3-Dimensional Blood Clot Simulation On Plastic Arterial Catheter using GAMBIT & FLUENT

By

AFZAL REZA BIN ALI RAHMAN

Dissertation submitted in partial fulfillment of the requirements for the

Bachelor of Engineering (Hons)

(Chemical Engineering)

MAY 2011

FINAL YEAR PROJECT II

DISSERTATION REPORT

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

3-Dimensional Blood Clot Simulation On Plastic Arterial Catheter using GAMBIT & FLUENT

by

Afzal Reza Bin Ali Rahman

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,

____

(Dr Ku Zilati Ku Shaari)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

MAY 2011

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, and the original work is produced on my own except as specified in the references and acknowledgements, and it has not been undertaken or done by unspecified sources or person.

Produced by,

AFZAL REZA BIN ALI RAHMAN

ABSTRACT

The Volume Of Fluid (VOF) model is a surface-tracking technique applied to a fixed Eulerian mesh. It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. The fluids share a single set of momentum equations, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. As such, VOF is an advection scheme that acts as 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 movement of one fluid with regards to its interface is studied to help the researchers and engineers in deciding certain parameters such as pressure and velocity in plastic arterial catheter in order to reduce error, computational cost, and save simulation time. Good resemblance between CFD predictions with the experimental data in certain locations was obtained with the factor of species (blood clot) transport and pressure profile, where dependence of VOF models and grid sizes were discussed in details. The results show us that, the demand in grid study is vital to obtain accurate results with minimal computational cost. On the other hand, wall adhesion is solved in an iterative way, modifying holdups at the wall until the specified wall contact angle had been satisfied. Since the VOF method is a Direct Numerical Simulation (DNS) approach, the time and length scales on which the equations are being solved should be sufficiently small to directly take fluctuating fluid motion due to turbulence into account. Therefore, VOF simulations do not incorporate any other turbulence models, thus only applicable to laminar models.

ACKNOWLEDGEMENT

The author would like to thank various people involved in making this project a success.

First and foremost, the author would like to thank his supervisor, Dr. Ku Zilati Ku Shaari who had given an enlightening and full supervision, guidance, and training. The author is deeply grateful for her advice, encouragement and patience throughout the project and preparation of this thesis. The author have very much benefited from her insightful understanding, profound knowledge, valuable experiences, professional and personal advice.

Secondly, the author would like to extend his uttermost gratitude to the lecturers of Chemical Engineering Program of University Technology of PETRONAS (UTP), who have been very helpful and resourceful while guiding him especially to the Final Year Research Project (FYP) Coordinator, Dr. Mohanad El-Harbawi and Dr. Farooq Hussein for the briefing conducted which was very helpful in constructing the research project and preparing the thesis. Not to forget, endless thanks to Mrs. Anis Suhaila Shuib, who are willingly to share her expertise and knowledge while providing necessary information and required parameters in order to maneuver the project.

Thirdly, endless thanks to the other FYP students for sharing their ideas and information. The thanks extend to the other related lecturers for their endless support in term of welfare and well being throughout completing this project.

Last but not least, the author is grateful to the lecturers and UTP staffs, who had dedicatedly provided him with additional support and encouragement throughout this project. Not to forget to all related parties, who have been giving help, support, courage and advice directly and indirectly, throughout this course in order to make this project a success.

To all individuals that has helped the author in any way, but whose name is not mentioned here, the author would like to thanks and express his gratitude for the wonderful supports.

Thank you very much.

TABLE OF CONTENT

CERTIFICATE OF APPROVAL..i
CERTIFICATE OF ORIGINALITY..ii
ABSTRACT...iii
ACKNOWLEDGEMENT..iv
TABLE OF CONTENT...v
LIST OF FIGURES...vi
LIST OF TABLES...vi
ABBREVIATIONS AND NOMENCLATURES...................................vii
CHAPTER 1 : INTRODUCTION...1
1.1 Background of Study...1
1.2 Problem Statement...2
1.3 Objectives and Scope of Study..5
CHAPTER 2 : LITERATURE REVIEW...7
2.1 Theory on Navier Stokes Equation..................................7
2.2 Theory on Blood Clot Simulation Modeling....................8
CHAPTER 3 : METHODOLOGY...13
3.1 Overview...13
3.2 Pre-Processing..14
3.3 Running/Solving...17
3.4 Post-Processing..19
CHAPTER 4 : RESULTS AND DISCUSSIONS...21
4.1 Grid Size Study...21
4.2 Blood Clot Behavior Simulation......................................23
CHAPTER 5 : CONCLUSIONS..29
REFERENCES...31
APPENDIX A: GEOMETRY FOR PLASTIC ARTERIAL CATHETER IN GAMBIT..33
APPENDIX B: GEOMETRY FOR PLASTIC ARTERIAL CATHETER IN FLUENT (WITH PATCHED BLOOD CLOT)..........................34

LIST OF FIGURES

Figure 1-1 : Phenomena of thrombosis..3

Figure 1-2 : Schematic view of GP clot removal device..5

Figure 2-1 : Non-occlusive clot (red) fills in the middle section of a vessel with diameter 2R except in the small flow channel (blue) of diameter 2r next to the vessel's wall...8

Figure 2-2 : Forces of the flowing blood to the surface of the clot in the flow channel..9

Figure 3-1 : Basic geometry for 3-dimensional plastic arterial catheter.....................16

Figure 3-2 : Example of grid size (fine mesh)..16

Figure 3-3 : Adaption region of patched blood clot..19

Figure 4-1 : Graph of Velocity vs Time at Point A Using 5 Different Type of Grid Size...22

Figure 4-2 : Graph of Volume Fraction of Blood Clot at Pressure Inlet vs Time......28

LIST OF TABLES

Table 3-1 : Blood parameters & properties..17

Table 4-1 : Image of Blood Clot In The Helical Spiral Device At Time, t, with P = 40kPa...24

Table 4-2 : Image of Blood Clot In The Helical Spiral Device At Time, t, with P = 50kPa...25

Table 4-3 : Image of Blood Clot In The Helical Spiral Device At Time, t, with P = 60kPa...26

ABBREVIATIONS AND NOMENCLATURES

Symbols

n	Number of phases	-
p	Pressure	N m^{-2}
Re	Reynolds number	-
t	Time	s
ρ	Density	kg m^{-3}

Abbreviations

CAE	Computer-Aided Engineering
CFD	Computational Fluid Dynamics
CVA	Cerebro-Vascular Accident
GP	Gwen Pearce
MCA	Middle Cerebral Artery
NATF	American Thrombosis Forum
VOF	Volume Of Fluid
WSS	Wall Shear Stress
DNS	Direct Numerical Simulation
RANS	Reynolds Average Navier Stokes

CHAPTER 1

INTRODUCTION

1.1 Background of Study

Approximately 17 million people worldwide suffer annually from strokes. A stroke (also called a cerebrovascular accident, CVA) can be defined as the sudden death of some brain cells due to a lack of oxygen when the blood flow to the brain is impaired by blockage or rupture of an artery to the brain (Medicine Net, 1999). Atherosclerosis is one particular disease which causes the formation of deposits (plaque) on the inner lining of an artery. Plaque rupture may result in emboli, which in turn may lead to myocardial infarction and ischemic strokes. A secondary concern by J. Bernsdorf, 2005 is that of flow disturbances associated with disease related narrowing of the vessel lumen (vessel stenosis). It is likely that areas of stagnant or recirculating flow will develop downstream of a stenosed artery and if activated blood remains in such region for a prolonged period of time, thrombosis may occur.

Coagulation can be initiated by shear rates of sufficient magnitude to cause cell lysis and release clotting factors (L. Badimon & J. Clin, 1989). Following activation, the route to coagulation involves a unique cascade of reactions. Several attempts have been made to model the relevant molecular pathways but these fail to consider realistic flow fields and their development with the growth of the thrombus (V.I Zarnitsina, A.V Pokhilko & F.I Ataullakhanov, 1996). Bernsdorf (2005) suggests that, a comprehensive understanding of thrombosis requires full consideration of the three entities of Virchow's triad; blood chemistry, vessel wall properties and fluid mechanics. For instance, the first triad explains on the phenomena on the blood coagulation (hypercoagulability) alterations in the constitution of blood, which has numerous possible risk factors. The second triad explains on the vessel wall properties, which is any phenomena associated with irritation of the vessel and its vicinity, or also known as "endothelial injury" or "vessel wall injury" arising from shear stress or hypertension. The third triad then explains on the fluid mechanics properties of a blood, which shows that alterations in normal blood flow, might lead to several situations. These include turbulence, stasis, mitral stenosis, and varicose veins.

Non-occlusive blood clots only fill blood vessels partially and form a channel together with the adjacent wall through which blood flows at usually much higher velocities than in normal vessels (I.Sersa, G. Tratar, A. Blinc 2005). I. Sersa et al. (2005) stated that a 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. This is because, blood flow through the channel increases transport of dissolution agents to the clot, thus exerting large forces to the surface of the clot along the channel.

Fluid flow, particularly fluid phase's flows has been a subject of investigations for long time because of their practical applications in many engineering's devices such as plastic arterial catheter, flow meter, venturi, etc. Current studies of the fluid flow in medical devices such as plastic arterial catheter have received considerable attention because the behavior of blood clot causes a greater number of physical phenomena to appear. Among them is the properties of the blood clot itself which is viscous and adhere to the wall of the devices.

1.2 Problem Statement

Thrombosis is the formation of a blood clot (thrombus) inside a blood vessel, obstructing the flow of blood through the circulatory system. When a blood vessel is injured, the body uses platelets and fibrin to form a blood clot to prevent blood loss. If the clotting is too severe and the clot breaks free, the traveling clot is now know as an embolus (North American Thrombosis Forum [NATF], 2010).

Thromboembolism is the combination of thrombosis and its main complication, embolism. When a thrombus occupies more than 75% of surface area of the lumen of an artery, blood flow to the tissue supplied is reduced enough to cause symptoms because of decreased oxygen (hypoxia) and accumulation of metabolic products like lactic acid. More than 90% obstruction can result in anoxia, the complete deprivation of oxygen, and infarction, a mode of cell death (NATF, 2010).

