Non-Newtonian Computational Fluid Dynamics (CFD) Modeling on Blood Clot Extraction

By

KATHIRAVEN MUNIANDY

Dissertation submitted in partial fulfillment of the requirements for the
Bachelor of Engineering (Hons)
(Chemical Engineering)

MAY 2013

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Non-Newtonian Computational Fluid Dynamics (CFD) Modeling on Blood Clot Extraction

by

KATHIRAVEN MUNIANDY

A project dissertation submitted to the
Chemical Engineering Programme
Universiti Teknologi PETRONAS
In partial fulfillment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(CHEMICAL ENGINEERING)

Approved by,

____

(AP Dr. Ku Zilati Ku Shaari)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
AUG 2013

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgments, and that the original work contained herein have not been undertaken nor done unspecified sources or persons.

Produced by,

____

KATHIRAVEN MUNIANDY

ABSTRACT

Several diseases such as atherosclerosis, strokes and heart attacks cases are alarming. One of the major contributors for these diseases is formation of thrombus in blood artery. Medical advancement has seen various efforts to overcome the cause, one of the method used is by removing the blood clot with GP device whereby the clot is removed without damaging the arterial wall. Since it requires the knowledge of the fluid behavior at certain parameters (viscosity), which is certainly difficult to determine in real time. Hence, computational fluid dynamics (CFD) comes in handy to solve the problem. A simple, but powerful method is the Volume of Fluid (VOF), is a sub of computational fluid dynamics. This VOF model functions as tracking and locating the free surface which belongs to the class of Eulerian. It is a two-phase problem consisting of blood and blood clot.

Blood is often assumed to be Newtonian fluid for modeling purposes. In reality, blood viscosity varies upon certain factors and individuals, which means it exhibits non-Newtonian fluid properties. Since blood exhibits non-Newtonian behavior, Power Law is used to define the viscosity of the blood flow and blood clot. This leads to a blood clotting modeling by using the Ansys Fluent. Several equations are used to set the boundary conditions and parameters such as Navier-Stokes equation. During the grid size selection, the finer grids are used to result in higher accuracy calculation which is at 0.1mm grid size.

Next, the GP device model will be analyzed by comparing the Newtonian fluid and non-Newtonian fluid at pressures of 40kPa, 50kPa and 60kPa. Theoretically, blood is a shear thinning fluid which means its viscosity decreases as the shear stress increases which mean it will result in faster blood clot extraction time. Based on the results obtained, it is parallel to the theoretical study whereby the non-Newtonian blood clot extraction time decreases as the pressure of suction increases. In addition to that, the clot extraction time for non-Newtonian model is relatively faster compared to Newtonian models at 40 kPa, 50 kPa and 60 kPa.

ACKNOWLEDGEMENT

First and foremost, I would like to express my deepest gratitude to my supervisor, AP Dr. Ku Zilati Ku Shaari for guiding, supporting and motivating me throughout the period of completion of this project. Moreover, my supervisor has really taught me to steadfastly stand on my own feet with the project given which in a way I have become more certain in making decisions and learning new subject.

Next, I would like to thank Mr Javed Khan for his patience and invaluable input to help me to solve the Ansys FLUENT and extracting results during the solving and post processing stages.

Besides that, a special thanks to the Final Year Project Coordinators, Dr Anis Suhaila bt Shuib (FYP I) and Dr Norhayati Mellon (FYP II). Thank you to both of you for coordinating some series of lectures, talks, trainings and guidance which acts as a catalyst for the completion of this project.

Without the assistance and guidance from these individuals, it would be really troublesome and challenging task for me to complete this project successfully. Hence, I thank all the staffs of Chemical Engineering Department who has supported me intentionally and unintentionally to complete my project.

Last but not least, I thank my friends and family whom lift me up and keep the spirit up and burning to complete this project.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

ABBREVIATIONS AND NOMENCLATURES

Symbols

μ	Viscosity
τ	Yield Stress
n	Measure of Deviation of Fluid from Newtonian
γ	Shear Rate
T	Temperature
ρ	Density

Abbreviations

CVA	Cerebrovascular Accident
CFD	Computational Fluid Dynamics
VOF	Volume of Fluid
GP	Gwen Pearce
WSS	Wall Shear Stress
RBC	Red Blood Cell
MTD	Mechanical Thrombectomy Device

CHAPTER 1

INTRODUCTION

1.1 Project Background

Stroke or cerebrovascular accident (CVA) can be defined as the rapid loss of brain function due to disturbance in the blood supply to the brain. This is caused mainly by ischemia which means lack of blood flow and another cause could be by blockages in the flow for instance, thrombosis, arterial embolism and hemorrhage. In thrombotic stroke, a thrombus usually forms around atherosclerotic plaques. A thrombus itself can lead to an embolic stroke, and if it breaks off, it is called as embolus. There are two types of thrombosis which causes stroke which are large vessels disease involves carotids, vertebral and one more cause is small vessel disease in smaller arteries in brain. Statistics shows that only in United States of America there are about 60 000 to 100 000 people dies due to thrombosis. This is a severe complication that occurs to almost anyone.

A thrombus is a blood clot phenomena anchored to damaged vascular wall. Thrombus forms in the blood vessel because of the occurrence of various hemodynamic and biomechanical processes. These processes can be divided into two main sub-processes namely platelet aggregation/activation and coagulation (fibrin gel formation). The main cause of thrombosis occurrence is due to quality of vessel, composition of the blood and the nature of the blood flow (Salvatore Cito, 2013).

A device namely Gwen Pearce (GP) device may overcome these problems associated with the thrombosis. The primary function of this device is assisting in the blood clot removal in the artery. It acts as the best device in this medical application compared to other similarly functioning devices such as 4F catheter and Solitaire. One of the added benefits of using the GP Device is it reduces the risk of device fracture; it does not disrupt the clot during the clot retrieval and also reducing the risk of loose fragments. Besides that, it has no moving parts; it relies on the suction physics of vortex creation which is mostly 30 kPa and above and perhaps permitting less downstream embolization (Christopher Tennuci, 2011).

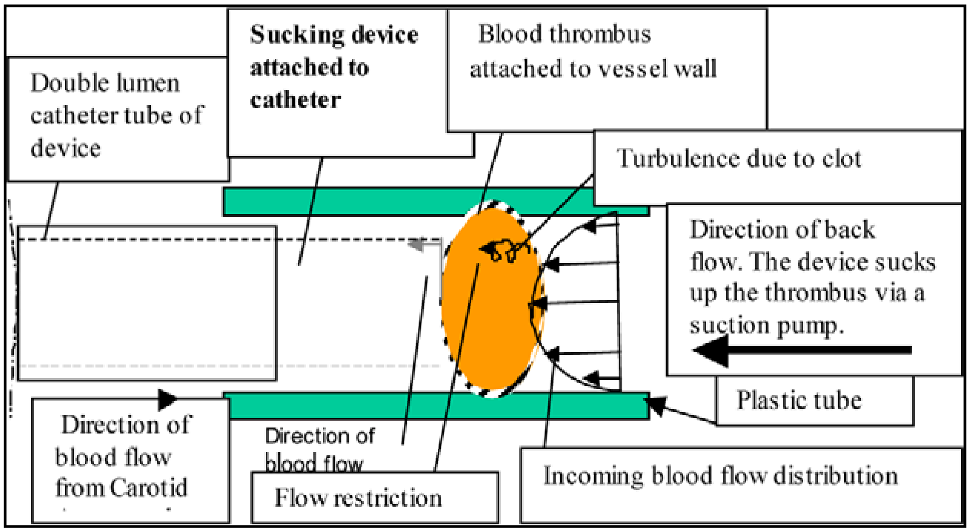


Figure 1: Clot removal device (G. Pearce, 2007)

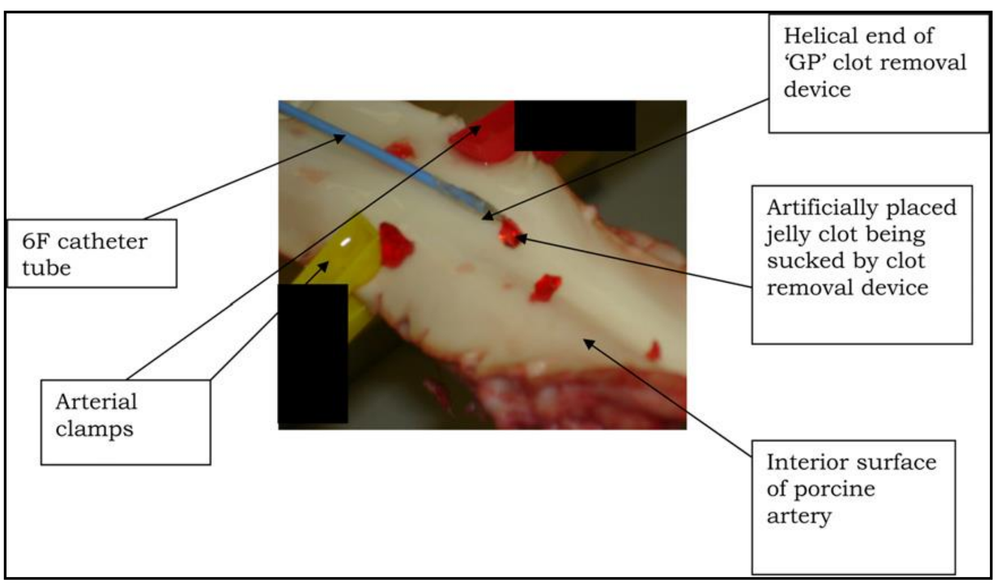


Figure 2: Capture of artificial clot by Gwen Pearce (GP) clot removal device (G. Pearce, 2007)

CFD is a computer based tool for simulating the performance of systems involving fluid flow and heat transfer mainly. It works by solving the equations of fluid flow over a region of interest, with known conditions on the boundary of the region. Since mid-1970's, complex mathematics vital to generalize algorithms began to be understood, and so general purpose of CFD solvers were developed. Currently, CFD is a reputable industrial design tool, with reduced design time scales and improve processes throughout the engineering world. In addition to that, CFD also provides a cost effective and accurate alternative to scale model testing. Some of the distinctive applications of CFD used by engineers and scientists around the world are process industry, building services, health and safety, motor industry, electronics, environmental, energy and medical (Katharine H. Fraser, M. Ertan Taskin, Bartley P. Griffith, Zhongjun J. Wu, 2011).

.

Therefore, CFD has proven to be a sensible, competent and consistent tool for investigating blood flow cases due to the improvement of low cost and simplicity of testing different physiological circumstances. The performance of CFD models is however directly dependent on blood rheology and the local environment of flow medium such as physiological flow conditions, mechanical properties of blood vessels, interactive forces between blood plasma and blood cells. The measurement of blood properties under different hemodynamic conditions has therefore gained great importance to obtain proper rheological models for CFD analysis (Finnigan P, Hathaway A, Lorensen W, 1990).

1.2 Problem Statement

Thrombus formation is quite dangerous since its formation in arteries could lead to heart attacks or ischemic strokes if the affected arteries are coronary or the carotids. The whole blood clotting process is a series of closely related and integrated sub-processes which form from the homeostasis triad. Blood clotting process can be divided into four major stages; endothelial injury, primary and secondary homeostasis, coagulation cascade and fibrinolysis (Bedekar A, Pant K, Ventikos YP, Sundaram S., 2005).

