SIMULATION OF BLOOD FLOW THROUGH MECHANICAL HEART VALVE USING ANSYS FLUID STRUCTURE INTERACTION

By:

CHAI JIEN WEI

(Matrix no: 120363)

Supervisor:

Dr. Mohamad Aizat Abas

JUNE 2017

This dissertation is submitted to
Universiti Sains Malaysia
As partial fulfillment of the requirement to graduate with honors degree in
BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

School of Mechanical Engineering
Engineering Campus
Universiti Sains Malaysia

Declaration

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed... (CHAI JIEN WEI)

Date...

STATEMENT 1

This thesis is the result of my own investigations, except where otherwise stated. Other sources are acknowledged by giving explicit references. Bibliography/references are appended.

Signed... (CHAI JIEN WEI)

Date..

STATEMENT 2

I hereby give consent for my thesis, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available outside organizations.

Signed... (CHAI JIEN WEI)

Date...

Acknowledgement

First and foremost, I would like to express my deepest appreciation to my supervisor Dr. Mohamad Aizat Abas for his continuous supervision, assistance and support from the initial to the final stage of my final project. This dissertation would not be possible without his patient guidance, enthusiastic encouragement and constructive suggestions throughout the whole period. The valuable lessons learnt not only helped me in enhancing my knowledge, but the interpersonal skill and the positive attitude as well.

Besides, I would also like to thank to master student, kak Hafizah for providing me useful advices and assistance when doing experiment. Sincere thanks are also extended to all the technicians for the given technical support. They are keen in teaching and assisting me when dealing with the machines and various equipment during my laboratory work. Their careful and precious guidance were extremely valuable for my study both theoretically and practically.

Next I would like to express my sincere gratitude to my family who constantly support and encourage me spiritually and mentally. A big thanks to my fellow coursemates for sharing their knowledge and ideas in the construction of this project.

Last but not least, I wish to express my sincere thanks to all the lecturers involved in my final year project for their precious guidance and opinions which are extremely valuable for my study both theoretically and practically. I perceive this research project as a big milestone in my career development. I will strive to use gained skills and knowledge in the best possible way, and I will continue to work on their improvement, in order to attain desired career objectives.

Table of Contents

List of Tables

List of Figures

List of Abbreviation

FSI	Fluid Structure Interaction/ Fluid Solid Interface
PIV	Particle Image Velocimetry
MHS	Mechanical Heart Valve
ATS	A company’s name
AHV	Artificial Heart Valve
CFD	Computational Fluid Dynamics
PHV	Prosthetic Heart Valve
BMHV	Bileaflet Mechanical Heart Valve
Re	Reynold’s Number
CAD	Computer Aided Design
UDF	User Defined Function
FEA	Finite Element Analysis
CNC	Computer Numerical Control
STL	STereoLithography
DVI	Doppler Velocity Index
LVO	Left Ventricular Outflow

Abstrak

Katup bileaflet merupakan katup jantung buatan yang unggul dan telah diimplan buat kali pertama sejak daripada 20 tahun yang lalu. Ciri signifikan katup bileaflet ialah geometri dua leaflet yang memainkan peranan penting dalam menentukan medan aliran. Tujuan penyelidikan ini adalah untuk memantau corak aliran darah semasa darah melepasi katup jantung bagi kelengkungan leaflets yang berlainan. Tambahan pula, analisis struktur leaflets turut dikaji. Simulasi dilaksanakan menggunakan ANSYS Fluid Structure Interaction (FSI). Experimen Particle Image Velocimetry jua akan dijalankan bagi mengesahkan keputusan simulasi. Akhirnya, keputusan seperti kontur halaju dan kepusaran dibandingkan dan mendapati bahawa gerakan leaflet dan medan aliran antara simulasi numerikal dengan experimen mempunyai persamaan yang agak banyak. Dalam kajian ini, leaflets yang melengkung ke arah dalam dengan sudut yang besar merupakan configurasi terbaik disebabkan dinamik aliran darah yang lancar. Di samping itu, tekanan von mises adalah paling tinggi di bahagian engsel. Semua keputusan akan berfungsi sebagai asas dalam usaha mencipta katup jantung yang dapat meningkatkan hemodinamik jantung.

Abstract

A successful mechanical prosthetic heart valve design is the bileaflet valve, which has been implanted for the first time more than 20 years ago. A key feature of bileaflet valves is the geometry of the two leaflets, which can be very important in determining the flow field. The aim of this research is to observe the flow pattern of the blood through the heart valve with different leaflets curvatures. In addition, the structural analysis of the leaflets was investigated. The simulation is carried out using ANSYS Fluid Structural Interaction(FSI). Particle image velocimetry experiment will also be conducted to validate the simulation results. Finally, the results such as velocity contour and vorticity are compared, revealing great similarity in leaflet motion and flow fields between the numerical simulation and the experimental test. Also, in this study, the leaflets which curved inwards with greater degrees are found to be the best configuration as it allows the greatest blood flow dynamic. Moreover, the maximum von mises stress is found to be at the hinge region. These results will serve as a basis for valve design to improve the hemodynamic of the heart.

Chapter 1 : Introduction

The heart is a vital muscular organ which pumps blood throughout the body. Heart valves are passive tissues of the heart which react to inertial forces exerted by surrounding blood by opening and closing action. When functioning properly, heart valves as fluidic control components of the heart will ensure unidirectional blood flow during the cardiac cycle, with highest flow rate and lowest flow resistance. The four valves of the human heart as shown in Figure 1.1 are the aortic, mitral, pulmonary and tricuspid valves. In this project, mitral valve will be the prime focus.

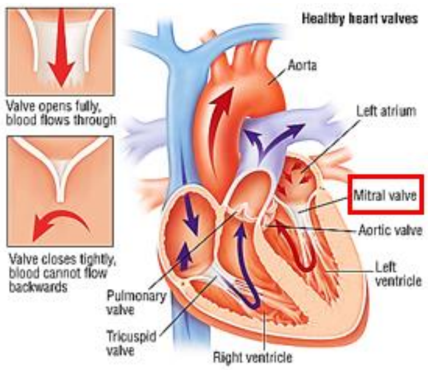


Figure 1.1. Four chambers and four valves of human heart [1]

1.1 Research Background

Valvular stensosis, valvular regurgitation and valvular atresia are valvular heart disease that are life-threatening and afflicts millions of people worldwide. Approximately 250,000 valves are repaired or replaced each year.[2] Malfunction of a native valve impairs its efficient fluid mechanic and hemodynamic performance. Artificial heart valves have been used since 1960 to replace diseased native valves and have saved millions of lives. Unfortunately, despite five decades of use, these devices are less than ideal and lead to many complications such as thrombosis, vegetation and hemolysis.

Thrombosis is the formation of a blood clot inside a blood vessel, obstructing the flow of blood through the circulatory system. Thrombosis of a prosthetic valve is potentially life threatening, resulting in hemodynamically severe stenosis or regurgitation. According to Vuyisile T. Nkomo, M.D., M.P.H., director of the Valvular Heart Disease Clinic at Mayo Clinic in Rochester, Minnesota, thrombotic risk is related to the type of valve, position of the valve and adequacy of anticoagulation.[3]

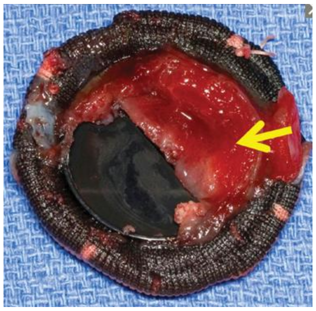


Figure 1.2. Thrombosis on mitral valve prosthesis entirely covering inflow aspect of one flap [4]

In medicine, vegetation is often associated with endocarditis. Endocarditis is an inflammation of the valves of the heart and it is often caused by the growth of bacteria on one of the heart valves, leading to a mass known as a vegetation.[5] The big lump of mass will bring serious complications as it impedes the flow of blood through the circulatory system.

