A Comprehensive Study on CFD Analysis of Blood Flow in Arteries

M.Tech THERMAL ENGINEERING AJAY MANN 2022

A PROJECT REPORT

SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR THE AWARD OF THE DEGREE

OF

MASTER OF TECHNOLOGY

IN **THERMAL ENGINEERING**

Submitted by

AJAY MANN

2K20/THE/02

Under the supervision of

DR. M. ZUNAID

DEPARTMENT OF MECHANICAL ENGINEERING
DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)
Bawana Road, Delhi-110042

MAY, 2022

DEPARTMENT OF MECHANICAL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)
Bawana Road, Delhi-110042

CANDIDATE'S DECLARATION

I, **AJAY MANN**, **2K20/THE/02** student of M.Tech Thermal engineering, hereby declare that the project Dissertation titled " A Comprehensive Study on CFD Analysis of Blood Flow in Arteries" which is submitted by me to the Department of Delhi Technological University, Delhi in partial fulfillment of the requirement for the award of the degree of Master of Technology/Bachelor of Technology, is original and not copied from any source without proper citation. This work has not previously formed the basis for the award of any Degree, Diploma Associateship, Fellowship or other similar title or recognition.

Place: Delhi

Date:30 MAY 2022

AJAY MANN
2K20/THE/02

DEPARTMENT OF MECHANICAL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)
Bawana Road, Delhi-110042

CERTIFICATE

I hereby certify that the Project Dissertation titled " A Comprehensive Study on CFD Analysis of Blood Flow in Arteries" which is submitted by AJAY MANN, 2K20/THE/02 Department of Mechanical Engineering, Delhi Technological University, Delhi in partial fulfillment of the requirement for the award of the degree of Master of Technology, is a record of the project work carried out by the students under my supervision. To the best of my knowledge this work has not been submitted in part or full for any Degree or Diploma to this University or elsewhere.

Place: Delhi

Date:30 MAY 2022

Dr. M. Zunaid
Assistant Professor,
Department of Mechanical Engineering,
Delhi Technological University
(Formerly Delhi College of Engineering)
Bawana Road, Delhi-110042
(SUPERVISOR)

ABSTRACT

Blood flow in a living organism is an unsteady and complex phenomenon. Blood vessels have geometrical features like curvature, bends, branching, Y-intersection, variations in the diameter of the vessel and the most important, pulsatile nature of the blood. Cardiovascular diseases are most Common in the world today that is fatal for mankind. Therefore, it is very important disease from public health and individual standpoint. So, it is vital to study the parameters that affect flow of blood in arteries [1]. We study the Newtonian nature of blood using patient specific data extracted using MRI or CT scan and applying these boundary condition in the flow simulation. ANSYS Fluent software is used to examine all of the cases comprising bend in artery, Y-intersection and bifurcating artery. We examine the wall shear stress profile and velocity profile for all these cases separately before we integrate them together.

Keywords: Blood flow; Coronary artery; Computational Fluid Dynamics (CFD); MRI Images.

Acknowledgement

I would like to thank my supervisor **Dr. M. Zunaid**, (Asst. Professor, Department of Mechanical Engineering) for his consistent guidance, motivation and supporting this research throughout this journey. I consider myself to be fortunate to learn from him and work under his mentorship.

I would also like to thank my teachers and colleagues at Delhi Technological University for assisting me in the pursuit of this project.

Finally, I am greatly thankful to my family and friends for their non-stop support and motivation. This work would have not been possible without them.

THANK YOU

AJAY MANN
2K20/THE/02

Table of Contents

LIST OF FIGURES PAGE

LIST OF ABBREVIATIONS

MRI	Magnetic Resonance Imaging
CT scan	Computed Tomography Scan
CAD	Coronary Artery Disease
CFD	Computational Fluid Dynamics
FEM	Finite Element Method
WSS	Wall Shear Stress

NOMENCLATURE

ρ	Density of the fluid, Kg/m^3
x, y, z	Cartesian co-ordinates
u, v, z	Velocity components of the flow field in x, y, z directions, m/s
$\vec{V}$	Velocity vector, m/s
t	Time, seconds
b_i	Body force component, N
τ_{ij}	Stress Tensor, N/m^2
λ	Volumetric dilation coefficient
δ_{ij}	Kronecker delta
P	Pressure, KPa
n	Power-law index
K	Consistency index
$\dot{\gamma}$	Rate of shear
η_0	Zero shear rate viscosity, Pa-s
η_∞	Infinite shear rate viscosity, Pa-s
λ	Characteristic relaxation time
Re	Reynolds Number
V_m	Mean Velocity of the flow, m/s
$\dot{m}$	Mass flow rate of blood, Kg/hr

CHAPTER 1

INTRODUCTION

1.1) INTRODUCTION

Within our own bodies, we also have blood that has to circulate to every organ, to every single cell to keep us alive, and at the very centre of that is the heart that beats 70 times a minute, 24 hours a day from many months before we were born to the second that we die. So it is very important task to circulate blood, oxygen, nutrients to the rest of body And not only it pumps the blood to the whole of the body but also to itself to make sure it can provide the energy to pump. And the pipes that lead from the heart to the body are the arteries, the capillaries and the veins. Average human does not have a smooth vascular system. A little lumps of cholesterol that are continuously developing in vascular system due to unhealthy lifestyle. And this process happens throughout life. About one in three human have a risk of heart attack or stroke at some point who are older than 35 [2]. India stands at fifth globally in number of premature heart attack. In India, 13.1% men, and 6.6% women between the ages of 30 and 69 have the probability of dying from cardio vascular diseases (CVD) as of 2015 [3]. Flow development in aorta is complex due to pulsatile nature of blood flow and geometric variations like strong curvature, twisting, bifurcating and intersecting arteries. Various computational analysis has been done in past and is still going on to study flow patterns in patient specific models [4–10]. This work has been further extended to study blood flow patterns for different characteristics in aorta computationally.

1.2) BLOOD

Blood is a colloid which means it is basically a suspension of particles including red blood cells or erythrocytes, the white blood cells, platelets (important in forming clots) and all of these elements are suspended in plasma which is the liquid that has sodium chloride and a little bit of potassium. So, it is a colloid. Blood is a non-Newtonian fluid[11]. It is a shear thinning fluid meaning that its viscosity decreases with an increasing rate of shear. There are a couple of phenomenons which add to the non-Newtonian behavior of blood. At low shear rates there is recites tend to aggregate and it causes blood viscosity to increase exponentially at low shear rates.

Role of blood viscosity is overlooked clinically. It is measured very rarely and for all practical purposes it's invisible. Viscosity is a fundamental property of all fluids including blood. In particular, elevated blood viscosity imparts an increased tendency for blood to clot are more properly thrombose (clot) and complications of thrombosis include coronary heart disease. Perfusion which is the amount of blood which actually goes to tissue organs and tissue is inversely proportional to blood viscosity. The higher the blood viscosity, the less perfusion of all tissue. This is very important in peak athletic performance. To supply the maximum amount of oxygen and glucose to skeletal muscle you need maximum perfusion and increase the blood viscosity decreases perfusion. And that decreases glucose uptake and can result in Hyperglycemia. Thicker the blood, the more resistance it will impart to flow. So, blood viscosity contributes to high blood pressure or hypertension[12-13]. But in this case due to high shear rate blood is assumed to be Newtonian. Hence effect of change is viscosity is neglected by taking viscosity constant.

1.3) Background

For the cardiologists working in a in a hospital, to examine of the patient, a number of invasive and non-invasive diagnostic tools are present by which patient conditions can be determine. The non-invasive tools are related to the medical imaging techniques like x-ray computer, tomography or magnetic resonance imaging. In the result of those tests is a two-dimensional radiology called images. This image shows what is inside of the skin. There is also a very popular method called electrocardiography which is measuring of heart beats. By the interpretation of the result of this procedure we can even check if the heart is in a good condition or if it have a good good shape.

There are also some non-invasive tools. They are angiography and fractional flow

Resolve. In this in those tests we are performing a measure of certain parameters of the blood flow in the heart.

A series of studies shows that the value of some parameter is crucial for treatment of a patient and indicates the need of more radical intervention in the therapy instead of just drug therapy. In the opposite to that invasive tools, there is a method to measure that parameters in a in a non-invasive way by using the computer fluid simulations.

1.4) Computational fluid dynamics

computational fluid dynamics is a subdivision of fluid mechanics by which we can analyze the problem related to the fluid flows and there are many approaches to

to perform such simulator simulations but the framework looks always the same.

So the first step of performing the fluid dynamics simulation is the processing. In this step we have to define the geometry of the problem, the shape through it or around which our fluid will be flowing. Then we have to define the volume occupied by the fluid (It can be inside or outside the geometry). This volume is divided into discrete set of cells called the volumetric mesh. After that we have to define the physical model of our simulation. It is usually some transport equation like navier-stocks equation and characteristic of the fluid, like its density, viscosity, heat capacity and others. The next step is the solution of our problem by application of the iterative solvers. And after the solution we are analyzing them in some post processing modules.

1.5) Objectives

Present work deals with CFD simulation of human artery. Using a set of MRI images of a healthy patient, a smooth surface of the artery is constructed. Coronary wall boundary condition is used for simulation to visualize the velocity field inside of artery having some geometrical features. The model includes sudden bend, Y-intersection and bifurcating artery. Parameters such as Velocity magnitude and pressure contours were studied in CFD simulation using ANSYS, based on finite element method. Inlet velocity is varied as 0.05, 0.1, 0.2, 0.3 and 0.4 m/s and exit velocity and pressure is obtained using simulation.

1.6) Application

As we go in the context of computational science there are a number of different phenomena in the cardiovascular system that would be interesting to highlight and these are all areas where there is a lot of potential. This type of modelling is applicable in a number of problems involving treatment planning and virtual surgery. That is the most obvious application of these methods and kind of what the field was set out to do from the beginning. So, the goal there is to pick the optimal treatment, out of variety of different surgical options. Another application of this concept is if some treatment is too risky to do in an animal study or a patient study then computational models can be used as sort of a sandbox for trying out different out-of-the-box ideas.

Applications of these simulations can be extended to learn more about the fundamentals of disease progressions. There are complex interaction of mechanical forces and biological signalling that was not fully understood. These models can make more fundamental contributions scientifically in those areas. Another area of application is to do risk stratification for patients. Such as to identify patients' risk for vein graft failure. So, if a patient who just had a surgery want to classify his risks so we will be able to know how closely to follow them over time.

