CFD SIMULATION OF NON-NEWTONIAN EFFECT ON HEMODYNAMICS CHARACTERISTICS OF BLOOD FLOW

SITI HAJAR BINTI ZAINUDIN

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

CFD SIMULATION OF NON-NEWTONIAN EFFECT ON HEMODYNAMICS CHARACTERISTICS OF BLOOD FLOW

SITI HAJAR BINTI ZAINUDIN

This report is submitted
in fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering with Honours

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2019

DECLARATION

I declare that this project report entitled "CFD Simulation of non-Newtonian Effect on Hemodynamics Characteristics of Blood Flow" is the result of my own work except as cited in the references

Signature : ....

Name : Siti Hajar Binti Zainudin

Date : ....

SUPERVISOR DECLARATION

I hereby declare that I have read this project report and in my opinion, this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Honours.

Signature	:	…................................
Name of Supervisor	:	Dr. Mohamad Shukri Bin Zakaria
Date	:	…................................

ACKNOWLEDGEMENT

It would not have been conceivable without the sort of help and genuine direction of numerous people and that have been involved directly or indirectly during Projek Sarjana Muda (PSM) period. I might want to stretch out earnest gratitude to every one of them.

First of all, I want to express my most profound thankfulness to my chief, Dr. Mohamad Shukri for giving me the opportunity to do a final year project with him. He generally gives me advise and give a direction whenever I faced issues or problem. I am grateful for his understanding and guidance that he gives to me through this project. I want to express my gratitude to my course mates for giving me their help, tolerance, and consolation. At last, I want to thanks to my family for their persistent help and support.

ABSTRACT

Computational fluid dynamics (CFD) is a part of engineering in the mechanical field to study the pattern flow of fluid, heat transfer and anything that related to the operation of computer-based simulation. This project is to study Newtonian and non-Newtonian model of viscosity consequence on hemodynamic characteristics when blood flows through the nozzle by using CFD simulation. For the current study, it focusses on low Reynolds number (laminar) only. The current study focusses to compare between Newtonian and non-Newtonian and select which model is close to blood behavior. The results for this study will be on the axial velocity along the centerline of the nozzle, wall shear stress and pressure. For the validation process, a previously published paper was selected to performed test for validation by using Computational Fluid Dynamics software ANSYS FLUENT. After validation is done, it can be concluded that the work shows in this report served as early goals to provide simulation to use patient data by using implanted in-body medical devices that can obtain from CT scans or MRI technique. Hence, this report shows that CFD predictions can be validate and verify by axial velocity along centreline of the nozzle. Axial velocity for Newtonian viscosity is higher than non-Newtonian viscosity model. Besides that, Newtonian blood viscosity has a high number of wall shear stress and it will cause blood clots and thrombosis to the patient. When blood clot and thrombosis occurred, it can disturb the flow of blood from the recirculating system. When the process of transporting oxygen to the heart and the release of carbon dioxide from our body is being obstructed it can cause high blood pressure, high cholesterol and a heart condition, the rhythm of heart disturbance.

ABSTRAK

Dinamik bendalir komputasi (CFD) adalah sebahagian daripada kejuruteraan mekanikal yang diajukan untuk menganalisis aliran bendalir, pemindahan haba dan berkaitan dengan penggunaan simulasi berasaskan komputer. Projek ini adalah untuk mengkaji kesan Newtonian dan non-Newton pada ciri hemodinamik aliran darah dengan menggunakan simulasi CFD. Untuk kajian ini, ia memberi tumpuan kepada nombor Reynolds yang rendah (lamina) sahaja. Kajian ini memberi tumpuan untuk membandingkan antara Newtonian dan bukan Newtonian dan memilih model mana yang hampir dengan tingkah laku darah. Keputusan untuk kajian ini akan berada pada halaju paksi di sepanjang pusat muncung, tegasan dan tekanan geseran dinding. Untuk proses pengesahan, kertas yang telah diterbitkan sebelum ini dipilih untuk ujian pengesahan yang dilakukan dengan menggunakan perisian ANSYS FLUENT Computational Fluid Dynamics. Selepas pengesahan dilakukan, dapat disimpulkan bahawa kerja yang ditunjukkan dalam laporan ini berfungsi sebagai tujuan awal untuk menyediakan simulasi untuk alat-alat medis dalam tubuh yang diimplan dengan menggunakan data pesakit yang dapat diperolehi dari scan CT atau teknik MRI. Oleh itu, laporan ini menunjukkan bahawa ramalan CFD boleh diverifikasi dan disahkan oleh halaju paksi di sepanjang pusat muncung. Halaju aksial bagi halaju Newton adalah lebih tinggi daripada model kelikatan non-Newtonian. Selain itu, kelikatan darah Newton mempunyai banyak tekanan ricih dinding dan ia akan menyebabkan bekuan darah dan trombosis kepada pesakit. Apabila bekuan darah dan trombosis berlaku, ia boleh menyekat aliran darah dari sistem peredaran darah. Apabila proses mengangkut oksigen ke jantung dan pelepasan karbon dioksida dari badan kita yang terhalang dapat menyebabkan tekanan darah tinggi, kolesterol tinggi dan keadaan jantung, irama gangguan jantung.

CONTENT

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 Background

Computational fluid dynamics (CFD) is a branch of engineering field in mechanical for widely studying the flow pattern of fluid, heat transfer, and related studies with the use of computer-based simulation (Lee, 2011). Examples are streamlined features and hydrodynamics of vehicles, turbines for power generation plants, engineering for electronics, chemical engineering, external and internal environmental architectural design, marine, and environmental engineering. In the biomechanical field, CFD is still in a phase of developing technologies. The primary cause why CFD in the biomedical field has collapsed behind is the incredible complexity of human life structures and behavior fluid in the human body. Lately, biomedical research with CFD is more open since the technologies have great performance equipment and software product are effortlessly accessible with advances in software engineering. This project aims to study the Computational Fluid Dynamic (CFD) Simulation of Non-Newtonian consequence blood flow on hemodynamics characteristic.

CFD is generally committed to liquids that are in movement, and how the liquid flow performance impacts process. Furthermore, physical attributes of fluid movement can ordinarily be defined through the basic mathematical equation, normally in fractional form, which governs the procedure of interest and is regularly named as governing equations. CFD has been used to created or analyze the medical device (Borse & Giri, 2016), (Izraelev et al., 2009). The mathematical equation is tackled by being changed over by computer scientist utilizing service of high-level computer programming languages. The calculations reflected the investigation of fluid pattern through simulation of numerical, which includes utilizing high-speed digital computers to achieve numerical solutions. Utilizing CFD, medical researchers can pick up expanded information on how body fluids and systems parts are expected to perform, to make the essential upgrades for bio-liquid physiology studies, and to make a medical device (Vierendeels, 2007). CFD gives suggestions for simulations before a genuine responsibility is attempted to implement any medicinal design alteration and may give the right direction to create medical involvements.

The FDA interlaboratory study (Paterson, Burgreen, Hariharan, Food, & Day, 2012) state that an idealized medical equipment must have of a small nozzle, which can contribute same characteristics as others blood-carrying medical devices, such as blood tubing, hemodialysis sets, catheters, cannulas, syringes, and hypodermic needles. The nozzle device was made to contain the blood flow with accelerating flow, decelerating flow, variations in shear stress, velocities, recirculating flow and all that may be related to blood damage in medical devices. These sizes of the nozzle features make complex flow phenomena including adverse pressure gradients, recirculating flow, and high and low shear stress area, which have same pattern features of cardiovascular devices such as found in the diffuser areas of blood pumps and ventricular support equipment (Hariharan et al., 2017).

In early research, the research found that the blood is a Newtonian fluid, yet blood vessel stream and stress pattern can be influenced by rheological properties of the blood. Anyway, in late research (Gijsen, Vosse, & Janssen, 1999) found from their examination on bloodstream in a carotid bifurcation demonstrate the speed dispersion of stream could be influenced by shear diminishing the characteristics blood for non- Newtonian viscosity. Based on variable applied stress or force, fluid viscosity for Non- Newtonian fluid can be determined. The physical behaviour of non-Newtonian fluid depends on the forces acting on it from time to time.

1.2 Problem Statement

Every researcher and industry of the medical device keep doing research about the medical device to produce the latest technology capable of addressing the problems faced by the previous medical device. The existing medical device for the hemodynamics of blood flow causes blood to clot (Zakaria et al., 2017). Hemolysis and thrombosis can happen when blood flowing through medical devices. The harmful effects of high shear stresses on blood have been widely known for many years. Relationship between hemolysis and time often stated that shear stress/exposure time relationship, which shows that increasing shear stresses can only be endured for short times before blood damage occurs, while decreasing shear stresses can be endured for longer times (Paterson et al., 2012). Therefore, the effect of hemodynamic characteristics of blood flow under CFD simulation will be a focus on this project

Computational fluid dynamics (CFD) is normally utilized for medicinal device advancement, its helpfulness for exhibiting device safety has not been proven yet. Dependable institutionalized strategies for detailed need are lacking and are constraining the use of computational methods in the controlling review of medical devices. To address this issue, participants from academia, industry, and the U.S. Food and Drug Administration recently finished a computational interlaboratory study to determine the suitability and approach for simulating fluid pattern in an ideal medical device (Paterson et al., 2012). This project will be studying the relationship between FDA nozzle and blood clot.

Regardless of whether the presumption that blood can remain demonstrated as a Newtonian fluid is permissible is under debate. A few numerical studies show that the impact of shear diminishing properties of blood is not effective for the low in large arteries (Gijsen et al., 1999). When to improve or change the current medical device to the new one, many aspects need to analyses and to consider. Since the technology still growing in this field, many parameters need to be considered in this research such as Non- Newtonian fluid. The representative of Non-Newtonian will be studied in this research.

1.3 Objective

The research focuses on the CFD simulation of Non-Newtonian effect on hemodynamics characteristics of blood flow. The main objectives of this study are:

i. To study the effect of Hemodynamics characteristic of blood flow in the FDA Nozzle.

ii. To simulate Newtonian and Non-Newtonian

iii. To compare between Newtonian and Non-Newtonian effect on hemodynamics characteristics of blood flow and suggest which model close to blood behavior.

1.4 Scope of Project

The scope or limitation of this project will be listed in this project. The first scope is, this paper state that condition for this blood flows through the nozzle to be in low Reynold number which is Re=500 in the laminar regime. This is because when to study the non-Newtonian performance (non-constant viscosity) the condition must be in low Reynold number. Secondly, using non-Newtonian performance on the viscosity of blood is the Carreau-Yasuda. This model was chosen because of the ability to keep the rise in blood viscosity when at low shear rates. This model also has a steady conversion role between low viscosity at the high shear amount and the high shear amount at decreasing shear rates. Thirdly, the numerical simulation that solves and provide the answer for behavior of the blood when the mathematical model is too complicated to solve it. Specifically, this paper used the Navier Stokes equations for numerical simulation. Navier Stokes equation increases the opportunity to add the viscosity of the blood. Lastly, the scope of this paper is the condition of the bloodstream. In this paper, the condition of the bloodstream will be transient flow.

CHAPTER 2

LITERATURE REVIEW

2.1 Blood Properties

Blood is a part body fluid that flows in individual and other creatures that can transfers essential substances, for example, supplements and oxygen to the cells and carriage metabolic waste things from those equivalent cells. Blood is coursed around the body through veins by the drawing activity of the heart. Creatures function for lungs, transfer oxygen through blood vessel blood from breathing in fresh air to the tissues of the body, and venous blood carries carbon dioxide, a waste result of absorption delivered by cells, to be breath out from the lungs. Many types of research define blood is modeled as an incompressible fluid, which kinematic constraint on velocity profile from (dynamics) continuity equation. However, to obtain a profile for the pressure it will go through difficulty process (J. H. Ferziger, 2002) specifically when running for finite-difference or finite volume method.