Figure 1.3 Mitral valve vegetation [6]

Hemolysis which is the rupturing of red blood cells is one of the potentially serious complications of prosthetic heart valves. It is usually associated with either structural deterioration or paravalvular leak. Therefore, the damage threshold of blood components by fluid shear stress must be considered when designing a prosthetic heart valve.

Generally, a mechanical heart valve consists of a ring outer body, inner ring, leaflets, orifice and hinges. Most mechanical heart valves are made of titanium, graphite, pyrolytic carbon, and polyester. The titanium is used for the housing or outer ring, graphite coated with pyrolytic carbon is used for the leaflet and 100% pyrolytic carbon is used for the inner ring. Figure 1.4 shows one of the existing mechanical heart valve in the market which is the On-X prosthetic bileaflet valve.

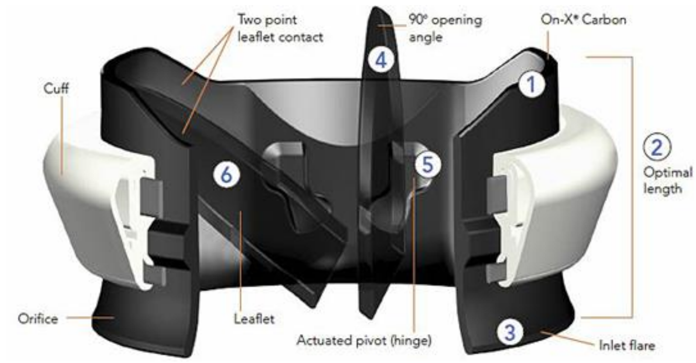


Figure 1.4. On-X prosthetic valve structure [7]

1.2 Problem Statement

Today, many people live long and healthy lives and never realize they have a mild valve problem. However, valve disease can seriously increase a persons' risk for sudden death or cause rapid development of problems in and around the heart that can become fatal without treatment. The creation of mechanical heart valve has made an incredible impact in biomedical field due to its lifetime durability. However, there are still problems in mechanical valves such as thrombosis, vegetation, and many more. Hence, a better valve design has to be developed to maximize the survival rate of patients.

1.3 Objectives

- To design a mechanical heart valve that meet forward and backward flow shear, reduce thrombosis, hemolysis and cavitation.
- To validate the simulation results by conducting in vitro experiment

1.4 Scope of Project

Basically, the scope of work includes the design phase, simulation phase and experiment. For the design phase, left heart model with five different leaflets configurations are established using SolidWorks. Next, the created geometries are used in the subsequent simulation phase. The simulation is carried out using ANSYS Fluid Structure Interaction (FSI) since multiphysics problem is involved. Finally, PIV experiment is conducted to compare the results with the simulation for validation purpose. The left heart model used in both simulation and experiment are similar to the actual heart of human to ensure that the results obtained are applicable to the real world.

Chapter 2 : Literature Review

History of Mechanical Heart Valve

The past 50 years have witnessed remarkable progress in the development of safe, hemodynamically favorable mechanical heart valves. The need for prosthetic heart valves was long recognized but seemed an impossible dream before 1952 when Dr Charles Hufnagel clinically introduced a ball valve that he placed into the descending thoracic aorta for treatment of aortic valvular insufficiency. This valve was implanted in a 30-year-old woman who could lead a normal life after the surgery. [8]

According to Dr Charles Hufnagel research, the drawback of this design was that it could only be placed in the descending aorta instead of the heart itself. Consequently, it did not fully correct the valve problem but only alleviate the symptoms. However, it was a significant progress because it proved that synthetic materials could be used to create heart valves. [9]

In 1960, a new type of valve was successfully implanted which is the Starr-Edwards ball valve. This valve was a modification of Hufnagel's original valve. [10] The ball of the valve was slightly smaller and caged from both sides so it could be inserted into the heart itself.

Tilting discs were introduced in the later 1960s. These valves were a major improvement over the ball designs as they allowed blood to flow in a more natural way while reducing damage to blood cells from mechanical forces. Regrettably, the struts of these valves tended to fracture from fatigue over time. [11]

Bileaflet valves were introduced in 1979. Blood flows directly through the center of these valves (like in an intact heart valve) which makes these valves superior to other designs. The major downside of this design is that it allows some backflow.

According to Journal of the Practice of Cardiovascular Sciences [12], the thrombogenicity in caged ball is the highest followed by tilting disc and bileaflet and this is why a vast majority of mechanical heart valves used today have bileaflet design.

Design Optimization of Mechanical Heart Valve

The paper by Yared Alemu [13] has previously compared two MHV, ATS and the St. Jude Medical, and demonstrated that owing to its non-recessed hinge design, the ATS valve offers improved thrombogenic performance. In the study, they further optimize the ATS valve thrombogenic performance by modifying various design features of the valve, intended to achieve reduced thrombogenicity. The modifications include optimizing the leaflet-housing gap clearance, increasing the effective maximum opening angle of the valve and introducing a streamlined channel between the leaflet stops of the valve that increases the effective flow area.

Analysis of Flow Field in Mechanical Heart Valves Using Finite Volume Method

In the paper by Feng Zhou [14], the impact of valve structure on blood flow in aorta, including hemodynamic aspects are investigated. The regional distribution of flow shear stress in an artificial heart valve (AHV) is analyzed using computational fluid dynamics and the AHV performance is evaluated in terms of variation of flow velocity and pressure when blood passes the leaflets in the aortic valve. The findings indicate that for the design of a mechanical AHV, the maximum opening angle and internal orifice diameter should be increased to improve the blood flow dynamics and reduce the likelihood of damage to blood components.

Besides, Mushtak Al-Atabi [15] investigated on blood flow through the mitral valve. The study is done numerically using computational fluid dynamics. A two-dimensional, 2D model of the experimental design was simulated using ANSYS FLUENT. From the results, it is seen that the flow through an open mitral valve produces two vortices. Vortices are vital as they help propel the flow from the ventricle. The absence and presence of the vortices and the structure of the vortices can be used to identify possible mitral valve anomalies. This can be used as the baseline for the protocol.

Computational Modelling of Bileaflet Mechanical Valves Using Fluid-Structure Interaction Approach

The study by Han Hung Yeh [16] presents numerical simulations of a minimally constrained mechanical valve model using a fully coupled fluid-structure interaction method with COMSOL Multiphysics. The model employs a physiological pulsatile pressure gradient across an aortic valve with an approximately symmetric aortic root. The complex hinge from the exact model is simplified with a pin joint and weak constraints to control the designated valve leaflet positions. Also, arbitrary Lagrangian-Eulerian method is exercised in order to accommodate large mesh displacements due to leaflet motion. It is found that vortices were generated with higher blood velocity passing through the unconstrained leaflet, which may lead to diagnostic confusion.

PIV Measurements of Flows in Artificial Heart Valves

The proposed study by Costantino Del Gaudio [17] presents the fabrication and in vitro characterization of a biodegradable electrospun heart valve prosthesis using the particle image velocimetry technique either in physiological and pathological fluid dynamic conditions. The scaffold was designed to reproduce the aortic valve geometry, also mimicking the fibrous structure of the natural extracellular matrix. To evaluate its performances for possible implantation, the flow fields downstream the valve were accurately investigated and compared. The experimental results showed a correct functionality of the device, supported by the formation of vortex structures at the edge of the three cusps, with Reynolds stress values below the threshold for the risk of hemolysis (which can be comprised in the range 400–4000 N/m^2 depending on the exposure period), and a good structural resistance to the mechanical loads generated by the driving pressure difference.