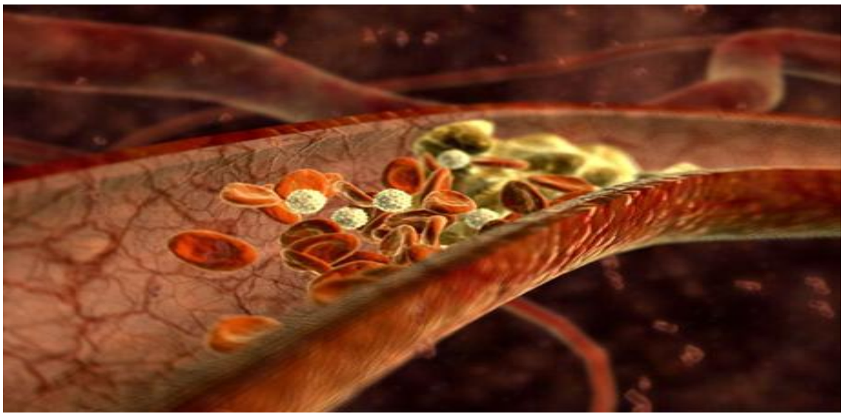


Figure 1-1 : Phenomena of thrombosis

I. Sersa et al. (2005) suggest that properties of thrombolytic agent, structure of the thrombus, and characteristics of molecular transport into the thrombus are three major conditions that determine the success of thrombolysis. Non-occlusive clots have one or more channels created by slowly penetrating plasma containing thrombolytic agents. The clots are almost never completely dissolved when blood flow is re-established along the remaining clot.

Computational Fluid Dynamics (CFD) is a branch of a computational technology that enables user to study the dynamics fluid flow. It is able to build a computational model that represents a system or device. User can apply fluid flow physics and chemistry to this virtual prototype, and it will output a prediction of the fluid dynamics and related physical phenomena.

Similarly, the blood clot phenomena in the artery can be simulated using the CFD technology. As an example, the use of CFD technology led to major design improvements in the heart assist device, enabling its human implantation (C. Kiris & D. Kwak, 2007). Because blood is the operating fluid, it is important that the device propel the blood gently, which means that it must minimize the damage to the red blood cells itself and also to the artery wall. This can be done by designing a device which is able to avoid regions of high shear stress. C. Kiris and D. Kwak (2007) stated that, the blood must be properly washed out of the pump since the formation of blood clots may appear within stagnation regions as a result of previously damaged blood cells. Since the device is small and the operating conditions severe, instrumentation for making necessary flow measurements is extremely difficult to design. Therefore it became necessary to look at the flow by computational means.

The detailed computational flow analysis now affords the designers with a view of the complicated fluid dynamic process inside their devices.

Since the fluid being used in the simulation is blood, there is a need for us to obtain the blood characteristics. Then the blood characteristics will be applied to the designed model to ensure the nature of the flow remains the same with the actual case data, thus, showing better and accurate results. I. Sersa (2005) stated that fast flowing blood through the channel has a large effect on the increase of clot dissolution rate compared to the dissolution rate in the absence of flow. This shows us that an effective flow is vital in the process of dissolving the blood clot. However, a precaution steps must be taken to ensure that the flow does not exceed the critical flow, which might damage the artery wall. I. Sersa et al. (2005) has proposed 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.

A medical device used in thrombolytic therapy, which is known as plastic arterial catheter has been invented. The Gwen Pearce device (or also known as GP device) is basically a clot removal device in which the application of suction, through a micro catheter, allows removal of emboli/thrombi and any subsequent debris without the necessity for direct contact with the clot surface or the intima (G. Pearce, J. H. Patrick & N. D. Perkinson, 2007) . The GP device has a helical design that, with a sectional force applied, produces a vortex at the catheter tip. It is this that removes the soft clot, potentially avoiding the need for balloon application and thus, downstream embolization. Furthermore, the design of the device itself which has less moving parts and engineering simplicity also make it easy to be used. Based on G. Pearce et al. (2007), even with conventional catheter opening as small as 1 mm, the unique mechanism of the GP clot removal device was successful as were previous demonstrations at larger diameters (2mm). Theoretic modelling of the device using 3-dimensional FLUENT simulations has also been undertaken by G. Pearce et al.(2007) A schematic view of the GP clot removal device is shown in Figure 2.

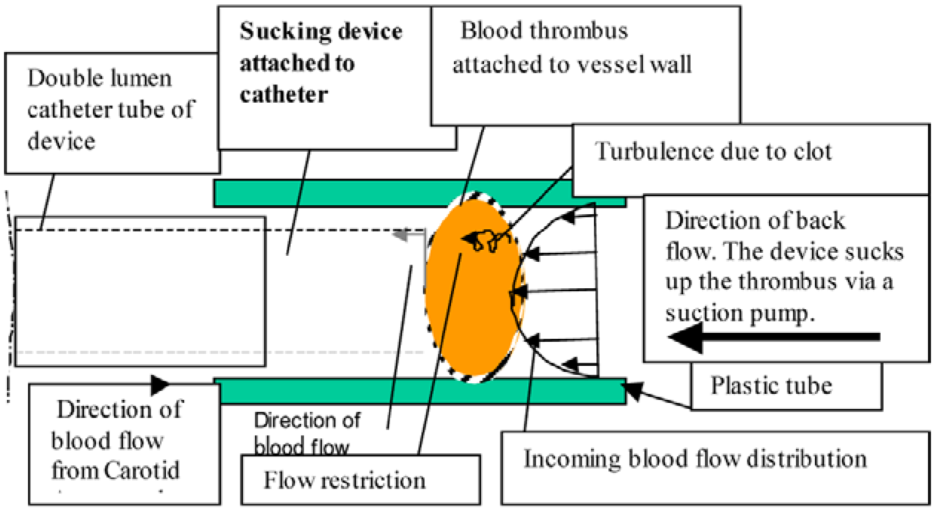


Figure 1-2 : Schematic view of GP clot removal device from "A New Device For the Treatment of Thromboembolic Strokes" by Pearce G, Patrick J. H. , & Perkinson N. D.,2007, Journal of Stroke and Cerebrovascular Diseases, Vol. 16, No. 4 (July-August): pp 167-172.

This device has several advantages over currently available devices because it has no moving parts, relying on the suction physics of vortex creation, which, in addition, may permit less downstream embolization.

Hence, there is also a need for us to determine the pressure required by the medical device and time taken for it to gently remove the blood clot safely (without damaging the artery wall), thus calculating its effectiveness towards modern medical treatment.

1.3 Objectives and Scope of Study

Recent advances in numerical simulations, such as CFD techniques, have been recognized as an alternative to detailed experimental investigation and traditional mathematical modeling. Simulation of blood clot in an artery at a laminar flow on the modern medical device is an important engineering problem. Since such simulation has not been developed yet, it is essential to execute since it can be used to enhance and results obtained can be applied in the modern medical treatment. The goal of this work is to determine the pressure required and time taken to remove the blood clot for the treatment of Thromboembolic strokes in Middle Cerebral Artery (MCA).

For this project, it is needed to model (in 3-Dimensional) the removal of a blood clot in the middle cerebral artery (MCA) in the brain, an artery in which clots commonly occur causing stroke. The MCA is 3mm in diameter. The spiral device is 0.89mm internal diameter and the wall of the arterial catheter is about 0.08mm thick. The one end contains a helical spiral that is 20mm long embossed onto the inside of the catheter, in the last 20mm of its length. The helical spiral is inclined at 30 degrees to the central axis of the catheter, since this angle gives the best flow rate. The catheter, which contains the helical spiral, is positioned within 3mm of the blood clot that is occluding the artery by 100%, thus, no blood is flowing through the artery. The other end of the catheter is attached to a vacuum pump that creates vacuum suction. As the vacuum is applied, the clot is gently sucked into the spiral at the other end of the catheter. This happens because the fluid between the spiral and the blood clot is caused to move in a spiral (a vortex) that in turn then gently absorbs the blood clot off the artery wall.

The pressure required needs to be determined for the removal of a clot of length 4cm and diameter 3mm, so it blocks the artery wall by 100%, which means that no blood flows past the blood clot. Besides, there is also a need to obtain the pressure for clot removal and the time taken to remove the blood clot for devices of 0.89mm, 1mm 2mm and 2.5mm internal diameters.

A comparison of the obtained result with the model of an arterial catheter that does not have any helical spiral inside is to be made, since the previous experiments shown that the spiral at the end of the device be able to reduce the risk of tearing the artery wall.

CHAPTER 2

LITERATURE REVIEW

2.1 Theory on Navier Stokes Equations

Navier Stokes equations describe the motion of fluid substances. This particular equation arise from applying Newton's second law to fluid motion, together with the assumption that the fluid stress is the sum of a diffusing viscous term, plus a pressure term. A solution of the Navier Stokes equations is called the velocity field of flow fluid, it describes the velocity of fluid at a given point in space and time, it dictates not position but rather velocity. The other quantities of interest may be found once the velocity field is solved. Instead of position, studying velocity makes more sense for a fluid; one can compute various trajectories for visualization purposes.

Xu et al. (2008) stated that blood plasma is treated as an incompressible fluid, which is modeled by the two-dimensional incompressible Navier Stokes equations that take the form

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} + \frac{1}{\rho}\nabla p = \frac{\mu}{\rho}\nabla^2 \mathbf{u} + \vec{f}\,,$$
$$\nabla \mathbf{u} = 0,$$

where u = (u, v) is the flow velocity, ρ is the density of the blood plasma, p is the pressure, μ is the viscosity. A Dirichlet boundary condition, where u = 0 on no-slip boundaries is imposed in the equation.

A simplification of the resulting flow equations is obtained when considering an incompressible flow of a Newtonian fluid. The assumption of incompressibility rules out the possibility of sound or shock waves to occur; so this simplification is invalid if these phenomena are important. Taking the incompressible flow assumption into account and assuming constant viscosity, the Navier Stokes equations will read, in vector form,

$$\rho \left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = -\nabla p + \mu \nabla^2 \mathbf{v} + \mathbf{f}.$$

Here **f** represents "other" body forces (forces per unit volume), such as gravity or centrifugal force. The shear stress term becomes the useful quantity when the fluid is assumed incompressible, homogeneous and Newtonian, where μ is the (constant) dynamic viscosity.