1.7) Newtonian and Non-Newtonian fluids

According to Newton's law of viscosity, at constant temperature and pressure, the simple shear stress (τ) is proportional to the shear rate ($\dot{\gamma}$) and the constant of proportionality is commonly known as the dynamic viscosity (μ). It infers that the shear stress increases linearly with an increase in the shear rate. The fluids which demonstrate this type of behaviour are called Newtonian fluids. Most of the fluids which possess low molecular weight such as various organic and inorganic liquids, molten metals, water, and a wide variety of salt solutions exhibit Newtonian behaviour when acted upon by the shear stress causing the fluid flow. It is observed that with an increase in either temperature or pressure, the viscosity of gases tends to increase due to the enhancement in the molecular collision as a result of the increased energy content of the gas, whereas in liquids the viscosity decreases with the increase in temperature due to the weakening of the force of attraction between the molecules present in the liquid[15]. On the whole, the higher the viscosity of a substance, the more resistance it will exhibit to the flow and thus will require more power to pump the fluid and transport it from one location to the other. Contrary to the Newtonian fluids, non-Newtonian fluids do not display a linear relationship between the stress-strain rate and a deviation is observed as depicted in figure 1 which shows stress-strain curves for different non-Newtonian fluid. It means that the apparent viscosity of the fluid defined as $(^{\tau}/_{\dot{\gamma}})$ is not constant and is a function of either τ or $\dot{\gamma}$[16. Additionally, the viscosity of some fluids also depends on the time period, and hence non-Newtonian can be classified into two broad categories of time-dependent and time-independent fluids. Since our study is limited to modelling the hemodynamic of blood, which is a shear-thinning time-independent non-Newtonian fluid, the discussion is restricted to just the time-independent fluids and these fluids can further be classified into shear-thickening, shear-thinning, and visco-plastic behaviour type.

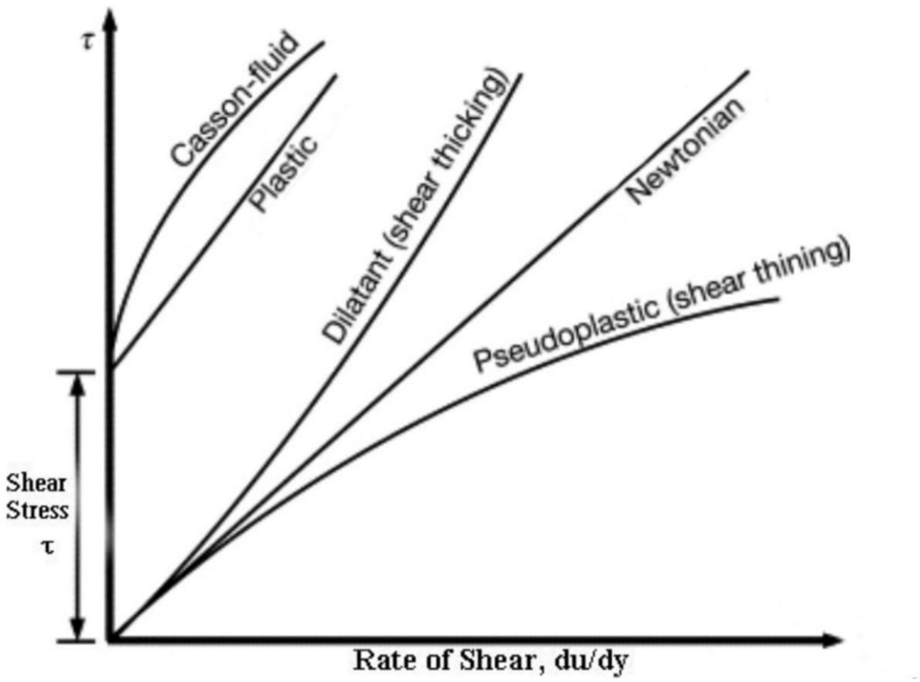


Figure 1: Rheological behavior of different fluids.[13]

The most common model employed to model the non-Newtonian characteristics is the power-law model or also known as the Ostwald de Waele Equation given as

$$\tau = K(\dot{\gamma})^n \tag{13}$$

Where K is the consistency index and n is the power-law index. The apparent viscosity, η is obtained by dividing the shear stress with the rate of shear.

$$\eta = \frac{\tau}{\dot{\gamma}} = K(\dot{\gamma})^{n-1} \tag{14}$$

It is well understood that for 0< n<1, $\left({}^{d\eta}/_{d\dot{\gamma}} \right)$ will always yield a negative value which implies that with the increase of the shear rate, there will be a reduction in the shear stress component of the fluid. The fluids presenting this type of behaviour are termed as shear-thinning fluids. Smaller the value of n, more shear-thinning is the fluid. Blood, various polymer solutions, and suspensions like bentonite-in-water are good examples of shear-thinning fluids. However, when the value of n is greater than 1, the behaviour tends to reverse, i.e. with an increase in shear strain, the value of shear stress also increases. This type of fluids falls under the category of shear-thickening behaviour which can be readily witnessed in a corn starch and water mixture. For the case of n=1, the fluid showcase Newtonian rheology.

CHAPTER 2

METHADOLOGY

- **Geometric model construction**: The approach for doing this research is a patient specific modelling. Patient specific information from Magnetic Resonance Imaging (MRI) and CT scans was extracted so the models became as precise and robust as possible. So, image data of an individual patient with normal heart rate is used to construct a 3D aorta geometric model using a 3D medical image processing software i.e., materialise mimics. Artery is dissected in three different parts based on geometrical features.
- **Mesh Generation**: Extracted geometry in (.stl) format is then exported to a 3D design software (ANSYS). For this computational fluid dynamics problem, a finite element method is used because of the complex geometry of this vascular model[21]. And each of the geometrical features such as curvature, intersecting arteries and bifurcating arteries will be studied separately.
- **Fluid properties**: Blood flow is considered to be homogeneous and Newtonian fluid because of moderately high shear rate (~300 sec-1) in arteries. Blood flow in aorta is assume to be Laminar and incompressible with no slip condition at the wall. [14–16].
- **Governing equations**:

  The general equation of continuity in differential form cartesian coordinates is given as

$$\frac{\partial(\rho)}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \tag{1}$$

Where ρ is the density of the fluid. u, v, w represents the velocity components of the flow field in x, y, z directions respectively and t is the time. Although equation 1 represents the general form, the equation simplifies due to several assumptions including steady and incompressible flow in the case of Newtonian fluids.

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{2}$$

The equation can also be written in an alternative form using the ∇ representing the divergence of the velocity vector is given by equation 3.

$$\nabla.\vec{V} = 0 \tag{3}$$

Where, $\vec{V}$ is the velocity vector and ∇ is the del operator

$$\nabla = \frac{\partial}{\partial x}\hat{\imath} + \frac{\partial}{\partial y}\hat{\jmath} + \frac{\partial}{\partial z}\hat{k} \tag{4}$$

$$\vec{V} = u\hat{\imath} + v\hat{\jmath} + w\hat{k} \tag{5}$$

The dot product of ∇ and velocity is known as the divergence of the velocity field.

The Navier-stokes equations are considered to be the fundamental equation of fluid dynamics as it is derived from the basic law of conservation of momentum. This equation consist of the body force terms that act on a differential fluid element when it is placed in a gravitational field, electrical field, magnetic field, etc or combination of these and the surface forces including both normal & shear stresses arising due to the pressure and viscid behavior of the fluid, respectively. The most generalised form of the momentum equation is as follows written with the index notation.

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(u_i u_j)}{\partial x_j} = \frac{\partial(\tau_{ij})}{\partial x_j} + b_i \tag{6}$$

The right hand of the equation represents the sum of all the forces acting on the fluid element. Where, b_i is the body forces i.e., the force which is acting on every single particle of the fluid, τ_{ij} is the stress tensor given by:

$$\tau_{ij} = -p\delta_{ij} + \lambda\frac{\partial u_k}{\partial x_k}\delta_{ij} + \mu\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right] \tag{7}$$

Where δ_{ij} is Kronecker delta, whose value is 1 when i=j and zero (0) for i≠j. λ is the volumetric dilation coefficient, μ is the coefficient of dynamic viscosity and $\frac{\partial u_k}{\partial x_k}$ signifies the volumetric deformation, whereas $\left[\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right]$ is the deformation caused due to shear stresses which act on the fluid element.

Since the fluid flow in the present work is assumed to be steady incompressible flow, the force due to the gravitational field is neglected and implying the stokes hypothesis, the equation 6 can be written for x, y and z-direction respectively as

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z} \tag{8}$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z} \tag{9}$$

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z} \tag{10}$$

The Navier-Stokes equation utilizes the divergence of the stress tensor, $\nabla.\tau$, The stress tensor can be represented in the form of a 3X3 matrix as

$$\tau=\begin{bmatrix}\tau_{xx} & \tau_{xy} & \tau_{xz}\\ \tau_{yx} & \tau_{yy} & \tau_{yz}\\ \tau_{zx} & \tau_{zy} & \tau_{zz}\end{bmatrix}=\begin{bmatrix}-p+2\mu\frac{\partial u}{\partial x} & \mu\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right) & \mu\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)\\ \mu\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right) & -p+2\mu\frac{\partial v}{\partial y} & \mu\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)\\ \mu\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right) & \mu\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right) & -p+2\mu\frac{\partial w}{\partial z}\end{bmatrix} \tag{11}$$

The viscosity, μ is a property of the fluid. It can be a constant as in the case Newtonian fluids, however, it may vary with the temperature, time, and the shear strain rate. These types of fluids fall under the category of non-Newtonian fluids[23].

- **Boundary condition**: One of the challenging areas in this context is how to choose and apply the boundary conditions because those have to be representative of the physiology of the circulatory system as a whole. So, information that is clinically relevant is extracted to make this model more realistic to what we see in coronary arteries. Such as the density of blood was taken as 1080 kg m-3 and dynamic viscosity as 0.0035 Ns m-2 [1]. At Inlet, single velocity waveform as a function of time, Outlets (Pressure outlet 13335 Pa). Unlike in civil, mechanical or aerospace applications where flow is uniformly coming, here is a stop and go i.e., pulsating in nature.