A different method is to assumed blood as a slightly compressible method, where the continuity equation was applied to this. An equation of state can calculate the value for pressure and from the continuity equation, it can determine the value for the density of blood. By following the FDA's regulation blood is with a density of 1056 kg/m^3. It is simulated with a dynamic viscosity of 0.0035 N.s/m^2 for Newtonian fluid as per the guidelines from the late research (Stewart, 2011). References value for pressure is 1200 Pa (Min et al., 2014). The setting for boundary condition for the slightly compressible fluid model is recommended for non-reflecting boundary condition. By evaluating separately the outgoing and incoming over the boundaries were the results that obtain from non-reflecting boundary conditions (Yoo, Wang, Trouvé, & Im, 2005)

2.2 Newtonian vs Non-Newtonian

Newtonian fluids are the stream performance of fluids with a simple linear relation between shear stress [mPa] and shear rate [1/s]. Newtonian fluids the stress feedback to a basic shear rate is linear. The linearity steady μ relies upon the thermophysical states of the fluids (temperature and pressure) (Aldi, Buratto, Pinelli, Ruggero, & Casari, 2016).

Water, natural solvents, and liquor are some case of Newtonian liquids. Those fluids viscosity is just reliant on temperature. Based on the graph of shear stress versus shear rate (See Figure 1) we can see a straight increment in stress with expanding shear rates, where the grade is given by the viscosity of the liquid. This infers the viscosity of Newtonian liquids will remain an unfaltering (see Figure 2) regardless of how fast they are constrained to course through a pipe or channel (for example viscosity is free of the rate of shear).

An exception to the standard is Bingham plastics, which are liquids that require the least stress to be associated before they stream. These are carefully for non- Newtonian, yet once the stream starts, they act fundamentally as Newtonian fluids (for example shear pressure is direct with shear rate). Mayonnaise is one of the instances of this sort of conduct.

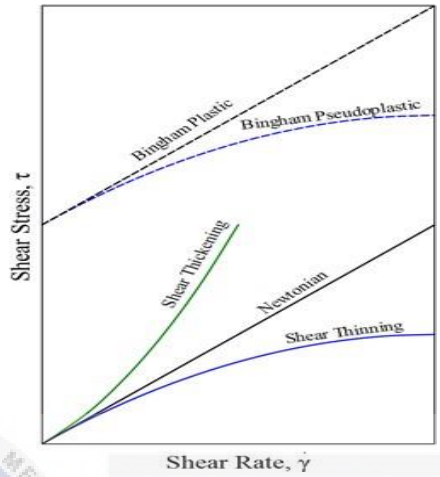

Figure 2.1: Shear stress as a component of shear rate for several kinds of fluids

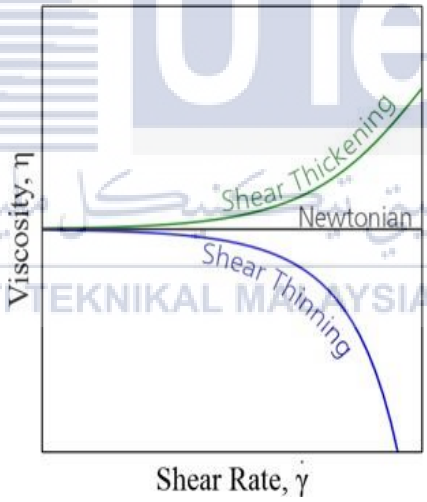

Figure 2.2: Viscosity of Newtonian, Shear Thinning and Shear thickening as an element of shear rate

In presence, most liquids are non-Newtonian, which infers that their viscosity is liable to shear rate (Shear Thinning or Thickening). As opposed to Newtonian liquids, non-Newtonian liquids show either a non-straight association between shear pressure and shear rate (see Figure 1), have yield stress, or viscosity that is needy to time or distortion history (or a blend of all the above mentioned).

The non-Newtonian liquid dealt with is a shear-diminishing fluid (the conspicuous viscosity reduces with increment stress) (Aldi et al., 2016). A liquid is shear thickening if the viscosity of the liquids augments as the shear rate increment (see Figure 2). A typical instance of shear thickening liquids is a blend of corn starch and water. Fluids are shear diminishing if the consistency lessens as the shear rate increases. Shear diminishing liquids, generally called pseudo-plastics, are universal in modern and organic procedures. Regular models incorporate ketchup, paints, and blood (Boyd, Buick, & Green, 2007).

Non-Newtonian sort of liquids can be brought about by a couple of components, all of them identified with structural reorganization of the fluid's molecules because of flow. In polymer melts and courses of action, it is the plan of the exceptionally anisotropic chains what results in a diminished viscosity. In colloids, it is the detachment of the various stages in the stream that causes a shear diminishing conduct.

The complex nature of the blood flows in the individual body is a principal cause for concern for the accuracy of the outcomes of bloodstream simulation. For the simulation of the human right arteries supply routes, (Johnston et al., 2004; Chen and Lu, 2006; O'Callaghan et al., 2006) demonstrated that "when studying the wall shear stress distribution for transient bloodstream in arteries, the application of a Newtonian bloodstream model is a reasonably good approximation. However, to study the stream inside the artery in greater detail, a non-Newtonian model is more appropriate". The non-Newtonian features are commonly validated at low shear rates which can exist close bifurcation sites and at recirculation zones creating in the arteries. Since the generation of atherosclerosis, i.e. atherogenesis, has been related with low (< 0.5 Pa) WSS values, it turns out to be certain that non-Newtonian models should be utilized to acquire more reliable estimates. Many types of research about blood have been simulated as a Newtonian fluid, however, due to rheological properties of blood, it will effect arterial flow and stress pattern (Gijsen et al., 1999) find that shear diminishing of non-Newtonian can be damaged by velocity transportation.

Blood is a non-Newtonian liquid, particularly at low shear rates. A couple of studies show that the numerical aftereffect of divider shear pressure (WSS) circulation in a vein might be unmistakable when blood was dealt with in an unexpected way (for example Newtonian or non-Newtonian), especially at low shear rates (Johnston et al., 2004; Chen and Lu, 2006; O'Callaghan et al., 2006).

Non-Newtonian models have been used with rising recurrence for stream simulation of other vascular segments, for instance, the aorta, the cerebral, and the coronary arteries. In every one of these cases, summed up Newtonian expressions are grasped which can more likely catch the complex behavior of the fluid under steady state condition. Regardless, while there is strong confirmation to demonstrate that the Carreau-Yasuda model is the most proper among constitutive equation for the depiction of bloodstream reliably, repeating the two its yield pressure and shear diminishing viscoplastic qualities, specialists routinely use various models, for instance, the Carreau–Yasuda or a general power law. The distinctions in the forecasts of different rheological models are introduced in a comparative study of (Karimi et al., 2014)

2.3 Medical Device Replicate

Hemolysis (the elimination of red blood cells) and thrombosis (development of blood clot) can be found in the medical device. For a long time of studies, the harmful of high shear stress has been knowing (Sutera, 2015). The connection between shear stress and exposure time always linked each other for hemolysis. Blood damage can happen in a short time when the blood in high shear stress, while for long time duration low shear stress blood damage can tolerate with it (Maruyama et al., 2005).

The problem that has been identified causes blood harm in the medical equipment was modeled to incorporate accelerating flow, decelerating flow, varieties in shear stress and velocities, and recycling stream. The hemodialysis sets, catheters, cannulas, and syringes are blood contacting equipment and the nozzle has an attribute like blood contacting device. Geometrically, the model incorporates a progressively convergent section, a throat area and a sharp-edged of sudden extension (Hariharan et al., 2017).

Participants in FDA study were given a choice to decide flow solver, mesh density, element shape, inlet/outlet, length, boundary condition details, and laminar or, turbulence models while model dimensions, volumetric flow rates, and fluid properties were stated by FDA (Toma & Toma, 2018). For the improvement of the safety of the nozzle, the ratio of hemolysis is an important aspect to consider. Blood cell rupture causes due to high shear stress was called hemolysis. The previous study has been done for medical device by experimental, simulation and experiment for blood damage (Paterson et al., 2012), (Fallon, Dasi, Marzec, Hanson, & Yoganathan, 2008), (Tremmel et al., 2010), (Vierendeels, 2007), (Tamagawa, Kaneda, Hiramoto, & Nagahama, 2009).

The dimensions of the nozzle geometry were achieved from the published literature (Stewart, 2011). The proposed idealized nozzle model comprises of four divided area comprising features of some blood flow on medical devices. The inlet and outlet tubes were stated with diameter 0.012 m and a nozzle throat with diameter 0.004 m followed by a sudden contraction and 20° conical diffuser. There is additionally a cone-formed tube interfacing the channel tube with the nozzle throat. For this geometry, flow in one direction is a sudden expansion issue, while flow in the opposite direction is a conical diffuser issue.

Non-Newtonian impacts on the axial velocity are little in the district upstream the sudden expansion, yet they end up being more progressively downstream that point, in the distribution area. The Newtonian model underestimates of blood viscosity at low shear pressure rates, or, in other words why Newtonian velocity is higher than the one procured with interchange models (Min et al., 2014).

The flow encounters a gradual contraction of zone from the inlet area to the throat area, at that point a sudden expansion relating to laminar will be examined

2.4 CFD for Blood Flow

In spite of the way that the U.S. Food and Drug Administration (FDA) does not require CFD reproductions in the appraisal of blood-reaching medical devices, overall standard for heart valves and implantable circulatory assistance device perceive that experimentally approved for CFD simulations may be connected to describe the stream fields inside and around these gadgets, and to assess their hemolytic and thrombogenic conceivable outcomes. Presently, the computational technique can be used to resemble both the solid and the fluid mechanics of a device, the transport of blood elements, and the transport and chemical reactions of molecular species. In addition, with increasing simulation speeds and computational capacities, CFD techniques can be utilized to evaluate an expanded variety of physiologic and design parameters and can possibly reduce the extent of animal testing and clinical preliminaries. One of the essential clarifications behind the limited use of CFD (and other physics-based simulations) is the test of satisfactorily setting up the ability for making safety claims (Hariharan et al., 2017).

CFD is a generally embraced methodology for solving complex issues in various advanced engineering fields. The value of CFD is developing new and better- quality devices and systems designs, and upgrading is conducted on the existing device through computational simulations, resulting in upgraded efficiency and lower working expenses. These days, CFD biomedical research is more reachable, since high-performance equipment and software are effectively available with advances in programming designing. All CFD procedure contains three fundamental components to give helpful data, for example, pre-preparing, solving mathematical equations, and post-processing. Initial accurate geometric modeling and boundary conditions are fundamental to acceptable results.

The most generally common type of validation approach found in the medical device writing is the difference of mean data from validation experiments with a single representative simulation. Normally, the examination between this information is subjective and statements like "sensible agreement" are utilized to depict the correlation. This technique does not give enough validation to a majority of operation cases since variety and vulnerability in the input parameters, for example, device geometry, fluid properties, and inlet and outlet boundary conditions, have not been expressed but rather are crucial for duplicating the approval tests.