2.2 Theory on Blood Clot Simulation Modeling

Blood is known as a complex mixture of proteins, lipoproteins, cells and ions by which nutrients and wastes are transported. Red blood cells are made up of approximately 40% of blood by volume. Blood does not exhibit a constant viscosity at all flow rates compared to water and is non-Newtonian in the microcirculatory system (in small branches and capillaries). The normal blood flow is laminar with secondary flows generated at curves or branches. The Reynolds number varies from 1 in small arterioles to approximately 4000 in the largest artery (Xu et al., 2008). The heart creates pulsating conditions in all arteries, making the blood flow and pressure unsteady.

Since we model the development of thrombus inside a blood vessel of length approximately 10 times larger than its diameter, in our model, we simplify the vessel to be a straight pipe. The vessel wall is assumed to be a rigid wall. Blood plasma is treated as an incompressible fluid, which is modeled by the two-dimensional incompressible Navier Stokes equations explained in Section 2.1.

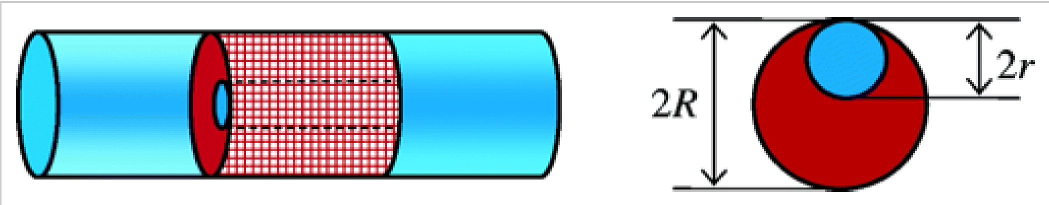


Figure 2-1 : Non-occlusive clot (red) fills in the middle section of a vessel with diameter 2R except in the small flow channel (blue) of diameter 2r next to the vessel's wall

A study done by I. Serza (2005) assumed that a cylindrical blood vessel is narrowed in one segment by a non-occlusive blood clot. The cylindrical clot with a diameter *R* has initially a cylindrical blood flow channel of a radius r_0 that is parallel to the clot axis and is positioned next to the vessel wall. After a pharmacologic concentration of a thrombolytic agent is added to the blood that flows in our idealized two-segment vessel the non-occlusive clot begins to dissolve, and the non-obstructed channel along the clot expands as thin layers of the clot are gradually removed by the flowing blood. The channel keeps its tangential position with the vessel border also when the clot dissolves, i.e., the channel widens to radius *r* (Figure 2-1).

The volume flow of blood through the vessel is assumed to be constant during the entire dissolution process. This may not be true at the beginning as the clot strongly impedes blood flow and may pose a larger resistance to blood flow than other blood vessels. However, the assumption significantly simplifies the model and is not very far from the real situation especially when the clot is well canalized (Sersa I., Tratar G., Blinc A.,)

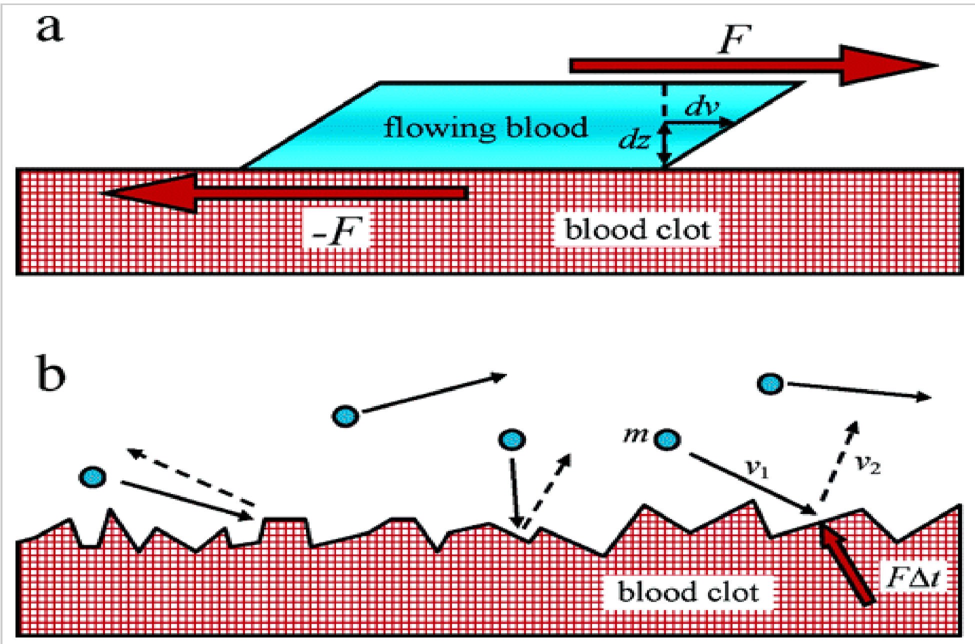


Figure 2-2 : Forces of the flowing blood to the surface of the clot in the flow channel are (a) viscous when flow is slow-laminar and (b) become kinematic, i.e., increase with the blood velocity squared, when flow is fast-turbulent. Viscous forces originate in the strain velocity of blood, whereas kinematic forces originate in the change of momentum of molecules as they bounce from the rough surface of the clot.

It is stated by I. Serza et.al (2005) that the rate of thrombolysis, i.e., the progression rate of lysing front dr/dt is proportional to the average mechanical power of the flowing blood to the surface of the flow channel (Figure 2-2). The power P is equal to the force F of the flowing blood to the clot surface multiplied by the average blood velocity in the flow channel v. The force F is also equal to the product of the pressure drop Δp across the flow channel multiplied by the cross-section area of the flow channel S, from where follows $P = Fv = \Delta pSv$. The pressure drop is equal to $\Delta p = 8\pi\eta vl/S$ when flow is laminar and is equal to the $\Delta p = f\rho lv^2/\sqrt{S}$ when flow is turbulent. Here η is the blood viscosity, ρ is the blood density, and f is a frictional coefficient, which was experimentally determined to be approximately equal to 0.011 for most round tubes. The mechanical power of flowing blood is therefore equal to $P = 8\pi\eta v^2 l$ when flow is laminar and is equal to $P = f\rho lv^3$ when flow is turbulent, from where it may be obtained :

$$dr/dt = a_l v^2$$

and

$$dr/dt = a_l v^3 \sqrt{S}$$

for laminar and turbulent flow, respectively. Other parameters η, ρ, l, and f do not change during dissolution and are therefore not included in the model equations for the dissolution. The flow channel radius r, its cross-section area S, and the average blood velocity v are not independent. They are related by the following relations: $S = \pi r^2$ and $v = \varphi_V/S = \varphi_V/\pi r^2$. Using these relations the two differential equations for clot dissolution change to dr/dt = $a_l(\varphi_V/\pi r^2)^2 = R^5/(5\tau_5 r^4)$ and to dr/dt = $a_t(\varphi_V/\pi r^2)^3 = R^6/(6\tau_6 r^5)$. These differential equations can be easily integrated, and the following dependence of the channel radius on time can be obtained:

$$r(t) = R\sqrt[n]{(r_0/R)^n + t/\tau_n}$$

where $n = 5$ for laminar flow and $n = 6$ for turbulent flow; here r_0 denotes the initial radius of the flow channel. For further analysis it is convenient to define a new variable $x \equiv 1 - S/S_\infty = 1 - (r/R)^2$, which corresponds to the relative clot area, i.e., the ratio between the clot and the vessel area. The quantity x also denotes the level of obstruction for blood flow through the clot. In addition to parameters τ_5 and τ_6 which regulate the dynamics of clot dissolution another parameter *T* may be introduced. *T* corresponds to the delay between injection of the thrombolytic agent into a blood circulation system to the time when it reaches the clot and starts chemically degrading it. Finally, the following model equation for the relative clot area as a function of time is obtained (I. Serza et al.)

$$x(t) = \begin{cases} 1 - (r_0/R)^2 & t < T_n \\ 1 - ((r_0/R)^n + (t - T_n)/\tau_n)^{2/n} & t \geq T_n \end{cases}$$

The aim of thrombolytic therapy is to dissolve blood clots and restore normal vessel functions. I. Serza (2005) stated that an experimental results of Skharov and Rijken (2000), as well as G. Tratar, A. Blinc, M Strukelj, U. Mikac and I. Serza (2004) show that fast blood plasma flow significantly enhances blood clot dissolution under optimal biochemical conditions, but a consistent theoretical model has not been presented yet. Their hypothesis shown that mechanical forces of flowing blood are high enough to play an important role in the dissolution of nonocclusive blood clots in addition to the biochemical reactions that responsible for chemically degrading the clot. The forces may do mechanical work, cause the strain deformations and surface vibrations, and may help the dissolution agent to more efficiently migrate into the clot and faster chemically degrade it. Mechanical forces have a viscous origin when blood flow is slow – laminar, and a kinematic origin when blood flow is fast – turbulent.

It is found that the parameter that is strongly related to the blood clot phenomena is pressure. This is because, from the Virchow's triad that has been mentioned in the earlier part of this report, the fluid mechanics properties of the blood in the blood vessel such as its nature of flow is determined by the radial pressure inside the vessel itself. This led to the findings of Wall Shear Stress (WSS) inside the blood vessel. WSS is the central to the vascular responses to hemodynamics. Wootton, D.M. and D.N. Ku suggest that, the blood WSS modulates diameter adaptive responses, intimal thickening and platelet thrombosis. For instance, if the artery senses increase in flow, it will dilate and remodel with a larger diameter. On the other hand, if low WSS is detected, intimal thickening mechanism will be triggered to re-establish a normal

WSS. Based on the findings from Glagov, S., et al., a baseline for normal WSS level for artery is between 1.5 to 2.0 Pa. Under abnormal shear stress, endothelial dysfunction can occur (Shin, S., et al). Disordered endothelial function triggers inflammatory response, which contributes to the development of atheromatous plaque thus trigger thrombosis. A normal and healthy artery can withstand a pressure within 1.5 to 2.0 Pa range. The maximum shear rate could reach to 1200s-1 which equals to 4.8Pa (Wootton, D.M. and D.N. Ku).