Rodoslav Kaminsky [18] proposed various modifications of PIV in order to describe, compare and realize which method is the most suitable for the quantification of such flows. The choice of the experimental procedure for testing the PHVs is strongly dependent on the optical access of the designed in-vitro testing loops simulating the human heart and vascular system. The hardware demand and its configuration such as stereoscopic PIV is much more complex than standard 2D PIV, thus the conditions and design of the testing loop have to be aware of to allow the desired flow measurement. The flow in heart valves as an unsteady periodically generated flow, can be attained by averaged phase locked or measurements with high temporal.

Moreover, Majid Y. Yousif [19] revealed that for accurate particle image velocimetry measurements in hemodynamics studies it is important to use a fluid with a refractive index (n) matching that of the vascular models (phantoms) and ideally a dynamic viscosity matching human blood.

Experimentally Validated Hemodynamics Simulations of Mechanical Heart Valves

The paper by Vinh-Tan Nguyen [20] proposed a numerical method for simulations of full three dimensional MHV with moving leaflets in a typical human cardiac cycle. A cell-centered finite volume method is employed to model incompressible flows in MHV. The numerical results for laminar and turbulent flows are then validated against experimental data using Particle Image Velocimetry technique. The schematic diagram of the experimental setup is shown in Figure 2.1. In the CFD simulations, the flows over the BMHV at different Reynolds numbers ranging from 350 to 5000 corresponding to the parabolic laminar flow seen at late systole and during diastole as well as peak systolic velocity of the cardiac cycle are investigated. The simulations for both Re = 350, Re = 750 and Re = 1000 were validated through comparisons with experimental data. In both simulations, the most experimental trends and magnitude are captured with reasonable accuracy. As for the finding, a triple jet structure is observed in the simulations together with a switching of central orifice jet flow from horizontal axis to vertical axis downstream of the leaflets and the results are well compared with the experimental data.

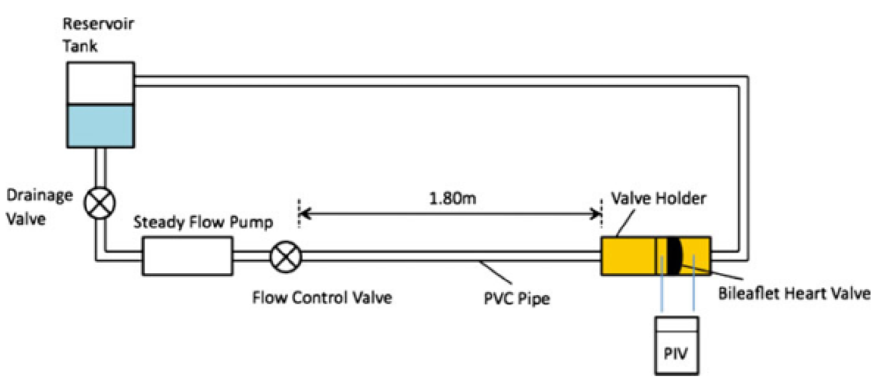


Figure 2.1. Schematic diagram of steady flow loop setup

Besides, the paper by S. Annerel [21] discussed on the validation of a recently developed fluid–structure interaction (FSI) coupling algorithm to simulate numerically the dynamics of an aortic bileaflet mechanical heart valve (BMHV). This validation is done by comparing the numerical simulation results with in vitro experiments. For the in vitro experiments, the leaflet kinematics and flow fields are obtained via PIV technique. The results reveal great similarity in leaflet motion and flow fields between the numerical simulation and the experimental test. Therefore, it is concluded that the developed algorithm is able to capture very accurately all the major leaflet kinematics and dynamics and can be used to study and optimize the design of BMHV.

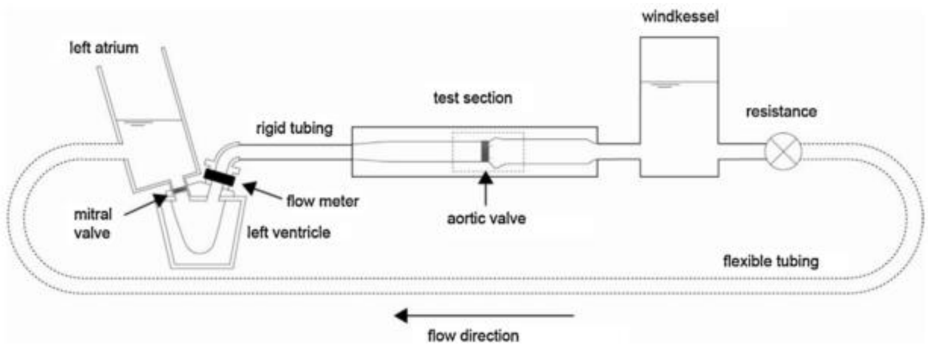


Figure 2.2. Schematic overview of the experimental setup

Comparison of Previous Studies and Current Study

To summarize, majority of mechanical heart valves used today are bileaflet valves because they allow the least resistance to flow and the least blood damage. In terms of numerical solution, FSI is a good approach in hemodynamic analysis of mechanical heart valve as physical parameters such as wall shear stress and leaflet deformation can be evaluated. Furthermore, PIV measurement is an excellent method to trace the flow pattern and measure the velocity of the fluid.

Up until now, there are limited studies that combine both FSI and PVI to study the hemodynamic of mechanical bileaflet heart valve especially in mitral valve position. For example, the paper by S. Annerel [21] discussed both FSI and PIV but foucs on the aortic valve rather than the mitral valve. Next, Vinh-Tan Nguyen paper [20] mentioned about limitations such as neglecting the hinge mechanism of the valve leaflets and the simulation was started with the leaflets in fully open position due to meshing issues. Thus, to make a breakthrough, this research focus on the mitral BMHV, implements a simplified hinge mechanism and the simulation is started with the leaflets in fully closed position.

Chapter 3 : Methodology

The research focuses on the application of mechanical heart valve as an alternative solution when natural heart valves become dysfunctional. The target audiences of this research are patients with valvular heart disease, doctors and researchers who are in the field of cardiology. The research work is meant to be hemodynamic of mechanical heart valve in the left heart in three dimension using engineering simulation software to analyze the results. Experiment, too, will be conducted to validate the simulation results.

3.1 Simulation using ANSYS Fluid Structure Interaction (FSI)

Fluid-structure interaction (FSI) is a multiphysics coupling between the laws that describe fluid dynamics and structural mechanics. This happening is characterized by interactions between a deformable or moving structure and a surrounding or internal fluid flow. In the simulation, 2-way FSI is used as the mechanical model and CFD model analysis results are correlated. A general 2-way FSI setup procedure is outlined in Figure 3.1.

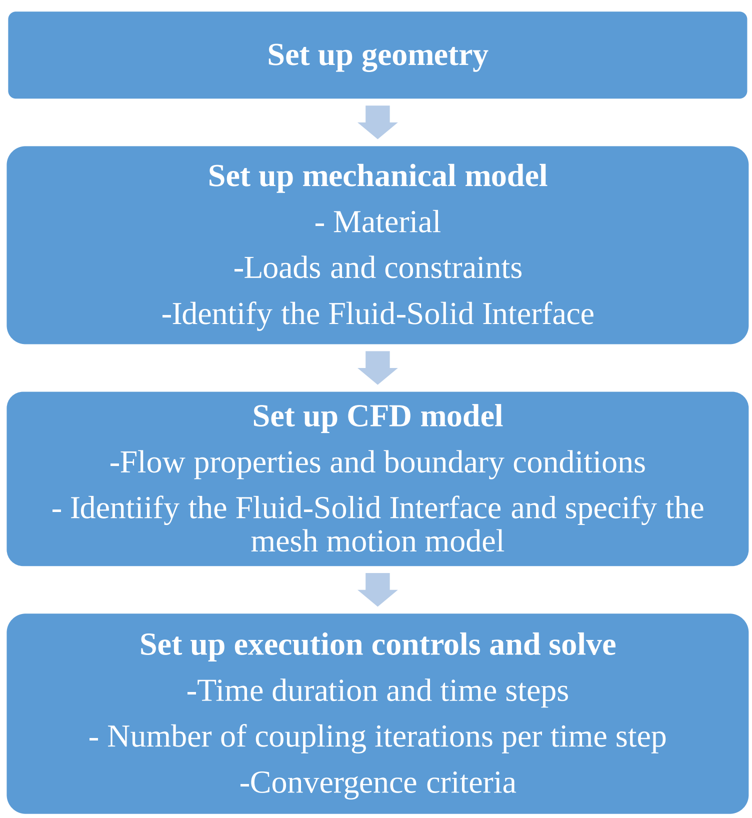


Figure 3.1. General 2-way FSI setup procedure

3.1.1 Pre-Analysis

In the Pre-Analysis step, three aspects will be reviewed.