CHAPTER 3

METHODOLOGY

3.1 Introduction

The methodology flowchart which appeared in figure 3 represents the flow of procedures which has been taken for the progressing of the project in PSM 1. For the starting phase of PSM 1, the struggle is to focus on the literature review which is related to the FDA nozzle and CFD simulation. Since the experiment is not cover in this research, so the validation outcome is done by associating it to the closest published journal by comparing the axial velocity in the nozzle under the same parameter. For the preprocessing, geometric of the nozzle is drawn by using CATIA V5R20 software with the same parameter of the closest journal and then being import into the Ansys workbench 16.0 software for creating meshing. Next, for the solver process, a number of parameters set for the simulation are to be done as required. After the validation process is completed, the simulation is repeated by changing the density of the blood by using a non-Newtonian equation. The result for both viscosity model are deliberated and recommendation the closest viscosity is close to blood behavior and increase the efficiency of the nozzle. Table 2 shows the flow of procedures which had taken in PSM 2.

3.1.1 Gantt chart

i) Gantt Chart

Table 3.1: Gantt Chart for PSM 1

Week **Activities**		2	3	4	5	6	7	8	9	10	11	12	13	14
Research Title Selection	■	■												
Literature review	■	■	■	■	■	■	■	■	■	■	■	■	■	■
Design of model (FDA geometry)	■	■	■	■										
Mathematical formulation				■	■	■	■	■						
Meshing							■	■	■	■	■			
Numerical setup									■	■	■			
Report writing										■	■	■		
Report submission													■	
PSM 1 Seminar														■

Table 3.2: Gant Chart for PSM 2

Week / **Activities**	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Literature review														
Numerical setup for Non- Newtonian blood properties														
Report writing														
Report submission														
PSM 2 Seminar														

3.1.2 Flow Chart

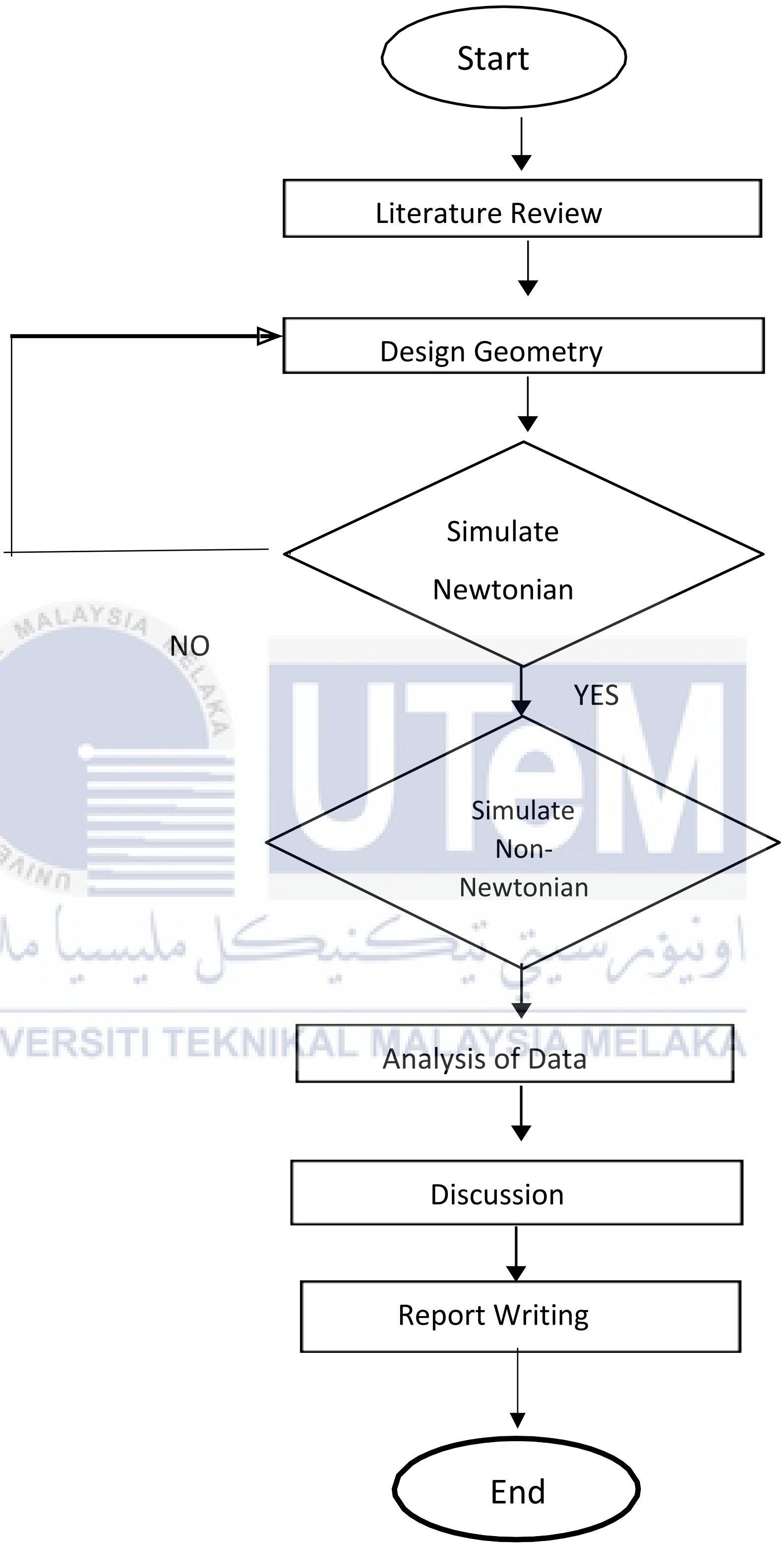

Figure 3.1: Flow Chart of the methodology

3.2 Mathematical model

3.2.1 Governing Equation

All flows of the Conservation Equations for Mass, Momentum and Transport equations for fluid in each flow geometry can be determined numerically by FLUENT. This case, the governing equations which involved continuity equation and momentum equation are introduced.

The continuity equation is used to determine the conservation equation of mass. The equation for the continuity equation is reported as

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

In this study, the model uses a momentum equation with pressure-based solver.

The momentum equation of the flow field is given by:

$$\frac{\partial u}{\partial} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + v\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) + f_x \quad (2)$$

$$\frac{\partial v}{\partial} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + v\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial x^2}\right) + f_y \quad (3)$$

$$\frac{\partial w}{\partial} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + v\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) + f_z \quad (4)$$

Where ρ is Density, v is Velocity and ν is Kinematic viscosity

In connection for ideal fluids is use direct Euler equations, Navier-Stokes equations include the possibility of including the viscosity of the fluids. In the specific case of hemodynamic, viscosity is an important part; additionally, the viscosity coefficient can't generally be thought to be consistent, as blood essentially has a non- Newtonian behavior.

3.2.2 Non-Newtonian model

Blood is having complicated behavior of fluid ("Flow-Dependent Rheological of Blood in Capillaries," 1987). A realistic and quite simple model of non-Newtonian effects on blood viscosity is the Carreau-Yasuda (CY) model. This model considers the rise of blood viscosity at low shear rates, including a smooth transition function between the high viscosity value at low shear rates and the low one at high shear rates (Boyd et al., 2007). In particular, the viscosity model reads as follows:

$$\mu = \mu_\infty + (\mu_0 - \mu_\infty)[1 + (k\,|\dot{\gamma}|)^{a}]^{\frac{n-1}{n}} \tag{5}$$

Where $|\dot{\gamma}| = \sqrt{\frac{1}{2}\dot{\gamma}_{ij}\dot{\gamma}_{ij}}$ is the effective shear stress rate; $\dot{\gamma}_{ij} = (\partial_i u_j + \partial_j u_i)$ is the shear stress tensor, u_j being the velocity field; μ_0 and μ_∞ are the viscosities at zero and infinity shear rates; and {k,n,a} are constant parameters. Using calibrated results for blood from [18], we have that: μ_0 =0.16 Pa.s, μ_∞ =0:0035 Pa.s, k =8.2s, a =0.64, and n =0.2128. The Newtonian approximation consists in using a constant viscosity value, equal to µ μ_∞ =0.0035 Pa.s.

3.3 Validation

The study of (Min et al., 2014) is selected as acceptance references. Table 3 shows the parameter of the FDA nozzle model that described an idealized medical device; it is shown in Fig 4 and 5 with all suitable dimensions. A 3D geometric model of the nozzle was generated using the commercial geometry creation software package (Design Modeler v.16.0, ANSYS). Full data for the previous past of nozzle study, geometry and experimental data can be retrieved from https://fdacfd.nci.nih.gov/.

Table 3.3: Parameter of the simulated nozzle by (Min et al., 2014)

Parameters	**Dimensions**
Inlet diameter, Di	**12 mm**
Throat diameter, Td	**4 mm**
Throat length, Tl	**40 mm**
Conical angle, Ca	**20°**
Conical length, Cl	**22.69 mm**
Exit diameter, Do	**12 mm**

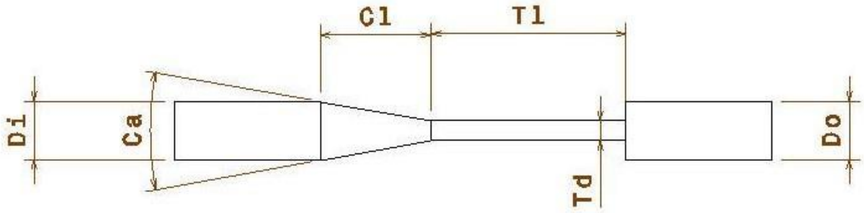

Figure 3.2: The dimension of the nozzle

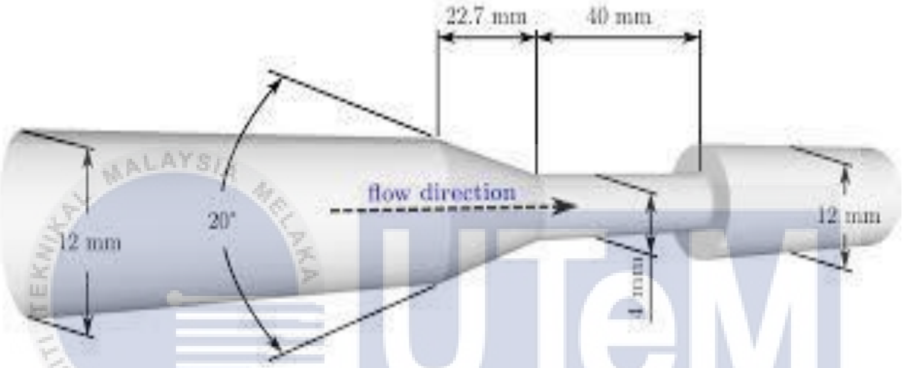

Figure 3.3: The 3D geometry of the nozzle

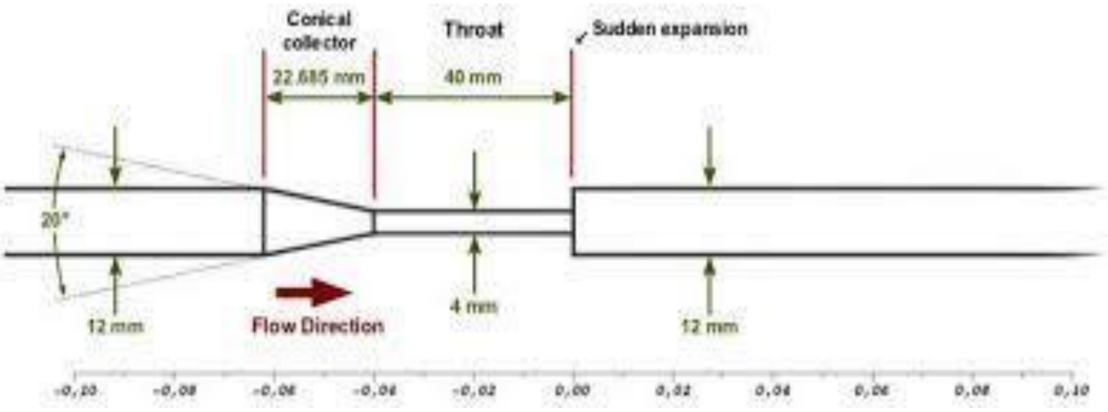

Figure 3.4: Specification for nozzle geometry (Min et al., 2014

3.4 CFD simulation

The instruction to handling simulation by using CFD must have three processes which are required in arrangement to be achieved which are pre-processing, solver, and post-processing.