The first improvement of the GP device was the addition of an inducer that spins with the impeller, drawing the blood in and out of the device, thus preventing a back flow (C. Kiris & D. Kwak, 2007).Additionally, the inducer provides enough pressure rise to eliminate back flow in the impeller hub region. The front edges of the blades were slanted forward, allowing blood to flow at the correct angle with the impeller, thereby increasing the efficiency of flow through the device. Second, CFD results suggested that the original design of the device caused clotting in the front bearing area where the blood passes over the flow straightener and meets the impeller blades. Expanding the hub area's width increased the circulation of blood, eliminating stagnant sections where clotting was known to occur. Additionally, researchers tapered the hub surface, accelerating blood flow, and thus creating good wall washing. And third, the exiting flow angle of the blood was examined and the diffuser angle was repositioned. Changing the diffuser blade angle aligns it with the blood flowing through the device, creating a smoother transition of blood over pump surfaces, and reducing the shear stress that causes cell damage.

CHAPTER 3
METHODOLOGY

3.1 Overview

A meshing software called GAMBIT, which is developed by Fluent Inc. will be used as a preprocessor for CFD analysis. GAMBIT is a state-of-the-art preprocessor for engineering analysis. With advanced geometry and meshing tools in a powerful, flexible, tightly-integrated, and easy to use interface, GAMBIT can dramatically reduce preprocessing times for many applications. Complex models can be built directly within GAMBIT's solid geometry modeler, or imported from any major CAD/CAE system. Using a virtual geometry overlay and advanced cleanup tools, imported geometries are quickly converted into suitable flow domains. A comprehensive set of highly-automated and size function driven meshing tools ensures that the best mesh can be generated, whether structured, multiblock, unstructured, or hybrid.

Then, another software is used to continue the simulation. Software called FLUENT provides complete mesh flexibility, including the ability to solve flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include triangular, quadrilateral, tetrahedral, hexahedral, pyramid, prism (wedge), and polyhedral meshes. In particular, the automatic nature of the techniques used to create polyhedral meshes saves time, and, since a polyhedral mesh contains many fewer cells than the corresponding tetrahedral mesh, convergence is faster.

Sophisticated numerics ensure accurate results on any combination of mesh types, including (hybrid) meshes with hanging nodes and non-matching mesh interfaces. FLUENT also allows user to refine or coarsen your mesh based on the flow solution. FLUENT runs robustly and efficiently for all physical models and flow types, steady-state or transient, incompressible or compressible flows (from low subsonic to hypersonic), laminar or turbulent flows, Newtonian or non-Newtonian flows, ideal or real gases, etc.

FLUENT software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants. Special models that give the software the ability to model in-cylinder combustion, aeroacoustics, turbomachinery, and multiphase systems have served to broaden its reach.

Today, thousands of companies throughout the world benefit from the use of FLUENT software as an integral part of their design and optimization phases of product development. Advanced solver technology provides fast, accurate CFD results, flexible moving and deforming meshes, and superior parallel scalability. User-defined functions allow the implementation of new user models and the extensive customization of existing ones. Furthermore, FLUENT’s interactive solver set-up, solution, and post-processing make it easy to pause a calculation, examine results with integrated post-processing, change any setting, and then continue the calculation within a single application. Case and data files can also be read for further analysis with advanced post-processing tools and to compare results from different cases side-by-side.

3.2 Pre-Processing

Before starting with the simulation, the first step is to make a geometry file named *mesh files* (.msh) by using GAMBIT software. First, a desired geometry is drawn by using the tools in GAMBIT. The geometry can be draw either in 2-dimensional or 3-dimensional. It is important to know the fact that drawing the geometry in 2-dimensional can reduce the computational cost. However, since the blood clot deformation need to be observed during the later stage of the simulation, drawing a 3-dimensional geometry is preferable.

In this particular project, the drawn geometry is taken from the work of previous student. However, there is a slight modifications need to be done with the drawing since the geometry need to be united first. On the other hand, the design of the geometry itself need to be amended to suit with the problem given and to improve the accuracy of the result.The modified geometry and the results will be discussed in the next chapter.

Once the geometry is drawn in GAMBIT, then only we can adapt the geometry's grid. This is the most important part of designing a mesh file, since the grid adaption can affect the simulation results. Grid independency is said to be achieved when any further increase in the number of cells does 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.

After the grids have been adapted, we have to define the boundary conditions at each of the faces and edges of the geometry. This is vital since the boundary conditions that have been set up in this stage will be defined for its parameters according to the problem, in the later solving stage.

Next, we have to examine the geometry and its mesh. It is important to check the quality of the resulting mesh, because properties such as skewness can greatly affect the accuracy and robustness of the CFD solution. It is also important to verify that all of the elements in the mesh have positive area or volume, to show the existence of the geometry drawn.

The MCA is 3mm in diameter. The spiral device is 0.89mm internal diameter and the wall of the arterial catheter is about 0.08mm thick. The one end contains a helical spiral that is 20mm long embossed onto the inside of the catheter, in the last 20mm of its length. The helical spiral is inclined at 30 degrees to the central axis of the catheter, since this angle gives the best flow rate. The catheter, which contains the helical spiral, is positioned within 3mm of the blood clot that is occluding the artery by 100%, thus, no blood is flowing through the artery.

The example of geometry is shown in the picture as below:

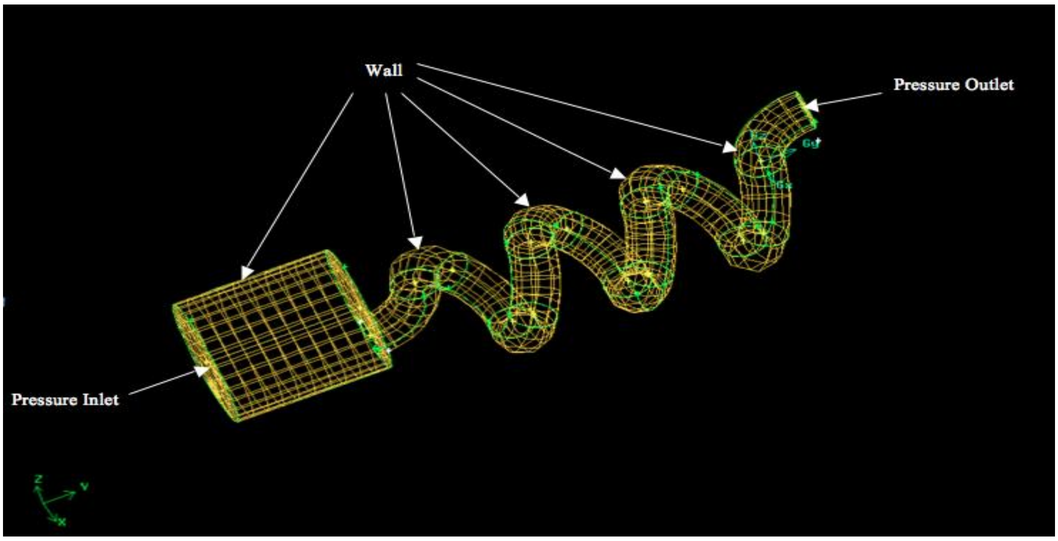


Figure 3-1 : Basic geometry for 3-dimensional plastic arterial catheter

Thus, grid sensitivity study also was conducted by assessing the effect of the grid size on pressure and laminar properties. In each case, the grid density was varied from coarse to fine by increasing the number of cells in the whole device generally, including the wall, since a lot of important data need to be calculated at these region.

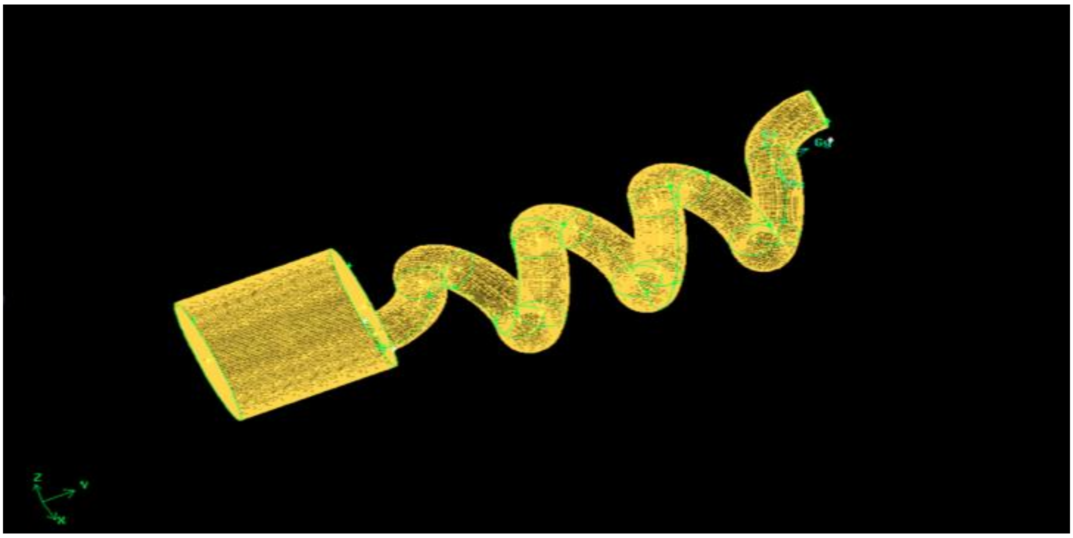


Figure 3-2 : Example of grid size (fine mesh)

The example grid density of the geometry is illustrated in Figure 3-2. Grid independency is said to be achieved when any further increase in the number of cells did not adversely affect the simulation results; the optimum grid size avoided any unnecessary prolonge computational effort required for the simulations with large number of cells. The grid sensitivity study will be discussed more into detail in Chapter 4 of Results and Discussion.

