Blood Hammer Model-Modeling and Analysis of Coronary Artery with Vascular Occlusion using Computational Fluid Dynamics Analysis

Jameela N K[1], Linda Mathew[2] , Hemalatha Karnan[3]*
[1],[2] Student/Biomedical Department [3]*Assistant Professor/Biomedical Department,
Dhanalakshmi Srinivasan Engineering College,Perambalur

Abstract **- Blood vessels (veins and arteries) are the super highways that circulate blood throughout the body with the muscular pumps. Any obstruction in the circulatory path by clot (or) plague inside the blood vessels, disturb the constant flow of blood thereby disturbing the hemodynamic of blood. This leads to fatal condition. This paper aims to study and analyze the hemodynamic mechanism of a single segment of stiffened coronary artery due to sudden blockage (or) stagnant blood flow and the relation between mechanical and rheological parameters in vascular occlusion. The changes in oscillating pressure, pressure profile, velocity, and skin friction coefficient and wall shear stress are studied and analyzed for various dimension of vascular occlusion and for different shapes of blood clot. We develop a "Blood Hammer Model" keeping in mind the basics of "water hammer" which is described in hydraulics. Modeling and analysis of the blood vessel with various dimension of vascular occlusion was done using the ANSYS Design Modeler Software and Computational Fluid Dynamics (CFD) methodology. From the complete theory of the designed 'blood hammer model' along with the derived explicit expressions on oscillating pressure and wall shear stress and the Computational Fluid Dynamics simulation of blood vessel gives relevant information for the diagnosis of the circulatory barricade.**

Keywords: Arteries, Computational fluid dynamics, Hemodynamics, Rheological parameters.

1.INTRODUCTION

As estimated 17.7 million people died from CVD's in 2015, representing 31% of all global deaths. Of these deaths, an estimated 7.4 million were died due to coronary heart disease and 6.7 million were died due to stroke. Heart attacks and strokes are usually acute events and are mainly caused by a blockage that prevents blood from flowing to heart or brain. The blood hammer phenomenon is a sudden increase of the upstream blood pressure in a blood vessel (especially artery or arteriole) when the bloodstream is abruptly blocked by vessel obstruction. The term "blood-hammer" was introduced in cerebral hemodynamics by analogy with the hydraulic expression "water hammer," already used in vascular physiology to designate an arterial pulse variety, the "water-hammer pulse." Complete understanding of the relationship between mechanical parameters in vascular occlusions is a critical issue, which can play an important role in the future diagnosis, understanding and treatment of vascular diseases.Wall shear stress has been proven to play a critical role in the formation and development of atherosclerotic plaques. Numerous studies have demonstrated that hemodynamic changes in arteries have a strong impact on cardiovascular disease. The oscillating pressure, pressure drop and blood flow velocity are the other predominant haemodynamic factors.

The present work aims to describe and provide interpretation of basic mechanical and haemodynamic phenomena related to forces applied to arterial walls and especially wall shear stress and oscillating pressure. It has long been a dream of science to apply the leverage of its more mathematical disciplines to its biological, especially medical, investigations. Biology is messy, however, and will not be shoehorned into the same kind of mathematical formalization as (say) physics or information theory or even chemistry. Yet a mathematical approach may very well help us better to understand a biological process, or to manipulate it, or to connect changes in its structure to changes in its function and make predictions on that basis; and hence to diagnose and treat diseases.

The object of this study is human blood clotting, a process exhibiting complexity typical of biological systems: no single set of differential equations or discrete-process model can completely describe the coagulation cascade (at least not given the current state of knowledge); yet on the other hand it is obviously of crucial clinical importance.