- Governing equations that need to be solved
- Boundary conditions that are applied
- Dynamic mesh that is employed

3.1.1.1 Governing Equations

Before commencing a CFD simulation, it is always good to look at the governing equations underlying the physics. In this case, although there are additional complexities such as pulsatile flow and non-newtonian fluids, the governing equations are the same as any other fluids problem. The most fundamental governing equations are the continuity equation and the Navier-Stokes equations.

Continuity Equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathrm{v}) = 0 \qquad \text{Eq. 1}$$

However, since blood is an incompressible fluid, the rate of change of density is zero, thus the continuity equation above, Eq. 1 can be further simplified in the form below:

$$\nabla \cdot \mathrm{v} = 0 \qquad \text{Eq. 2}$$

Navier-Stokes Equation

$$\rho\left(\frac{d\mathrm{v}}{dt} + \mathrm{v} \cdot \nabla \mathrm{v}\right) = -\nabla \mathrm{p} + \mu \nabla^2 \mathrm{v} + \mathrm{f} \qquad \text{Eq. 3}$$

One thing to note in the Navier-Stokes equation is that the viscosity coefficient, μ is not a constant but instead a function of shear rate. Blood gets less viscous as the shear rate increases. This phenomenon is known as shear thinning. Hence, Carreau fluids model is employed to model the blood viscosity. The mathematical formulation of Carreau model is as follows:

$$\mu_{eff}(\gamma) = \mu_{inf} + \left(\mu_0 - \mu_{inf}\right)\left(1 + (\lambda\gamma)^2\right)^{\frac{n-1}{2}} \qquad \text{Eq. 4}$$

where

μ_{eff} = effective viscosity

$$\left.\begin{array}{l} \mu_0 = \text{viscosity at zero shear rate (Pa}\cdot\text{s)} \\ \mu_{inf} = \text{viscosity at infinite shear rate (Pa}\cdot\text{s)} \\ \lambda = \text{relaxation time (s)} \\ \text{n} = \text{power index} \end{array}\right\} \text{material coefficients}$$

For the case of blood,

μ_0 = 0.056 (kg/m·s)

μ_{inf} = 0.0035 (kg/m·s)

λ = 3.313 (s)

n = 0.3568

3.1.1.2 Boundary Conditions

Inlet

To the best of our knowledge, mammalian blood flow is pulsatile and cyclic in nature. Therefore, the velocity at the inlet is not set to be a constant, but instead, in this case, it is a time varying periodic profile. The pulsatile profile within each period is a combination of two phases namely the systolic and the diastolic phase. During the systolic phase, the velocity at the inlet varies in a sinusoidal pattern. The sine wave during the systolic phase has a peak of 0.5 m/s and a minimum velocity of 0.1 m/s. Assuming a rapid heartbeat rate of 120 per minute, the duration of each period is 0.5s. This model for pulsatile blood flow is proposed by Sinnott et, al. [22] A figure of the profile within two periods is presented in Figure 3.2.

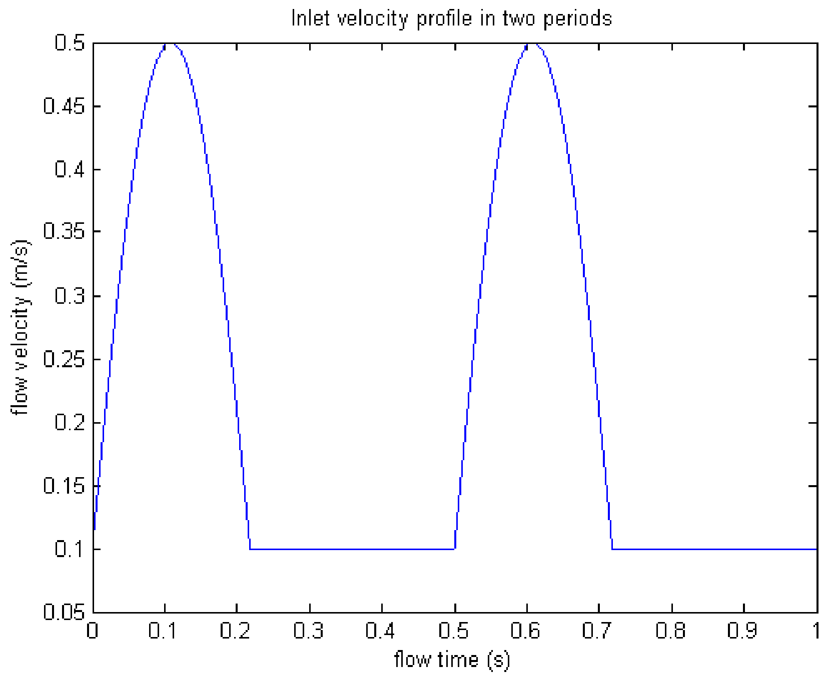


Figure 3.2. Inlet velocity profile in two periods [23]

To illustrate the profile with clarity, a mathematical description is also given below.

$$v_{inlet}(t) = \begin{cases} 0.5\sin[4\pi(t + 0.0160236)] & 0.5n < n \leq 0.5n + 0.218 \\ 0.1 & 0.5n + 0.218 < n \leq 0.5(n+1) \end{cases}$$

Where n = 0,1,2…

Outlet

The systolic pressure and diastolic pressure of a healthy human is around 120 mmHg and around 80 mmHg respectively. Thus, taking the average pressure of the two phases, 100 mmHg (around 13332 Pascal) is used as the static gauge pressure at the outlets.

Wall

The easiest boundary condition to determine is the left atrial and ventricular wall. The wall regions of this model are simply defined and set to “wall”. From a physical viewpoint, the “wall” condition dictates that the velocity at the wall is zero.

3.1.1.3 Dynamic Mesh

Smoothing Method

Diffusion based smoothing is integrated in the dynamic mesh. It tends to produce better quality meshes than spring-based smoothing and often allows larger boundary deformations before breaking down. For diffusion-based smoothing, the mesh motion is governed by the diffusion equation $\nabla \cdot (\gamma \nabla u) = 0$ where u is the mesh displacement velocity. The diffusion coefficient can be used to control how the boundary motion affects the interior mesh motion. In this simulation, the formulation for the diffusion coefficient, γ is a function of the boundary distance with a formula of $\gamma = \frac{1}{V^{\alpha}}$ where α is the diffusion parameter. In the simulation, diffusion parameter of $\alpha = 1.4$ is applied.

Remeshing Method

Remeshing is also exerted as an insurance because when the boundary displacement is large compared to the local cell sizes, the cell quality can deteriorate or the cells can become degenerate. This will invalidate the mesh (for example, result in negative cell volumes) and consequently, will lead to convergence problems when the solution is updated to the next time step. To circumvent this problem, remeshing option is activated. By using remeshing, it agglomerates cells that violate the skewness or size criteria and locally remeshes the agglomerated cells or faces. If the new cells or faces satisfy the skewness criterion, the mesh is locally updated with the new cells (with the solution interpolated from the old cells). Otherwise, the new cells are discarded and the old cells are retained.