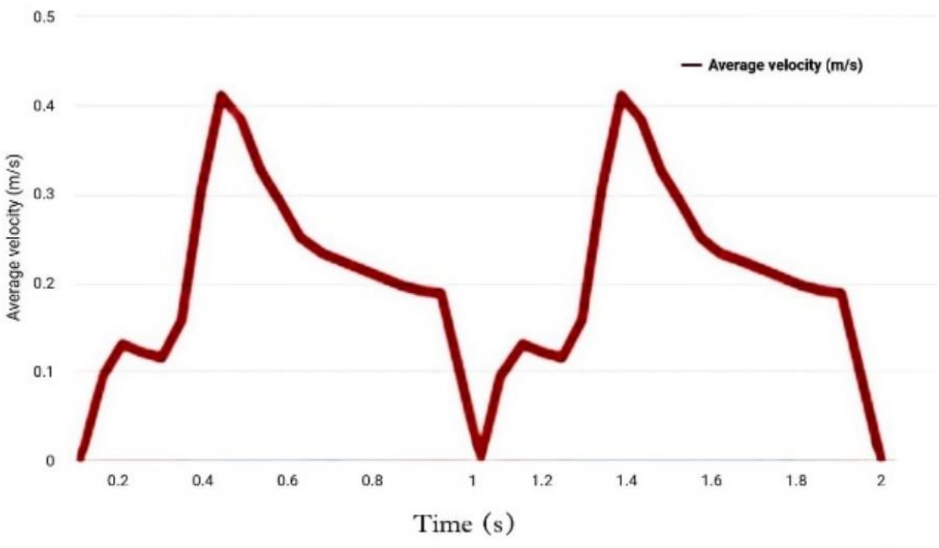
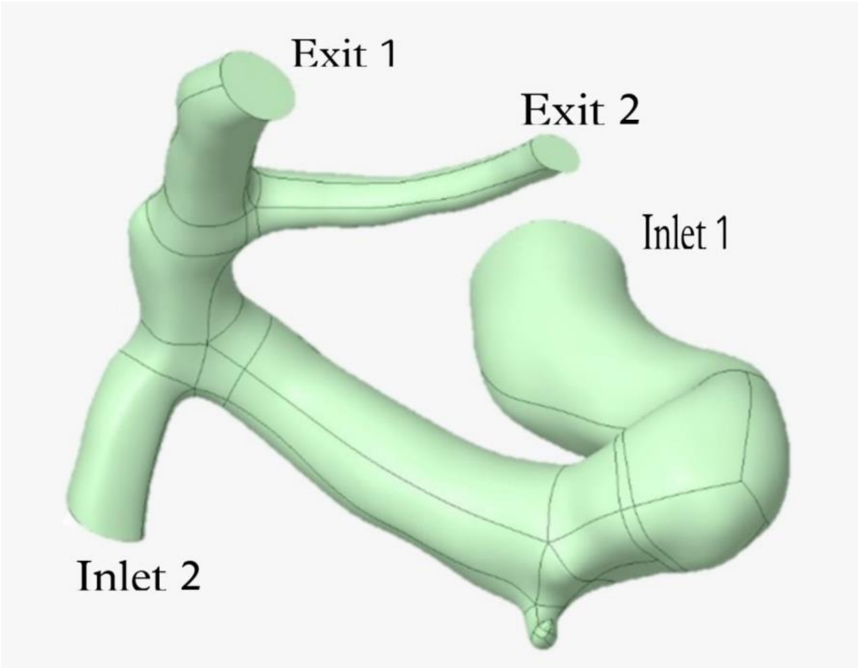


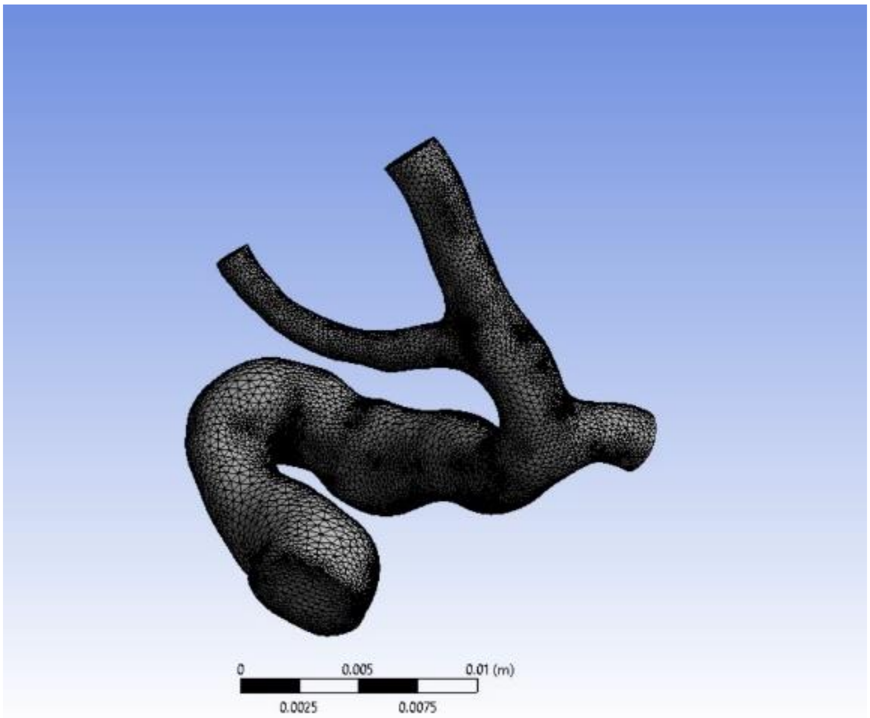
Figure 2: velocity at the entrance of artery

Thus, at the inlet the velocity is not constant. So, by assuming five velocities (say 0.05, 0.1, 0.2, 0.3 and 0.4 m/s) this research has been carried out.

- **Geometry and Mesh**



(a)



(b)

Figure 3: geometry and mesh generation of the artery

CHAPTER 3

MODEL VALIDATION AND GRID INDEPENDENCE

Some initial validation is done to ensure that the model is reliable and the results are accurate. The computational results of Aorta model were compared with that of the results by P. D. Dwidmuthey, C. S. Mathpati and J. B. Joshi [7]. These results are shown in Figure 4. It can be seen that the maximum error is 4.256%. Hence, the computational results comply with that of the experimental results. Therefore, the model and the boundary conditions chosen are accurate and reliable.

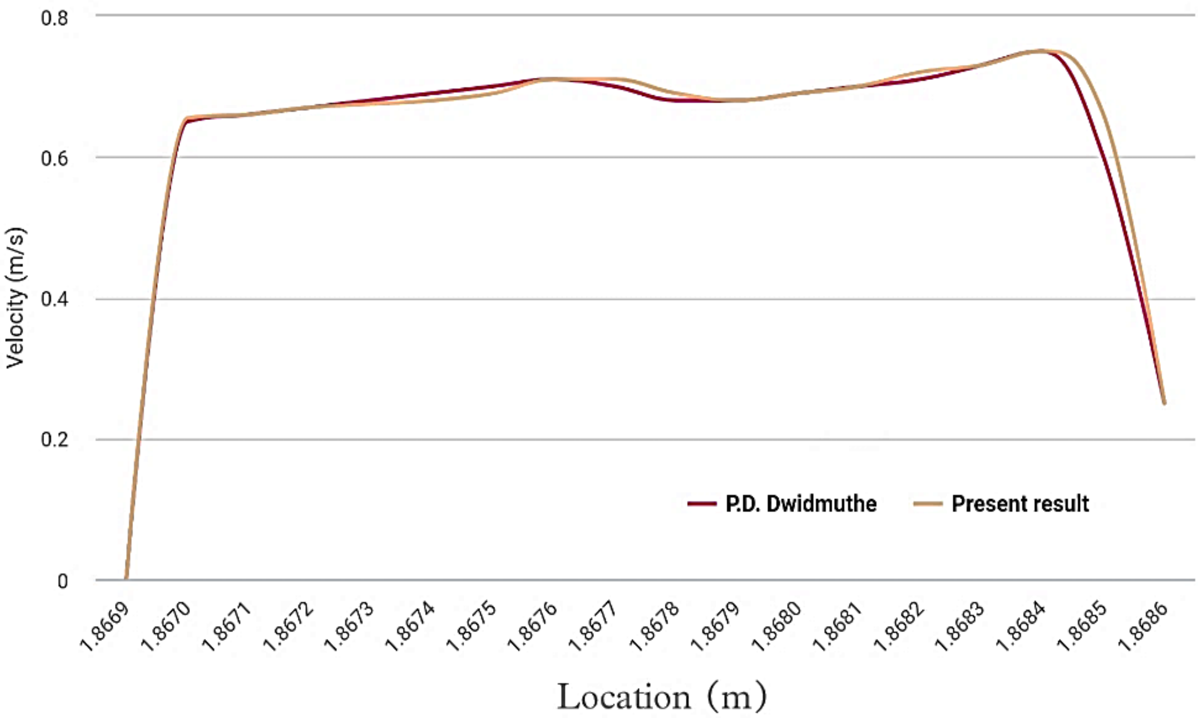


Figure 4: Validation of current study with the existing literature [7]

Grid generation is a crucial step in a CFD study. The quality and size of the elements used to divide the flow domain have a huge impact on the numerical results and its accuracy. Coarse mesh takes less computational time but has poor numerical prediction capability whereas fine mesh is proven to give better accuracy with higher computational time. Thus, selecting an optimized grid element size is necessary keeping both the accuracy and the computational time in mind. A grid sensitivity analysis is performed (Figure 5) and found that above 2 million mesh size there is almost no changes in results.
So, the optimum number of elements should be above 2 millions elements.