II.LITERATURE SURVEY

CFD Simulation of Blood Clot Behavior Using GP Device Kuzilati Kushaari, Computer Modeling and Simulation (2012)

There are 130,000 strokes in the UK alone each year, causing death or servere disability to those who are affected. Advances in recent times have enabled clot extraction to be undertaken using mechanical thrombectomy devices. In this paper we present a model of such an extraction device, and investigate the value of such modelling in predicting the behavior of the device under given conditions. A 3-dimensional blood clot simulation associated with a plastic arterial catheter is studied by applying CFD simulations with different size of grid, using the Volume of Fluid (VOF) model. Grid size study shows that the smaller grid size (finer mesh) results in higher accuracy of the output result of the simulation. It is noted that grid independency is achieved when any further increase in the number of cells did not adversely affect the simulation results, the optimum grid size avoided any unnecessary prolonged computational effort required for the simulations with large number of cells. From the results obtained in the grid size study, it can be concluded that mesh 0.3 has reached its asymptotic level. It is believed that it can predict the right clot deformation and blood flow in the device. All the pressures used in this study are able to remove the blood clot. It is found that the higher the pressure applied the faster the removal. Different deformation patterns are also observed when different pressures are applied. It is found that the fastest time to remove the blood clot with the range of pressure used in this work is 0.006 s, when 60 kPa of suction pressure is applied.

Analysis of the “GPATD”: Geometrical Influence on Blood Clot Extraction using CFD Simulation Gregorio Romero, Computer Modelling and Simulation (2014)

In this work, we present the study of the influence of geometry on an experimental device recently developed in the UK, called the “GP” Thrombus Aspiration Device (GPTAD). This device has been designed to remove blood clots without the need to make contact with the clot itself, thereby potentially reducing the risk of problems such as downstream embolisation. To obtain the minimum pressure necessary to extract the clot and to optimize blood clot extraction, we simulate the performance of the GPTAD analyzing the pressure losses and pressure distribution taking into account the geometrical effects. Previous full models have been undertaken using the Bond Graph technique. However in this paper we include the analysis of the influence of the of the geometry device using Computational Fluid Dynamics simulation. We model a range of diameters for the GPTAD considering a 95% occlusion case, different lengths and diameters of catheter, and different GP geometric characteristics. In each case we determine the pressure losses and distribution just in front of the attached blood clot.

"The Use of Computational Fluid Dynamics in the Development of Ventricular Assist Devices" Fraser KH1, Taskin ME, Griffith BP, Wu ZJ, 'American society of artificial internal organs (2011)'

Progress in the field of prosthetic cardiovascular devices has significantly contributed to the rapid advancements in cardiac therapy during the last four decades. The concept of mechanical circulatory assistance was established with the first successful clinical use of heart-lung machines for cardiopulmonary bypass. Since then a variety of devices have been developed to replace or assist diseased components of the cardiovascular system. Ventricular assist devices (VADs) are basically mechanical pumps designed to augment or replace the function of one or more chambers of the failing heart.

"Computational Fluid Dynamics Analysis of Thrombosis Potential In Left Ventricular Assist Device Drainage Cannulae" Fraser KH1, Zhang T, Taskin ME, Griffith BP, Wu ZJ, 'American society of artificial internal organs (2010)'

Cannulation is necessary when blood is removed from the body, for example in hem dialysis, cardiopulmonary bypass, blood oxygenators, and ventricular assist devices. Artificial blood contacting surfaces are prone to thrombosis, especially in the presence of stagnant or recirculation flow. In this work, computational fluid dynamics was used to investigate the blood flow fields in three clinically available cannulae (Medtronic DLP 12, 16 and 24 F), used as drainage for pediatric circulatory support, and to calculate parameters which may be indicative of thrombosis potential. The results show that using the 24 F cannula below flow rates of about 0.75 l/min produces hemodynamic conditions which may increase the risk of blood clotting within the cannula. No reasons are indicated for not using the 12 or 16 F cannulae with flow rates between 0.25 and 3.0 l/ min

"COMPUTATIONAL FLUID DYNAMICS SIMULATION OF EARLY DIAGNOSIS OF DEEP VEIN THROMBOSIS" Nur Shazilah bt Aziz, Nabilah bt Ibrahim 'ARPN Journal of Engineering and Applied Sciences'

This paper presents a validation of in vivo experiment of early diagnosis of Deep Vein Thrombosis (DVT) using Computational Fluid Dynamics (CFD) method. This study was focusing on the pressure and also velocity of blood along the popliteal vein distribution. It is important to study the early stage of DVT as it could prevent the fatal injury to the patients. By using Ansys-CFX, the blood movement in the vein can be further analysed. The result of pressure shows that, the highest velocity value was 15.45 cm/s and the lowest velocity recorded was 0.73 cm/s.