3.1.2 Establishment of CAD Model

Generally, the CAD model consists of two major parts, namely the left heart and the mechanical bileaflet heart valve. The model is constructed using SolidWorks instead of using the real left heart which is scanned by 3D scanner. This is because the real scanned model has lots of uneven surfaces on its irregular geometry which increase the difficulty to assemble both the left heart and mechanical heart valve together. Moreover, the complexity of the mesh due to highly irregular geometry will also increase the computational time. Since the aim of this research is to investigate the effect of different leaflets curvature on blood flow, therefore a simplified model is enough to carry out the simulation.

The simplified model is constructed using SolidWorks 2016. Loft features are widely used to create the irregular geometry. The left heart model consists of one inlet and one outlet and the left atrium and ventricle length are 50mm and 80mm respectively. The diameter of the outer ring is 31mm. Regarding the leaflets, the leaflets are initially in closed position where the position is 60˚ from the horizontal axis. The simplified model used in the simulation is shown in Figure 3.3. Overall, there are five different leaflets curvature that have been designed. The leaflets curvature modifications are explained in Figure 3.4 and the five different leaflets curvature are shown in Figure 3.5.

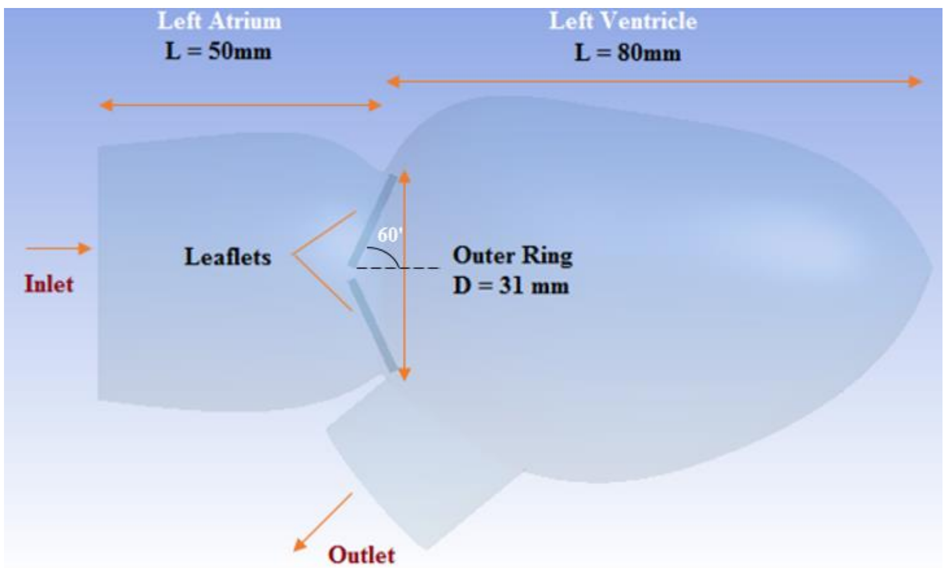


Figure 3.3. Simplified CAD model for simulation with predefined dimensions

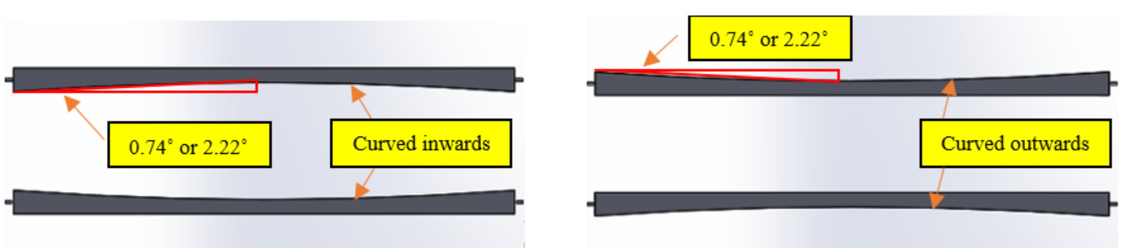
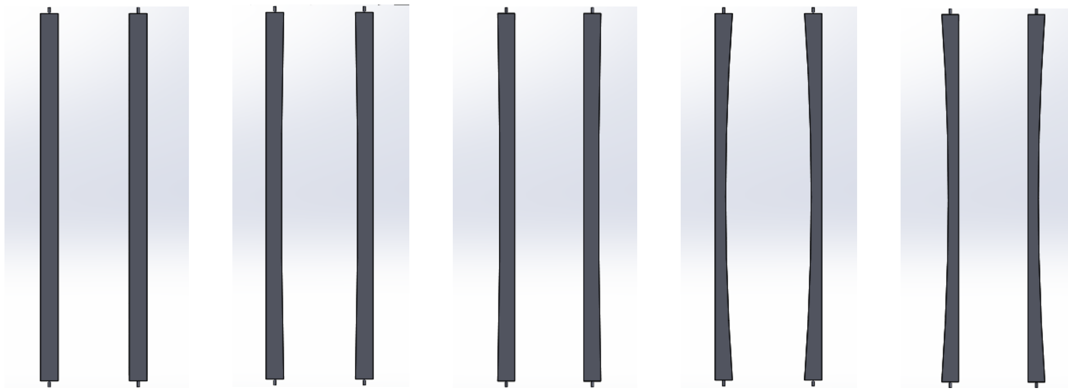


Figure 3.4. Explanation of leaflets curvature modifications



(a) (b) (c) (d) (e)

Figure 3.5. Different leaflets curvature of the mechanical heart valve. (a) No curved (original), (b) 0.74˚ curved inwards, (c) 0.74˚ curved outwards, (d) 2.22˚ curved inwards, (e) 2.22˚ curved outwards

The five CAD models are then exported to STEP file format which is suitable to be read by ANSYS Design Modeler for further processing.

3.1.3 Transient Structural

Before importing the STEP file, the material properties of pyrolytic carbon are defined as the leaflets of mechanical heart valve are made from pyrolytic carbon. The general physical properties of pyrolytic carbon are listed below in Figure 3.6.

-	Pyrocarbone
Densité (g.cm^{-3})	1.7 - 2
Module d'Young (GPa)	21 - 28
Coefficient de Poisson	0.22
Allongement à la rupture (%)	1.2 – 1.5
Résistance à la flexion (MPa)	280 – 560
Résistance à la compression (MPa)	1350
Limite élastique en traction (MPa)	345
Dureté (Vickers)	150 – 250

Figure 3.6. General physical properties of pyrolytic carbon [24]

The physical properties that are used in ANSYS simulation are shown in Figure 3.7.

Density = 1850 kg / m^3

Young's Modulus = 24.5 GPa

Poisson's ratio = 0.22

Elastic limit = 345 MPa

Properties of Outline Row 3: pyro_carbon

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	1850	kg m^-3		
3	Isotropic Elasticity				
4	Derive from	Young'...			
5	Young's Modulus	2.45E+10	Pa		
6	Poisson's Ratio	0.22			
7	Bulk Modulus	1.4583E+10	Pa		
8	Shear Modulus	1.0041E+10	Pa		

Figure 3.7. Physical properties of pyrolytic carbon that used in ANSYS simulation

3.1.3.1 Geometry

The STEP file is imported to ANSYS Design Modeler in ZX plane. Generally, there are five parts that make up the whole model namely rod-x, rod+x, leaf+x, leaf-x and heart. Here, Boolean (subtract) operation is applied to create a cavity as the fluid solid interface for the fluid domain. The target body will be the heart while the remaining four parts will be the tool bodies. The geometry imported to the Design Modeler is shown in Figure 3.8.