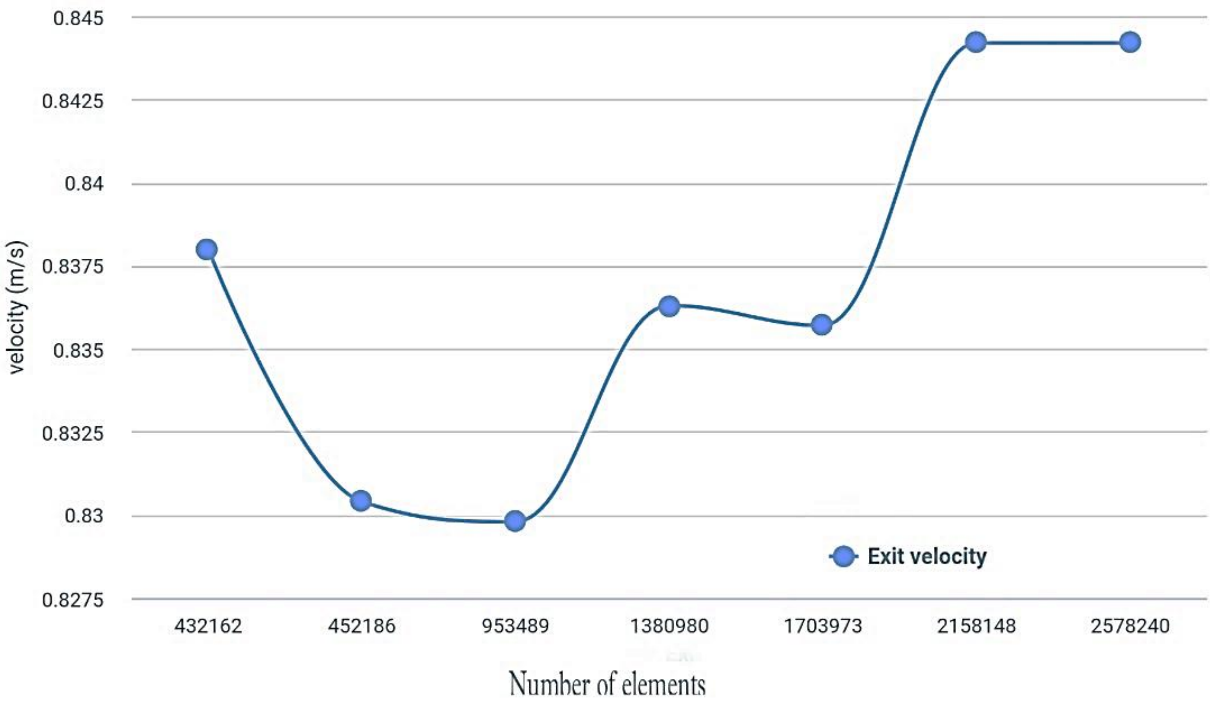


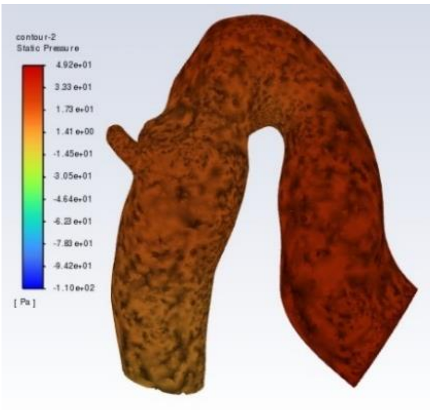
Figure 5: Grid sensitivity study

CHAPTER 4

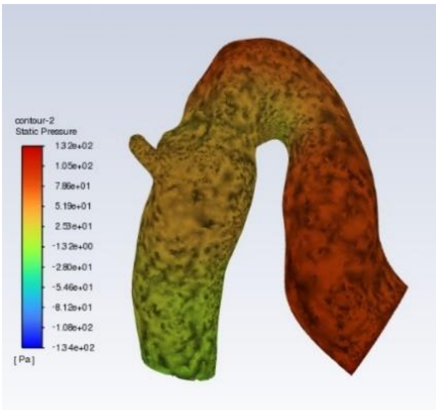
RESULTS

The contours of pressure and velocity of the 3D aorta having bent, Y bifurcation and Y intersection blood vessel, for different inlet inputs resulted as follows:

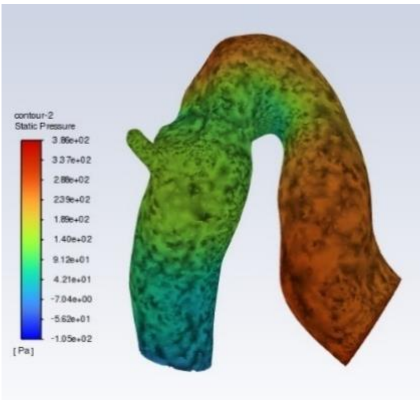
4.1) **Curvature in aorta**



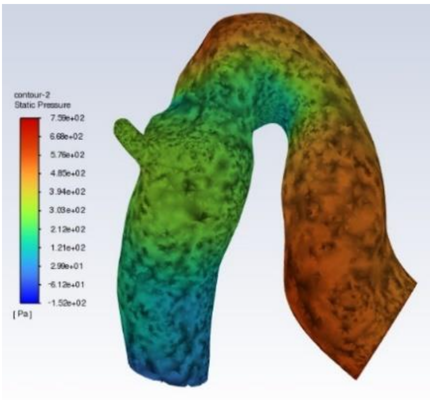
a



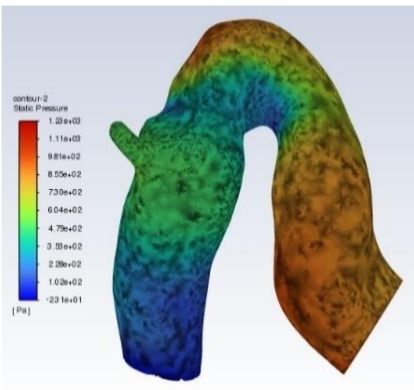
b



c



d



e

Figure 6: Pressure contour of bent aorta at inlet velocity (a) 0.05m/s, (b) 0.1m/s, (c) 0.2m/s, (d) 0.3m/s, (e) 0.4m/s

The above pictures depict that for the inlet velocity of 0.05m/s the pressure of bent in blood vessel can reaches up to 49.2 Pascal and at velocity of 0.4m/s pressure in bent can reach up to 1230 Pascal in artery. Due to curvature a cross-flow motion can be observed in aorta[22]. As a result, maximum pressure difference can be observed between inner and outer wall of the curvature.

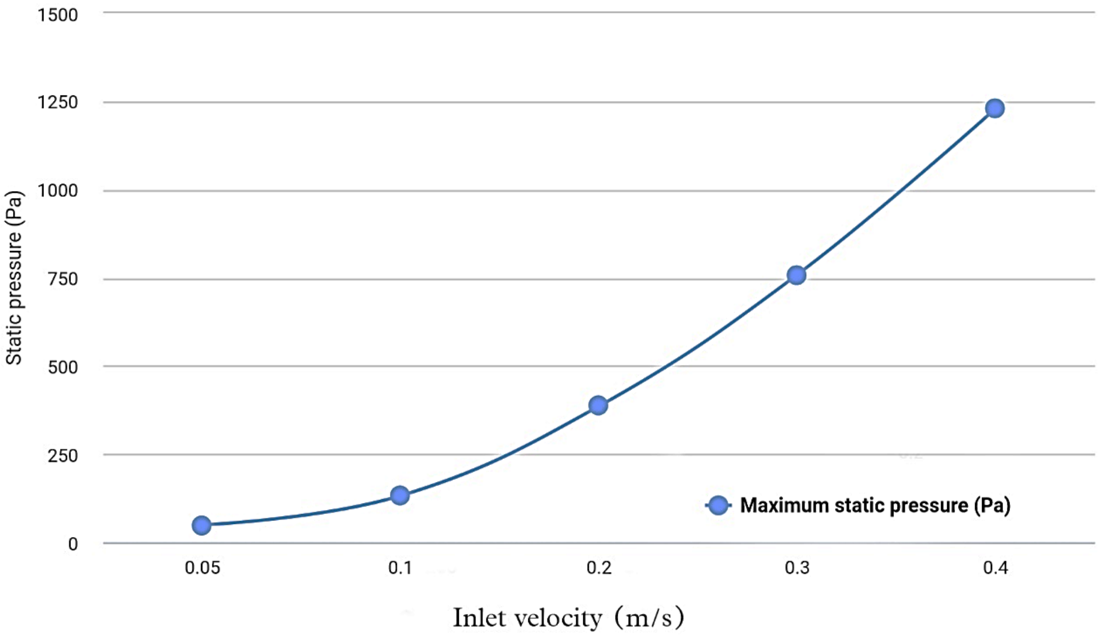
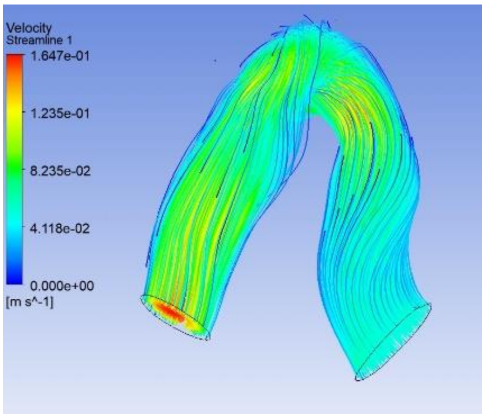
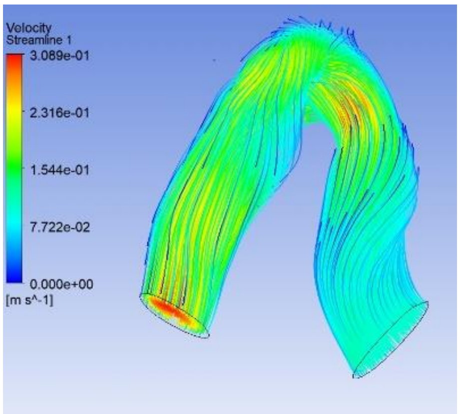


Figure 7: Static pressure change with inlet velocity

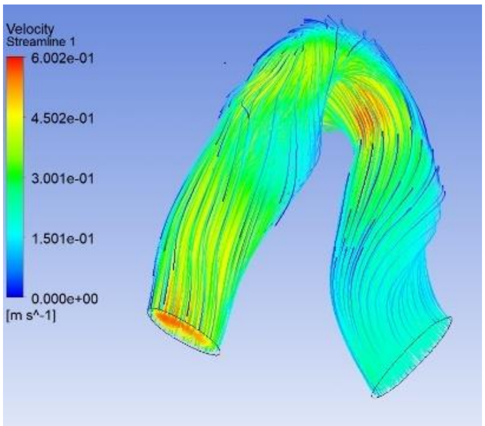
It is clear from Figure that when inlet velocity is varied from 0.05 m/s to 0.4 m/s, then static pressure increases and a maximum pressure drop is observed at outlet when there is a bent in artery.