Although a number of parameters such as pressure, lumen diameter, compliance of vessels, peripheral resistance are well known physiological parameters that affect the blood flow. On top of that, whole blood viscosity is also a significant key physiological parameter. Many researches and reviews have been made on the blood viscosity affecting the blood flow only. A significant simulation model has not been developed on blood viscosity affecting the blood clot in artery.

Rheology study on the blood clot viscosity and the blood flow viscosity will be conducted and a CFD analysis to be done. This is vital because a group of researchers reported that whole blood viscosity was significantly higher in patients with peripheral arterial disease than that in healthy controls. In addition to that, blood is often assumed as Newtonian fluid to study the behavior of the blood but actually it behaves as a non-Newtonian fluid. Therefore, a real time CFD modeling is required to understand the actual behavior of the fluid at its actual viscosity. Hence, the CFD model will provide a better image on the conditions of the blood flow and blood clot at the varying viscosities. Besides that, there is also a need to determine the time taken for the medical device to gently remove the blood clot safely, without damaging the artery wall. The CFD model obtained from this study can be useful in the diagnostics and modern medical treatment.

1.3 Objective and Scope of Study

Simulation of blood clot model in an artery on the modern medical device is a vital engineering problem. This project has a series of objectives which should lead to a successful outcome of a CFD modeling on the non-Newtonian blood clot extraction in artery. The objectives of this project are as follow:

i) To study the effect of viscosity (non-Newtonian) on the blood clot extraction.
ii) To understand the relationship between non-Newtonian blood flow and the pressure required for blood clot extraction.
iii) To compare the difference of the Newtonian and the non-Newtonian blood clot extraction.

Since such simulation model has not been developed yet, it is essential to execute it, so it can be to enhance the application of modern medical treatment. This objective will be achieved with the assistance of the Ansys FLUENT solver. This project aims to simulate the CFD modeling on the non-Newtonian blood clot extraction by using the FLUENT software. This will be a breakthrough in the medical field as it will be fundamental review to dissolute the thrombus at specified viscosity of blood. The scopes of this project are:

- Create and mesh the geometry domain using the DesignModeler.
- To perform simulation with the FLUENT solver.
- To perform grid sensitivity study to identify the most appropriate one, so it will minimize the computational time without affecting the accuracy of the results.
- To run the simulation in Newtonian conditions after validating the model.
- To run the simulation in the non-Newtonian conditions of blood and blood clot.
- To study the blood clot volume fraction movement via contour analysis.
- To compare the differences of Newtonian and non-Newtonian blood clot.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Computer-aided design and Computational Fluid Dynamics (CFD) have initiated their way into all branches of engineering and sciences. These techniques can enhance performances, decrease concept-to-prototype time, create huge savings in equipment and energy costs, as well as reducing environmental impact. CFD is used as a gizmo to eliminate several physical experimentation cycles of the design process (Sandro De Gruttola, Kevin Boomsa and Dimos Poulikakos, 2005). Due to its capability, the CFD method has been mostly used for studying the complex behavior of the blood flow. The first CFD studies used idealized geometries to calculate the blood flow characteristics and properties like wall shear stress (WSS) and residence time. Later, due to development of medical imaging techniques, more accurate and realistic geometries have been used in blood flow simulation studies (Ferraris, S. Pierucci and G. Buzzi, 2010).

2.2 Newtonian Fluid

Fluid such as air, water, ethanol and benzene are Newtonian. The shear stress is plotted against shear rate at a given temperature; the plot shows a straight line with a constant slope that is independent of shear rate. (Gruttola, 2005). This slope is called the viscosity of the fluid. The simplest constitutive equation is Newton's law of viscosity;

$$\tau = \mu\dot{\gamma}$$

where μ = Newtonian viscosity and γ = shear rate or the rate of strain.

The Newtonian fluid is the basis for classical fluid mechanics. Gases and liquids like water and mineral oils exhibit characteristics of Newtonian viscosity. However, many important fluids, such as blood, polymers and paints show non- Newtonian viscosity (Gundogdu, 2008).

2.3 Non-Newtonian Fluid

Any fluids that do not obey the Newtonian relationship between shear stress and shear rate are non-Newtonian. The topic of rheology is dedicated to the study of the behavior of such fluid. The slope of shear stress versus shear rate curve is not constant. The viscosity of a fluid decreases with increasing shear rate, this is known as shear-thinning. Vice versa to that, where the viscosity increases as the fluid is subjected to a high shear rate, it is called shear-thickening (Subramaniam, 2002). For all flows, FLUENT solves conservation equations for mass and momentum. FLUENT provides four options for modeling non-Newtonian flow which are Power Law, Carreau model for pseudo-plastics, Cross model and Herschel-Bulkley model for Bingham plastics.

2.3.1 Power Law

The non-Newtonian viscosity is calculated as FLUENT 6.0 is shown below:

$$\eta = k\dot{\gamma}^{n-1}e^{T_0/T}$$

FLUENT allows upper and lower limits to be placed on the power law function, compliant the equation below:

$$\eta_{\min} < \eta = k\dot{\gamma}^{n-1}e^{T_0/T} < \eta_{\max}$$ (Svetla Petkova, 2003)

Where k= measure of the average viscosity of the fluid, n= measure of the deviation of the fluid from Newtonian, T_0 = reference temperature and n_{min} and n_{max} are the lower and upper limits respectively. Hence, the value of n determines the classes of the fluid which is as follow:

n = 1 (Newtonian Fluid)

n > 1 shear-thickening (Dilatant fluids)

n < 1 shear-thinning (pseudo-plastics)

Unfortunately, one of the noticeable disadvantages of the power-law model is that it fails to portray the viscosity of many non-Newtonian fluids in very low and very high shear rate regions. Viscosity for many suspensions and dilute polymer solutions becomes constant at a very high shear rate. This cannot be solved via the power-law model. (Kim, 2002)

2.3.2 Cross Model

In order to overcome this drawback of the power-law model, Cross (1965) proposed a model that can be described as:

$$\tau = \dot{\gamma}\left(\eta_\infty + \frac{\eta_0 - \eta_\infty}{1 + m\dot{\gamma}^n}\right)$$

Where

n_0 and $n\infty$ = viscosities at very low and high shear rate

m and n = model constants

Cross model produces Newtonian viscosities at very low and shear rates compared to the Power Law model (Kim, 2002).

2.3.3 Bingham Plastic Model

Many types of food stuffs exhibit a yield stress and are said to show a plastic or viscoplastic behavior. Bingham plastic model is the simplest form of viscoplastic model, and it can be expressed as follow (Kim, 2002):

$$\tau = m_B \dot{\gamma} + \tau_y$$

when $\tau \geq \tau y$

$$\dot{\gamma} = 0$$

When $\tau \leq \tau y$

where

τ_y = constant as yield stress

M_B =constant plastic viscosity

Bingham plastic model can describe the viscosity characteristics of a fluid with yield stress whose viscosity is independent of shear rate. Therefore, the Bingham plastic model does not have the ability to handle the shear-thinning characteristics of non-Newtonian fluids.

2.3.4 Herschel-Bulkley Model

The Herschel-Bulkley model resembles the simple power-law model to comprise the yield stress as below (Kim, 2002):

$\tau = m\gamma^n + \tau_y$ when $\tau \geq \tau_y$,

$\gamma = 0$ when $\tau \leq \tau_y$

where *m* and *n* are model constant.

It shows both the yield stress and the shear-thinning non-Newtonian viscosity (Ferguson J and Kemblowski Z, 1991).

2.4 Rheology of Blood

Blood behaves like a non-Newtonian fluid whose viscosity varies with shear rate. The non-Newtonian characteristics of blood come from the presence of various cells in the blood (typically making up 45% of the blood's volume), which make blood a suspension of particles (Kim, 2002). When the blood begins to move, these particles interact with plasma and among themselves. Hemorheologic parameters of blood include whole blood viscosity, plasma viscosity, red blood cell aggregation, and red blood cell deformability (or rigidity).

Much research has been performed to formulate a theory that accounts completely for the viscous properties of blood, and some of the key determinants have been identified (Guyton A.C. and Hall J.E, 1996). The four main determinants of whole blood viscosity are plasma viscosity, hematocrit, red blood cell (RBC) deformability and aggregation, and temperature. Hematocrit and RBC aggregations, mainly contribute to the non-Newtonian distinctiveness of shear-thinning viscosity and yield stress (M. Karsheva, P. Dinkova, I. Pentchev, T. Ivanova, 2009).

2.5 Blood Clotting Models and MTD

In the early stages, the finite difference and the finite element method was widely used. Its advantage, it is favorable to algebraic equations for efficient numerical problem solving. Unfortunately one of the drawback of this method is it requires more computing time and memory capacity. Later, space-marching method was developed, which was suitable for many fluid flow problem. It is divided into three sections, which are parabolic technique is used to solve the Navier-Stokes equations, second phase is elliptic technique is used to integrate the problem and finally the third phase is including the neglected terms and repeat the second phase steps. Later, finite volume method is introduced in the field of fluid dynamics for further improvise the 3-D modeling (Hou-Cheng Huang, Zheng-Hua Li and Asif S.Usmani, 1998)

Thrombus formation is a consequence of an abnormal functional behavior of hemostasis. Thrombosis is largely triggered by an endothelial damage or the stasis of the blood state. Flow conditions such as abnormal wall shear stress overcome in arterial thrombosis. It can be divided into four stages which are endothelial injury, primary and secondary hemostasis, coagulation cascade and fibrinolysis. (Salvatore Cito, 2013)

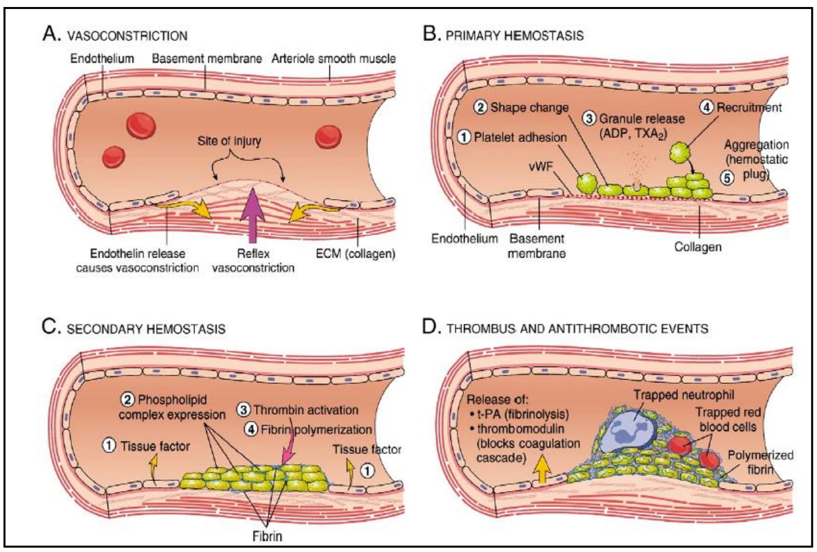


Figure 3: Blood Clotting Process of Hemostasis (Salvatore Cito, 2013)

Suspension fluid, plasma consists mainly of water and large concentration of cells in whole blood. This results in the non-Newtonian behavior of the blood. At amplified cell concentration blood exhibits shear-thinning behavior with higher overall viscosity at any shear rate. Most commonly the Power and modified Cross models are used in computational modeling of blood flow (Salvatore Cito, 2013). Navier-Stokes equation is used for 3-D theory of blood flow with certain assumptions such as incompressible, homogeneous, Newtonian fluid flow with suitable initial and boundary conditions. It is also assumed to be laminar flow as the baseline condition in the artery. The inlet surface for the flow, the outlet surface and the inner surface of the vascular wall are the basic surfaces of interest in hemodynamic (Salvatore Cito, 2013).