3.3 Running/ Solving

Once the pre-processing steps completed and meshing file is produced, running and solving processes of simulation take place. During this step, the boundary conditions, and operating conditions are specified based on the case of the simulation. Shown below is a table showing the blood parameters and properties needed in performing the simulation.

Table 3-1: Blood parameters & properties

Pressure	0 - [-30, -60] kPa
Blood Viscosity (η)	0,0035 Pa·s
Blood Density (ρ)	1.060 kg/m^3
Bulk's coefficient (B)	$2{,}2\cdot10^9$ N/m
Catheter length (L)	110 cm
Catheter diameter (D)	0,001 m
'GP' length (L)	0,020 m
'GP' diameter (D)	0,001 m
'GP' thickness (h)	0,0001 m
Artery Young modulus (E)	$2{,}8\cdot10^9$ N/m
Artery thickness (h)	0,0001 m
Artery diameter (D_a)	0,003 m
Artery length (La)	0,003 m
Load loss coefficient (ξ)	0,4
'GP'-artery mean diameter (D_m)	0.002 m
Domain change coefficient (r)	141.471,06
Static friction	$2{,}5\cdot10^{-6}$ N·s/m
Dynamic friction	$2{,}5\cdot10^{-7}$ N·s/m
Clot weight	0,6-1,0 gr

The main interest of this simulation was on the study of behavior of blood clot when certain pressure is applied at one end of the blood vessel and to determine the pressure required and time taken to remove the blood clot safely. Therefore, the solver type used to run this simulation was *FLUENT 5/6* solver, which is a solver for laminar, incompressible, unsteady-state flow of Newtonian fluid.

The simulation will be run until it reached a convergence or peak state, where there are no significant changes in the parameters. Therefore, the convergence check must be done first. Generally, there are two ways to perform a convergence check. First, is by making a residual plot and also check on the monitor points. Convergence can be achieved if the monitor point shows constant value after a few number of iteration steps. Based on the convergence check done in this simulation, the acceptable residual value can be near 1e-7. In this report, the second method will be discussed more into detail, which is by monitoring on the points desired. The monitor point was introduced inside the geometry where changes can be observed a plot of parameter versus iteration time can be produced.

In this project, FLUENT is used to perform the simulation. The solver being used is a Pressure-based type, which is typical for VOF's case. The formulation type used is *Implicit*. Since the operating fluid is blood and its flow is intermittent due to heart pumping, a *1st Order Implicit Unsteady State* is used as our time formulation. *Absolute* type is chosen for our velocity formulation.

Only then the multiphase model is set up. As being mentioned earlier, VOF is applied as the primary model, and Explicit is chosen for the VOF parameters. The Courant number is fixed at 0.25, and the number of phases is set at 2 phases, which are blood and blood clot in this case.

Then, the 2 phases are defined as 'blood' and 'blood clot', by putting the value of density and viscosity. Other parameters such as wall adhesion, shear stress and surface tension can be applied during the simulation to increase the accuracy of the simulation.

The operating condition such as pressure inside the vessel and its reference location in the geometry drawn earlier is then set according to the findings. Next, the boundary conditions are applied based on our findings, theoretical knowledge and assumptions.

Now, the blood clot in the blood vessel need to be justified. This can be done by adapting a region for this blood clot. Once the blood clot's region exists, it is then patched inside the blood vessel, resulting in 2 different phases with parameters that we have set earlier.

Shown below is the figure of the blood clot in the blood vessel:

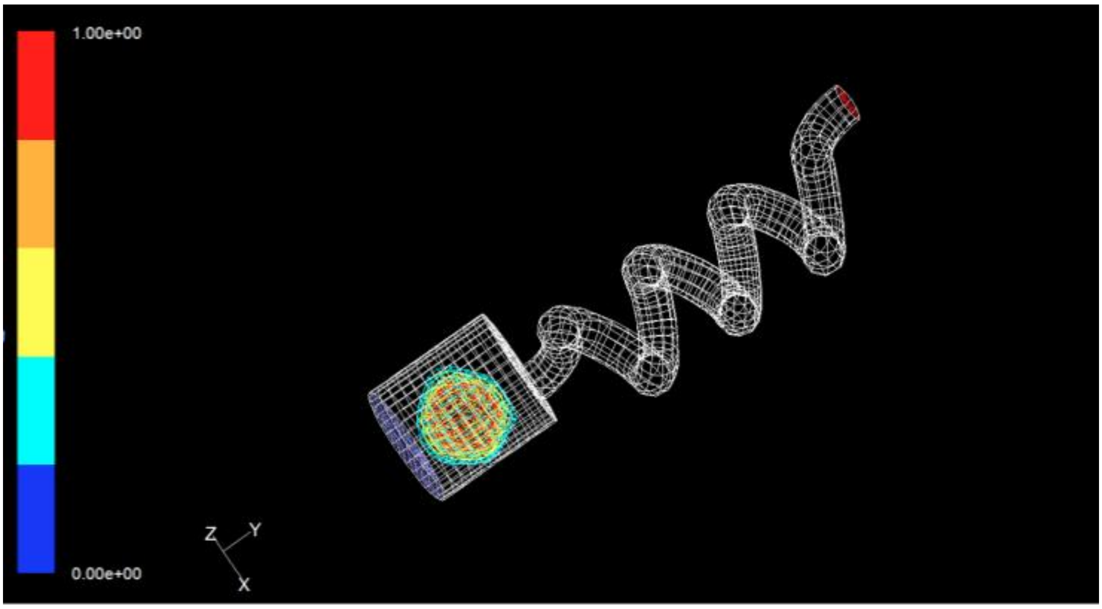


Figure 3-3 : Adaption region of patched blood clot

Once all the parameters of the simulation including the patched region have been set up,the case simulation now is ready to be solved and calculated prior proceeding to post-processing steps.

3.4 Post-Processing

Once the model is iterated and solved based on desired operating conditions and boundary conditions, the post-processing part now can be carried out. The objective of the post-processing steps is to display the results of the simulation in a desired manner based on cases. From post-processing steps, the results can be observed quantitatively. Countour, vectors, and line/rake pathlines are among the typical model that are usually used during the post-processing steps.

For this work, the post-processing step is one of the most vital step in the entire simulation. This is because, it is important to observe the blood clot deformation in the blood vessel when it is subjected to certain inlet and outlet pressure. This can be done by observing the movement of the blood clot the moment pressure is applied to it, by the help of moving frames or animation properties in FLUENT. After the solving stages is done, FLUENT will create a data files (*.dat*) at each of specified time steps. This data files are used as a frames and these frames is combined together to form an animation sequence. Once the animation sequence is ready, FLUENT will convert it into a media file format known as *.mpeg* file. The animation sequence will be discussed more into details in Chapter 4 of Resuls and Discussions.

From the blood clot behavior, then only the suction pressure, which is the pressure required to gently suck the blood clot from the blood vessel as well as the time taken to remove the blood clot safely and effectively, can be determined.

Several iterations have been made in order to perform the simulation. Once the data files (*.dat*) are saved, the behavior of the fluid flow and its effect towards desired boundary conditions under certain operating conditions now can be displayed.

CHAPTER 4

RESULTS AND DISCUSSIONS

In the nature of solving a physical fluid flow simulation, the technique used was by breaking down the physical domain into a large number of discrete control volumes, called elements or cells. Discretization of these elements or cells is probably the most crucial source of error for the accuracy of numerical fluid flow simulations (Michael Casey, 2000). Therefore, two parameters have been studied to control the numerical error, which are convergence check study and grid sensitivity.

4.1 Grid Sensitivity Study

Grid sensitivity tests were conducted by assessing the effects of grid sizes on the absolute and total pressure. The grid size was varied by increasing the number of cells in x, y, and z directions (since this is a 3D case). 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.

The location is evaluated along the geometry with coordinate as the same as the blood clot position, which is (0.2,-9.5,-0.17) for x, y, and z respectively. This point is selected because we want to perform a grid size study at the location that appears to have a big deformation on the blood clot, since the fluid at this coordinate is caused to move in a spiral (a vortex) that in turn gently absorbs the blood clot off the artery wall.

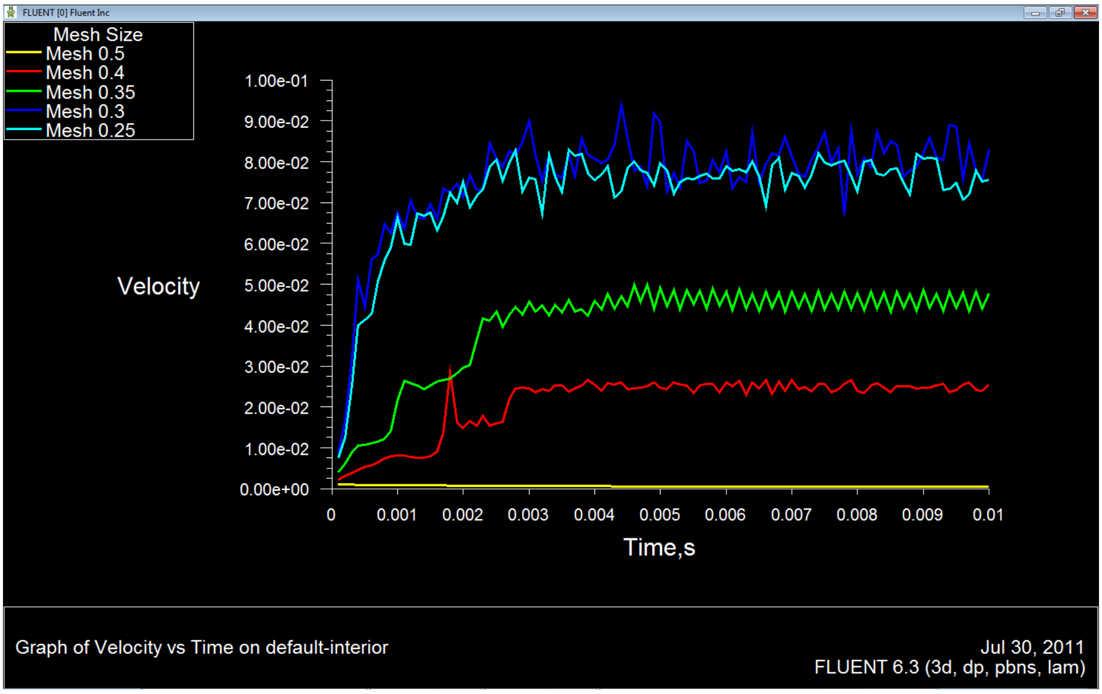


Figure 4-1 : Graph of Velocity vs Time at Point A Using 5 Different Type of Grid Size

Results for the line plot for the integral pressure using different grid size is shown in Figure 4-1. The 4 types of grid size meshes are 0.5, 0.4, 0.35, 0.3, and 0.25. In the graph above, mesh size 0.5, which is the coarsest behave very differently from the other meshes, and the static pressure continually behaving differently as the number of iterations increasing. It is repeatedly found in many CFD papers that the authors check only on one parameter, which is velocity component. It is not wrong to check the velocity component in this problem, but it is better to check on the pressure component since the problem relates on VOF model.