"Computational Study of Thrombus Formation and Clotting Factor Effects under Venous Flow Conditions" Govindarajan V1, Rakesh V1, Reifman J2, Mitrophanov AY1, 'Bio physical journal(2016)'

A comprehensive understanding of thrombus formation as a physicochemical process that has evolved to protect the integrity of the human vasculature is critical to our ability to predict and control pathological states caused by a malfunctioning blood coagulation system. Despite numerous investigations, the spatial and temporal details of thrombus growth as a multicomponent process are not fully understood. Here, we used computational modeling to investigate the temporal changes in the spatial distributions of the key enzymatic (i.e., thrombin) and structural (i.e., platelets and fibrin) components within a growing thrombus. Moreover, we investigated the interplay between clot structure and its mechanical properties, such as hydraulic resistance to flow. Our model relied on the coupling of computational fluid dynamics and biochemical kinetics, and was validated using flow-chamber data from a previous experimental study. The model allowed us to identify the distinct patterns characterizing the spatial distributions of thrombin, platelets, and fibrin accumulating within a thrombus. Our modeling results suggested that under the simulated conditions, thrombin kinetics was determined predominantly by prothrombinase. Furthermore, our simulations showed that thrombus resistance imparted by fibrin was ~30-fold higher than that imparted by platelets. Yet, thrombus-mediated blood flow occlusion was driven primarily by the platelet deposition process, because the height of the platelet accumulation domain was approximately twice that of the fibrin accumulation domain. Fibrinogen supplementation in normal blood resulted in a nonlinear increase in thrombus resistance, and for a supplemented fibrinogen level of 48%, the thrombus resistance increased by ~2.7-fold. Finally, our model predicted that restoring the normal levels of clotting factors II, IX, and X while simultaneously restoring fibrinogen (to 88% of its normal level) in diluted blood can restore fibrin generation to ~78% of its normal level and hence improve clot formation under dilution.

"Computational Fluid Dynamic Analysis of Blood Flow Pattern - A Review" Bharath Ganesan, Karthikeyan Mayakrishnan, International Journal of Science and Research(2012)

Computational Fluid Dynamics (CFD) has made impressive progress in the past decade and has evolved into a promising design tool for the development of biomedical devices. Rheology deals with the flow and deformations of fluids in a set of given conditions Blood being a biological fluid has its own behavior and rheology, flow pattern of blood varies during the flow of blood in the blood vessels in the body. The utilization of computational fluid dynamics in the analysis of blood behavior has been discussed and the potential for developing an instrumentation using flow pattern is analysed.

"Blood Clot Dissolution Dynamics Simulation during Thrombolytic Therapy" Sersa I, Tratar G, Blinc A. 'Journal of chemical engineering and modeling'

Nonocclusive blood clots only partially fill blood vessels and together with the adjacent vessel wall form a channel through which blood flows at usually much higher velocities than in normal vessels. Our aim was to find a theoretical explanation for the experimentally observed fact that fast flowing blood through the channel has a large effect on the increase of the clot dissolution rate compared to the dissolution rate in the absence of flow. Blood flow through the channel increases transport of dissolution agents to the clot and also exerts large forces to the surface of the clot along the channel. Proposed is a model for clot dissolution which assumes that the clot dissolution rate is proportional to the forces of flowing blood to the surface of the clot multiplied by the average blood velocity. The model has been verified by fitting to experimental magnetic resonance imaging data obtained by dynamical magnetic resonance microscopy of clots dissolved by recombinant tissue plasminogen activator in an artificial blood flow system.

III .CFD METHODOLOGY

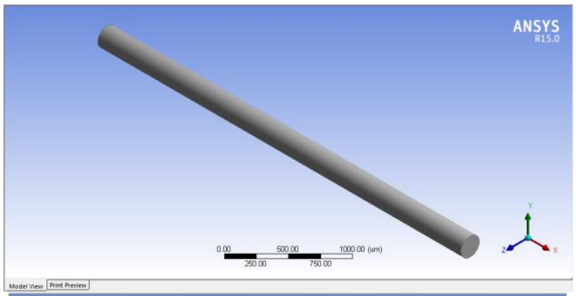
CFD is a method to numerically calculate heat transfer and fluid flow. Currently, its main application is as an engineering method, to provide data that is complementary to theoretical and experimental data. This is mainly the domain of commercially available codes and in-house codes at large companies.CFD can also be used for purely scientific studies, e.g. into the fundamentals of turbulence. This is more common in academic institutions and government research laboratories. Codes are usually developed to specifically study a certain problem.