In order to predict the non-Newtonian behavior of blood, Newtonian models are used as references with various "characteristics viscosity" assumptions are made. Wall shear rate in non-Newtonian models are generally higher than the Newtonian models. For instance, it is 10% higher at aorta, 20% higher along the outer wall of aorta and 40% higher at other part of vessels (G.C Bourantas, 2011).

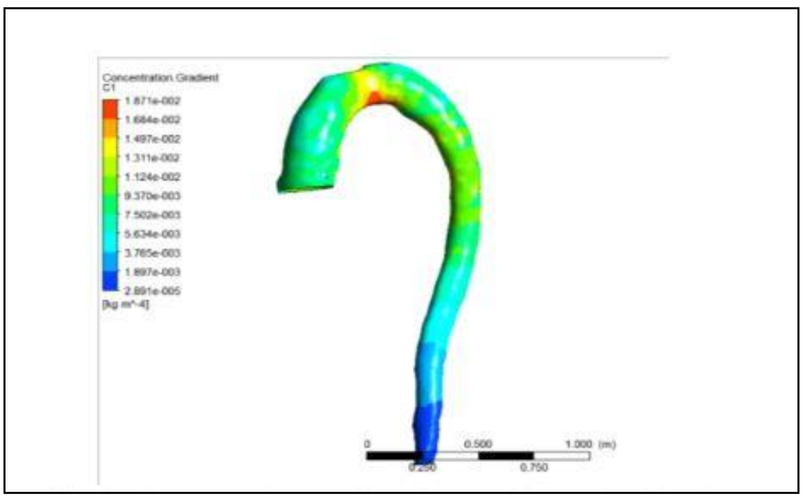


Figure 4: Concentration gradient for non-Newtonian blood (G.C Bourantas, 2011)

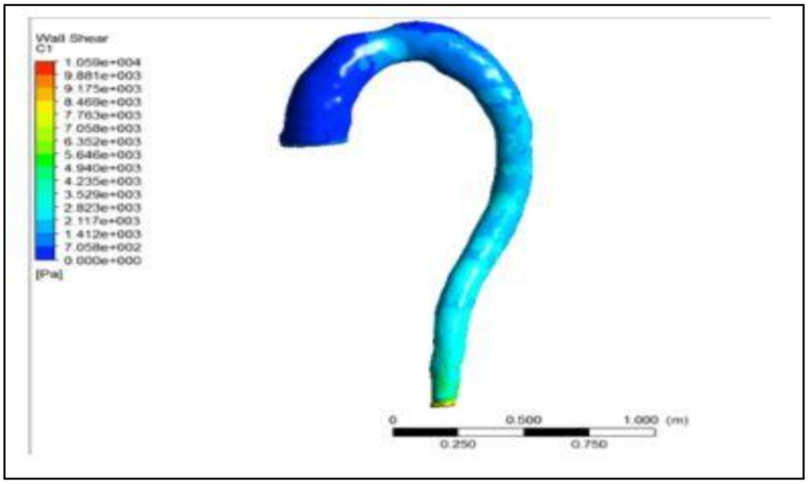


Figure 5: Wall shear stress distribution for non-Newtonian blood (G.C Bourantas, 2011)

In developing the blood clot model, certain assumptions need to be considered are blood flow is one dimensional and in axial direction, arteries are elastic cylindrical tubes and blood flow is laminar. Blood is also assumed as incompressible fluid, and the blood exhibit constant density which is $\rho = 1.06 \times 10^3$ kg/m^3 (Yaghoub Dabiri, 2005). Next, the surface tension,ɣ from the attraction forces of the molecules is defined as force by unit length. The surface tension used in the earlier studies is equivalent to 0.1N/m. This value is based on the blood clot size patched on the artery which is at about 3 to 5mm. (Nazehah, 2012)

Before moving any further into the analysis, a grid sensitivity study is conducted by assessing the effect of the grid size on pressure and the laminar properties (KuShaari, A.Rahman & Pearce, 2012). Hence, for each case the grid density is varied from coarse to fine by increasing the number of cells. The grid independency is achieved when any increase in the number of cells does not affect the accuracy of the results and also saves the computational time (KuShaari, A.Rahman & Pearce, 2012). Besides that, it is also found that the fastest time to remove blood clot is at 0.00498s, 60kPa for GP with cylindrical tubes rather than helical spiral tubes. Hence, these findings will be used as basis for the research works on the non-Newtonian modeling.

Recent advances of medical field have seen major improvement in the treatment of strokes. This includes application of the thrombolytic agent alteplase (tPA) to dissolve the clots in a circulatory system and mechanical clot removal devices. The major concern of mechanical device treatment is to prevent damage on the arterial wall. There are several devices such as MERCI clot retriever, penumbral device, angiography catheters, rheolytic catheters and GP device.

Among all these devices, the GP MTD device has been proved most successful with least problems and most advantageous. The plus point of this device are can be used in relatively smaller arteries, has no moving parts which means risk of vessel breaking can be prevented, reduces the risk of downstream embolization and permits aspiration during clot retrieval (KuShaari, A.Rahman & Pearce, 2012) Besides that, study was done to ensure the best application of this device. The result obtained from the study showed that clot removal is faster in the embedded ‘GP’ MTD’s (G. Pearce, 2007).

2.6 Ansys FLUENT

CFD modeling by using Ansys Fluent as the tool, for the non-Newtonian blood clot extraction modeling to be referred to multiphase flow regimes. These flow regimes consist of gas-liquid, liquid-liquid, solid-gas and three phase flows. The concern here is the liquid-liquid interaction. The examples that fall under this category are bubbly flow, droplet flow, slug flow, stratified flow and free surface flow. Basically, blood has the property of slug flow or stratified flow. Hence, the most suitable model to be used in solving the CFD model is the Volume of Fluid (VOF) model. It applies surface tracking method to a fixed Eulerian mesh. This model can be used for two or more immiscible fluids, learning of position of interface between fluids of interest. This technique is also known for the ability to preserve the mass of the traced fluid, when the fluid interface changes its topology. Here, the changes is easily traced, hence the interface can join or break. However, there are some convergence problem requirements to be noted in this model which are the time dependent and steady solutions. It is recommended to use small scale calculation for both time step and volume fraction, and increase it gradually (Inc, 2006).

Control volume is also known as the volume of the computational grid cell. For instance, if the n^{th} fluid's volume fraction in the cell is denoted as $\alpha_{n,}$ the following are the possible conditions:

$a_n = 0$, when the cell is empty with no traced fluid inside

$\alpha_n = 1$, when the cell is full

$0 < \alpha_n < 1$, when the cell contains the interface of fluids

The interface between the phases is tracked by using the volume fraction continuity equation. This volume fraction can be solved either via implicit or explicit time discretization. Next, in Fluent the reconstruction schemes available are Geo-Constructed and Donor Acceptor. The geometric construction uses the piecewise-line approach between the fluid interfaces, is the most recommended technique. This scheme assumes that there are linear slopes within each cell between two fluids whereas the donor-acceptor approach is used to determine the amount of fluid advected near the interface. For both these schemes, a time dependent solution must be computed (Inc, 2006).

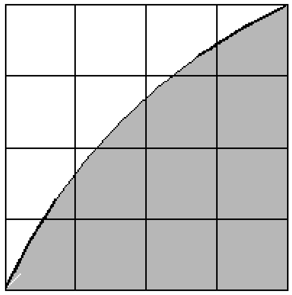


Figure 6: Actual Interface Shape (Inc, 2006)

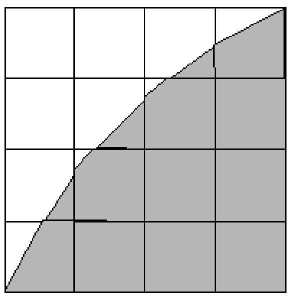


Figure 7: Geometric Reconstruction (Piecewise-Linear) Scheme (Inc, 2006)

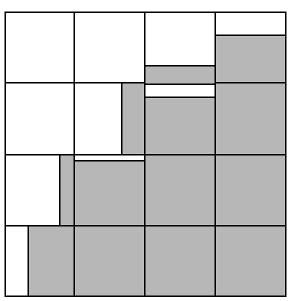


Figure 8: Donor-Acceptor Scheme (Inc, 2006)

The momentum equation that is solved in the domain will result in a velocity field, which is the description of the fluid at given space and time. This velocity will be shared among the phases. Therefore, if the velocity difference exists between the phases, the accuracy will be affected in huge scale. The equation responsible for this solution synthesis is the Navier-Stokes equation, which arises from applying Newton's second law

(Inc, 2006). The following is the general form of the equation which is used for conservation of momentum, mass and energy conservation as well;

$$\rho\left(\frac{dv}{dt} + v.\nabla v\right) = -\nabla p + \mu\nabla^2 v + f,$$

where f = other body forces,

v = velocity,

μ = viscosity,

ρ = density

This equation can further derived to compute material properties. Moreover, in a two phase system, the recommended formulation to identify parameters such as viscosity and density in each cell is as follow (Inc, 2006);

$$\rho = \alpha_2\rho_2 + (1 - \alpha_2)\rho_1$$

Similar to momentum equation, the energy equation also shared among the phases. The following is the equation used to compute the energy of the phase;

$$\frac{\partial}{\partial t}(\rho E) + \nabla.\left(\vec{v}(\rho E + p)\right) = \nabla.\left(k_{eff}\nabla T\right) + S_h$$

Other equally crucial parameters to be calculated are the scalar equation, surface tension, wall adhesion, and open channel flow (Inc, 2006).

CHAPTER 3

METHODOLOGY

This chapter simply elaborates on detail the methodology used to achieve the desired results.

3.1 Research Methodology

The methodology used to carry out this research is firstly by identifying the problem required to be solved and the objectives to be achieved. Hence, further research done on the literature review of this topic in terms of qualitative and quantitative analysis. This is a crucial step before moving into the project deeper so a solid understanding will be obtained from various scholars and sources. This project deals with the 3D CFD modeling. The solver that will be used throughout the study is Ansys FLUENT. This is a tool used to analysis the fluid flow troubles and the general well known term used for this branch of science is known as Computational Fluid Dynamics (CFD). The following is the diagram of the blood vessel:

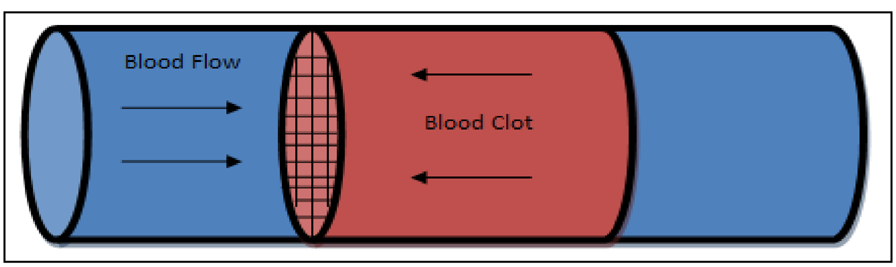


Figure 9: Blood artery (Blood flow and blood clot resistance)

Basically, DesignModeler is used during the **pre-processing** procedure. Pre-processing is a crucial step before the start of the fluid flow problem. Some of the key operations are the *geometry creations* which specify the domain of the fluid flow problem. Next is *mesh generation* which means discretization of domain to solve governing equations at each cell and later, specify the boundary zones to apply boundary conditions for problem. Finally, the pre–processed work will be saved at the Ansys workbench for future work continuation (Inc, 2006). The following is the basic geometry of the GP device cylindrical arterial catheter.