On the other hand, the graph for mesh 0.4 and 0.35 starts to vary and change in a consistent manner. This shows us that by changing the grid size, it will increase or change the effectiveness of the calculation by a substantial amount as shown in the graph.It is noted that when the grid size is decreased until certain mesh size (0.3 and 0.25) the graph behave similarly to each other. This indicates us that one of the size of the grid used in each of the mesh will be the ideal grid size,since a finer mesh will not give any significance improvement to the final results. From the analysis of the graph above, therefore, mesh 0.3 is said to reach the grid independent, and it will be used as a basis of grid size mesh in the further works of this project.

4.2 Blood Clot Behavior Simulation

In this work, the objective is to observe the blood clot deformation and its behavior when certain pressure applied to the inlet and the outlet of the plastic arterial catheter. Then, as discussed in Section 3.4, a media file containing the animation sequence of some frames throughout the simulation is saved. Table 4-1, 4-2, and 4-3 shows the image of blood clot in the helical spiral device at certain time,t, when a pressure of 40kPa, 50kPa and 60kPa are applied at the outlet of the device,respectively :

Table 4-1 Image of Blood Clot In The Helical Spiral Device At Time, t with P = 40 kPa

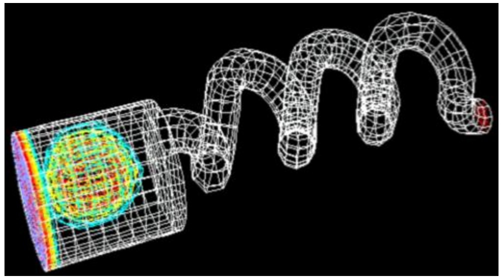
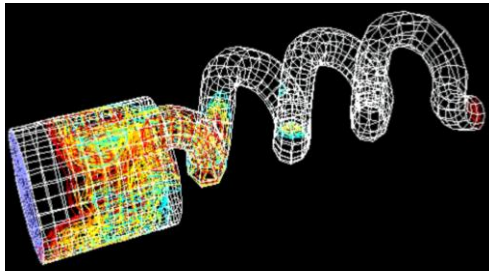
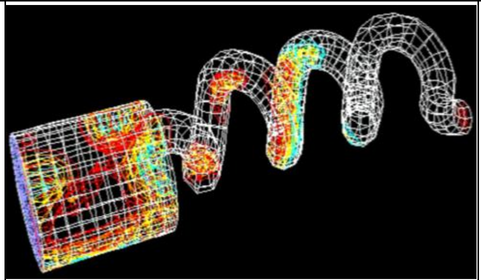
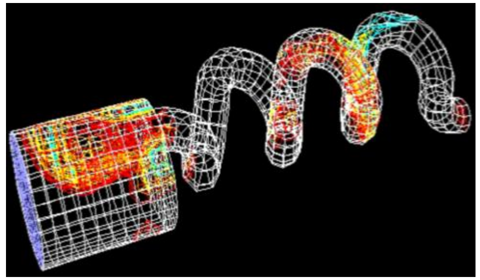
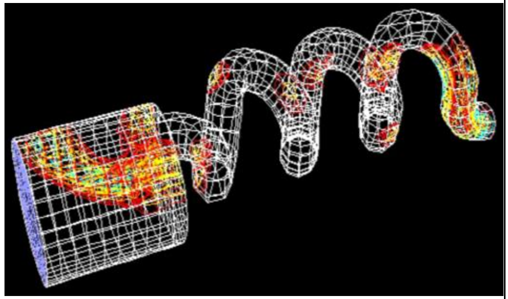
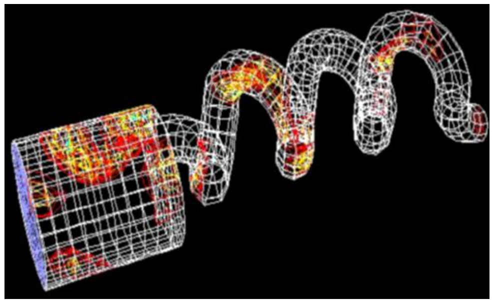
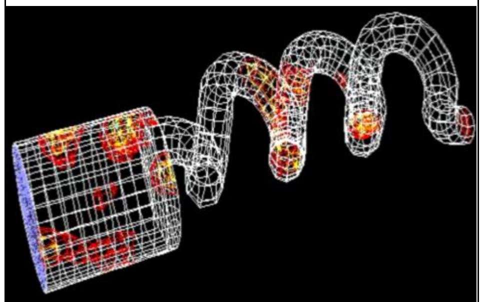
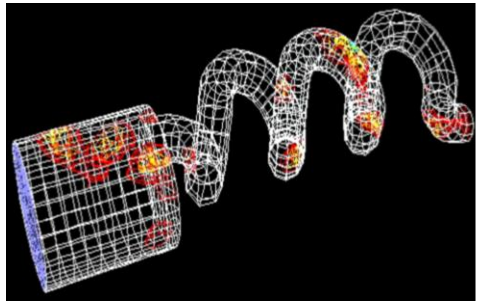
t / P	0 s	7.5×10^{-4} s	1.5×10^{-3} s	2.25×10^{-3} s
40 kPA				
	3.0×10^{-3} s	3.75×10^{-3} s	4.5×10^{-3} s	5.25×10^{-3} s
				

Table 4-2 Image of Blood Clot In The Helical Spiral Device At Time, t with P = 50 kPa