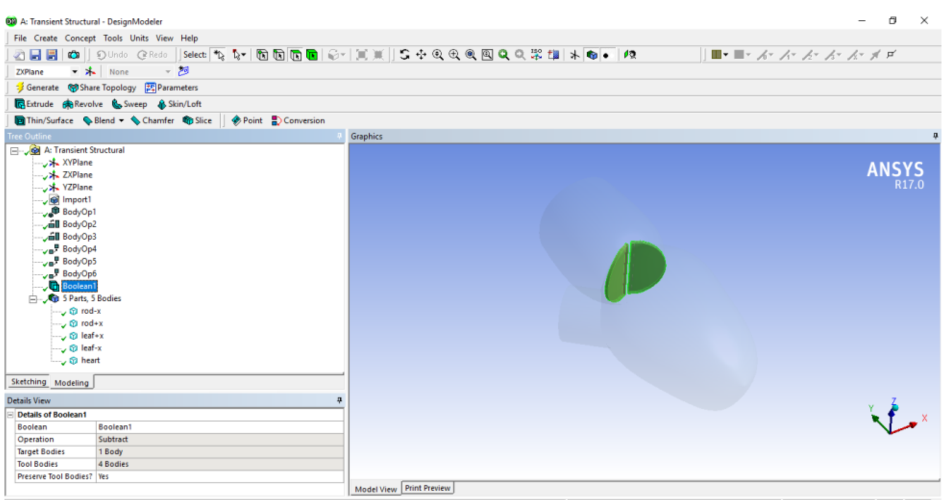


Figure 3.8. Geometry imported to the ANSYS Design Modeler

3.1.3.2 Mesh

Since in transient structural, the focus is the leaflets so the left heart model is suppressed. The meshing is only done on the leaflets. Here, adaptive size function with fine relevance center is adopted. In addition, refinement is applied at the 4 edges of the leaflets as shown in Figure 3.9. The meshed geometry is presented in Figure 3.10.

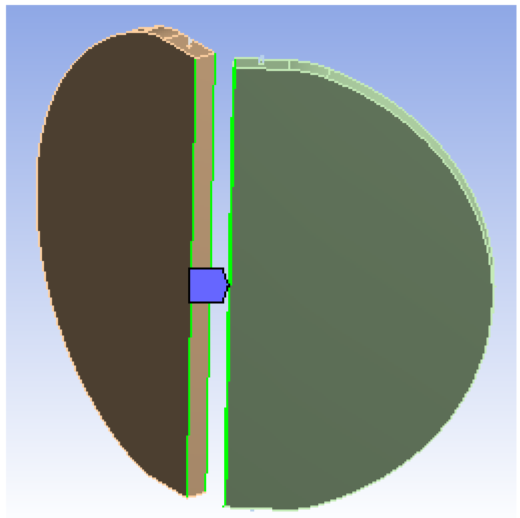


Figure 3.9. Four edges that are highlighted for refinement

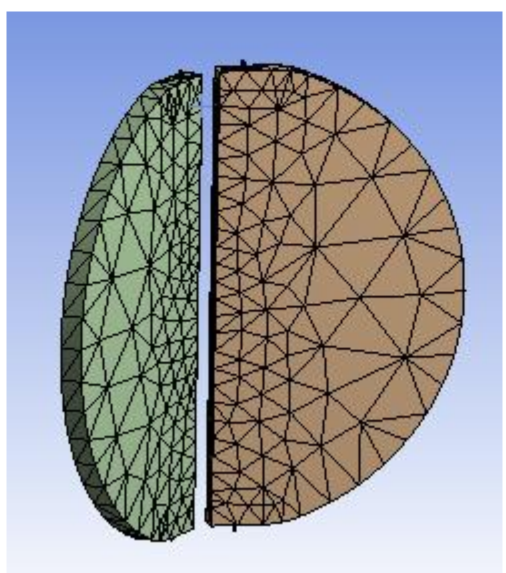


Figure 3.10. The meshed geometry(leaflets)

3.1.3.3 Physics Setup

The complex hinge from the exact model is simplified with a pin joint and weak constraints to control the designated valve leaflet positions. Here, the default contacts are not suitable to represent the hinge motion. Thus, new joints contacts are established. Here, the top and bottom faces of the hinge are fixed to the ground. Also, revolute joints are also defined to allow the leaflets to rotate in z direction. Figure 3.11 and Figure 3.12 shows the fixed and revolute joints respectively.

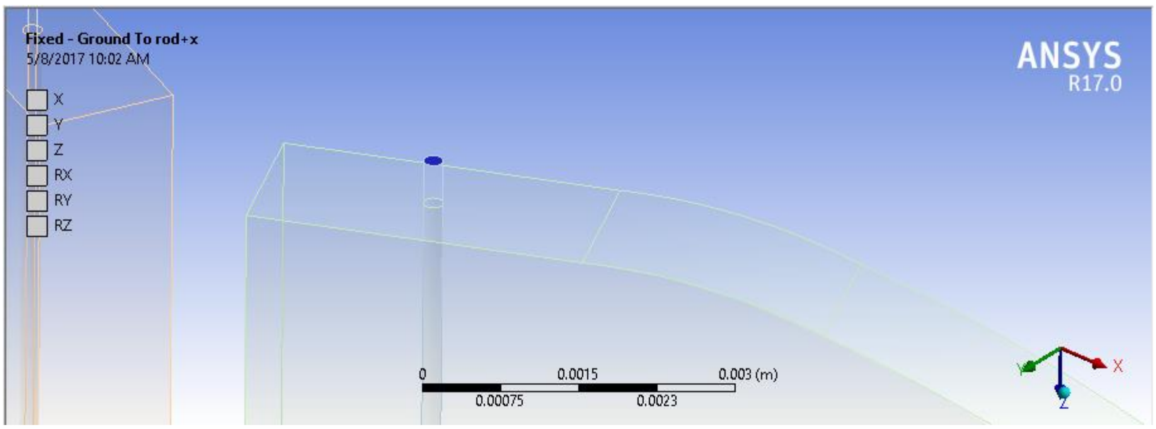


Figure 3.11. Face that is selected for the fixed to the ground joint

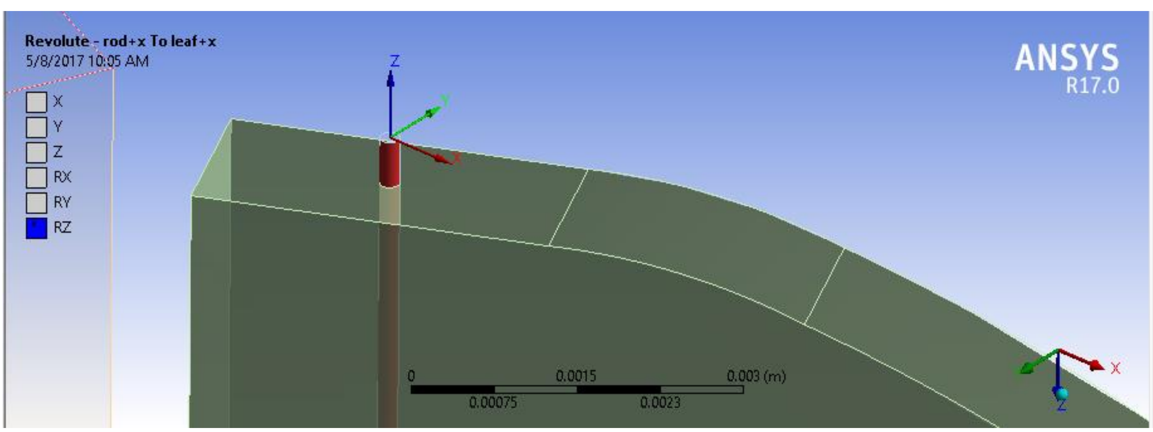


Figure 3.12. Faces that are chosen for the revolute joints

For the analysis settings, the step end time applied is 0.5s with auto time stepping disable. The time step is defined by 0.01s interval. About the loads, joints loads are applied to create the rotational motion in z direction. In addition, 18 faces of the leaflets are selected for the Fluid Solid Interface and shown in Figure 3.13.

