pulsation and number of time steps so maintained that steady periodic solution is obtained.

III. RESULTS AND DISCUSSION

Simulations were performed on ANSYS FLUENT. Comparison of these biomechanical characteristics for different cases will be used to various diagnostic purposes. Initial velocity given for tibial artery is 0.5 m/s. A parabolic velocity profile at the inlet with a maximum value of 1.47 m/s is carried until the mid of the geometry. This is because of almost uniform diameters and negligibly small differences of the axi symmetric geometry caused by the smaller tapering coefficient. High velocity gradients can be observed near the tapering region of the geometry. The velocities at the walls were zero. After passing through the tapering region, the velocity increases and reaches the maximum value of 0.5 m/s close to the outlet.

It is stated in a analysis of Healthy tibial artery model is used to analyze the flow variation, the non-Newtonian Power Law with carreau model is considered to approximate the blood viscosity model in a more satisfactory way. Steady analysis was performed to analyze the velocity and fine meshes to save computational time.. The following results obtained from the study that can be used as a reference model for analysis.

A. BERNOULLI EQUATION

By using Bernoulli's equation, the inner velocity and outer velocity of a given geometry model of tibial artery is calculated.

Initial Velocity (V_1) = 0.5 m/s

Radius of Inlet (r_1) = 0.013 m

The radius of the tibial artery model is reduced in between at some point to resemble the features of the anatomical structure of the artery (r_2) = 0.0075 m

The velocity of the tibial artery model is reduced in between at some point to resemble the features of the anatomical structure of the artery (v_2) =?

Blood Density ($\rho_{1=}\rho_2$) = 1060 Kg/m^3

$$\rho_1 A_1 V_{1=}\rho_2 A_2 V_2$$

$$(1060)\ (\pi r_1^2)\ (0.5) = (1060)\ (\pi r_2^2)\ (V_2)$$

$$V_2 = 1.5 \text{ m/s} \quad \ldots\ldots\ldots\ldots\text{.. (Theoretical value)}$$

B. ANALYSIS USING CFD

Inlet velocity (m/s)	0.5
Outlet velocity (m/s)	0.5

Max velocity of the tibial artery model is reduced in between at some point to resemble the features of the anatomical structure of the artery (m/s)	1.47

Table 3:Analysis value of velocity presents on modeled tibial artery

V_2=1.47 m/s ………….. (Calculated value)

1.5 m/s ≈ 1.47 m/s

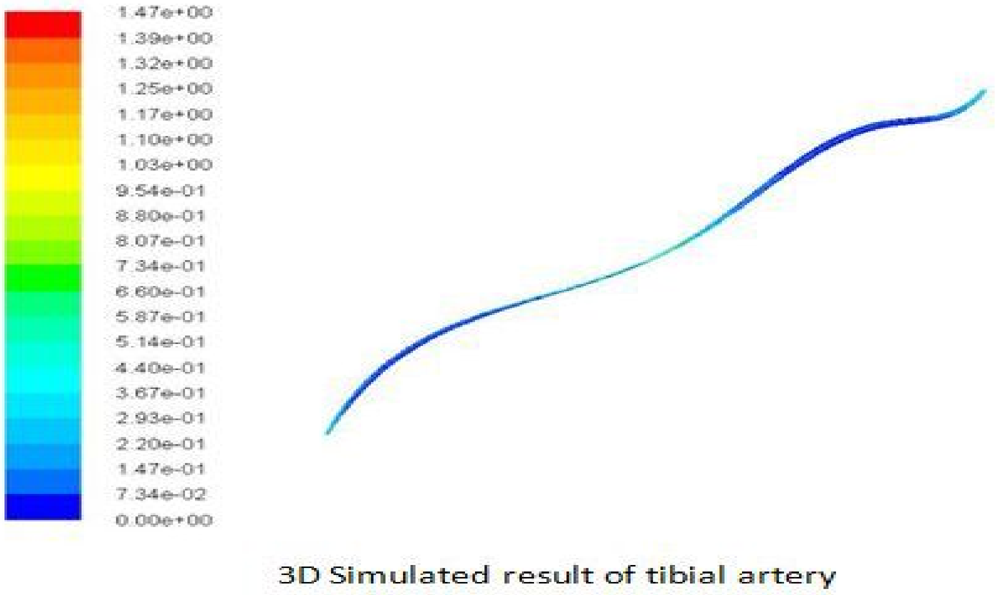

Fig 1: contours of velocity magnitude (m/s) presents on normal modeled tibial artery