3.4.1 Pre-Processing

In the pre-processing stage, the activities required to perform is geometric creating using CATIA V5R20, create boundary name and creating a mesh. Figure 3.3 shows the flow chart of the action in pre-processing.

3.4.1.1 **Boundary Name Defining**

The geometry of the nozzle was identified by construct name selection for inlet and outlet in order to clarify the procedure and future work. Table 4 and Figure 7 show the selection name for the part of the nozzle.

Table 3.4: Part selection

Part	A	B
Name selection	inlet	outlet

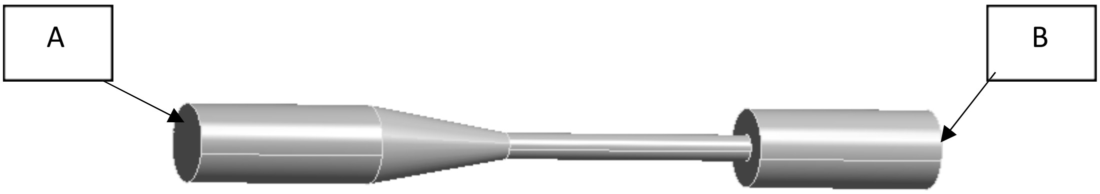

Figure 3.5: Computational domain of nozzle

3.4.1.2 Grid Generating

Pre-processing continues with grid creating after describing the name selection for each part. In the present study, fine significance center is selected to provide more certainty results. For meshing type, hexahedral is used. The total of meshing elements get in this present study is 314807 and the number of nodes is 333870. In numerical computation, the quality of the mesh plays an important role to control precision and constancy. The mesh quality attribute with mesh quality is skewness and aspect ratio. The maximum number skewness of this nozzle is 0.55917 and maximum number for aspect ratio is 8.7722. Figure 7 shows the front sight of the meshing model and figure 8 shows the inlet sight of the radial section of the meshing.

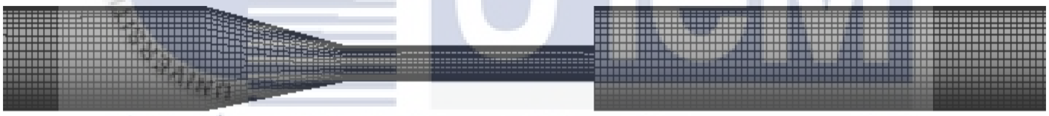

Figure 3.6: Front view Meshing Geometry

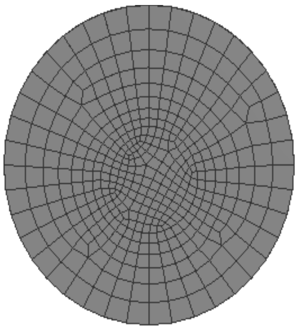

Figure 3.7: The inlet view of the radial section of mesh

3.4.2 Solver

For the solver setting, the time is set as steady as the fluid flows are assumed to be constant and incompressible. Due to this simulation only involved blood flow in the nozzle, therefore except for continuity equation, other equations are turned off. For the continuity equation, since the present study is focused on laminar flow, therefore the laminar equation with common wall function is turned on in this simulation. The properties of blood used in the present study are 1056 kg/m^3 and mass flow rate as 5.21 x 10^{-6} m^3/s.

The wall is treated as a static, non-deformable, inflexible body. Fluid interaction gets through the no-slip condition for viscous fluids, which express that at a solid boundary, the fluid will have zero velocity near to the wall. The density and pressure fields must be attuned accordingly. In precise, we must use the characteristics breakdown of the Navier-Strokes equations in order to separate between the received and departing characteristics fields, and only alter the approaching ones in such a way that the no-slip conditions are being fulfilled. For the two pressure of the inlet and outlet is set as zero pressure and outlet is set as 1200 pascal. The boundary condition of the following study as follows (Min et al., 2014)

Table 3.5: Parameter for boundary condition

Inlet	Flow rate= 5.21 x 10^{-6} m^3/s Reynolds Number= 500
Outlet	P =1200 Pa
Wall	Wall motion =Static wall Shear Condition =No slip boundary condition

The Semi-Implicit Method for Pressure-Linked Equations calculation is utilized for coupling pressure and velocity. For the spatial discretization, fewer squares cell-based is utilized for the angle and typical for the pressure. For the momentum, the second order upwind method is utilized.

The calculation is the last phase of the solver. The number of iterations can be expanded to a higher number in the following cases to get an increasingly precise outcome. The convergence criteria for each condition are set to 1 x 10^{-5} and are watched all throughout the calculation to ensure the solution come to convergence.

3.4.3 Post-Processing

After the calculation completed, the simulation result will be considered in both graphically and numerically. Axial velocity versus z-coordinate will be plotted to investigate the relationship between kinematic viscosity with the density of blood.

The number of element and node are critical to defining the accurateness of the results that gained. The excessive element will contribute to high computational consumption and longer estimation time spending, whereas the lacking number of elements will prompt to an incorrect result. By doing the grid independence test, a critical number of elements and nodes required obtaining the highest accuracy test, and less calculation time is required.

The simulation outcome gained from the current study is compared with the results of the previous work done by (Min et al., 2014) graphically for validity.

CHAPTER 4

RESULT AND DISCUSSION

Aim for this part is to explain the importance and effect of Newtonian and non- Newtonian blood response on different specification regenerate from CFD simulation, and validated the result obtained from a simulation with FDA's experiment result from their laboratory studies (Paterson et al., 2012).

4.1 Grid Independency Test

In order to conduct the grid dependency test, the number of nodes and element are being calculated until there is no difference as a previous result. After acknowledged grid generation, the independence test takes place before continuing to the last stage of simulation. The graph and result of grid independence test are shown in numerically in table 6. To conduct this test, three types of meshing was used such as coarse, medium and fine type. The number of elements for a coarse type of meshing the number of elements for 314807 and number of nodes 333870 for fine type meshing cell. To get more precise for simulation result, increasing factor rate was set to 1.20 and set to lower than the default value.

Table 4.1: Grid Independency Test

Number element of meshing	Axial velocity (m/s)
205617	0.173
314807	0.1703
617622	0.1523

4.2 Validation

4.2.1 Axial Velocity validation

This section is to validate the result that obtains from the present study with journal result. It is an important part to determine the success of the CFD simulation. The CFD analysis, solution method, meshing size, and laminar model were a parameter chosen to determine the success of CFD simulation. The important part of this validation to know the axial velocity flow through the centreline of the nozzle.

Figure 4.1 and 4.2 shows the validation result of the present study result and (Min et al., 2014) result. From validation result, there were slightly different between the present study and (Min et al., 2014) result. The difference between the present study and past study might be a few cases, for example, the geometry, and length of the nozzle, meshing size of the nozzle as the information were unable to gain in past study journal.

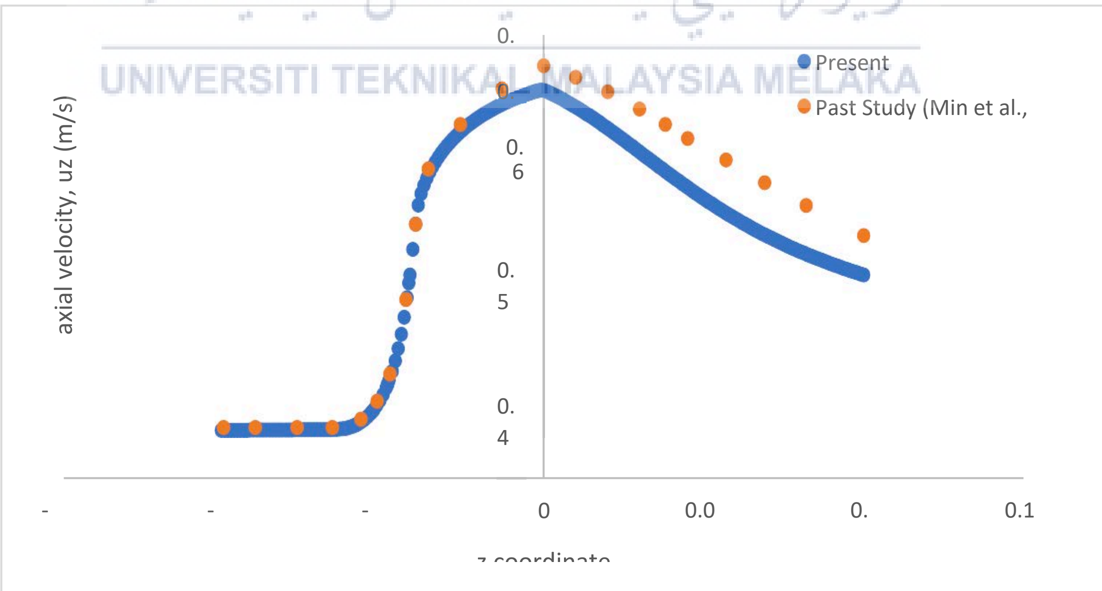

Figure 4.1: Axial velocity along the nozzle centreline for the present study and (Min et al., 2014) for Newtonian

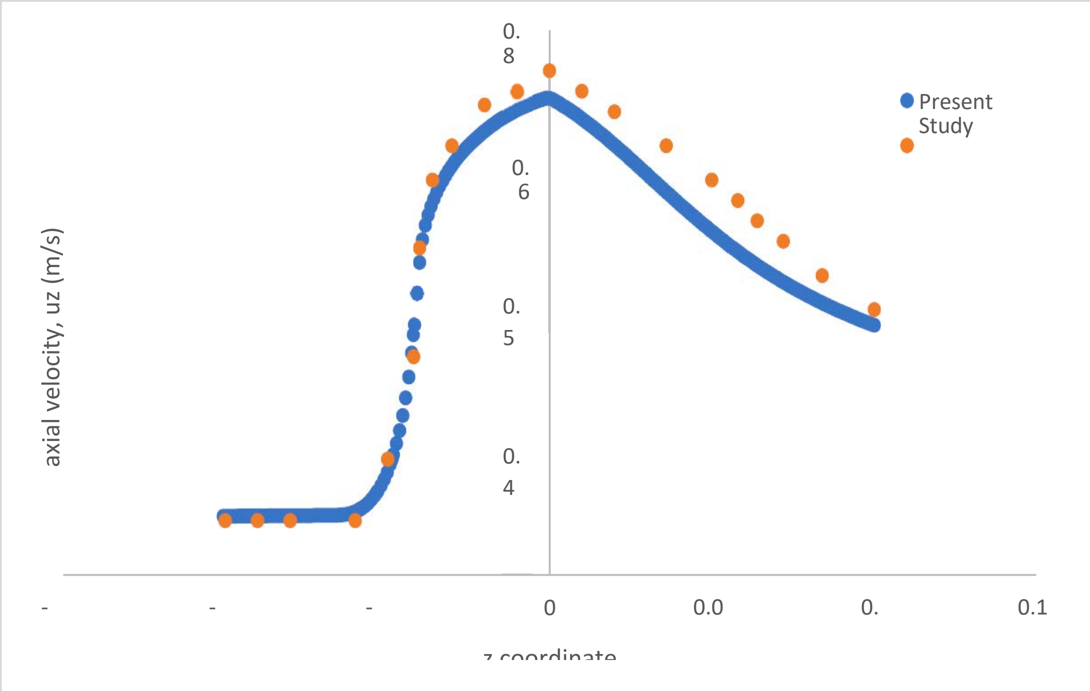

Figure 4.2: Axial velocity along the nozzle centreline for the present study and (Min et al., 2014) for Carreau-Yasuda

4.2.2 Wall shear stress (WSS) validation

This section is to validate the result that obtains from the present study with journal result. It is an important part to determine the success of the CFD simulation. The CFD analysis, solution method, meshing size, and laminar model were a parameter chosen to determine the success of CFD simulation. The important part of this validation to know the wall shear stress through the nozzle.