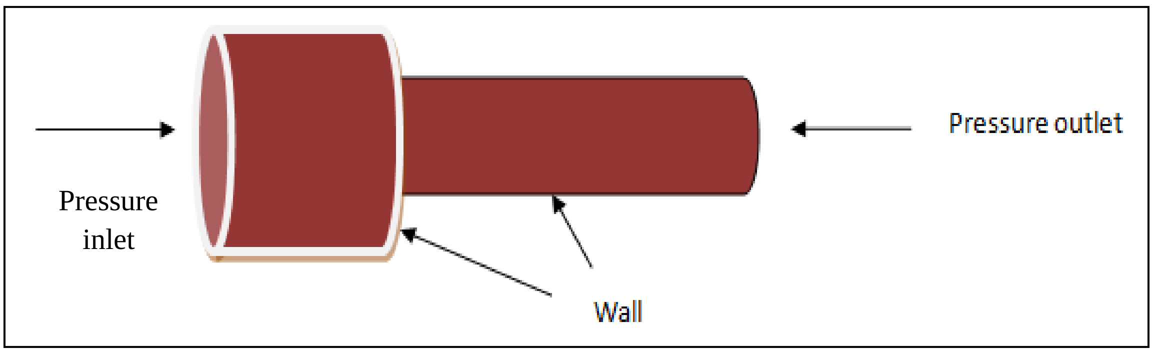


Figure 10: Basic Geometry of GP device (cylindrical)

3.1.1 Pre-Processing

There are two main steps to be completed in this section which are design geometry of the domain of interest and meshing of the geometry constructed. In this case, the domain of interest is the GP device and the blood artery. Hence, with the assistance of the DesignModeler from the Ansys Workbench, the designing of the geometry was constructed with less complication. There are several designs made with various numbers of tubes. The parameters that are used in order to setup the geometry are referred from the previous study that have been conducted and proved feasible (Nazehah, 2012). The 2D sketch of the geometry domain is as follow:

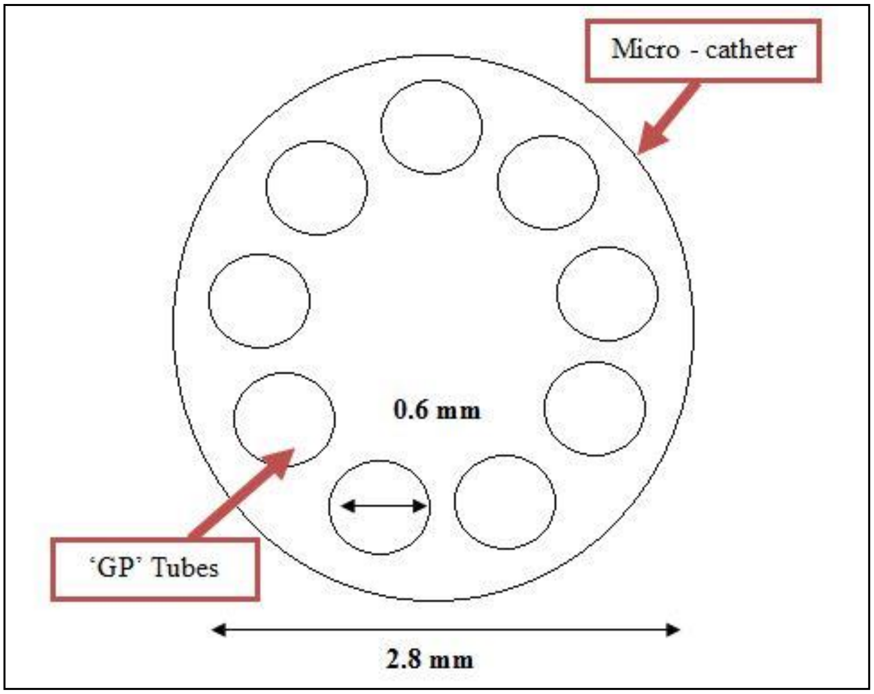


Figure 11: 2D Sketch of GP device (Front view) (Nazehah, 2012)

The parameters used to setup the geometry and the 3D geometry domain generated with DesignModeler software is as follow:

Table 1: GP Device parameters

Model A	GP Device
Number of tubes	9
Tube diameter (mm)	0.6
Horizontal length (mm)	20
Tube shape	Cylindrical

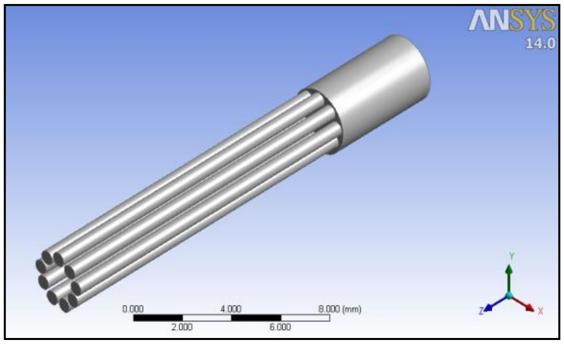


Figure 12: Isometric view of GP model

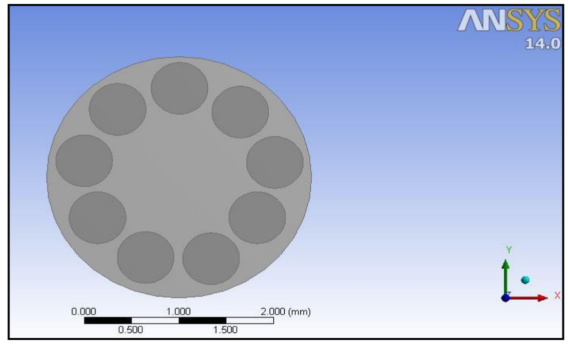


Figure 13: Front view of GP model

Then, after the 3D geometry is generated, the next step is meshing of the geometry. Meshing is a critical step prior to setting up the boundary conditions for solving purpose because the mesh influences the accuracy, convergence and speed of the solution. It is important to have an optimum number of cells so that the solver does not take long and also to prevent from obtaining inaccurate results. Meshing simply means discretize the boundary in such a way that well shaped elements can be created. It simply helps breaking down a physical domain into small discrete volumes where calculations of sets of equations can be done.

The meshing sizes are referred based on the mesh size range study done by (KuShaari, A.Rahman & Pearce, 2012). Hence the meshing size adapted ranges with minimum size of 0.1mm and maximum size of 0.4mm. The quality of the meshing determines the accuracy of later in the simulation. Hence it is very important the quality of meshing is prioritized. One of the aspects that determine the meshing quality is the mesh skewness. It is best to keep the value lower than 0.8 based on the standard practices. The following is the diagram of geometry that has undergone meshing procedure with its details of meshing:

Table 2: Meshing details

Meshing details of geometry	
Relevance center	**Fine**
Minimum size (m)	**0.0001**
Maximum size (m)	**0.0004**
Number of element	**395 088**
Skewness range	**0.00004 - 0.79**

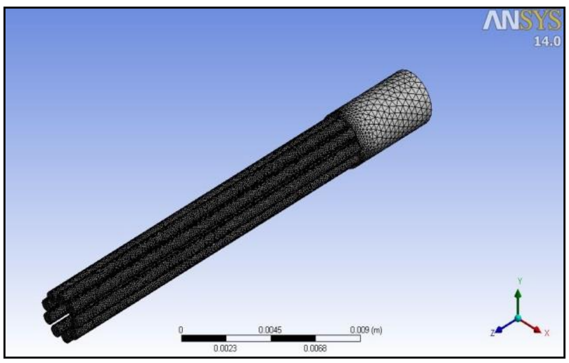


Figure 14: Isometric view of fine meshing GP model

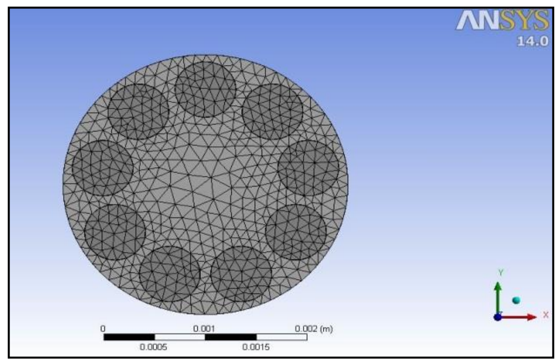


Figure 15: Front view of fine meshing GP model

3.1.2 Solving

In addition to that, FLUENT solver is used to solve and iterates the geometry constructed earlier. In order to solve the problem, FLUENT require certain data to setup the solution for the problem. The required data are solution method or model, material properties, boundary and operating conditions for given problem, initial conditions and number of iterations needed for solution process. Firstly in the solving step it is important to validate the model obtained with a previous model. In this case, it should be compared with a Newtonian model. Next, a grid sensitivity study should be followed to ensure the sensitivity chosen is the optimum one. Finally, parameters are tested, for this study it is the viscosity. Once the solution is performed, the next step is post-processing (Inc, 2006)

The modes of solving steps are assimilated from the earlier studies. The solver type used for this simulation is the pressure based, with the transient time setup. This is chosen as we need to analyze the blood clot extraction over time. As mentioned earlier, the Volume of Fluid (VOF) is chosen as the model as it involves 2 phase problem consisting of blood flow and blood clot. VOF model visualizes the blood clot suction clearly. Hence, the number of Eulerian phases is set two which are blood-primary; blood clot-secondary. The surface tension between both the phases is set at 0.1N/m at constant value; this is referred based on the earlier study on the similar subject. Since it is essential to validate the model to an existing model, the material properties and parameters are set to be similar. Hence, the next steps are to setup the material properties and boundary condition. The following are the details of the parameters.

Table 3: Material properties (Newtonian) (Nazehah, 2012)

Parameter/Phase	Blood	Blood Clot
Density (g/cm^3)	1.06	1.08
Viscosity (poise)	0.035	0.35

Table 4: Material properties (non-Newtonian) (Svetla Petkova, 2003)

Parameter/Phase	Blood	Blood Clot
Density (g/cm^3)	1.06	1.08
Power law index, n	0.4851	0.4851
Consistency index, k	0.2073	0.2073
Minimum viscosity limit, η_{max}	0.0125	0.125
Minimum viscosity limit, η_{min}	0.035	0.35

The boundary conditions are also set based on the literature review (KuShaari, A.Rahman & Pearce, 2012).

Table 5: Boundary conditions

Boundary Condition	Value (kPa)
Mixture Pressure Inlet	40, 50 & 60
Mixture Pressure Outlet	-40, -50 & -60

The solution method used is PISO and the Non-Iterative-Time-Advancement is chosen to increase the calculation speed. Next the blood clot adapted is to be in cylindrical shape with the radius of 0.00135m and 0.004m. This blood clot is patched at the end of artery. Next, the suitable time step size was identified to perform the calculation without diverging. Hence, the calculation was repeated with by varying the boundary conditions and repeated with both Newtonian model and non-Newtonian model. Once these simulations are completed, the results obtained are analyzed in the post-processing step.