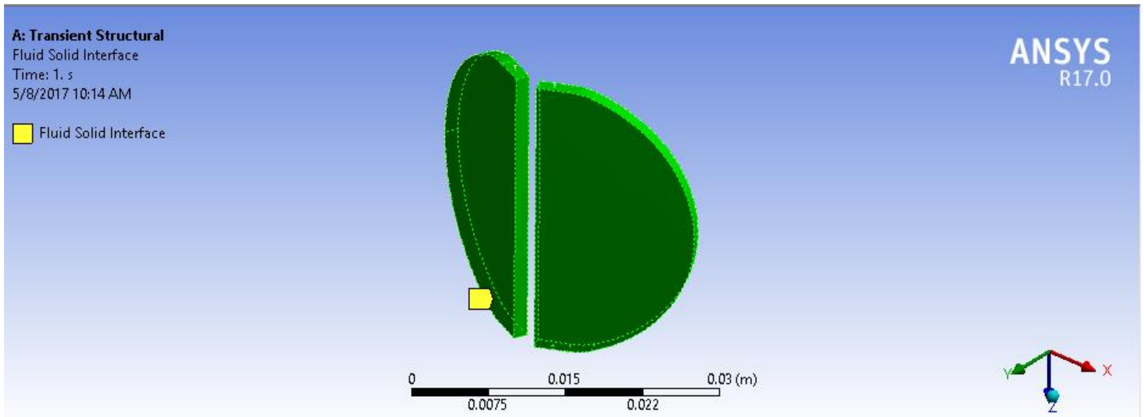


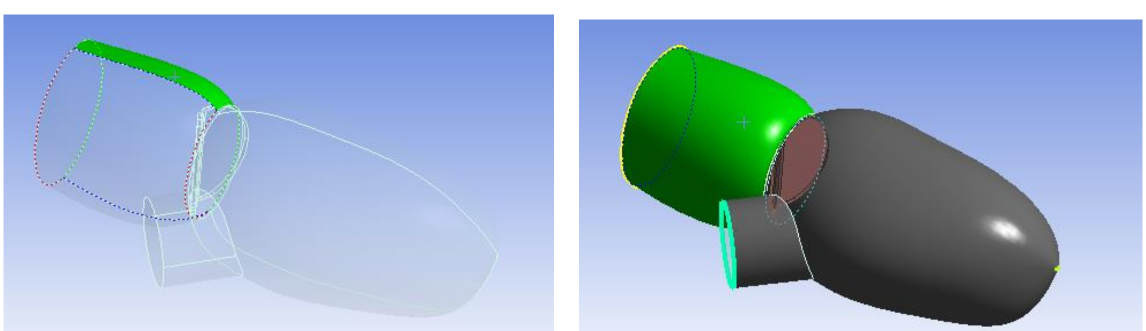
Figure 3.13. 18 faces that are selected for Fluid Solid Interface

3.1.4 Fluid Flow (FLUENT)

The geometry that is used previously in transient structural is also used in FLUENT.

3.1.4.1 Mesh

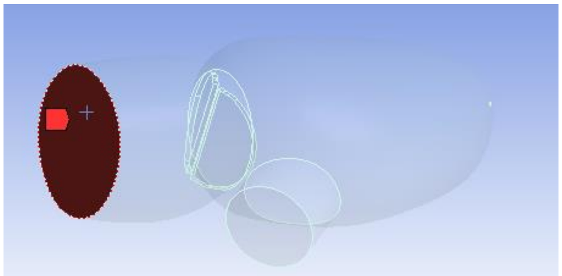
Similarly, in FLUENT the fluid domain which is the left heart is prioritized and the leaflets are suppressed. Virtual topology is created to merge the surfaces in order to simplify the geometry. The geometry before and after virtual topology is applied are shown in Figure 3.14. It is clearly shown that the surfaces are merged after applying virtual topology thereby optimizing the generated mesh.



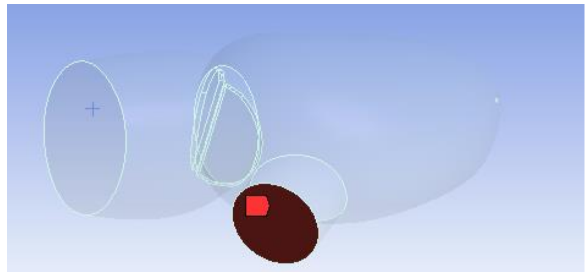
Before After

Figure 3.14. Geometry before and after apply virtual topology

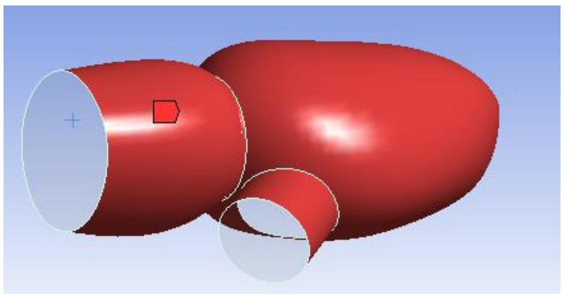
Next, named selections (inlet, outlet, wall_heart, fluid_domain and fsi) are created. Figure 3.15 displays the faces selected for the respective name.



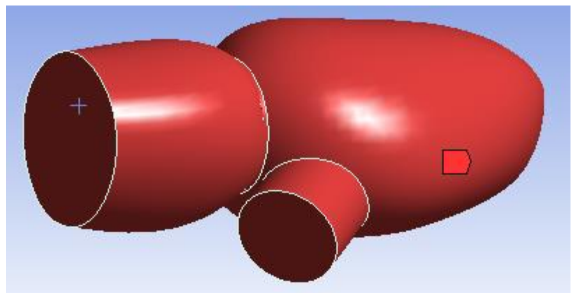
(a)



(b)



(c)



(d)

(e)

Figure 3.15. Faces selected for named selection. (a) Inlet, (b) Outlet, (c) Wall_heart, (d) fluid_domain, (e) fsi

Regarding the mesh, curvature size function with coarse relevance center is applied. Fine span angle center is also chosen. The mesh quality is justified based on skewness and orthogonal quality to ensure the mesh of the model is good for the simulation. Figure 3.16

shows the range of skewness and orthogonal quality to gauge the quality of the mesh. The cross-sectional view of the meshing of the model is shown in Figure 3.17.

Skewness:

Outstanding	Very Good	Good	Sufficient	Bad	Inappropriate
0-0.25	0.25-0.50	0.50-0.80	0.80-0.95	0.95-0.98	0.98-1.00

Orthogonal quality:

Inappropriate	Bad	Sufficient	Good	Very Good	Outstanding
0-0.001	0.001-0.15	0.15-0.20	0.20-0.70	0.70-0.95	0.95-1.00

Figure 3.16. Range of skewness and orthogonal quality to gauge the quality of mesh

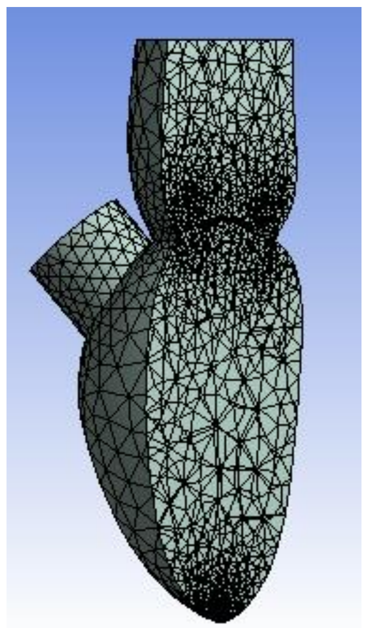


Figure 3.17. Cross-sectional view of the meshing of the model

3.1.4.2 Physics Setup

In FLUENT, the time is changed to transient since we wish to observe the blood flow pattern over time. Laminar viscous model is also chosen. For the material, blood with a density of 1060 kg/m^3 is applied. The viscosity is described using the carreau fluid model with predefined parameters as mentioned in section 3.1.1.1 Governing Equations.