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. The result of CFD analyses is relevant engineering data used in Conceptual studies of new designs, detailed product development, Troubleshooting, Redesign. CFD analysis complements testing and experimentation. Reduces the total effort required in the laboratory.

1. PRE-PROCESSING

1.1 MODELLING OF BLOOD CLOT

MODELLING OF blood clot using ANSYS Design Modeler software using the commands of sketcher and revolve



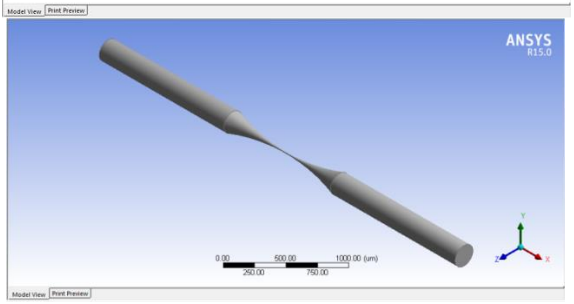
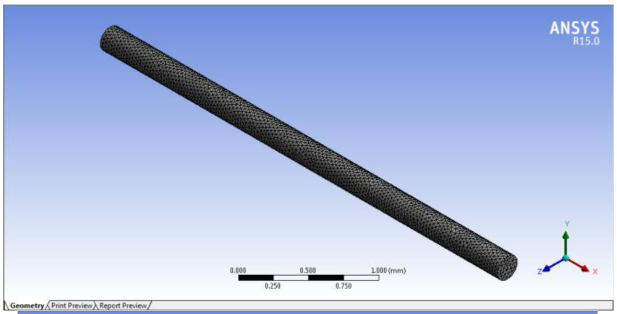


Fig 1 3D Model Of Veins Without Block

Fig 2 3D Model Of Veins With Full Surface Block

2,MESH GENERATION



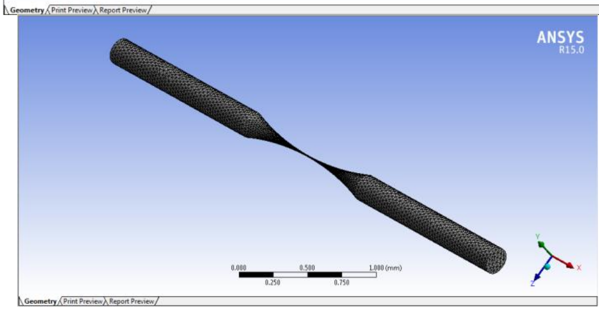


Fig 3 Mesh Model Of Veins Without Block

Fig 4 Mesh Model Of Veins With Full Block

Material Property of Blood
Density :1050 kg/m^3

Specific Heat Capacity: 3490 kj kg^{-1} ^{0}C^{-1}

Thermal Conductivity: 0.56

Viscosity: 0.00365

Boundary Conditions:

Pressure Based Solver

Turbulence Model :Standard K-epsilon

Fluid Material : Blood

Inlet Boundary conditions: 0.04 m/s of blood velocity

Solver Equation : SIMPLE (Semi Implicit Pressure Linked Equations)

3 SOLVER

3.1 The Fluent Code

Fluent is a computational fluid dynamics computer code developed and marketed by Fluent Inc. The code solves the equations for conservation of mass, momentum, energy and other relevant fluid variables using a cell-centred finite-volume method. First the fluid domain is divided into a large number of discrete control volumes (also known as cells) using a pre-processor code which creates a computational mesh on which the equations can be solved. Once the fluid domain has been meshed, the governing equations (in integral form) are applied to each discrete control volume and used to construct a set of non-linear algebraic equations for the discrete dependent variables. Fluent then offers the user a number of choices for the algorithm used to solve these equations, including coupled explicit, coupled implicit, and segregated solvers. In all the calculations reported here only the segregated solver has been used. In this approach the governing equations are solved sequentially. Since these equations are non-linear they are first linearized using an implicit method. This produces a scalar system of equations containing only one equation per computational cell per degree of freedom. A point implicit (Gauss-Siedel) linear equation solver is then used in conjunction with an algebraic multigrain (AMG) method to solve the resultant scalar system of equations for the dependent variable in each cell. Since the equations are non-linear several iterations of the solution loop must be performed before a converged solution is obtained.