3.1.3 Post-Processing

Post-processing is the final step upon completion of the study. Here, the blood clot deformation in the blood artery will be observed subjected to the viscosity of the blood since it is a non-Newtonian modeling. Contours are used to analyze the results obtained whereby the blood clot volume fraction is displayed at selected flow time. Besides that, a graph of blood clot volume fraction against the flow time in the artery is also plotted to elaborate more on the comparison of the Newtonian and non-Newtonian model quantitatively.

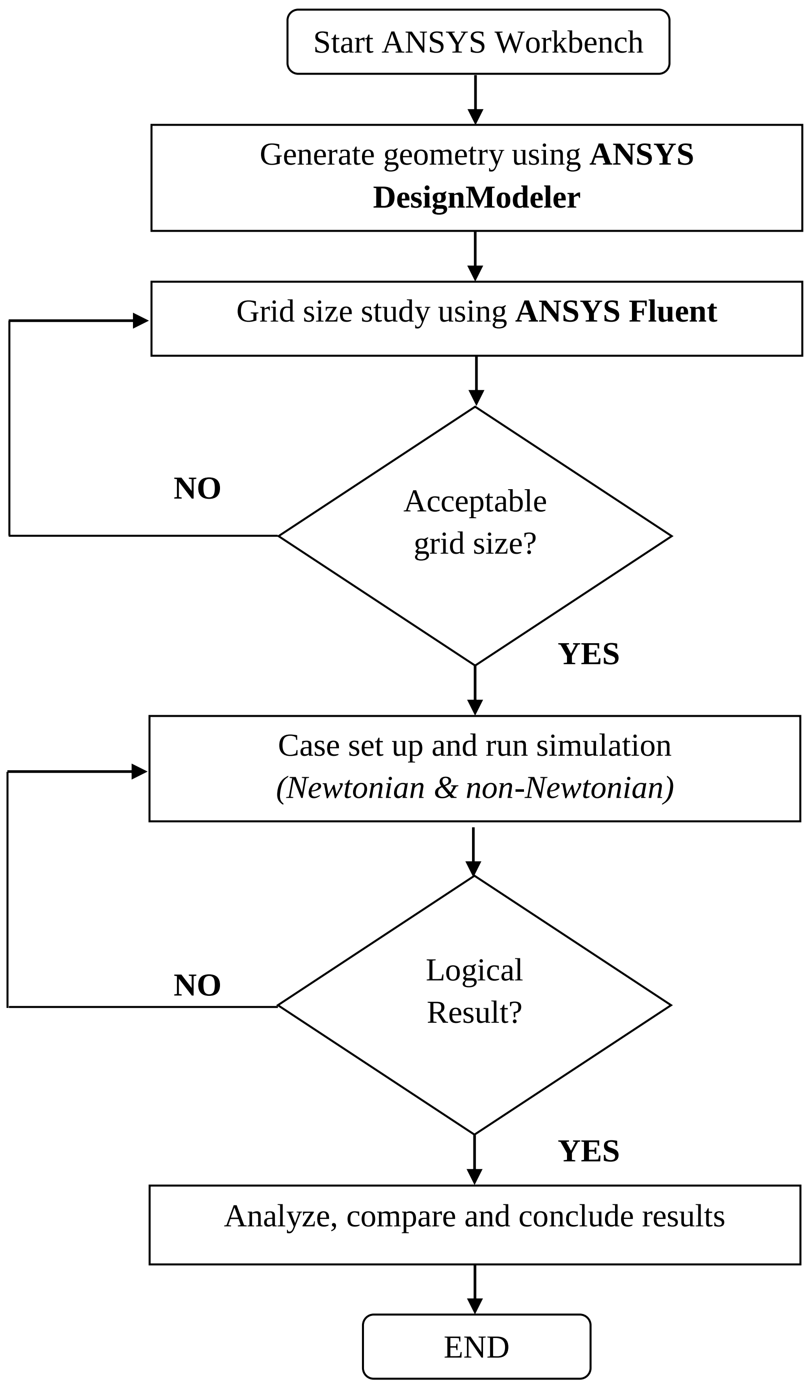
3.2 Tools/Software

The tools/software used to conduct this research is listed below:

- Ansys DesignModeler ™
- Ansys Fluent ™ 14.0

For detailed view of the Gantt chart during the commencement of this project, please refer to the Appendices.

3.3 Project Activities



3.4 Key Milestones

Some of the key events and activities that will take place during the Final Year Project I and II are as follow:

1. Finalizing the project title from project supervisor.
2. Thorough background study on the topic given via online articles, journals and books.
3. Identification of the topic problem statement and objectives of the project.
4. Plan the methodology to be used in this project and the flow of the project.
5. Completion and submission of Extended Proposal to project supervisor.
6. Proposal defense with project supervisor and panel of examiners.
7. Project work continues on FYP I. Creating domain and meshing of the domain by using Ansys DESIGN MODELER and FLUENT.
8. Modeling of the domain by using 2D and 3D models.
9. Completion and submission of FYP I Interim Report to be reviewed by project supervisor and panel of examiners.
10. Validation of the model to be reviewed by supervisor.
11. Proceed to solving steps and perform grid sensitivity study.
12. Project work continues on FYP II. Submission of Progress Report.
13. Parametric study on the modeling of non-Newtonian blood clot which are the viscosity, pressure inlet/outlet and time taken to blood clot extraction.
14. Result and data analysis made on the research. To be reviewed by supervisor.
15. Pre-SEDEX preparation.
16. Submission of draft report.
17. Submission of technical paper.
18. Oral presentation and submission of project dissertation (hard bound)
19. Completion of FYP II.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Grid Sensitivity Study

The function of grid sensitivity study is to identify the best meshing size by saving the computation time without neglecting the accuracy of the results. Basically, meshing is done to break down single unit geometry into small discrete control volumes where the set of equations are applied on each cell and computed. The accuracy of the iterations is fully dependent on the size of the cells. For instance, the smaller the size of the cells, the higher the accuracy of the results but significantly leads to a higher computational time. Hence, a grid size study is conducted over a range of grid size to compare and obtain the best accuracy and minimum computational time. The grid density is varied from coarse to fine by increasing the number of cells for the geometry. Based on earlier studies, it is mentioned that grid independency is achieved when any further increase in the number of cells does not affect the accuracy of the simulation results. In addition to that, an optimum grid size avoids any unnecessary prolonged computational time (KuShaari, A.Rahman & Pearce, 2012).

In order to conduct the study, earlier during the problem setup (solving) procedure, the surface monitor at pressure outlet for the velocity is selected. Hence, a graph can be constructed whereby the velocity at different flow time for range of grids.

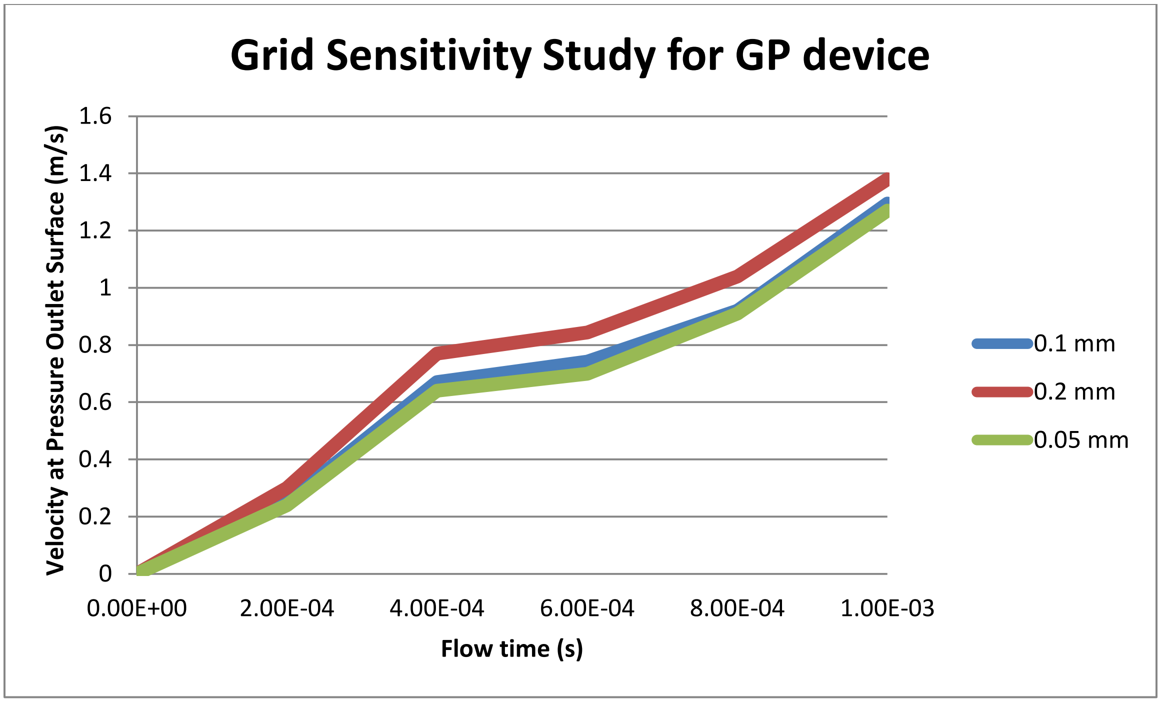


Figure 16: Graph of velocity against flow time at each grid size

The graph of velocity against the flow time is plotted for several grids and based on the result obtained the best grid is chosen. From the pattern above, 0.2 mm and 0.1 mm are almost similar. Commonly, the larger grids are always chosen at situations as it is. Hence, the 0.2 mm is preferable. However, in order to obtain a better blood clot shape upon patching, 0.1 mm grid is more preferable whereas 0.05 mm is also suitable but it will allow unnecessary computational time which should be avoided. Therefore, to fit the purpose, 0.1 mm grid size is used in this study to perform the simulation study.

4.2 Validation of Model

Once the grid size is selected, the next step is to validate the geometry constructed by matching all the parameters similarly. In this case, the Newtonian model is validated with the study conducted earlier. In order to validate the model, parameters such as the material properties, boundary condition and other basic parameters are maintained. Hence, the result obtained from the geometry constructed in this study is compared with the results from the earlier study. The result referred to the blood clot extraction time from the artery region. It is important to take note that there will be slight deviation in the result because only the GP model is maintained the same, not the size of the artery. The summary of the comparison of the result is shown in the table below.

Table 6: Model Validation

Pressure/Model	Blood Clot Extraction Time (Model A)	Blood Clot Extraction Time (Model B)
40 kPa	7 µs	8 µs
50 kPa	6 µs	7 µs
60 kPa	5 µs	6 µs

Model A is the model from the earlier published study

Model B is the model from this research

Based on the result obtained, there is slight deviation in the result due to the difference in the length of the artery embedded. The percentage of error in the model is shown below.

$$\text{Percentage error (\%)} = \frac{6-5}{5} x 100 = 20\%$$

Since the percentage of error is below 50%, hence the model constructed is accepted and validated to be used in this research whereby the deviation is assumed to be negligible as it is solely due to the length of the artery and not due to the GP model.

4.3 Blood Clot Extraction Study

The qualitative and quantitative analyses on the blood clot extraction by GP device at different pressure are tabulated and compared between the Newtonian and non-Newtonian models.