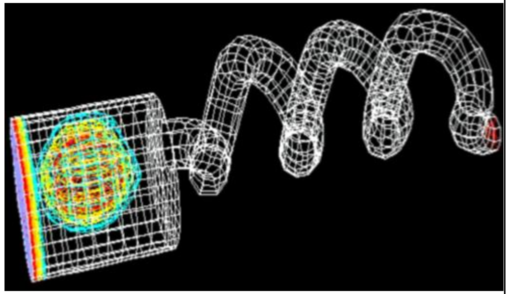
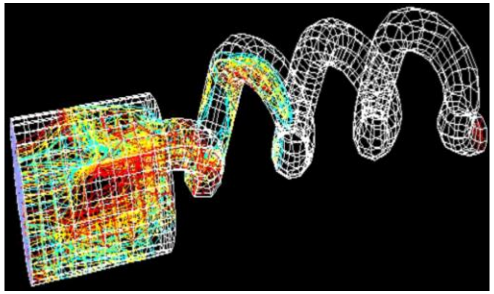
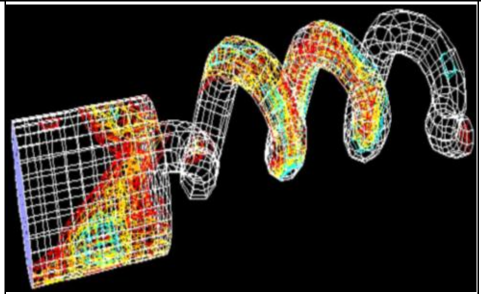
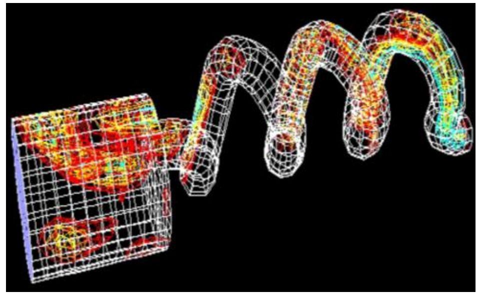
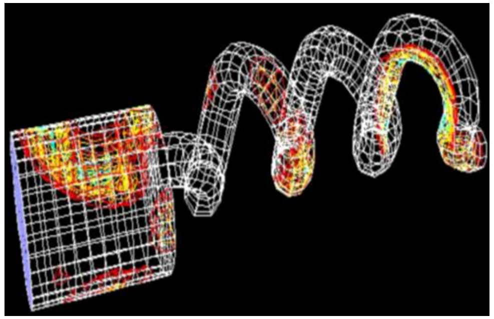
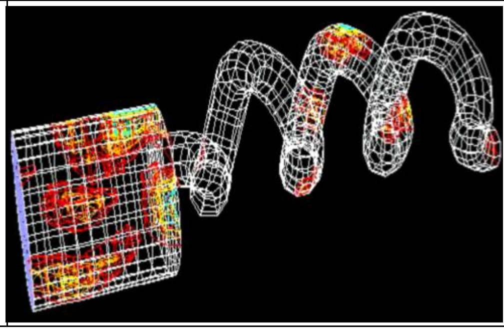
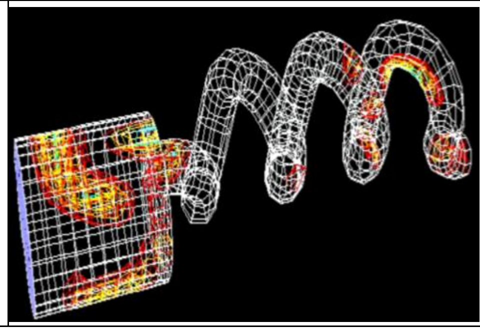
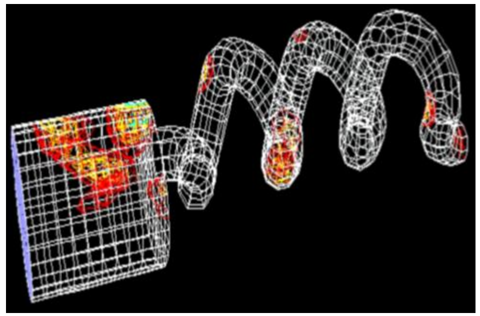
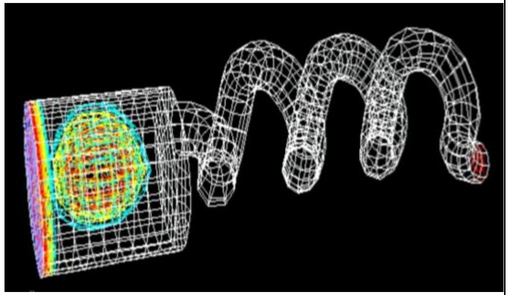
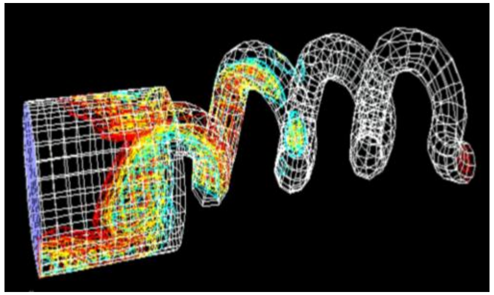
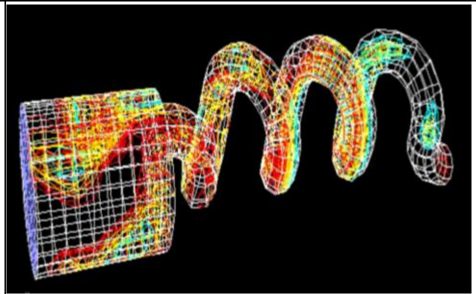
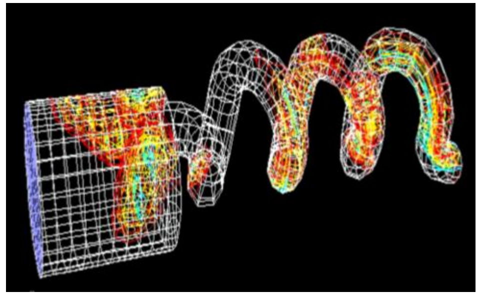
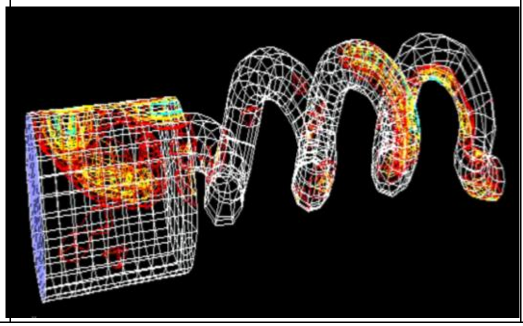
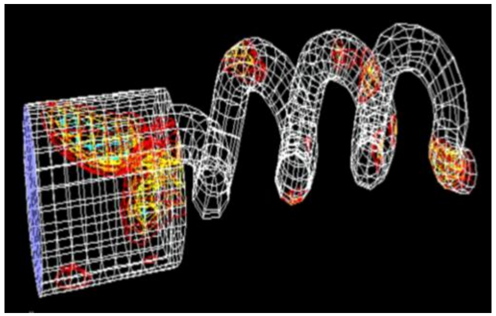
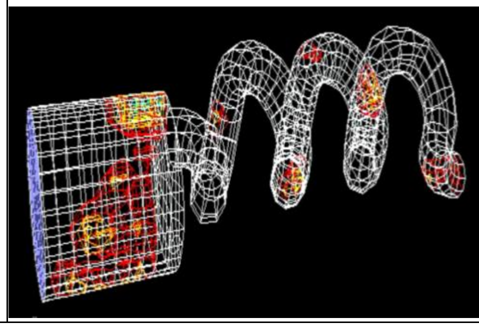
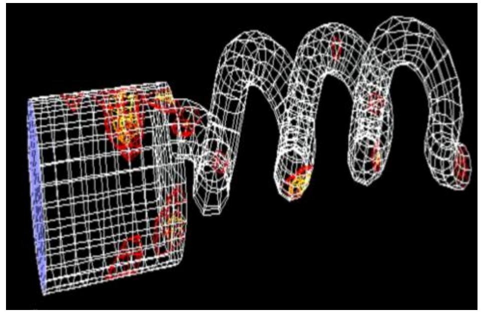
t / **P**	0 s	7.5 x 10^{-4} s	1.5 x 10^{-3} s	2.25 x 10^{-3} s
				
50 kPA	3.0 x 10^{-3} s	3.75 x 10^{-3} s	4.5 x 10^{-3} s	5.25 x 10^{-3} s
				

Table 4-3 Image of Blood Clot In The Helical Spiral Device At Time, t with P = 60 kPa

t / P	0 s	7.5×10^{-4} s	1.5×10^{-3} s	2.25×10^{-3} s
60 kPA				
	3.0×10^{-3} s	3.75×10^{-3} s	4.5×10^{-3} s	5.25×10^{-3} s
				

In the table above, it is clearly shown that the blood clot is having a deformation as soon as the pressure is applied at the inlet of the plastic arterial catheter. This shows us that the blood clot can be sucked out gently using a minimum pressure of 40kPa.

As we can see in the results above, the blood clot deformation behaves like a gel composition. This shows us that the parameters that we have set earlier during the solving stage in Section 3.3 are correct. It is clearly shown that the blood clot deformation rate is greater when higher pressure is applied at the end of the plastic arterial catheter. This is necessarily true since higher pressure tends to suck out with a greater force, hence, more blood clot can be sucked out at one point, with less time.

By observing the animation file generated by FLUENT for this particular modeling and simulation, it is shown that the spiral part of this plastic arterial catheter tends to "cut" the clot into a smaller part or portion during the blood clot removal process. This is not stated in any journal or works, but the spiral part seems to provide a safe precaution measurement to remove the blood clot safely.

A study of how rapid the blood clot is being dissolved over time using different pressure is also done throughout this work. A graph of volume fraction of blood clot versus time is plotted and the result can be seen in Figure 4-2. In this particular graph, we can see that the volume fraction of the blood clot is decreasing as time increasing. This shows us that the blood clot is being sucked continuously from time to time. During the halfway of the process, the volume fraction of the blood clot seems to drop significantly since the blood clot is dissolved into a smaller part or portion during this stage. The graph however decline slowly towards the end of the process because the applied pressure is trying to remove the residual clot that is remaining in the device.

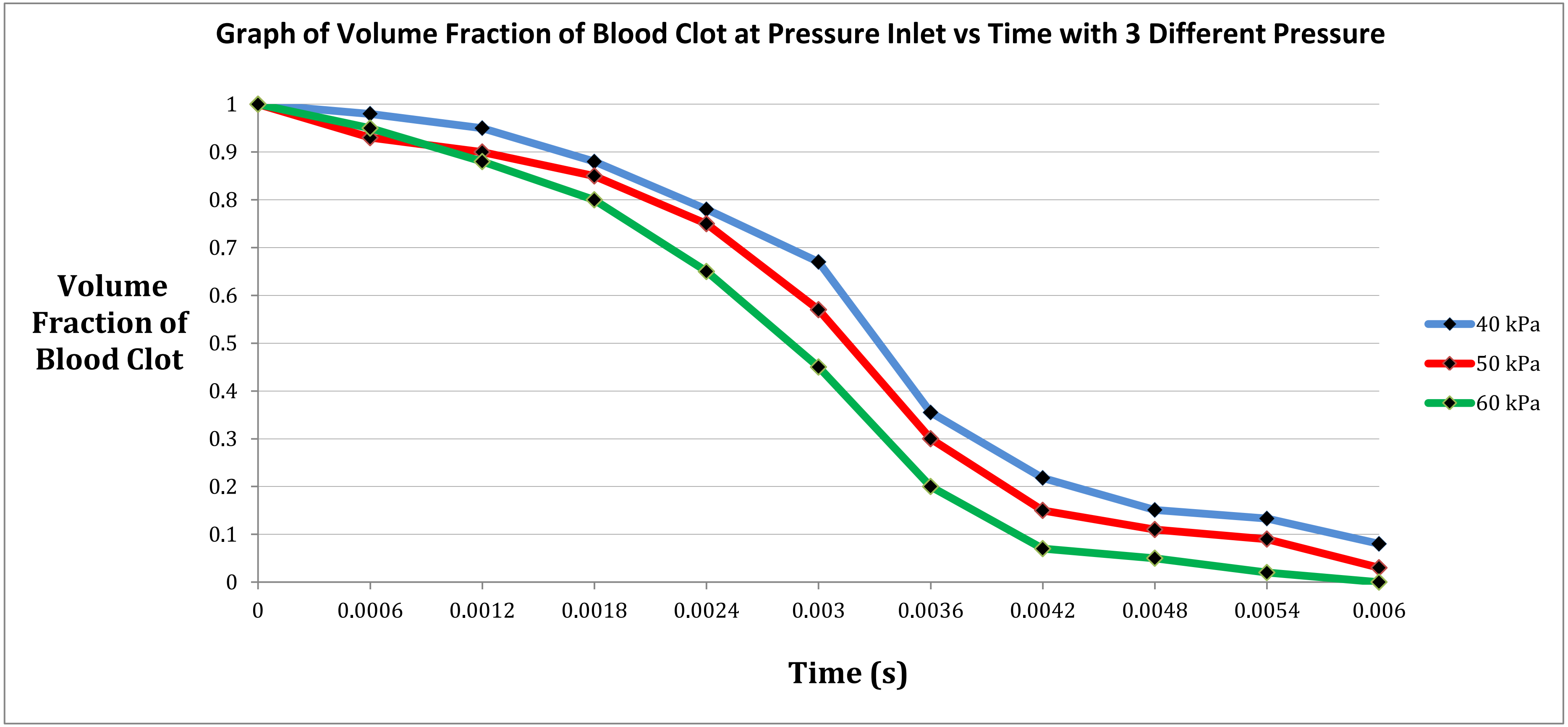


Figure 4-2 : Graph of Volume Fraction of Blood Clot at Pressure Inlet vs Time

CHAPTER 5

CONCLUSIONS

3-dimensional blood clot simulation on plastic arterial catheter is studied and done by applying the CFD simulations with different size of grid, using the Volume of Fluid (VOF) model. This is done to show the pressure recommended from the literature reviews or previous works is suitable to remove the blood clot safely. On the other hand, the time taken to remove the blood clot is recorded significantly throughout the simulation process.