3.2 Pressure-Velocity Coupling

Using this approach, an equation for each component of the momentum equation and then the continuity equation are solved sequentially. Once the three components of velocity have been calculated for each cell using this sequential system the velocities may not satisfy the continuity equation and so a Poisson-type equation for a pressure correction is derived from the continuity equation and the linearized momentum equations. This pressure correction equation is then solved to obtain the necessary corrections to the pressure and velocity fields such that continuity is satisfied. Although the pressure variable appears in each of the component momentum equations each of these equations is solved by treating the relevant component of velocity as the unknown variable and the pressure field is taken to be that from the previous iteration. In this sequential procedure the continuity equation is used as an equation for the pressure. However pressure does not appear explicitly in the continuity equation for incompressible flows (which are the only flows considered in this report) and so a procedure must be devised to introduce pressure into this equation. Fluent provides methods based on the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) family of algorithms to do this. The basic SIMPLE algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field.

3.3 Volume of fluid method

In computational fluid dynamics, the volume of fluid (VOF) method is a free-surface modelling technique, a numerical technique for tracking and locating the free surface (or fluid-fluid interface). It belongs to the class of Eulerian methods which are characterized by a mesh that is either stationary or is moving in a certain prescribed manner to accommodate the evolving shape of the interface. As such, VOF is an advection scheme a numerical recipe that allows the programmer to track the shape and position of the interface, but it is not a standalone flow solving algorithm. The NAVIER–Stokes equations describing the motion of the flow have to be solved separately. The same applies for all other advection algorithms.

3.4 K-epsilon turbulence model

K-epsilon (k-ε) turbulence model is the most common model used in Computational Fluid Dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. It is a two equation model which gives a general description of turbulence by means of two transport equations (PDEs). The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows. The first transported variable determines the energy in the turbulence and is called turbulent kinetic energy (k). The second transported variable is the turbulent dissipation (ε) which determines the rate of dissipation of the turbulent kinetic energy.

Realizable k–ε (RKE) Dissipation rate (ε) equation is derived from the mean-square vortices fluctuation, which is fundamentally different from the SKE. Several reliability conditions are enforced for Reynolds stresses. Accurately predicts the spreading rate of both planar and round jets Also likely to provide superior performance compared with the standard k-epsilon model for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation.

3.6 SOLVER SETTINGS

FLUENT Solver can be used for this project the flow chart give the brief explanation to use this solver

3.6.1 PRESSURE-BASED SOLVER

The pressure-based solver is applicable for a wide range of flow regimes from low speed incompressible flow to high-speed compressible flow. Requires less memory (storage).Allows flexibility in the solution procedure. Allows flexibility in the solution procedure. The pressure-based coupled solver (PBCS) is applicable for most single phase flows, and yields superior performance to the standard pressure-based solver. Not available for multiphase (EULERIAN), periodic mass-flow and NITA cases. Requires 1.5–2 times more memory than the segregated solver. The density-based coupled solver (DBCS) is applicable when there is a strong the density based coupled solver (DBCS) is applicable when there is a strong coupling, or interdependence, between density, energy, momentum, and/or species.

3.7 Standard Initialization

The solver works in an iterative manner. Therefore before the very first iteration, a value must exist for every quantity in every grid cell. Setting this value is called Initialization The more realistic the value, using the patch method the volume fraction of the water is defined in the initialization

3.8 Convergence

The solver should be given sufficient iterations such that the problem is converged at convergence, the following should be satisfied the solution no longer changes with subsequent iterations. Overall mass, momentum, energy, and scalar balances are achieved. All equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance Monitoring convergence using residual history generally; a decrease in residuals by three orders of magnitude indicates at least qualitative convergence.

At this point, the major flow features should be established. Scaled energy residual should decrease to 10-6 (for the pressure-based solver). Scaled species residual may need to decrease to 10-5 to achieve species balance. Tightening the Convergence Tolerance If solution monitors indicate that the solution is converged, but the solution is still changing or has a large mass/heat imbalance, this clearly indicates the solution is not yet converged. In this case, you need to reduce values of Convergence Criterion or disable Check Convergence in the Residual Monitors panel. Continue iterations until the solution converges.