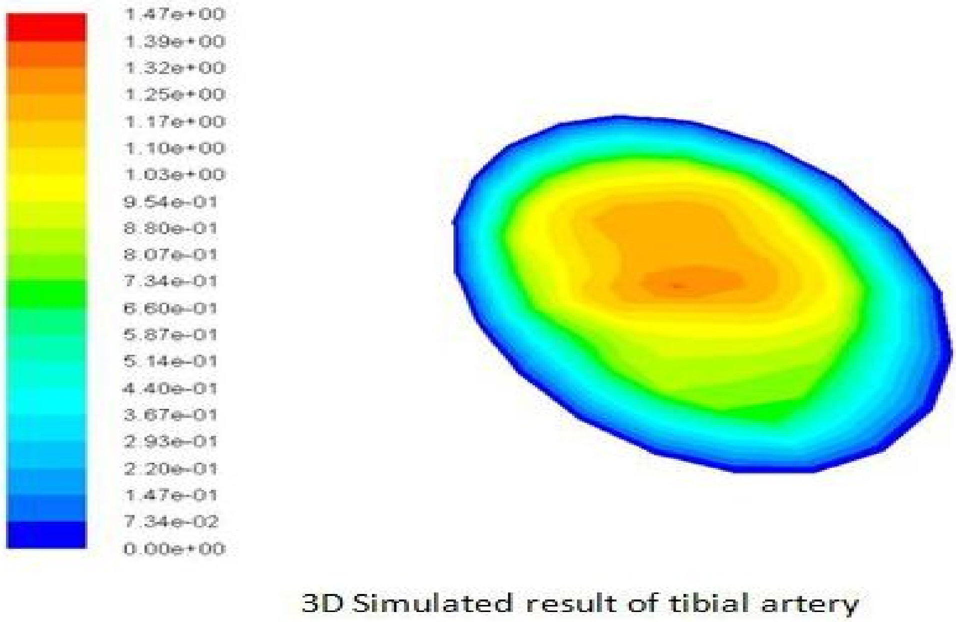

Fig 1.1: Sectional view of maximum velocity present on normal modeled tibial artery.

IV. CONCLUSION

The model that was developed satisfies the Bernoulli's condition. By using this as the reference value, the flow velocity from the patient's data will be compared for diagnosing method in the field of clinical research.

V. FUTURE WORK

This kind of study might be applied to a modified different arteries geometry model. Furthermore, the effect of various intervention and arteries bypass grafts can be simulated and suggest the best method of intervention and also useful for better stent applications. Various biomechanical characteristics on human artery will be analyzed using CFD like velocity, wall shear stress, pressure, wall fluxes etc.Comparison of the biomechanical characteristics with normal geometrical model of tibial artery and various patients tibial artery model from software like MIMICS, 3D DOCTOR, SurgiCase CMF. Comparison of these biomechanical characteristics for different cases will be used to various diagnostic purposes.

REFERENCES

[1] Shih, Sung-Tsun, and Kuan-Yu Chang. "Synchronous TBI and ABI measurement Scheme for Diabetes Patients Synchronous Sphygmomanometer of Four Limbs." *Life Science Journal* 6.4 (2009): 62-68.

[2] Turnipseed, William D., Christof Hurschler, and Ray Vanderby Jr. "The effects of elevated compartment pressure on tibial arteriovenous flow and relationship of mechanical and biochemical characteristics of fascia to genesis of chronic anterior compartment syndrome." *Journal of vascular surgery* 21.5 (1995): 810-817.

[3] Sherwin, S. J., et al. "Computational modeling of 1D blood flow with variable mechanical properties and its application to the simulation of wave propagation in the human arterial system." *International Journal for Numerical Methods in Fluids* 43.6-7 (2003): 673-700.

[4] Vignon-Clementel, Irene E., et al. "Outflow boundary conditions for three-dimensional finite element modeling of blood flow and pressure in arteries."*Computer methods in applied mechanics and engineering* 195.29 (2006): 3776-3796.

[5] Lee, Byoung-Kwon. "Computational fluid dynamics in cardiovascular disease." Korean circulation journal 41.8 (2011): 423-430.

[6] Pinto, S. I. S., et al. "Blood Analog Fluid Flow in Vessels with Stenosis: Development of an OpenFOAM Code to Simulate Pulsatile Flow and Elasticity of the Fluid." *APCBEE Procedia* 7 (2013): 73-79.