4.3.1 Qualitative Visualization of Blood Clot Behaviour at 40, 50 and 60kPa

In this section, the blood clot extraction is visualized at different flow times for both Newtonian and non-Newtonian model at 3 different pressures. The results obtained are tabulated in the table below:

Table 7: Blood clot behaviour visualization for both Newtonian and non-Newtonian models at 40kPa

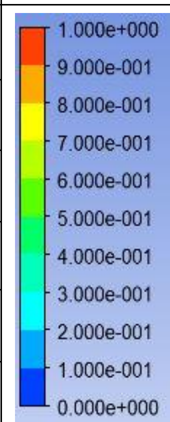
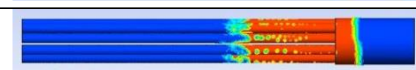
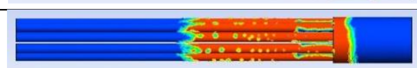
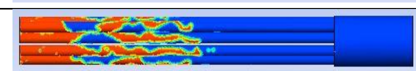
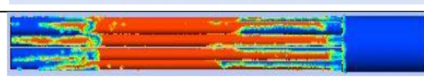
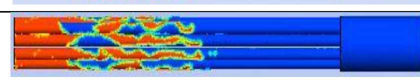
Time (s)	Pressure (40kPa)		Blood Clot Volume Fraction
	Newtonian	Non-Newtonian	
0			
0.001			
0.003			
0.005			
0.007			
0.009			

Table 8: Blood clot behaviour visualization for both Newtonian and non-Newtonian models at 50kPa

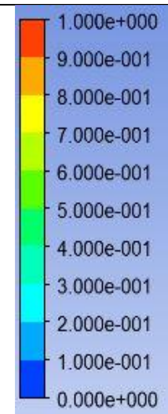
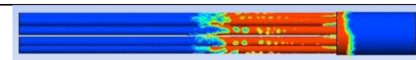
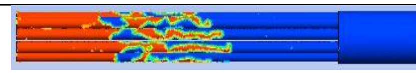
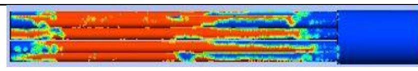
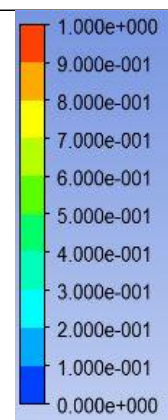
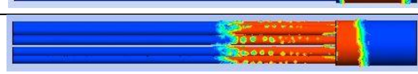
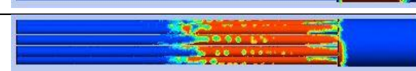
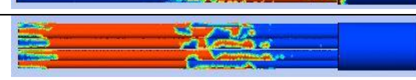
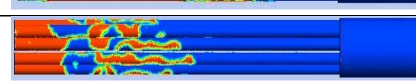
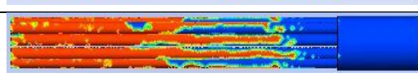
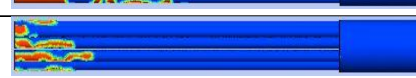
Time (s)	Pressure (50kPa)		Blood Clot Volume Fraction
	Newtonian	Non-Newtonian	
0			
0.001			
0.003			
0.005			
0.007			
0.009			

Table 9: Blood clot behaviour visualization for both Newtonian and non-Newtonian models at 60kPa

Time (s)	Pressure (60kPa)		Blood Clot Volume Fraction
	Newtonian	Non-Newtonian	
0			
0.001			
0.003			
0.005			
0.007			
0.009			

Based on the visualization shown on the table above, it is clear that non-Newtonian model has faster blood clot extraction time compared to the Newtonian model for all the three different pressures. Hence, an early conclusion could be drawn here. In addition to that, the blood clot extraction time also decreases as the pressure increases. This visualization is further analysed quantitatively to further prove the hypothesis.

4.3.2 Quantitative Analysis of Blood Clot Behaviour at 40, 50 and 60kPa

In this section, a graph of blood clot volume fraction in the artery region against flow time is plotted for both the Newtonian and non-Newtonian models at different pressures.

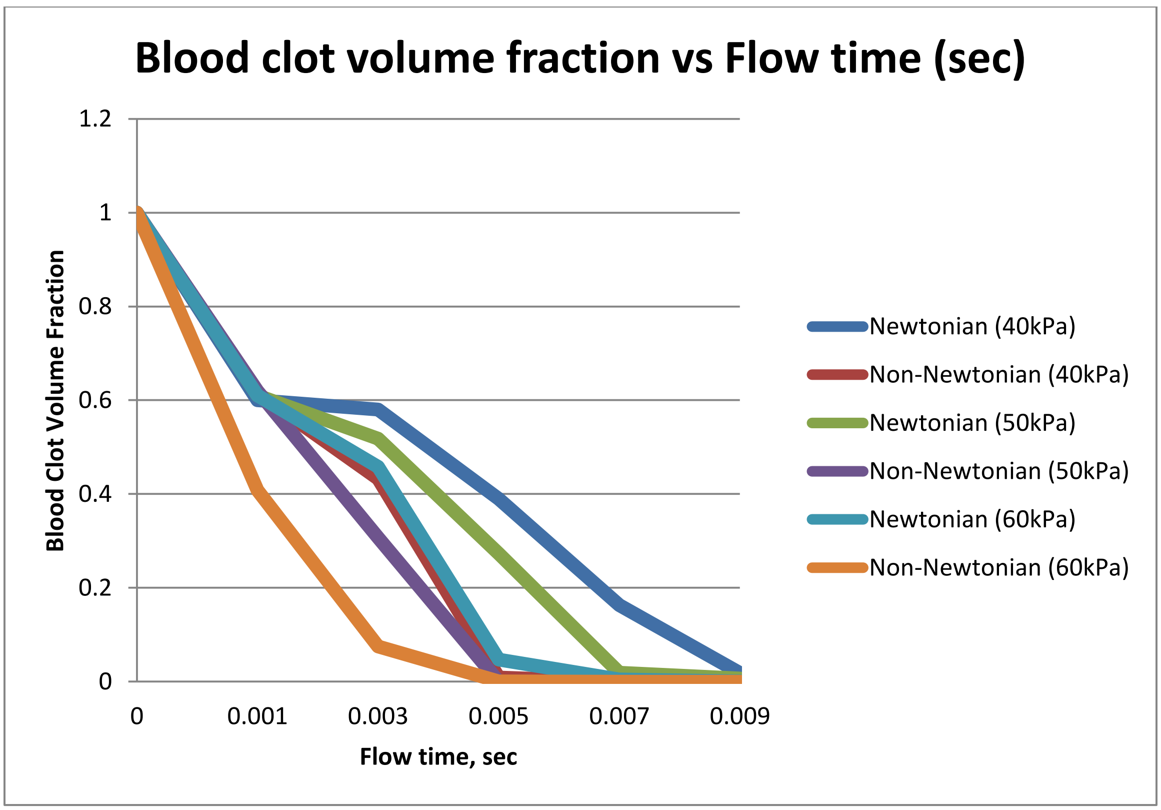


Figure 17: Graph of blood clot volume fraction in the artery region

The table below shows the time taken to completely extract the blood clot out of the artery region for both the Newtonian and non-Newtonian models at different pressures.

Table 10: Summary of blood clot extraction in the artery

Pressure/Model	**Blood Clot Extraction time (s)**	
	Newtonian	**Non-Newtonian**
40 kPa	8µs	6µs
50 kPa	7µs	5µs
60 kPa	6µs	4µs

Based on the results obtained qualitatively and quantitatively, it is shown that the non-Newtonian model has a shorter blood clot extraction time compared to the Newtonian model. Non-Newtonian relatively has shorter blood clot extraction time because theoretically, blood exhibits non-Newtonian property whereby to be more specific it has the shear thinning fluid property. This can be further proven with the results obtained. Non-Newtonian fluid has varying viscosity and it is dependent on the shear rate changes. From the data extracted, as the pressure increases, the blood clot extraction time decreases. An early conclusion can be drawn that blood is a shear thinning fluid because as the shear stress increases, the viscosity decreases which leads to a shorter blood clot extraction time. Hence, the shortest blood clot extraction time is at pressure applied 60kPa which is 4 µs.

Whereby for the Newtonian model, the blood clot extraction time is quite slower compared to non-Newtonian model and it is similar to the earlier studies which were conducted. This is because Newtonian fluid viscosity is proportional over shear rate changes which can also be defined as independent to shear rate changes. Hence, increment in the pressure applied will not affect much on the blood clot extraction time. Whereby, the decrease in the blood clot extraction time is gradual and not as seen in the non-Newtonian model. Furthermore, as mentioned earlier in this study blood is a type of non-Newtonian fluid, therefore it is more practical to use non-Newtonian model rather than Newtonian model for the blood clot extraction study.

It is stated in a study that comparing the Newtonian and non-Newtonian models that, the non-Newtonian Power Law is considered to approximate the blood viscosity model in a more satisfactory way (A. Razavi, E. Shirani, M.R. Sadeghi, 2011). The following figure shows the results obtained from the study that can be used as a reference to prove this non-Newtonian model which is parallel to the result obtained in this research. Hence, the non-Newtonian model in this research is validated.

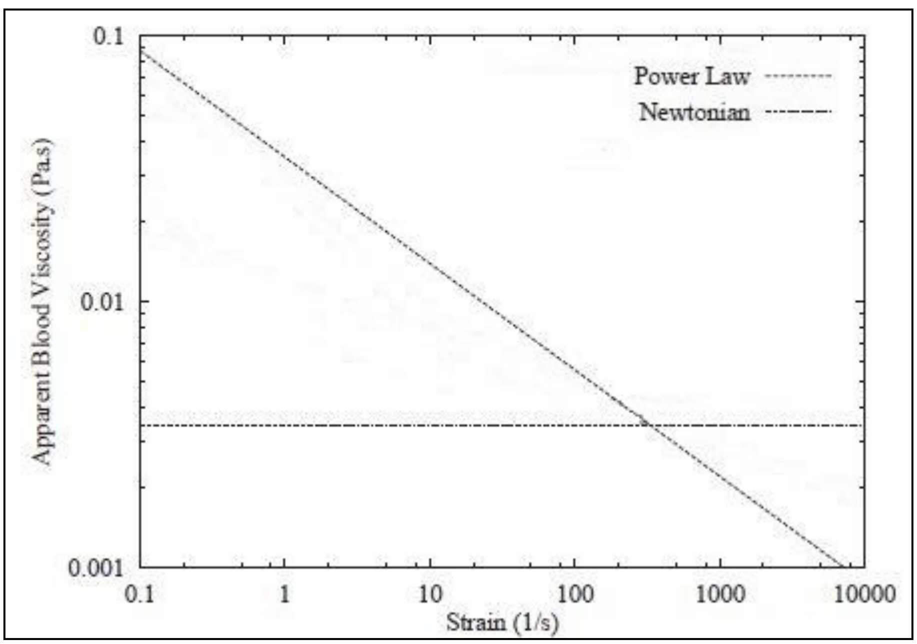


Figure 18: Power Law vs. Newtonian (A. Razavi, E. Shirani, M.R. Sadeghi, 2011)

CHAPTER 5

CONCLUSION

In a nut shell, the simulation study on the blood clot extraction using GP device is very much essential and parallel to the development of medical field to cure stokes. This research will further help on identifying the required parameters for the applicability of the device for clinical purpose.