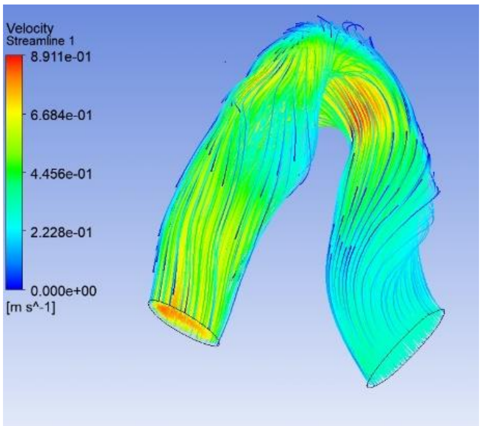
a



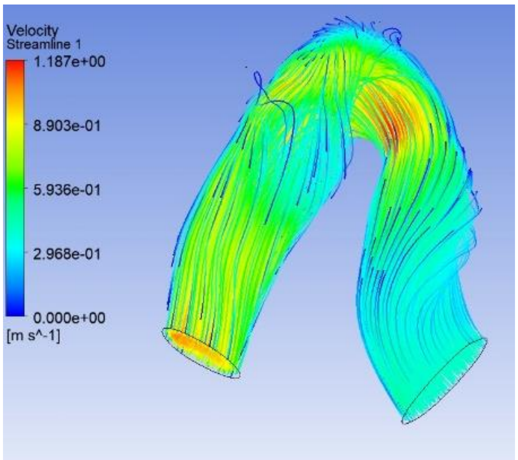
b



c



d



e

Figure 8: Velocity contour of bent aorta at inlet velocity (a) 0.05m/s, (b) 0.1m/s, (c) 0.2m/s, (d) 0.3m/s, (e) 0.4m/s

The above pictures depict that for the inlet velocity of 0.05 m/s the velocity at the exit of bend in blood vessel is in range of 0-0.1647 m/s but as we increase inlet velocity to 0.4 m/s the exit velocity reaches to 1.187 m/s.

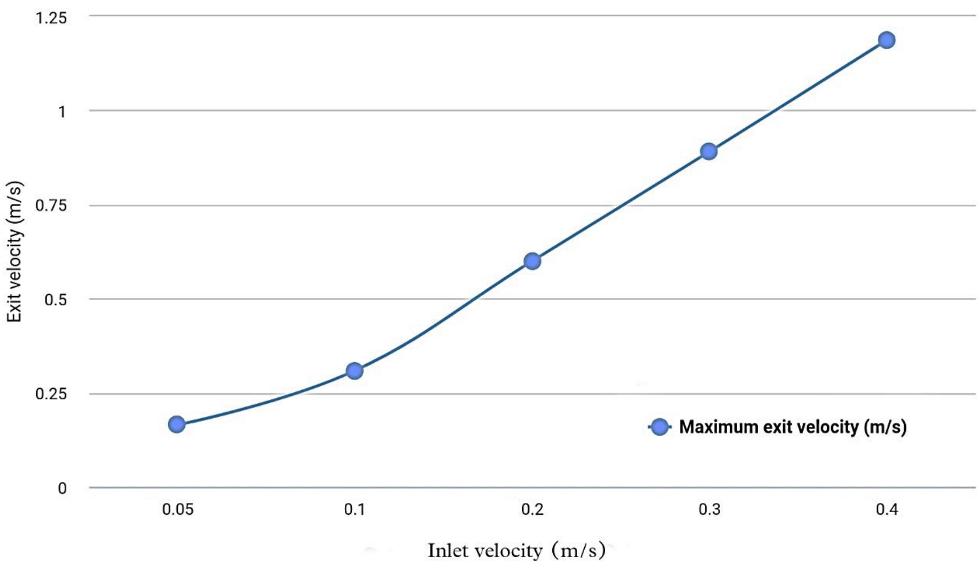
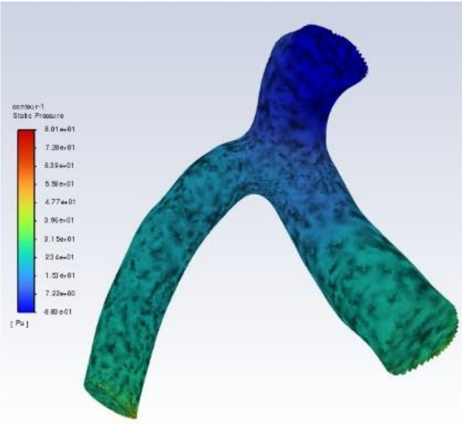


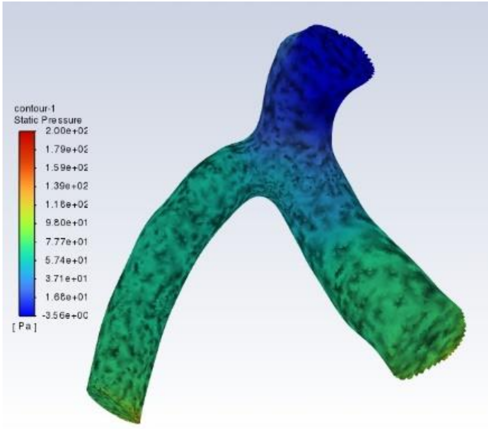
Figure 9: Exit velocity change with inlet velocity

- From the above results of curvature in artery it is observed that when inlet velocity is varied from 0.05 m/s to 0.4 m/s exit velocity at outlet also increases.

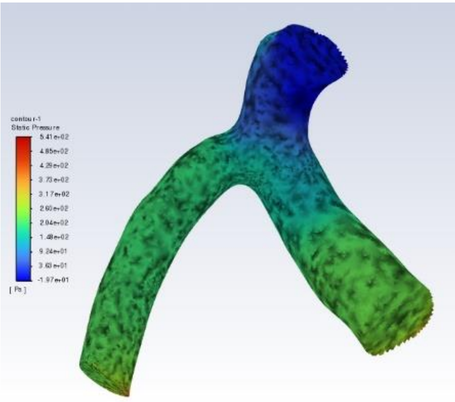
4.2) Y intersection



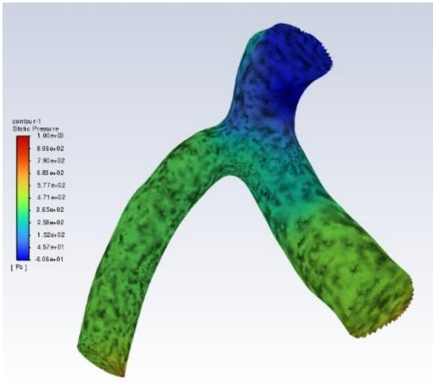
a



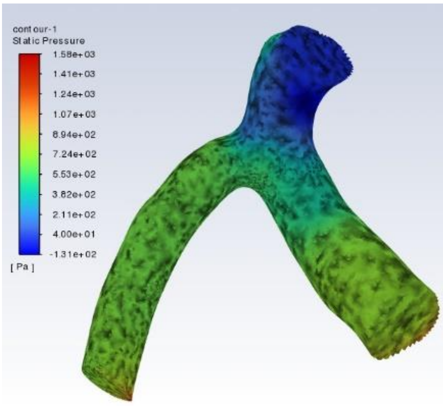
b



c



d



e

Figure 10: Pressure contour of Y intersection at inlet velocity (a) 0.05m/s, (b) 0.1m/s, (c) 0.2m/s, (d) 0.3m/s, (e) 0.4m/s

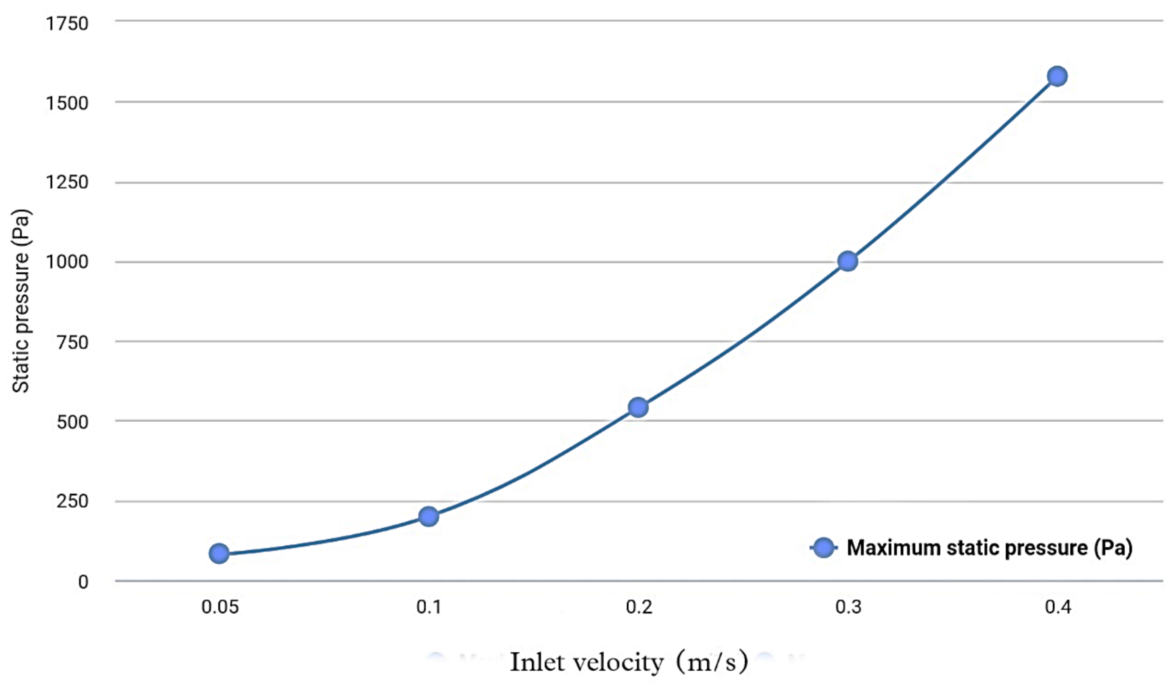
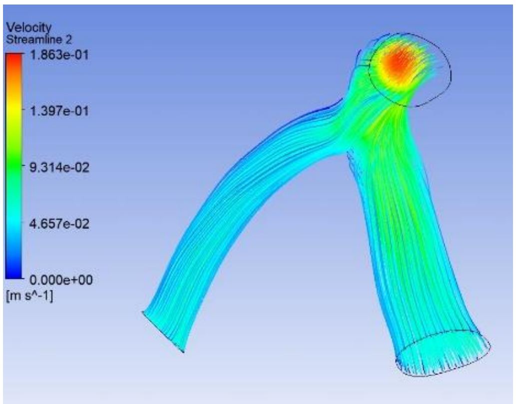
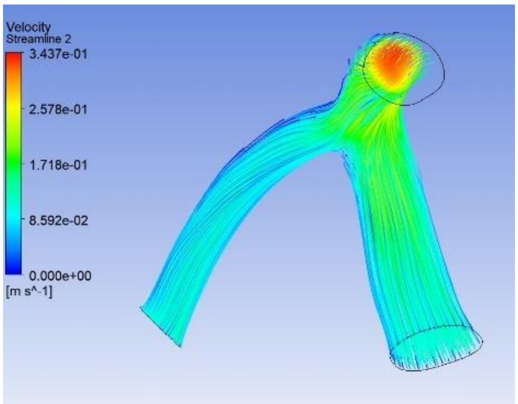