Figure 4.3 and 4.4 shows the validation result of the present study result and (Min et al., 2014) result. From validation result, there were slightly different between the present study and (Min et al., 2014) result. The difference between the present study and past study might be a few cases, for example, the geometry, and length of the nozzle, meshing size of the nozzle as the information were unable to gain in past study journa;

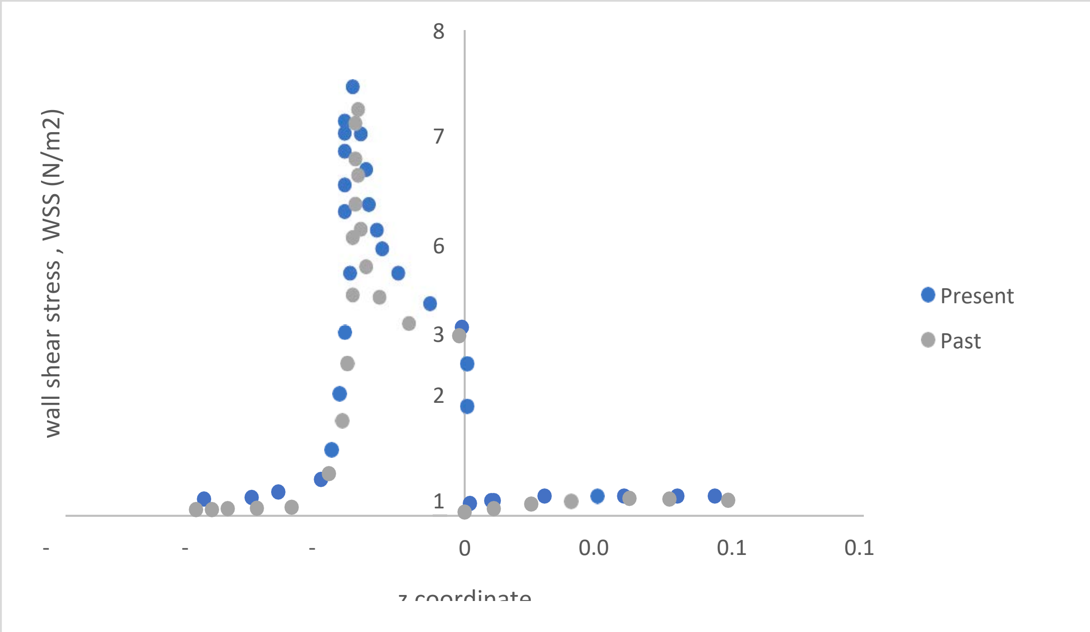

Figure 4.3: Wall shear stress of the axial position of the present study and (Min et al., 2014) for Newtonian

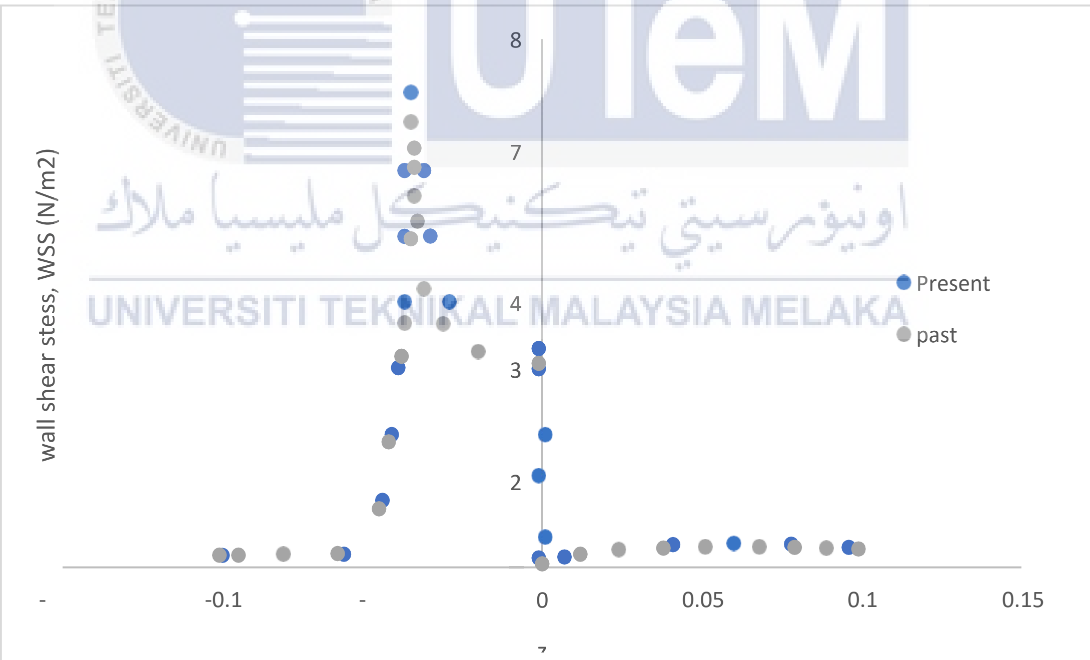

Figure 4.4 : Wall shear stress of the axial position of the present study and (Min et al., 2014) for non-Newtonian Carreau-Yasuda model

4.3 Axial velocity along the centerline

The axial velocity along centreline play an important part to validate and verify the assumption of CFD predictions. The results of axial velocity are on the Z coordinate axis. This is because the centreline includes the data from all different place and because FDA's experimental data for velocity were accurately obtained by using PIV approach. By using PIV approach, the flow was recognized to be fully laminar (Paterson et al., 2012). Figure 4.5 shows the results of CFD assumption for axial velocity along centreline with using 2 different viscosity models and attach with the result for axial velocity for experimental. Non-Newtonian result outcome for axial velocity in the inlet region of sudden expansion is small. However, after a sudden expansion region, the axial velocity has fallen continuously in the zone of recirculation. For Newtonian cases, the blood viscosity at low shear stress rate. This is the cause why the axial velocity for Newtonian is higher than others viscosity model. By differentiating with a Newtonian and non-Newtonian model that is Carreau-Yasuda data produce a difference in the axial velocity

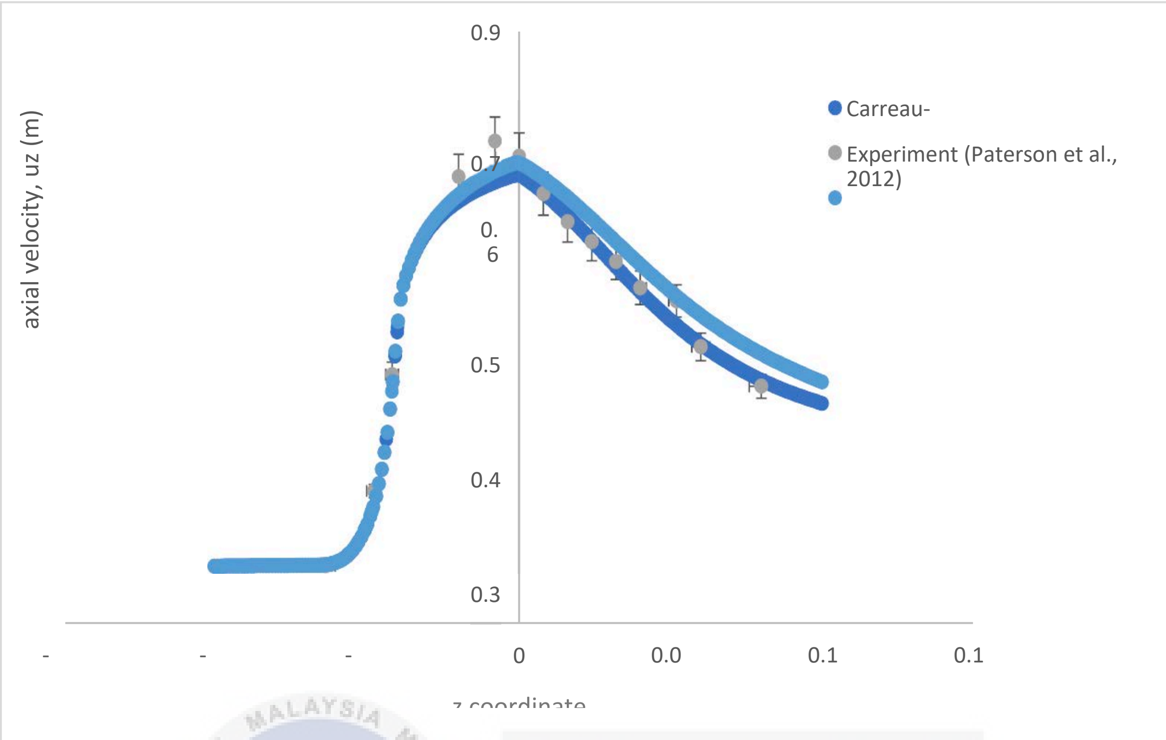

Figure 4.5: Axial velocity along the nozzle centreline for two different viscosity and experiment.