Firstly, the selection of the optimum grid size is vital before proceeding further into the simulation. Theoretically, the finer the grid size the higher the accuracy of the results but with prolonged computational time. Hence, from the grid sensitivity study performed earlier in the research the optimum grid size of 0.1mm is selected which is fine enough for accurate result and large enough to avoid unnecessary computational time.

One of the main objectives of this research is to identify the effect of viscosity which is non-Newtonian on the blood clot extraction. As per study, blood is a non-Newtonian fluid and it can be classified as shear thinning fluid. Hence, as the shear stress increases, the viscosity decreases. Then, it leads to a faster blood clot extraction. In addition to that, the relationship between the pressure applied and the blood clot extraction time is inversely proportional which means as the pressure applied increases, the shorter time taken to retrieve the blood clot. For instance, at 60 kPa for the non-Newtonian fluid the time taken is only 4 μs whereas at 40 kPa it is 6 μs.

Last but not least, it is vital to compare the Newtonian and non-Newtonian model for the blood clot extraction. Based on the comparison study conducted earlier in the research, it can be concluded that at 40 kPa, 50 kPa and 60 kPa the non-Newtonian models have faster blood clot extraction time compared to the Newtonian models. Therefore the entire objectives have been concluded and achieved.

Further work can be continued from this study in order to provide better accuracy and reliability to the results. Some of the works that can be done are comparison with the experimental data in order to validate this simulation results. This will prove the simulation results obtained from this research is valid. Besides that, this research is conducted with non-Newtonian Power Law, where a comparison study can also be done by comparing with the Cross models, Herschel-Bulkley and Carreau models.

REFERENCES

A. Razavi, E. Shirani, M.R. Sadeghi. (2011, April). Numerical simulation of blood pulsatile flow in a stenosed carotid artery using different rheological models. *Biomechanics*(44), 2021-2030.

Bedekar A, Pant K, Ventikos YP, Sundaram S. (2005). Food Bioproducts Process. *A computational model combining vascular biology and haemodynamics for thrombosis prediction in anatomically accurate cerebral aneurysms*, 118-126.

Cho Y.I, Kensey K.R. (1991). Effects of the Non-Newtonian viscosity of blood on hemodynamics of disease arterial flows. *Steady flow, Biorheol, 28*.

Christopher Tennuci, G. P. (2011). Comparison of the Effectiveness of Three Methods of Recanalization in a Model of the Middle Cerebral Artery. *Stroke Research and Treatment*, 6.

Ferguson J and Kemblowski Z. (1991). Applied Fluid Rheology,.

Ferraris, S. Pierucci and G. Buzzi. (2010). Computer Aided Chemical Engineering. *European Symposium of Computer Aided Process Engineering*.

Finnigan P, Hathaway A, Lorensen W. (1990). Mechanical Engineering. *Merging CAT and FEM*, 28-32.

G. Pearce, S. J. (2007). A New Device For the Treatment of Thromboembolic Strokes. *Journal of Stroke and Cerebrovascular Diseases*, 167-172.

G.C Bourantas, E. S. (2011). Two Phase blood flow modeling and mass transport in the human aorta. *IEEE.* Greece: School of Medicine, University of Patras.

Gruttola, S. d. (2005). *Particle separation processes with emphasis on blood separation.* Zurich: Swiss Federal Institute of Technology Zurich.

Gundogdu, F. Y. (2008). A critical review on blood flow in large arteries; relevance to blood rheology, viscosity models and physiologic conditions. *Korea-Australia Rheology Journal*, 197-211.

Guyton A.C. and Hall J.E. (1996). *Textbook of Medical Physiology 9th edition.* Philadelphia: W.B. Sanders Company.

Hou-Cheng Huang, Zheng-Hua Li and Asif S.Usmani. (1998). *Finite Element Analysis of Non-Newtonian Flow*. United Kingdom: Springer.

Inc, F. (2006, September 20). *ANSYS Fluent*. Retrieved February 20, 2013, from Fluent 6.3 User's Guide: http://cdlab2.fluid.tuwien.ac.at/LEHRE/TURB/Fluent.Inc/fluent6.3.26/help/html/ug/main_pre.htm

Katharine H. Fraser, M. Ertan Taskin, Bartley P. Griffith, Zhongjun J. Wu. (2011). Medical Engineering & Physics. *The use of computational fluid dynamics in the development of ventricular assist devices*, 263-280.

Kim, S. (2002). *A Study of Non-Newtonian Viscosity and Yield Stress of Blood in a Scanning Capillary-Tube Rheometer.* Drexel University.

KuShaari, A.Rahman & Pearce. (2012). CFD Simulation of Blood Clot Behaviour using GP Device. *14th International Conference on Modelling and Simulation* (pp. 77-80). Perak: Universiti Teknologi PETRONAS.

M. Karsheva, P. Dinkova, I. Pentchev, T. Ivanova. (2009). Blood Rheology-A key for blood circulation in human body. *Journal of the University of Chemical Technology and Metallurgy*, 50-54.

Nazehah, N. (2012). *CFD Simulation for the Extraction of Blood Clot in Middle Cerebral Artery using 'GP' 2 Device.* Perak: Universiti Teknologi PETRONAS.

Salvatore Cito, M. D. (2013). Thrombosis Research. *A Review of Macroscopic Thrombus Modeling Methods*, 116-124.

Sandro De Gruttola, Kevin Boomsa and Dimos Poulikakos. (2005). Computational Simulation of a Non-Newtonian Model of the Blood Separation Process. *Artifical Organs*, 949-959.

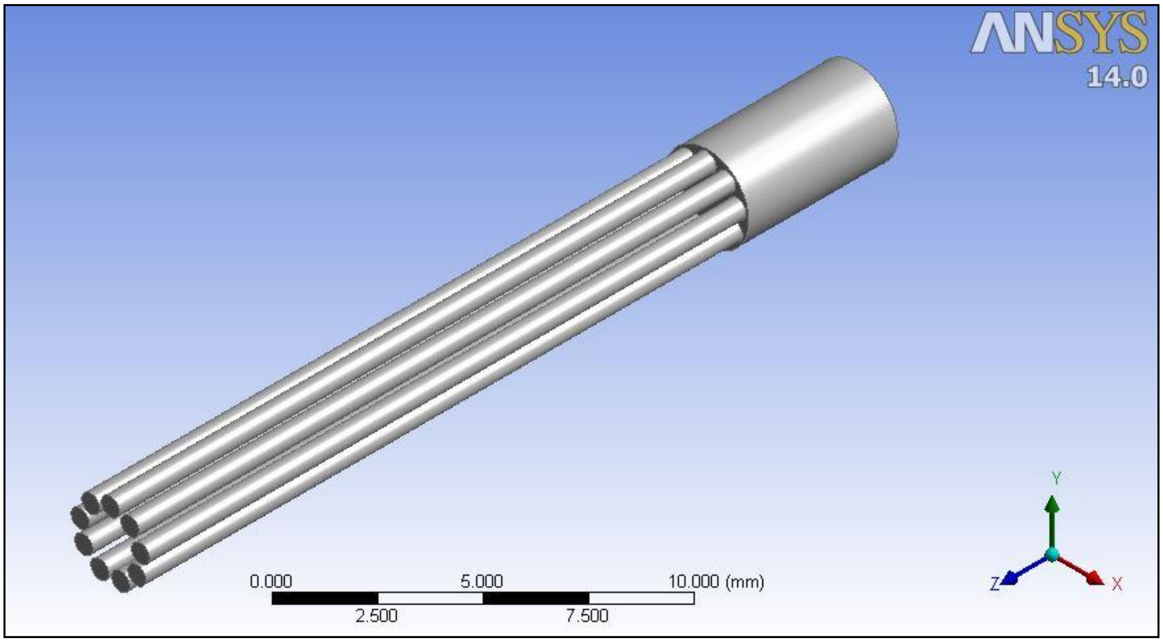
Subramaniam, R. (2002). *Non-Newtonian Flows.* Clarkson: Department of Chemical and Biomolecular Engineering.

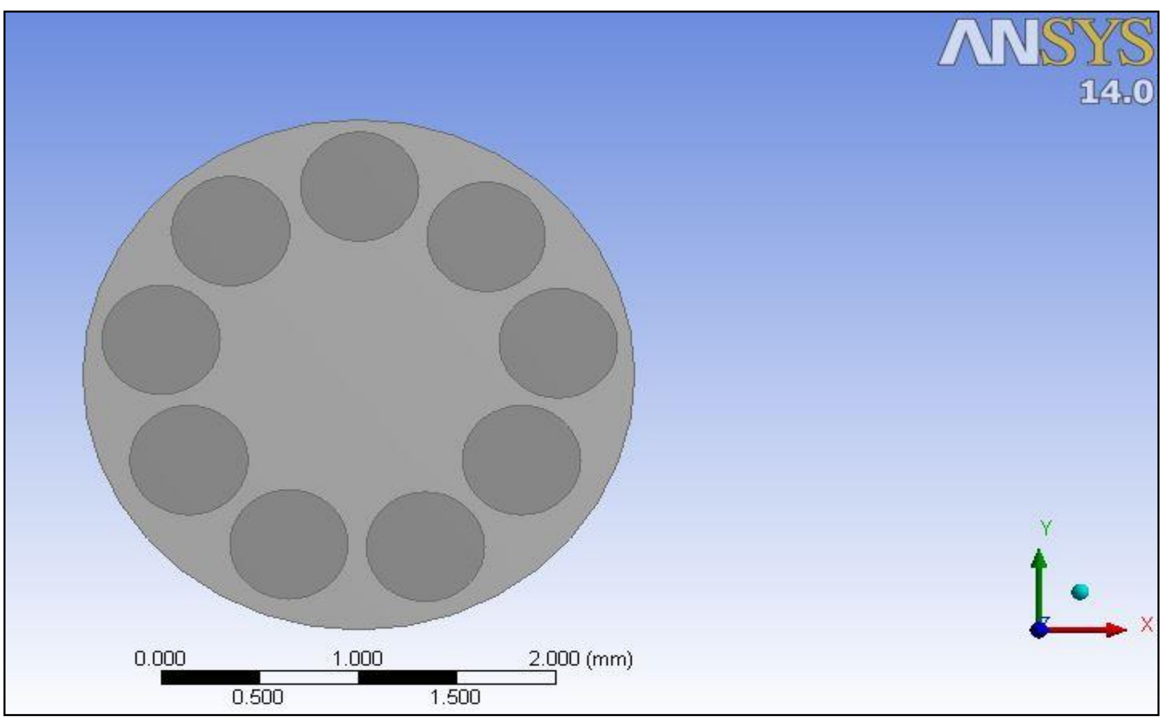
Svetla Petkova, A. H. (2003). CFD Modeling of Blood Flow in Portal Vein Hypertension with and without Thrombosis. *3rd International Conference on CFD in the Minerals and Process Industries.* Melbourne.

Yaghoub Dabiri, N. F. (2005). A Computer Simulation of Blood Flow in Arterial Networks, Including Blood Non-Newtonian Models and Arterial Stenosis. *Engineering in Medicine and Biology 27th Annual Conference* (pp. 2312-2315). Shanghai: IEEE.