Figure 11: Static pressure change with inlet velocity

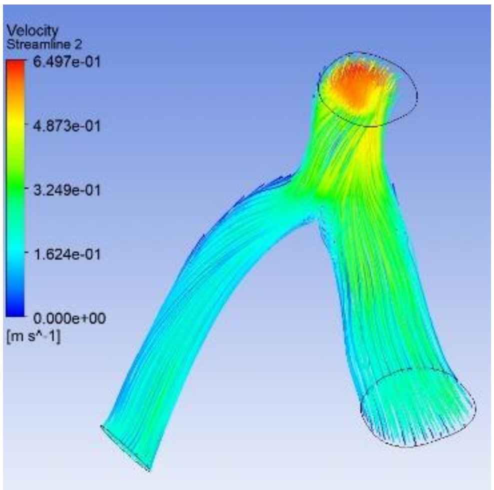
The above pictures depict that for the inlet velocity of 0.05 m/s the static pressure blood vessel is in range of 80.1 Pascal but as we increase inlet velocity to 0.4 m/s the static pressure reaches to 1580 Pascal and a very high-pressure drop is observed at outlet of intersecting artery.



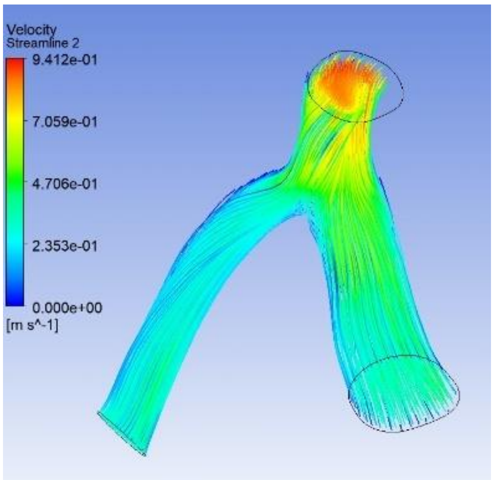
a



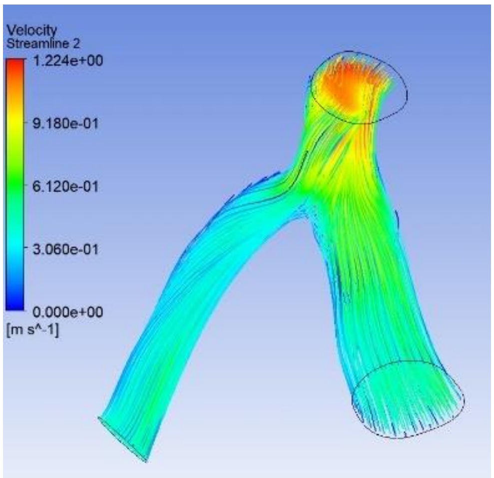
b



c



d



e

Figure 12: Velocity contour of Y intersection at inlet velocity (a) 0.05m/s, (b) 0.1m/s, (c) 0.2m/s, (d) 0.3m/s, (e) 0.4m/s

In the above picture we depict that the velocity of the blood flow is increasing from inlet of the Y intersection blood vessel to the outlet of the same blood vessel. The maximum velocity is found in the outlet i.e., 1.224 m/s when inlet velocity is 0.4 m/s.

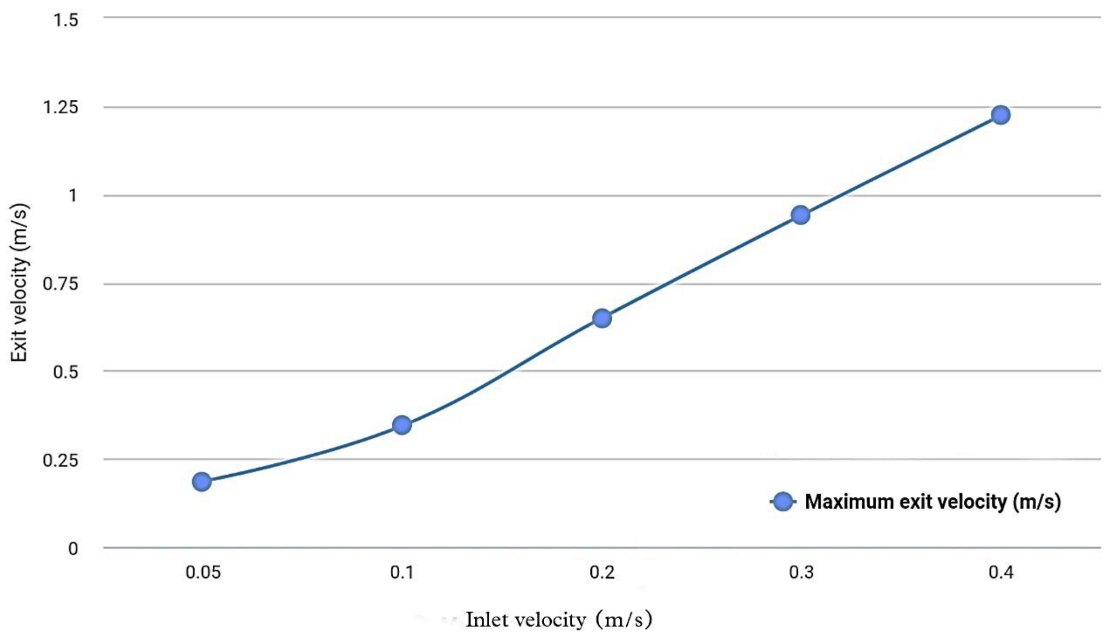


Figure 13: Exit velocity change with inlet velocity

In the above picture it is depicted that the velocity of the blood flow is increasing from inlet of the Y intersection blood vessel to the outlet. As expected, pressure drop drastically with increase in velocity at outlet of 1.224m/s.

4.3) Y bifurcation

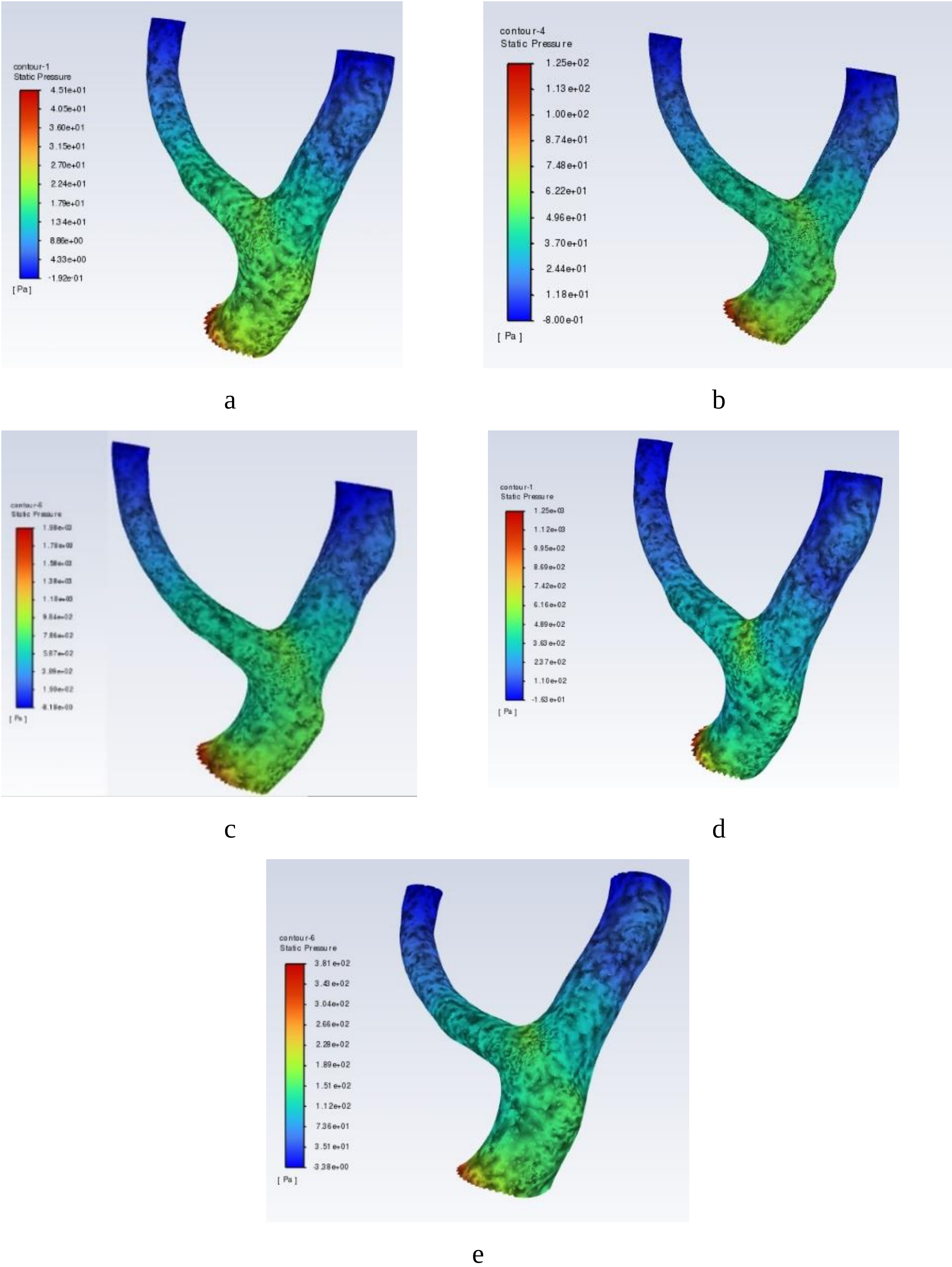


Figure 14: Pressure contour of Y-bifurcation at inlet velocity (a) 0.05m/s, (b) 0.1m/s, (c) 0.2m/s, (d) 0.3m/s, (e) 0.4m/s