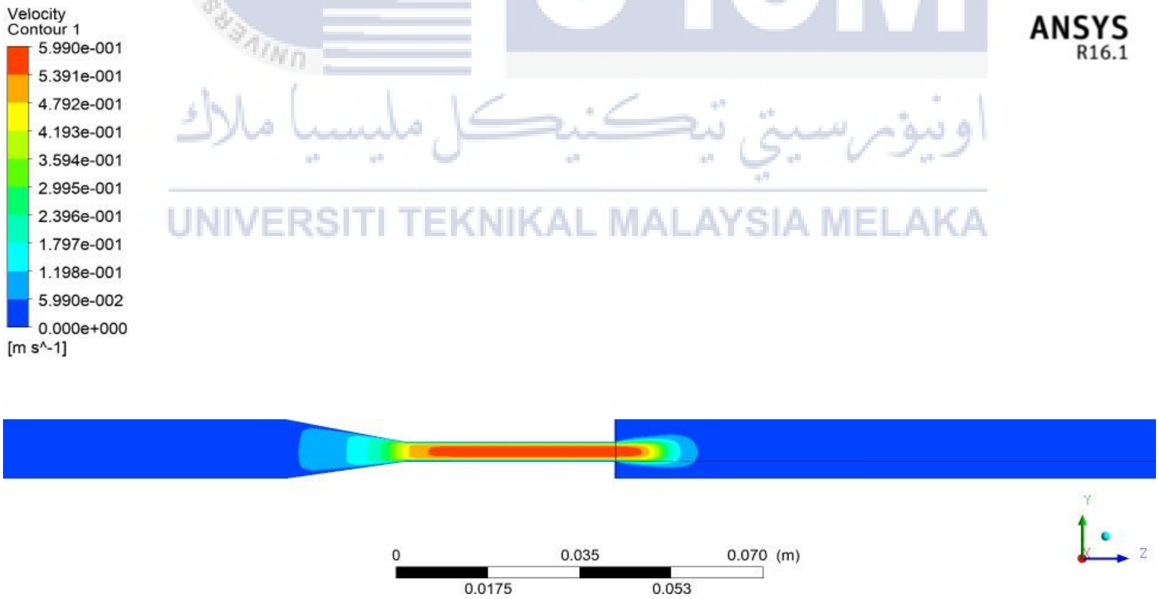

Figure 4.6: Velocity contour along the nozzle centreline for Newtonian viscosity

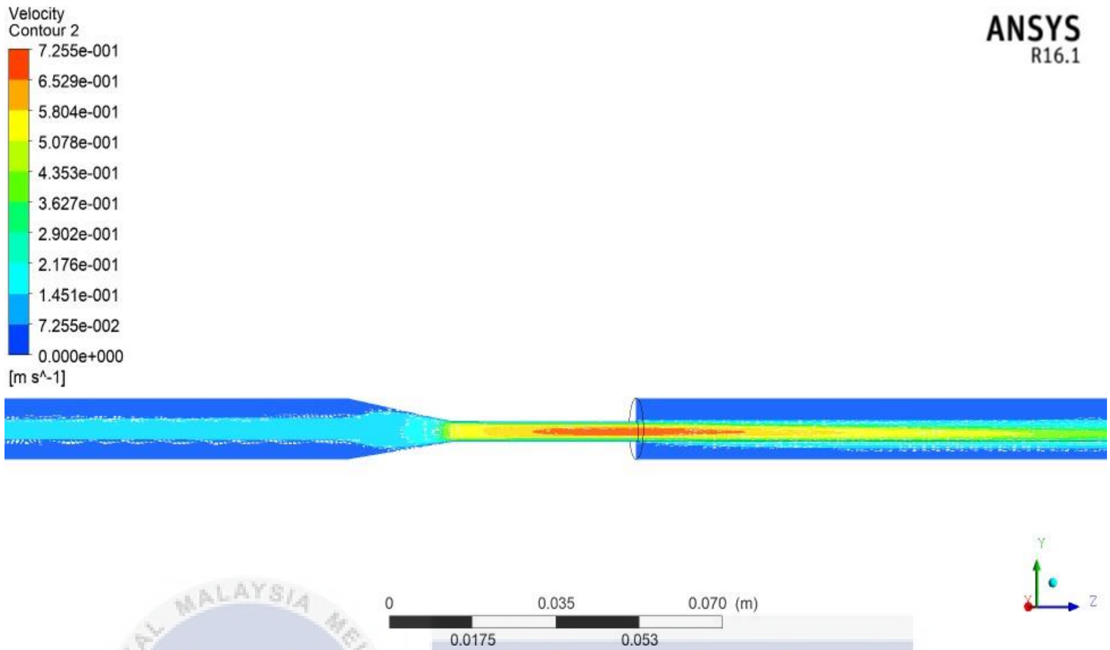

Figure 4.7: Velocity contour along the nozzle centreline for non-Newtonian Carreau-Yasuda viscosity

4.4 Velocity streamline

The fluid stream is identified by three-dimensional space in velocity vector range, within the structure of continuum mechanics. The result that obtains from the field line of the vector field called streamline. Streamline are the line that tangent with the velocity vector of the flow.

Figure 4.8 and 4.9 displays the velocity streamlines for the Newtonian viscosity model and non-Newtonian Carreau-Yasuda model. For the Newtonian model, the flow of the streamline is normal until it reaches after sudden expansion at the recirculation zone. The jet velocity that has been through the throat region was at the maximum of the velocity at the throat. The jet velocity was slow down after the sudden expansion region. For non-Newtonian model, jet velocity going past through the throat region and its flows through until the end of the sudden region.

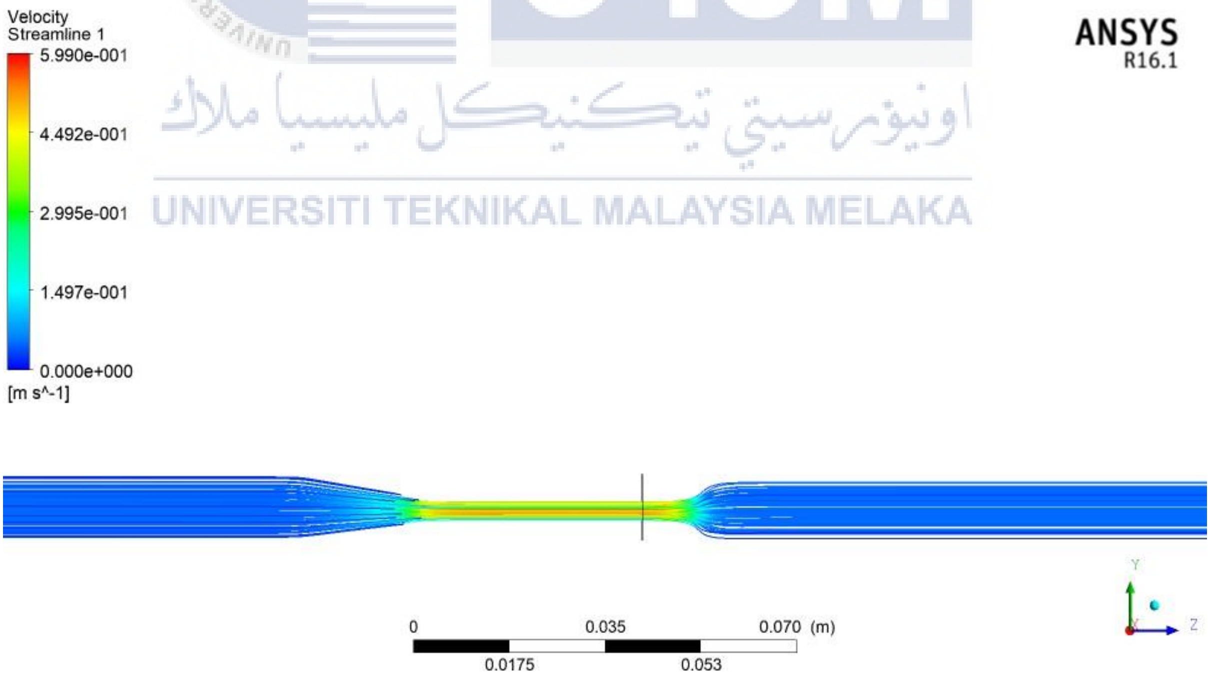

Figure 4.8: Streamline velocity for Newtonian viscosity model

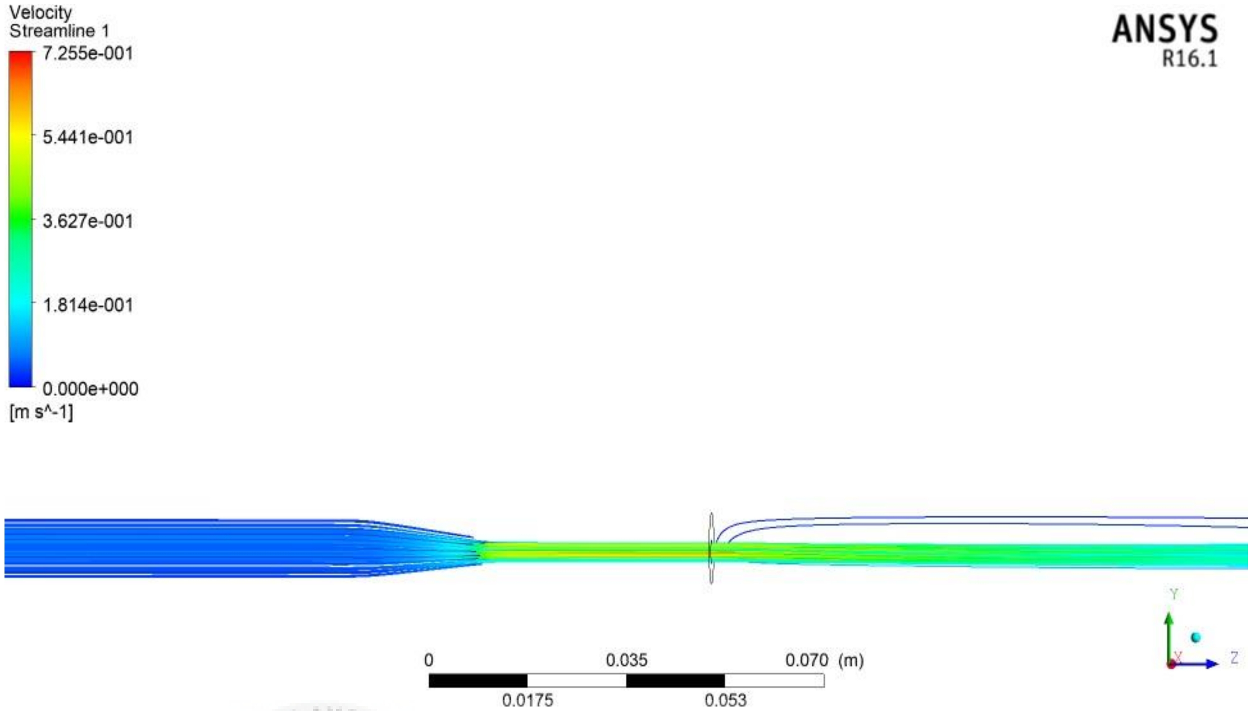

Figure 4.9: Streamline velocity for non-Newtonian model Carreau-Yasuda viscosity model

4.5 Pressure

When the researcher takes part in the FDA's trial, they used pressure drop along the centerline method to directly measure that obtained in experimental that they were running. In a medical perspective, this method was often used to detect the disease, for example, aortic valve stenosis. But in reality, with the actual patient, the pressure drop that happened along a valve or device was unable to precisely measured. Instead, by using Doppler techniques the physicians can get the velocity measurements and can estimate the pressure drop.

A sudden expansion region the expected pressure drops obtained for Carreau- Yasuda viscosity model is 323.74 Pa and 383.55 Pa for the Newtonian model. For the Newtonian model in figure 19, the lowest value for pressure is 1200 Pa and for the highest value for pressure is 1583.55. For the non-Newtonian model in figure 20, the lowest value for pressure is 1200 Pa and for highest value is 1523.74 Pa. In theoretical, by using the Bernoulli Equation, the value will get around 284.1-262.5 Pa. Based on this data, for this nozzle and the flow region, CFD forecast to predict pressure data have narrow effect for non-Newtonian effect. For every, miscalculate of pressure drop that happens to the patient can cause the potential stenosis (abnormal narrowing in a blood vessel) can be miscalculated as well. If the pressure drop is higher than what it should be, it could be harmful to the patient.