IV RESULTS

1. WALL SHEAR STRESS

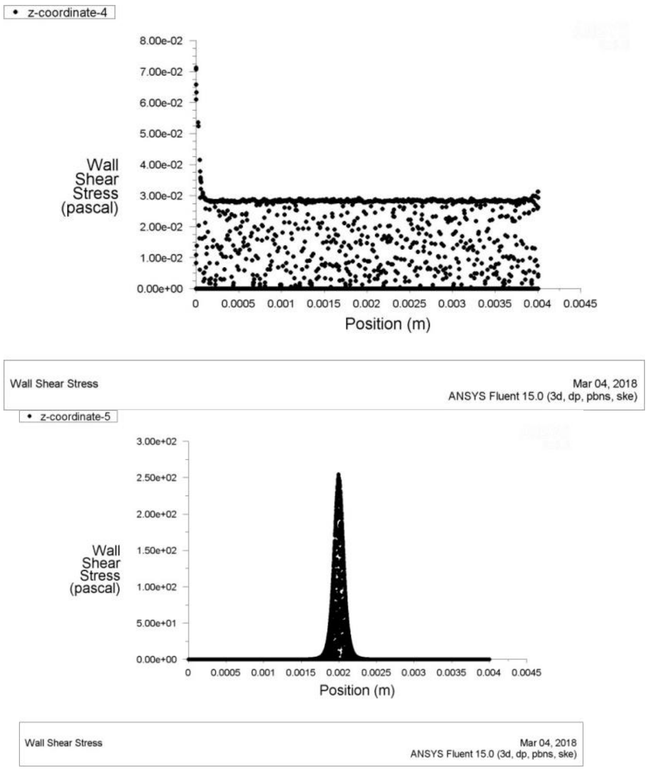
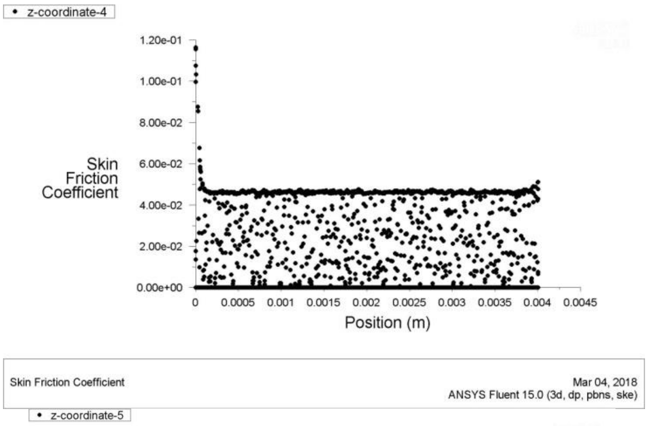


Fig5wall Shear Stress Without Blockage

2 . SKIN FRICTION COEFFICIENT



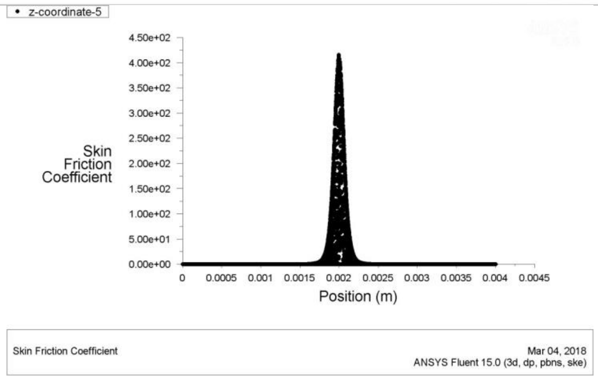
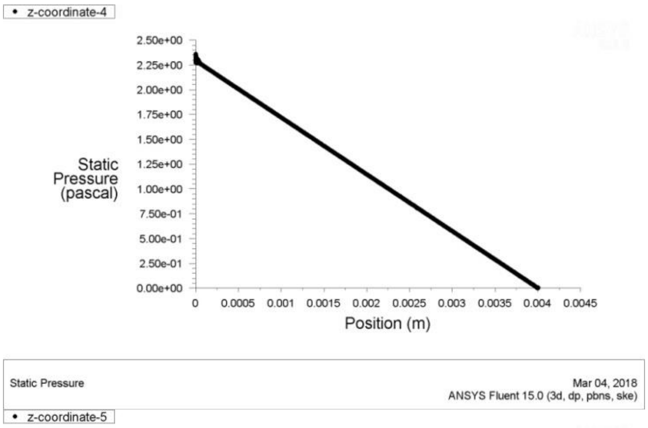