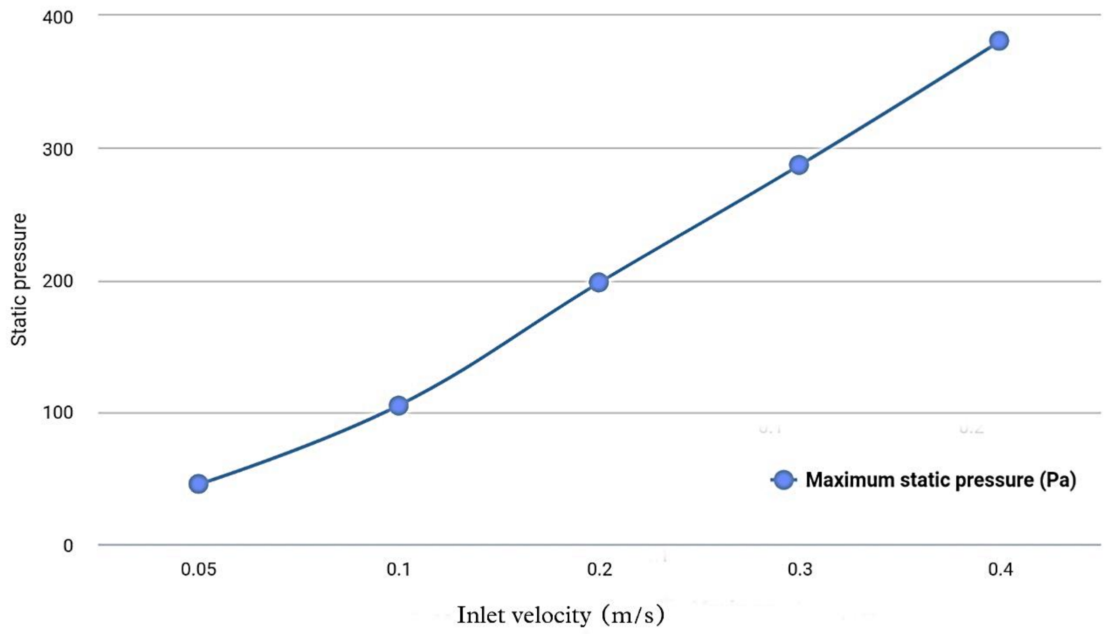
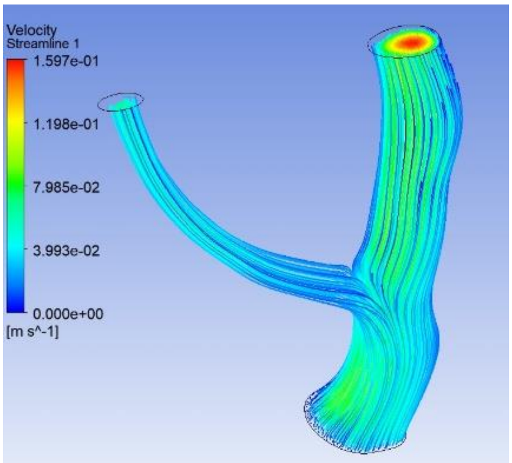
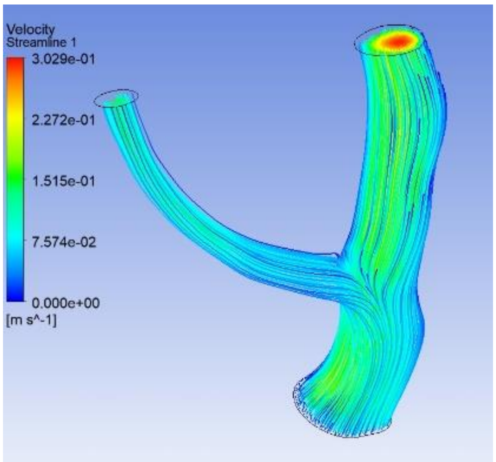


Figure 15: Static pressure change with inlet velocity

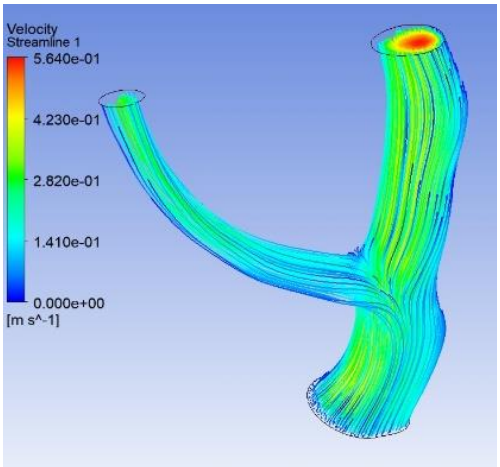
In the above picture it is depicted that the pressure drop is higher in narrow bifurcating artery as compared to the other vessel. The maximum static pressure is found to be 381 Pascal.



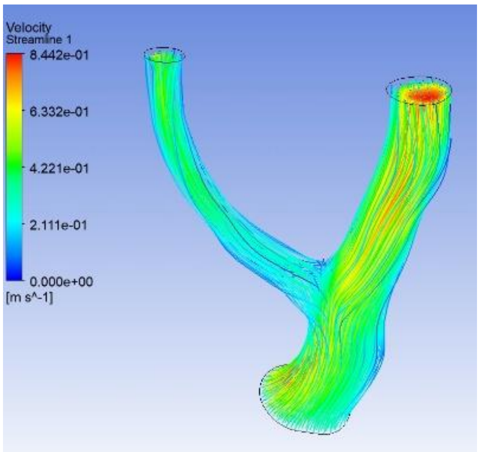
a



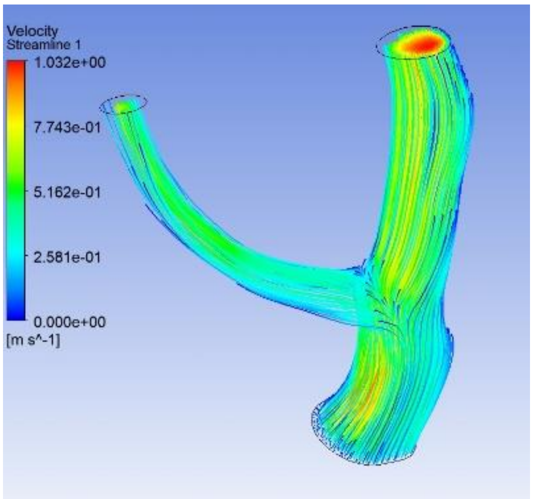
b



c



d



e

Figure 16: Velocity contour of Y-bifurcation at inlet velocity (a) 0.05m/s, (b) 0.1m/s, (c) 0.2m/s, (d) 0.3m/s, (e) 0.4m/s

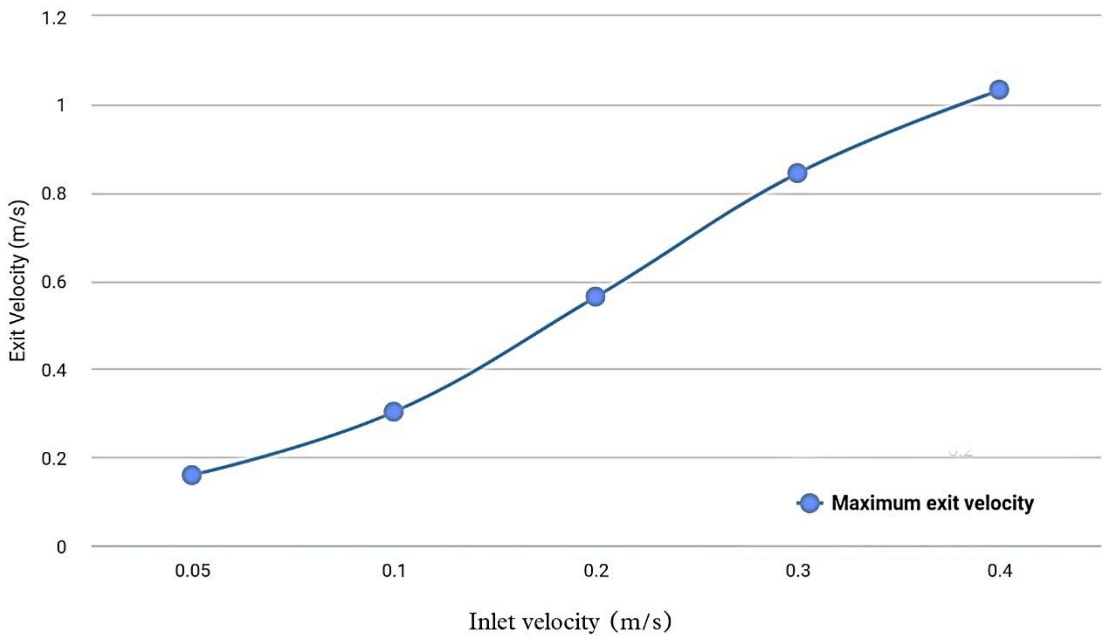
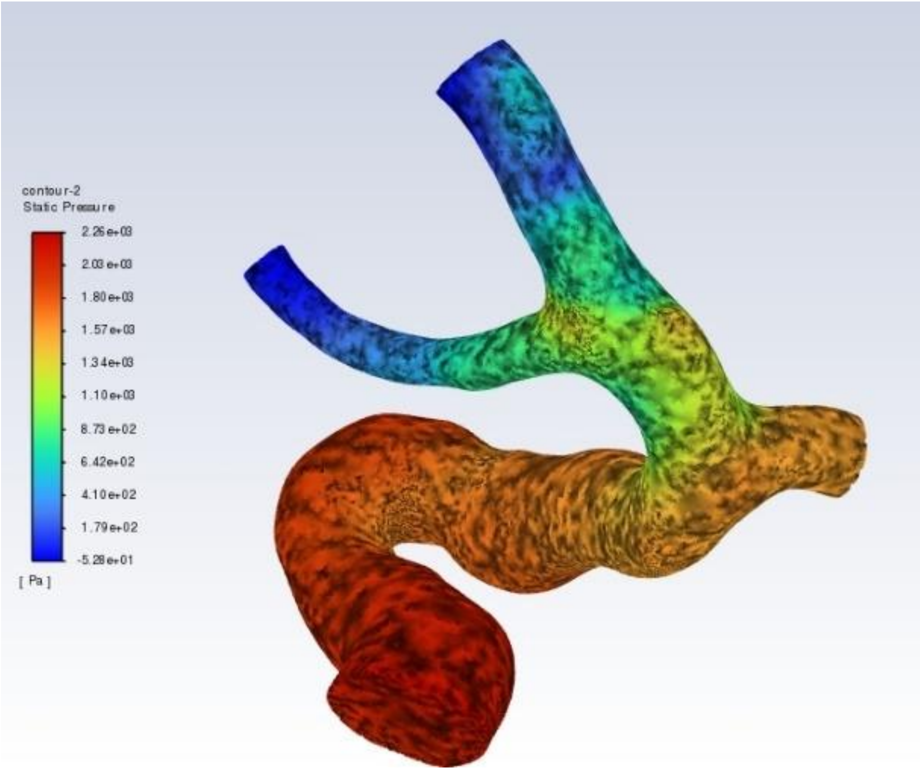