For the case of selecting grid size (grid size study), it is shown 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. Therefore, from the results obtained in the grid size study, it can be concluded that Mesh 0.3 has reach its asymptotic level and chosen as grid independent. It is believed that it can predict the right clot deformation and blood flow in the device.

From the animations that are created using FLUENT software as shown in Table 4-1, 4-2, 4-3, it is obvious that a certain range of pressure that is recommended by the literature reviews and previous works which are 40 kPa to 60 kPa are enough to remove the blood clot safely and effectively. The suction pressure however must be kept within this range to avoid under-pressure flow or turbulent flow.

Time also has been an important parameter throughout this work. A different time section is recorded in order to observe the blood clot deformation patterns at certain time. Then, this blood clot deformation patterns is compared to the patterns in other cases where different pressure is applied at one fixed time. From the animation sequence, it is proved that at certain fixed time, t, different deformation patterns are observed when different pressure are applied to different case. From the full simulation, the time that is needed to remove the blood clot safely in this particular case is to the extend of 6×10^{-3} seconds.

VOF model is capable of simulating the blood clot deformation in this particular work. This is important since it can become the foundation or basis of assumption in next works or experiment, hence contributing information to medical industry. This study can be very helpful to the medical practitioners to understand the nature, feature and characteristics of the device as well as the fluid flow inside the device itself. Knowing these facts and elements can provide a platform for further research of plastic arterial catheter in deciding the right model and parameters. This is vital in order to save the simulation time, reduce computational errors, hence producing more accurate results to be used in the future.

REFERENCES

MedicineNet.Website,URL

< http://www.medterms.com/script/main/art.asp?articlekey=9791>

North American Thrombosis Forum.Website, URL

<http://www.natfonline.org/what_is_thrombosis.php>

Wisegeek.Website, URL

<http://www.wisegeek.com/what-is-thrombosis.htm>

Badimon L.and Badimon JJ., J. Clin. Invest. **84**: 1134-1144 (1989).

Bernsdorf J., Harrison S. E. , Smith S. M., Lawford P. V. , Hose D. R., 2005, *Concurrent Numerical Simulation of Flow and Blood Clotting using the Lattice Boltzmann Technique*, 11th International Conference on Partial and Distributed System (ICPADS'05)-*7695-2281-5/05 IEEE.*

Glagov, S., et al., *Hemodynamics And Atherosclerosis - Insights And Perspectives Gained From Studies Of Human Arteries.* Archives Of Pathology & Laboratory Medicine, 1988. **112**(10): p. 1018-1031.

Kiris C. , Kwak D. , 2007, *Space Technology and CFD Applied to the Development of the DeBakey Heart Assist Device*, Revolutionary Technology,NASA.

Pearce G, Patrick J. H. , & Perkinson N. D. , 2007, *A New Device For the Treatment of Thromboembolic Strokes, Journal of Stroke and Cerebrovascular Diseases,* Vol. 16, No. 4 (July-August), 2007: pp 167-172

Sersa I., Tratar G., Blinc A., 2005, *Blood clot Dissolution dynamics Simulation during Thrombolytic Therapy*, 50th Anniversary Journal of Chemical Information and Modeling, ACS Publications, Journal of Chemical Information Model, 17 September 2005, 45 (6), pp 1686-1690, 10.1021/ci050162k.

Sakharov, D.V., Rijken D.C, *The effect of flow on lysisis of plasma clots in a plasma environment.*Thromb. Haemostasis 2000, 83, 469-474.

Shin, S., et al., *Blood flow resistance with vibration and its effect n blood cell migration.* Clinical Hemorheology And Microcirculation, 2004. 30(3-4): p. 353-358.

Wootton, D.M. and D.N. Ku, *Fluid mechanics of vascular systems, diseases, and thrombosis.* Annual Review Of Biomedical Engineering, 1999. **1**: p. 299-329.

Zarnitsina V.I., Pokhilko A.V. and Ataullakhanov F.I., Thromb. Res. 84: 225-236 (1996).

Michael Casey, Torsten Wintergerste, *Best Practice Guidelines, Version 1.0,* Fluid Dynamics Laboratory, January 2000.

Xu Z., Chen N., Shadden S., Marsden J. E., Kamocka M. M., Rosen E. D., Alber M., (2008 September 15), *Study Of Blood Flow Impact on Growth of Thrombi Using A Multiscale Model.*

Dryden, H. L.; Murnaghan, F. P.; Bateman, H. In *Hydrodynamics*; Dover Publications: New York, 1956; Part 3, Chapter 3, pp 438−444.

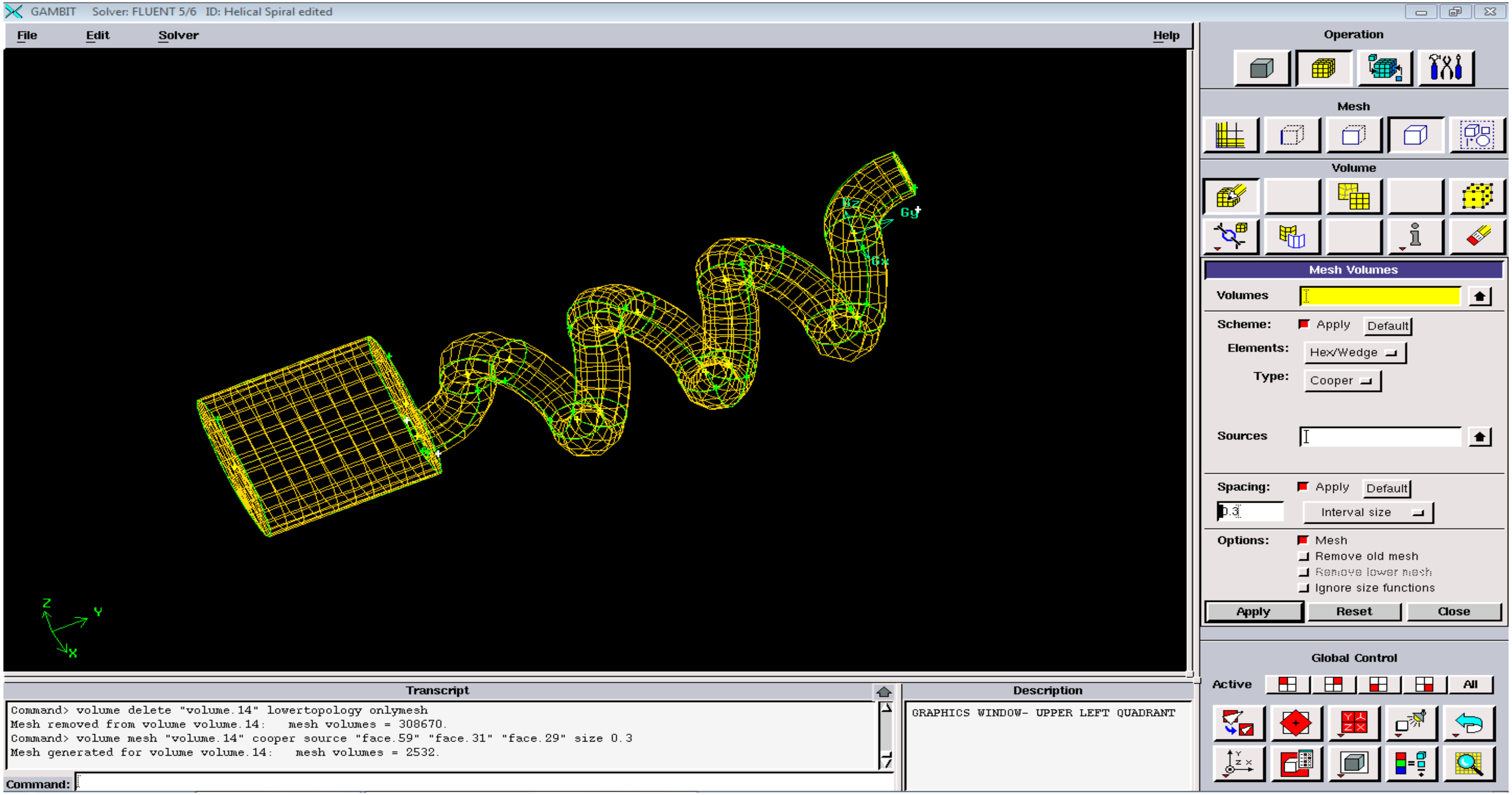
S. C. Shadden, K. Katija, M. Rosenfeld, J. E. Marsden and J. Dabiri 2007, *Journal of Fluid Mechanics*, 593, 315-331.

T. M. Cickovski, C. Huang, R. Chaturvedi, T. Glimm, H. Hentschel , M. S. Alber, J. A. Glazier, S. A. Newman, J. A. Izaguirre. 2005, IEEE/ACM Trans. Comput. Biol. *Bioinformatics*, 2(4):273-288. ISSN 1545-5963.

A. Anderson, M. Chaplain and Rejniak K, 2007, *Single Cell-Based Models in Biology and Medicine* Eds. Birkhaser

S. Amornsamankul, B. Wiwatanapataphee, Y. H. Wu and Y. Lenbury, 2006 *Intl. J. of Biomedical Sciences*, 1(1), 42–46.

APPENDIX A : GEOMETRY FOR PLASTIC ARTERIAL CATHETER IN GAMBIT



APPENDIX B : GEOMETRY FOR PLASTIC ARTERIAL CATHETER IN FLUENT (WITH PATCHED BLOOD CLOT)