Fig 7 WITHOUT BLOCKAGE



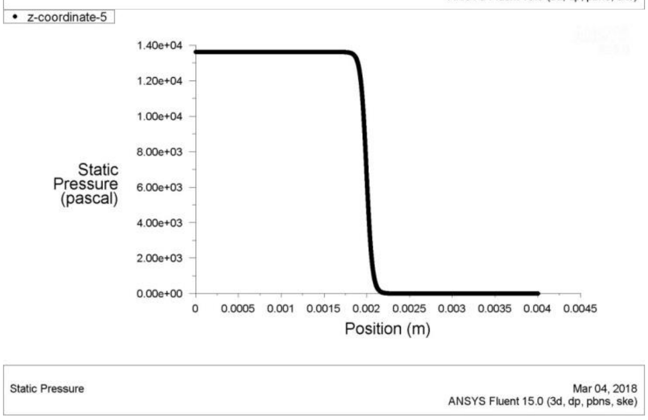
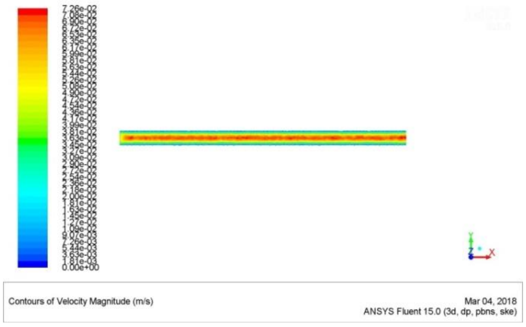


Fig 9 PRESSURE DROP FOR NO BLOCKAGE

4 VELOCITY PROFILE



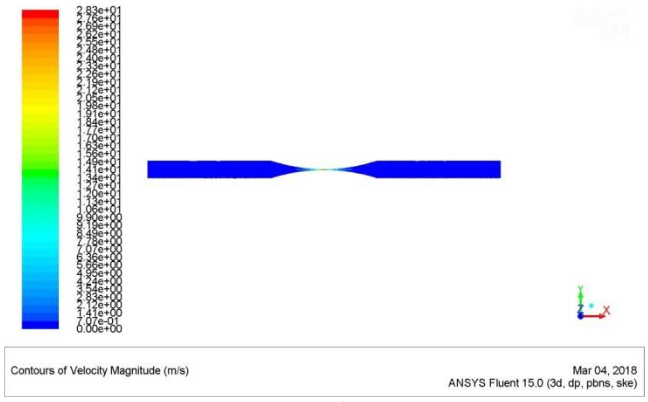
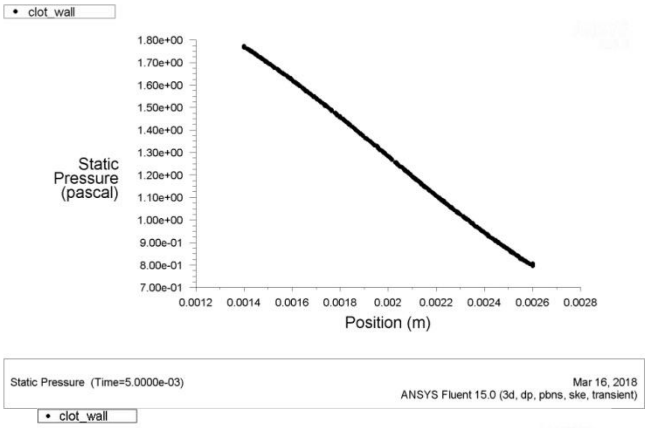