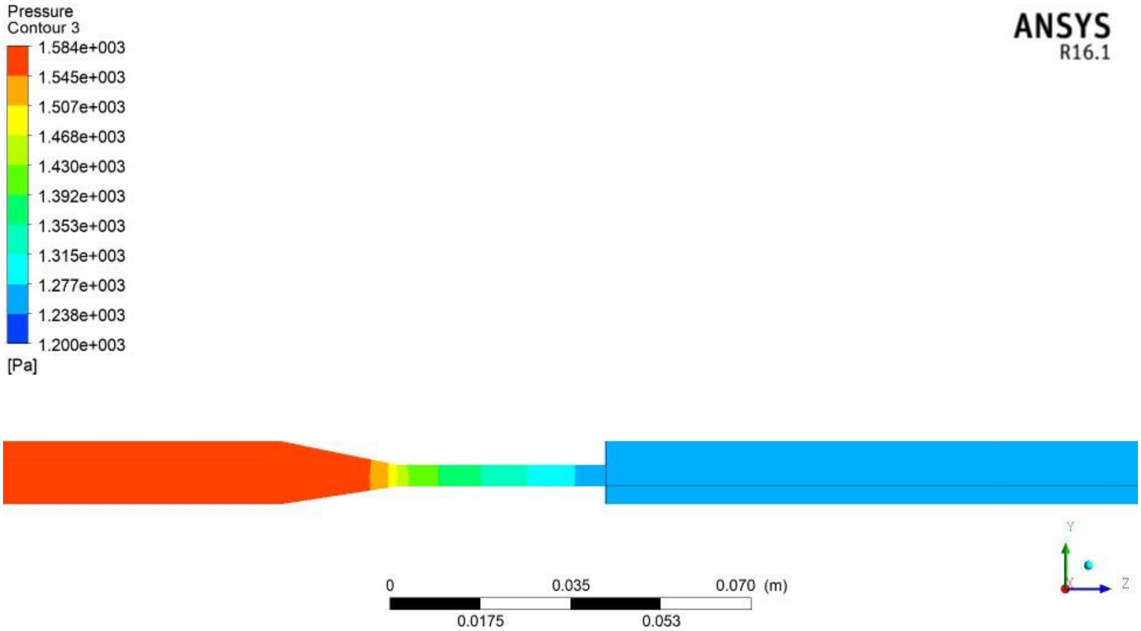

Figure 4.10: Contour for pressure Newtonian viscosity model

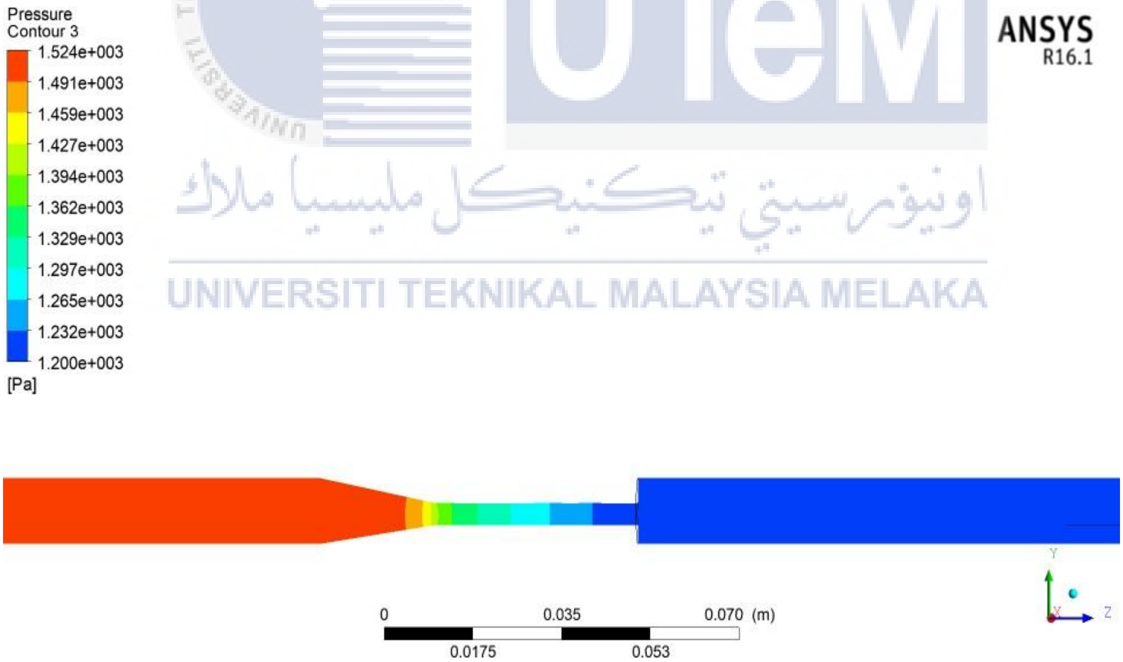

Figure 4.11: Contour for pressure Carreau-Yasuda viscosity model

4.6 Wall shear stress

Various study (Nixon, Gunel, & Sumpio, 2009), (Stalder et al., 2008) and man more find that the wall shear is the main sign of WSS because it is a guide to know atherosclerosis (formation of plaque inside arteries) and the exposure of aneurysm development (artery wall weakness). In experimental, the laboratories or researcher were faced difficulty to obtain the shear stress rate. The data for shear rate in FDA's interlaboratory experiment were obtained by velocity profile and changed to shear stress value to consider a specific viscosity model for blood (Paterson et al., 2012). When using the experimental method, it is difficult to get velocity near the wall which can cause a high percentage of error in the experiment. This is where the advantage of using CFD simulation contribute to help the researcher in the medical field and the bioengineering community to calculate and estimate the shear stress rate. Shear stress rate data can give benefit to provide information for aortic valve mechanical stress, blood damage indicator, etc. If the blood has high shear stress value, the duration for the destruction of red blood cell can happen in a short time. So, it is important to keep wall shear stress at artery as low as possible. For low shear stress, the hemolysis can endure a longer time period (Maruyama et al., 2005).

Figure 4.12 and 4.13 shows the contour of wall shear stress profile versus Z coordinate along with the nozzle from CFD simulation with two different viscosity model of blood and experimental data that have been calculated by considering the viscosity model. We can see in the graph that at the wall, shear stress experimental errors extensively rise. The maximum shear stress is recorded at the throat of the nozzle where the forward and reverse of flow occurs. By comparing the viscosity model, the graph displays that the Newtonian has the lowest WSS value along the centreline on the nozzle. The high contribution comes from the conical and throat region, where most blood damage always occurred in this section.

In the graph in figure 23, it displays that maximum WSS occurs between conical and throat regions.in CFD simulation, the researcher must well define the definition of the wall and its contact with blood, which becomes a challenge and the root of the difficulty in the experiment (see figure 13 of (Paterson et al., 2012)). By comparing with two different viscosity model, it shows that the model has relative different and they were larger at low wall shear stress value.