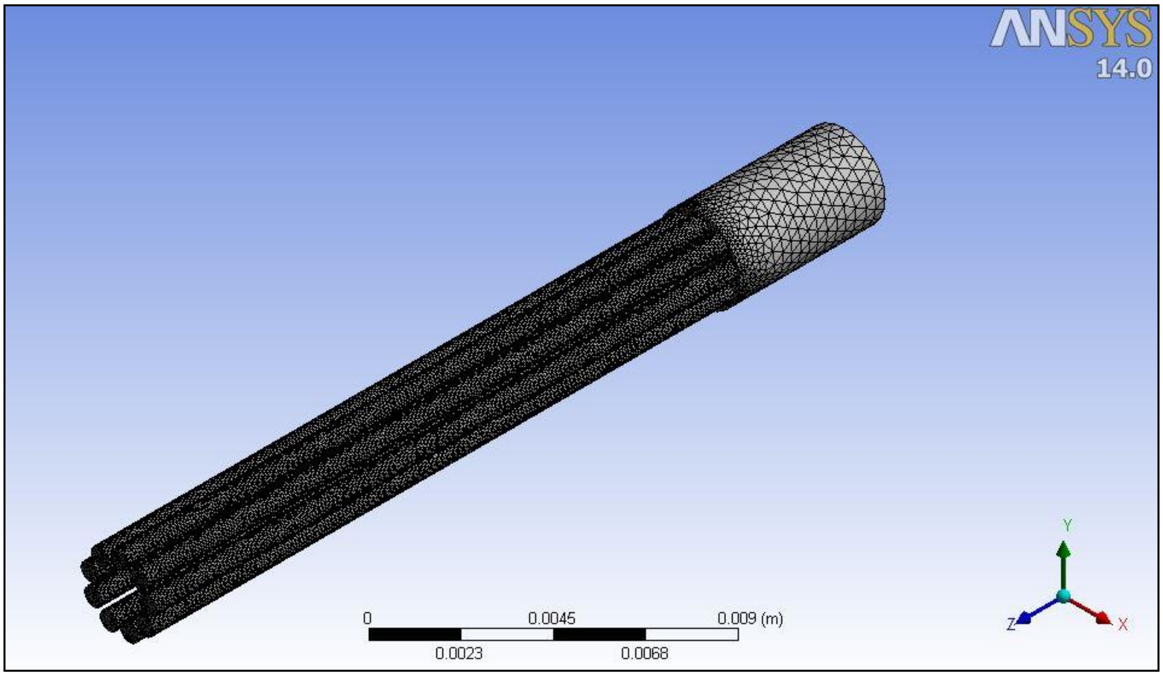
APPENDICES

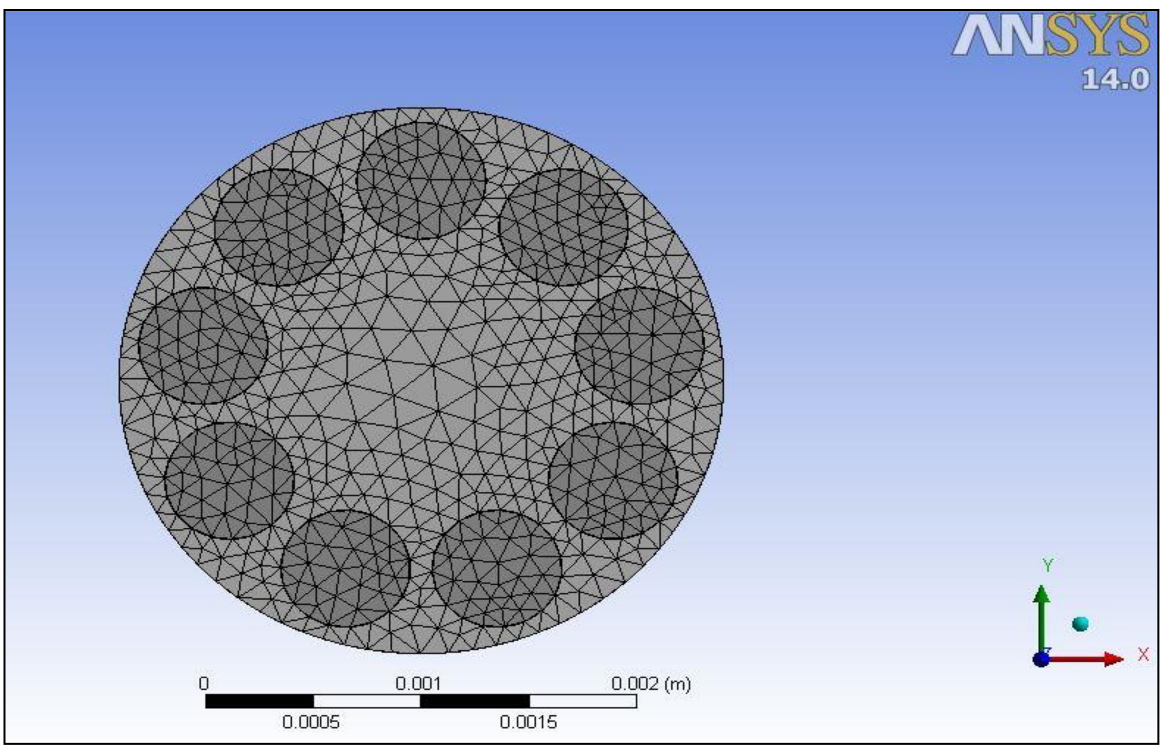
Appendix 1: Geometry Domain



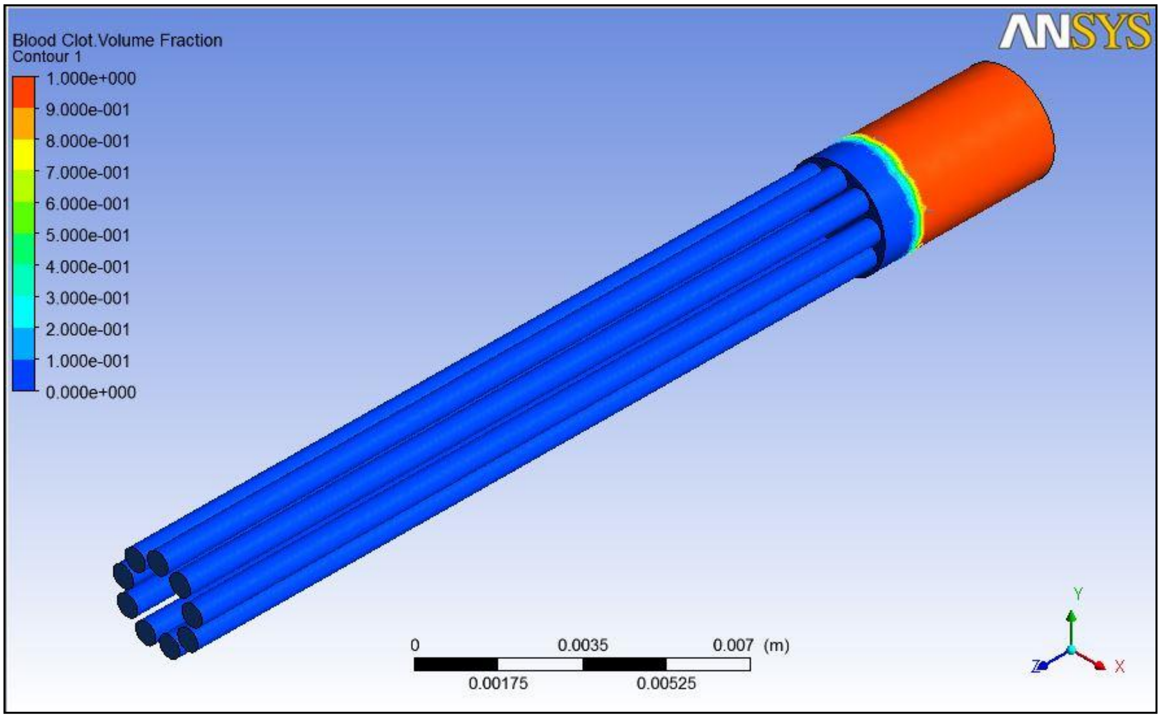


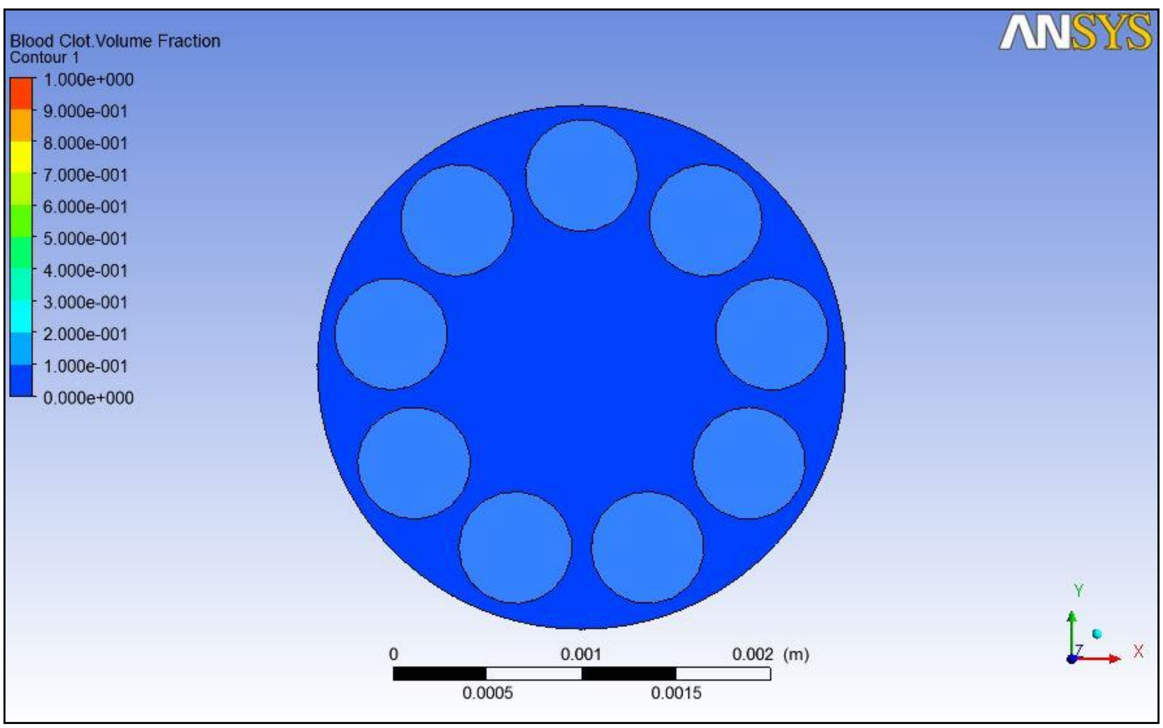
Appendix 2: Meshing of Geometry





Appendix 3: Patched Blood Clot in Geometry





Appendix 4: FYP 1 & FYP 11 Gantt Chart

Task (FYP I)	Week Number													
	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Selection of the Project Title														
Preliminary research work														
• Background study														
• Problem statement & objectives														
• Literature review														
Submission of extended proposal							*							
Proposal defense presentation								*						
Detailed literature review														
Geometry design phase														
Preparation of interim report														
Submission of interim report														*

Task (FYP II)	Week Number														
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
Project work continues															
Validate Geometry domain															
Meshing															
Grid sensitivity study															
FLUENT simulation															
Post processing															
Submission of Progress Report							*								
Pre-SEDEX											*				
Submit Draft Report												*			
Submit Dissertation (soft bound)													*		
Submit Technical Paper													*		
Viva														*	
Submit Dissertation (hard bound)															*