Fig 11 Velocity Profile For Without

5 OSCILLATING PRESSURE



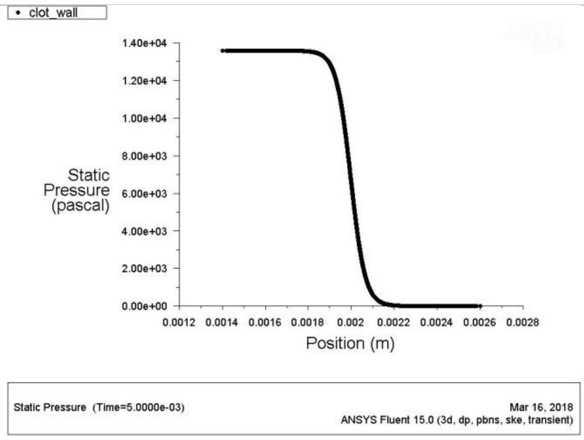


Fig 13 Oscillating Pressure For No Block
Fig 14 Oscillating Pressure For Full Block

V CONCLUSION

Blood hammer analysis was done in the CFD methodology Wall Shear Stress, Pressure drop results, Skin friction coefficient results and velocity profile results are studied in the CFD Analysis with various micrometer thickness of the blood clot was studied Wall shear stress value is linearly increased with the increase in thickness of the blood clot, Skin friction coefficient value also linearly increased with the increase in thickness of the blood clot, Pressure drop value also linearly increased with the increase in thickness of the blood clot, Velocity profile is linearly increased with the increase in thickness of the blood clot, but the velocity profile is getting very high in 50 micrometer height of the clot, so from 50 micrometer height clot and above height of the clot will affect the vein The comparison of the results are check with clot and without clot.

REFERENCE

[1] 1.Pressure and wall shear stress in blood hammer-Analytical theory - Chiang c.Mei , Haixiao Jing

[2] 2.Hemodynamic Analysis in an Idealized Artery Tree: Differences in Wall Shear Stress between Newtonian and Non Newtonian Blood Models- Jared C. Weddell , Jaehyuk Kwack , P.I. Imoukhuede, Arif Masud

[3] Assessment of Wall Shear stress in the Common Carotid Artery of Healthy Subjects Using 3.0-Tesla Magnectic Resonance - B. Sui, P. Gao, Y.Lin, B.Gao, L.Liu , J. AN Vascular Wall Shear Stress:Basic Principles and Methgods - Theodoros G, Papaioannou, Christodoulos Stefanadis

[4] .Models and Finite Element Techniques for Blood Flow Simulation - M.Behr, D.Arora, O.Coronodo and M. Pasquali

[5] CFD Simulation of Blood Clot Behavior Using GP Device Kuzilati Kushaari, Computer Modeling and Simulation (2012)

[6] Analysis of the “GPATD”: Geometrical Influence on Blood Clot Extraction using

[7] CFD Simulation Gregorio Romero, Computer Modelling and Simulation (2014)

[8] "The Use of Computational Fluid Dynamics in the Development of Ventricular Assist Devices" Fraser KH1, Taskin ME, Griffith BP, Wu ZJ, 'American society of artificial internal 2011)'

[9] "Computational Fluid Dynamic Analysis of Blood Flow Pattern - A Review" Bharath Ganesan, Karthikeyan Mayakrishnan, International Journal of Science and Research(2012)

[10] "Computational Fluid Dynamics Analysis of Thrombosis Potential In Left Ventricular Assist Device Drainage Cannulae" Fraser KH1, Zhang T, Taskin ME, Griffith BP, Wu ZJ, 'American society of artificial internal organs (2010)'

[11] "COMPUTATIONAL FLUID DYNAMICS SIMULATION OF EARLY DIAGNOSIS OF DEEP VEIN THROMBOSIS" Nur Shazilah bt Aziz, Nabilah bt Ibrahim 'ARPN Journal of Engineering and Applied Sciences'

[12] "Computational Study of Thrombus Formation and Clotting Factor Effects under Venous Flow Conditions" Govindarajan V1, Rakesh V1, Reifman J2, Mitrophanov AY1, 'Bio physical journal(2016)'

[13] "Blood Clot Dissolution Dynamics Simulation during Thrombolytic Therapy" Sersa I, Tratar G, Blinc A. 'Journal of chemical engineering and modeling'

[14] The Croonian Lecture. On the functions of the heart and arteries. -Thomas Young