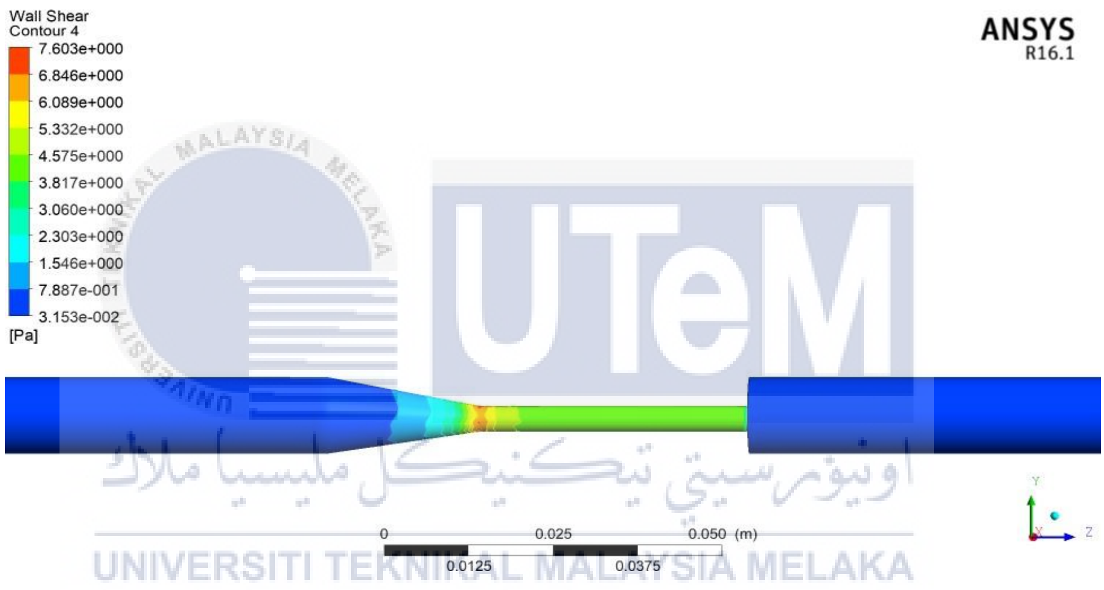

Figure 4.12: Contour for wall shear stress Newtonian viscosity model

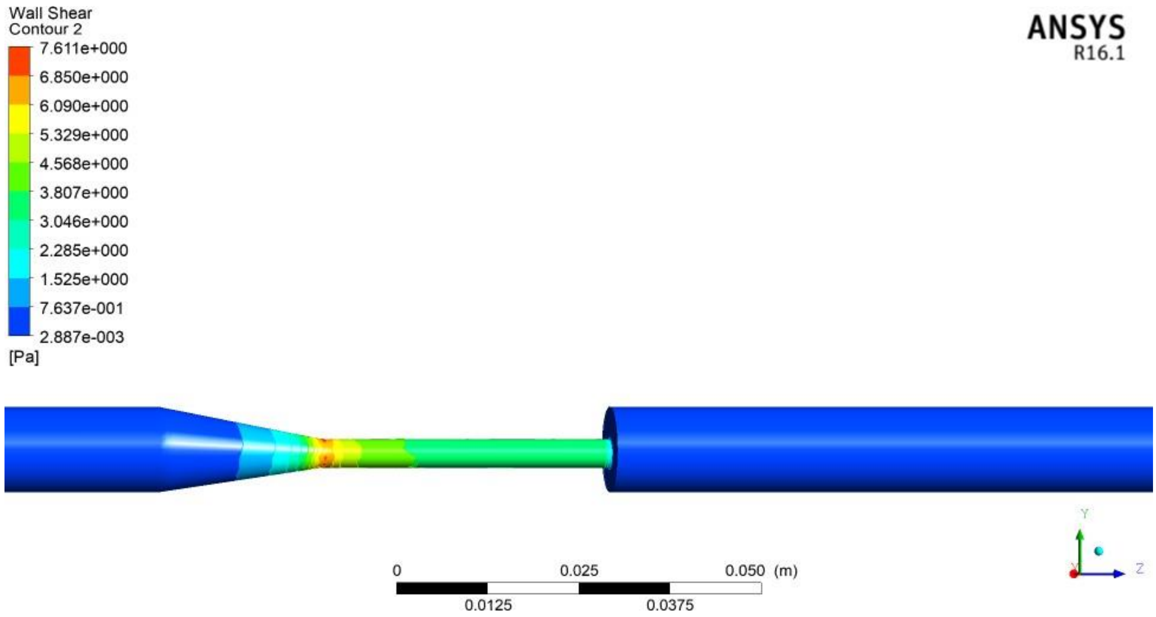

Figure 4.13: Contour for wall shear stress Non-Newtonian model Carreau-Yasuda

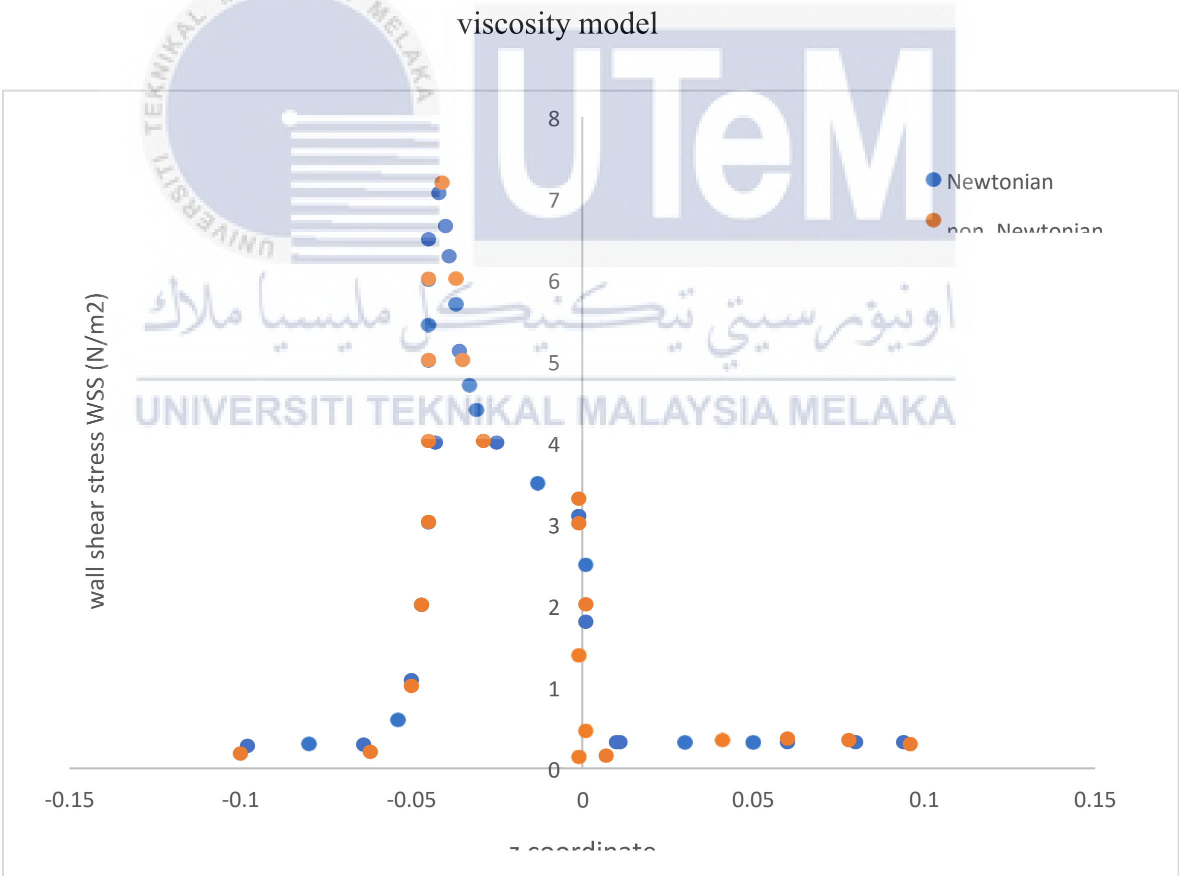

Figure 4.14: Graph for wall shear stress Newtonian viscosity model and non- Newtonian model Carreau-Yasuda

CHAPTER 5

CONCLUSION

In conclusion, I have compiled an ideal medical device for cardiovascular by performed CFD simulation to show the blood flow through the device for Laminar Reynold number, analyze the data for experimental by FDA (Paterson et al., 2012) and validation study from the journal (Min et al., 2014). This research has considered two rheological models for blood which is Newtonian and Carreau-Yasuda model. Besides that, the result that obtains by CFD simulation validated with the experimental method by FDA. The validation was conduct by validating the axial velocity along with the nozzle with two different model and one experiment. By using CFD simulation, the researcher can predict the data to give benefit in hemolysis studies wherein experimental there was a limitation to get the data measurement specifically at the throat region of the nozzle.

At early of this report was completed by introducing the basic of the idealized cardiovascular nozzle. The first objective is completed at a literature review which is to study the effect of hemodynamic characteristics of blood flow in the FDA nozzle. When at the pre-processing stage, the design of the nozzle is done by using the parameter provided by the FDA inside the paper of (Paterson et al., 2012). For the next part of this research in PSM II, the axial velocity along the centreline is further analysis and changing the blood viscosity model which is Newtonian and Carreau-Yasuda model.

This researcher work shows in this report served as early goals to provide patient data through simulation for implanted in-body medical devices that obtain from CT scans or MRI technique. Hence, this report shows that the Newtonian blood viscosity has a high number of wall shear stress and it will cause blood clots and thrombosis to the patient. When blood clot and thrombosis occurred, it can disturb the flow of blood from the recirculating system. Once the process of transporting oxygen to the heart and the release of carbon dioxide from our body is being obstructed it can cause high blood pressure, high cholesterol and a heart condition, the rhythm of heart disturbance.

CHAPTER 6

RECOMMENDATION

There are some recommendations that have been acknowledged during this study, which may be used for future studies. The recommendations are as follow:

1) The geometry of the nozzle must follow the parameter given. However, the length of the nozzle was not given. The length of the nozzle plays an important part to let the have a steady flow after sudden expansion region.

2) Increase the number of mesh number by changing the mesh relevant sizing and increasing refinement of the meshing for better result.

3) The convergent criteria are set to a smaller number for better result.

4) Do research for model-oriented with the flow in conical diffuse which is opposite direction flow from this study.

REFERENCE

Aldi, N., Buratto, C., Pinelli, M., Ruggero, P., & Casari, N. (2016). CFD Analysis of a non-Newtonian fluids processing pump. *Energy Procedia*, *101*(September), 742–749. https://doi.org/10.1016/j.egypro.2016.11.094

Apostolidis, A. J., Moyer, A. P., & Beris, A. N. (2015). Non-Newtonian effects in simulations of coronary arterial blood flow. *Journal of Non-Newtonian Fluid Mechanics*, *233*, 155–165. https://doi.org/10.1016/j.jnnfm.2016.03.008

Borse, Y. V, & Giri, P. S. A. (2016). Computational Fluid Dynamics Analysis of Blood Flow in Human Aorta, *3*(6). https://doi.org/10.17148/IARJSET

Boyd, J., Buick, J. M., & Green, S. (2007). Analysis of the Casson and Carreau- Yasuda non-Newtonian blood models in steady and oscillatory flows using the lattice Boltzmann method. *Physics of Fluids*, *19*(9). https://doi.org/10.1063/1.2772250

Fallon, A. M., Dasi, L. P., Marzec, U. M., Hanson, S. R., & Yoganathan, A. P. (2008). Procoagulant properties of flow fields in stenotic and expansive orifices. *Annals of Biomedical Engineering*, *36*(1), 1–13. https://doi.org/10.1007/s10439- 007-9398-3

Flow-Dependent Rheological of Blood in Capillaries. (1987), *58*, 46–58.

Gijsen, F. J. H., Vosse, F. N. Van De, & Janssen, J. D. (1999). The in # uence of the non-Newtonian properties of blood on the # ow in large arteries : steady # ow in a carotid bifurcation model, *32*, 601–608.

Hariharan, P., D’Souza, G. A., Horner, M., Morrison, T. M., Malinauskas, R. A., & Myers, M. R. (2017). Use of the FDA nozzle model to illustrate validation techniques in computational fluid dynamics (CFD) simulations. *PLoS ONE*, *12*(6), 1–25. https://doi.org/10.1371/journal.pone.0178749

Izraelev, V., Weiss, W. J., Fritz, B., Newswanger, R. K., Paterson, E. G., Snyder, A.,

… Rosenberg, G. (2009). A passively suspended tesla pump left ventricular assist device. *ASAIO Journal*, *55*(6), 556–561.https://doi.org/10.1097/MAT.0b013e3181bae73e

Karimi, S., Dabagh, M., Vasava, P., Dadvar, M., Dabir, B., & Jalali, P. (2014). CFD Study on CT Image-based Geometry. *JOURNAL OF NON-NEWTONIAN FLUID MECHANICS*. https://doi.org/10.1016/j.jnnfm.2014.03.007

Lee, B.-K. (2011). Computational Fluid Dynamics in Cardiovascular Disease.

Korean Circulation Journal, *41*(8), 423. https://doi.org/10.4070/kcj.2011.41.8.423

Maruyama, O., Numata, Y., Nishida, M., Yamane, D. T., Oshima, I., & Adachi, D.

Y. (2005). Hemolysis caused by surface roughness under shear flow, 228–236. https://doi.org/10.1007/s10047-005-0316-x

Min, B., Bona, C., Trias, M., Arbona, A., & Masso, J. (2014). FDA ’ s Nozzle Numerical Simulation Challenge : Non- Newtonian Fluid Effects and Blood Damage, *9*(3). https://doi.org/10.1371/journal.pone.0092638

Nixon, A. M., Gunel, M., & Sumpio, B. E. (2009). The critical role of hemodynamics in the development of cerebral vascular disease. *Journal of Neurosurgery*, *112*(6), 1240–1253. https://doi.org/10.3171/2009.10.jns09759

Paterson, E. G., Burgreen, G. W., Hariharan, P., Food, U. S., & Day, S. W. (2012).

Assessment of CFD Performance in Simulations of an Idealized Medical Device : Results of FDA ' s First Computational Interlaboratory Study Assessment of CFD Performance in Simulations of an Idealized Medical Device : Results of FDA ' s First Computational, (June). https://doi.org/10.1007/s13239-012-0087-5

Stalder, A. F., Russe, M. F., Frydrychowicz, A., Bock, J., Hennig, J., & Markl, M. (2008). Quantitative 2D and 3D phase contrast MRI: Optimized analysis of blood flow and vessel wall parameters. *Magnetic Resonance in Medicine*, *60*(5), 1218–1231. https://doi.org/10.1002/mrm.21778

Sutera, S. P. (2015). Flow-Induced Trauma to Blood Cells.

Tamagawa, M., Kaneda, H., Hiramoto, M., & Nagahama, S. (2009). Simulation of thrombus formation in shear flows using lattice boltzmann method. *Artificial Organs*, *33*(8), 604–610. https://doi.org/10.1111/j.1525-1594.2009.00782.x

Toma, M., & Toma, M. (2018). Biomedical Research and Reviews, 1–3.

Tremmel, M., Xiang, J., Natarajan, S. K., Hopkins, L. N., Siddiqui, A. H., Levy, E. I., & Meng, H. (2010). Alteration of intraaneurysmal hemodynamics for flow diversion using enterprise and vision stents. *World Neurosurgery*, *74*(2–3), 306– 315. https://doi.org/10.1016/j.wneu.2010.05.008

Vierendeels, J. (2007). Comparison of the Hemodynamic and Thrombogenic Performance of Two Bileaflet Mechanical Heart Valves Using a CFD/FSI Model. *Journal of Biomechanical Engineering*, *129*(4), 558. https://doi.org/10.1115/1.2746378

Yoo, C. S., Wang, Y., Trouvé, A., & Im, H. G. (2005). Characteristic boundary conditions for direct simulations of turbulent counterflow flames. *Combustion Theory and Modelling*, *9*(4), 617–646. https://doi.org/10.1080/13647830500307378

Zakaria, M. S., Ismail, F., Tamagawa, M., Aziz, A. F. A., Wiriadidjaja, S., Basri, A. A., & Ahmad, K. A. (2017). Review of numerical methods for simulation of mechanical heart valves and the potential for blood clotting. *Medical and Biological Engineering and Computing*, *55*(9), 1519–1548. https://doi.org/10.1007/s11517-017-1688-9

APPENDIX

Appendix A: the website for NCIP Hub for FDA Nozzle experimental data and previous study

Appendix B : data that obtain from experimental