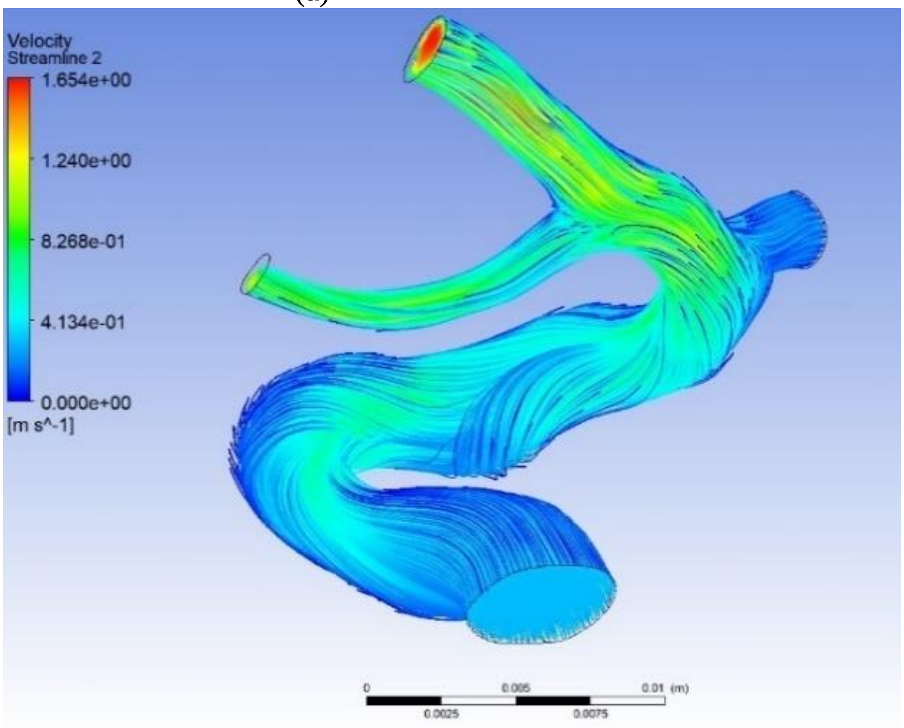
Figure 17: Exit velocity change with inlet velocity

In the above picture it is shown that the velocity of the blood flow is higher in narrow bifurcating artery as compared to the other vessel. The maximum velocity is found in the outlet is 1.032 m/s.

4.4) Combination of curvature, Y-intersection and bifurcating artery



(a)



(b)

Figure 18: (a) Pressure contour and (b) Velocity contour at inlet velocity= 0.3 m/s

In fig 18 it is shown that the velocity of the blood flow found in the outlet is 1.65 m/s and pressure drop to -52.8 Pascal when inlet velocity is 0.3 m/s.

CHAPTER 5
CONCLUSIONS

- For pulsatile flow velocity is not constant at two different time intervals so in this paper by taking two different velocities all the geometric features are observed.
- when all the geometric curves, bend, bifurcation and intersections are combined velocity reaches the maximum and a very pressure drop is observed
- As expected, an increase in flow velocity is observed when blood flows into a narrow artery as a result of conservation of mass. However, some of the profiles show a decrease in velocity in the artery due to cross flow as a result of curvature in geometry.
- The intensity of flow was concentrated in the center of the artery even when the velocity reaches its maximum. Hence, a low-flow-intensity regions form near the walls.
- CFD simulations shows that there is high pressure difference between outer and inner surface of the bend as a result secondary flow is created.

5.1) FUTURE STUDY

- There is a time for every heartbeat cycle where blood velocity decreases and next beat comes in and pushes it forward and we see the considerable build-up of viscosity. A constant viscosity model will not be able to predict that and Newtonian approach cannot be used in that case. It definitely challenges some long-standing Newtonian assumptions. Therefore, it will lead to a whole other set of new research directions.
- To take the patient data where a time resolved MRI scan is used to extract the motion of the vessels and apply that motion as a moving wall boundary condition in the flow simulation and assumption of rigid wall can be eliminated.

REFERENCES

[1] Yogesh V. Borse, Prof. S.A. Giri, 2016 “Computational Fluid Dynamics Analysis of Blood Flow in Human Aorta,” IARJSET, Volume: 3, Issue: 6.

[2] F. Sanchis-Gomar, C. Perez-Quilis, R. Leischik, A. Lucia, 2016, “Epidemiology of coronary heart disease and acute coronary syndrome,” Ann Transl Med, 4 (13) p. 256

[3] C. Ke, R. Gupta, D. Xavier, *et al.*, 2018, “Divergent trends in ischaemic heart disease and stroke mortality in India from 2000 to 2015” a nationally representative mortality study Lancet Glob Health, 6 (8) pp. e914-e923

[4] David, N. Ku., 1997, “Blood Flow in Arteries,” Annual reviews of fluid mechanics, 29, pp. 399-434.

[5] Silva, E., Teixeria, S., and Lobarinhas, P., 2010, “Computational Fluid Dynamics Simulations: an approach to Evaluate Cardiovascular Dysfunction,” University of Minho., Portugal, Europe, pp. 25-46.

[6] Taylor, C. A., and Draney, M. T., 2004, “Experimental and Computational Method in Cardiovascular Fluid Mechanics,” Annual Review of Fluid Mechanics, 36, pp.197-231.

[7] Dwidmuthe, Prachi & Mathpati, C & Joshi, J. (2018). CFD Simulation of Blood Flow inside the Human Artery: Aorta. FMFP2017–PAPER NO. 679

[8] Shafiullah Mohammad, M., 2012,”Computational Fluid Dynamics Analysis of Blood Flow through Stented Arteries”, Master’s Thesis, Northern Illinois University, DeKalb, IL

[9] Shaik, E., Hoffmann, K., Dietiker, J. F., 2006, “Numerical Flow Simulations of Blood in Arteries,” 44th American institute of aeronautics and astronautics, AIAA Paper No.2006 -294

[10] Robert F.W., Pritchard P.J., 2010, “Introduction to Fluid Mechanics, 8th ed., Wiley. [14] Series in Computational Methods in Mechanics and Thermal Sciences by Patankar

[11]. Panta Y.M And Harvey M, 2012, "Computational Fluid Dynamic (CFD) Analysis Of Blood Flow Through Human Arteries", Journal Of Computational And Modeling, ISSN,Vol 2,Issue 1,Pp.- 27-29.

[12]. K.P Beenuty , M Anburajan, 2011, "Simulation And Analysis Of Aortic Bypass Surgery Using Computational Fluid Dynamics", IEEE.

[13]. Sun, Z., & Xu, L. (2014). Computational fluid dynamics in coronary artery disease. *Computerized medical imaging and graphics*, *38*(8), 651-663.

[14] Luo Ke et al., Fluid-Solid Interaction Analysis on Iliac Bifurcation Artery: A Numerical Study, International Journal of Computational Methods, doi: 10.1142/S0219876218501128

[15] B.M. Johnston, P.R. Johnson., S. Corney., and D. Kilpatrick., 2004, “Non-Newtonian blood flow in human right coronary arteries: steady state

simulations," Journal of Biomechanics, 37, pp.709 – 720.

[16] L. Zheng, X. Zhang, in Modeling and Analysis of Modern Fluid Problems, 2017

[17] Shaik, E. (2007). Numerical simulations of blood flow In arteries using fluid-structure interactions (Doctoral dissertation, Wichita State University).

[18] Ahmad, R., Farooqi, A., Farooqi, R., Hamadneh, N. N., Fayz-Al-Asad, M., Khan, I., ... & Saleem Khan, M. F. (2021). An Analytical Approach to Study the Blood Flow over a Nonlinear Tapering Stenosed Artery in Flow of Carreau Fluid Model. *Complexity*, *2021*.

[19] Armstrong, M., Rook, K., Pulles, W., Deegan, M., & Corrigan, T. (2021). Importance of viscoelasticity in the thixotropic behavior of human blood. *Rheologica Acta*, *60*(2), 119-140.

[20] Satoshi Numata , Keiichi Itatani , et al., 2017, "Computational Fluid Dynamics Simulation Of The Right

Subclavian Artery Cannulation", The Journal Of Thoracic And Cardio Vascular Surgery, Vol 154, pp.- 480-487.

[21] Oshima, M., Torii, R., Kobayashi, T., Taniguchi, N., & Takagi, K. (2001). Finite element simulation of blood flow in the cerebral artery. *Computer methods in applied mechanics and engineering*, *191*(6-7), 661-671.

[22] Hoogstraten, H. W., Kootstra, J. G., Hillen, B., Krijger, J. K. B., & Wensing, P. J. W. (1996). Numerical simulation of blood flow in an artery with two successive bends. *Journal of biomechanics*, *29*(8), 1075-1083.

[23] Foong, L. K., Shirani, N., Toghraie, D., Zarringhalam, M., & Afrand, M. (2020). Numerical simulation of blood flow inside an artery under applying constant heat flux using Newtonian and non-Newtonian approaches for biomedical engineering. *Computer Methods and Programs in Biomedicine*, *190*, 105375.