For the boundary conditions, the inlet is set to velocity inlet. As mentioned in section 3.1.1.2 Boundary Conditions, blood flow is pulsatile so a udf as shown in Appendix 1 is used to describe the periodic profile at the inlet. At the same time, the outlet is set to pressure-outlet with a gauge pressure of 13332 Pa. The remaining named selection (fsi, wall_heart and fluid_domain) are set to wall.

Besides, dynamic mesh is implemented. Smoothing and remeshing mesh methods are activated. The parameters of both mesh methods are as shown in Figure 3.18. Next, to define the dynamic mesh zones it is important to defined the fsi as the system coupling and the remaining as stationary.

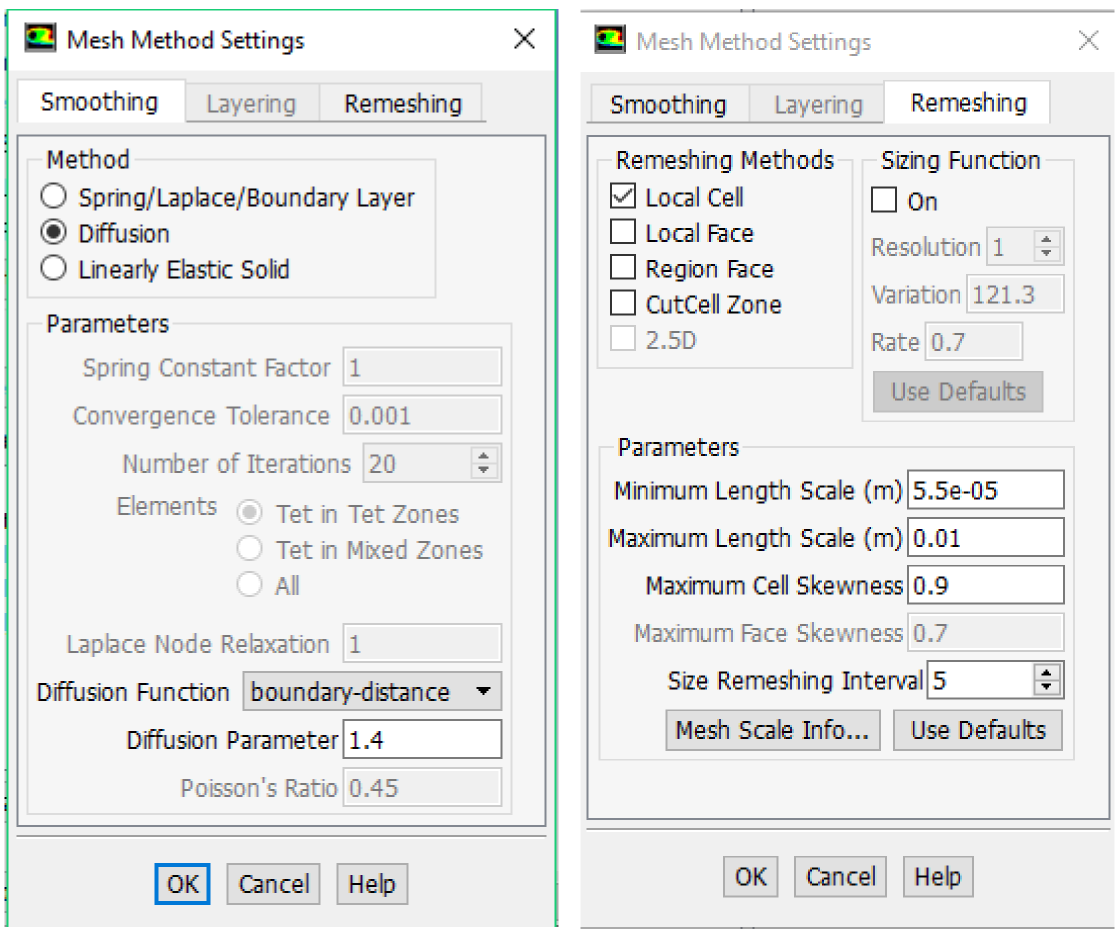


Figure 3.18. Parameters for smoothing(left) and remeshing(right) mesh method

Figure 3.19 shows the mesh distribution of the left heart model when the leaflets are fully closed and opened. It is observed that the deformation of the mesh from closed to opened position is not large. Thus, it does not encounter negative cell volume detected issue which means that the element does not have a big distortion. Not big enough to do some vertex, or edges, to penetrate an opposite face resulting in a negative volume. Therefore, the dynamic mesh used is appropriate.

(a) (b)

Figure 3.19. Mesh distribution of the left heart model. (a) Fully closed, (b) Fully opened

For the reference values, the area is changed to follow the body area in design modeler which is 0.023565 m^2. This value is obtained by using the analysis tool to study the entity information which will then display the details of the simulation model. The reference values of density and velocity are changed to 1060 kg/m^3 and 0.1 m/s respectively.

Upcoming will be the pressure-velocity coupling method. Here, the coupled scheme is applied. The pressure-based coupled algorithm obtains a more robust and efficient results. Also, the hybrid initialization is employed as solution initialization. Lastly, the time step size is set to 0.01s with 50 number of time steps and max iterations per time step is set to 30. However, it is important to but note that the system coupling's time step size will override FLUENT predefined time step value.

3.1.5 System Coupling

System coupling is the most vital part in fluid structural interaction simulation as it links both the transient structural and FLUENT together to provide a final solution. Physics are coupled by passing loads across the fluid structure interfaces where CFD transfer forces on structure surfaces to FEA while FEA transfer displacements of solid structures to CFD. Individual physics are solved separately and then coupled sequentially or simultaneously until equilibrium is achieved. Figure 3.20 shows the flow chart on how the system coupling works in ANSYS.

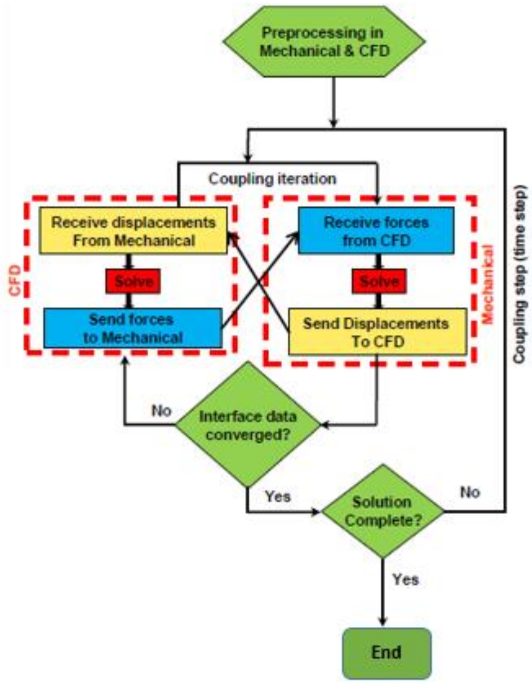


Figure 3.20. Flow chart showing how the system coupling works in ANSYS

In ANSYS, the two way FSI is set up as shown in Figure 3.21. The setup of transient structural and FLUENT are linked together to the system coupling for the solution purpose.

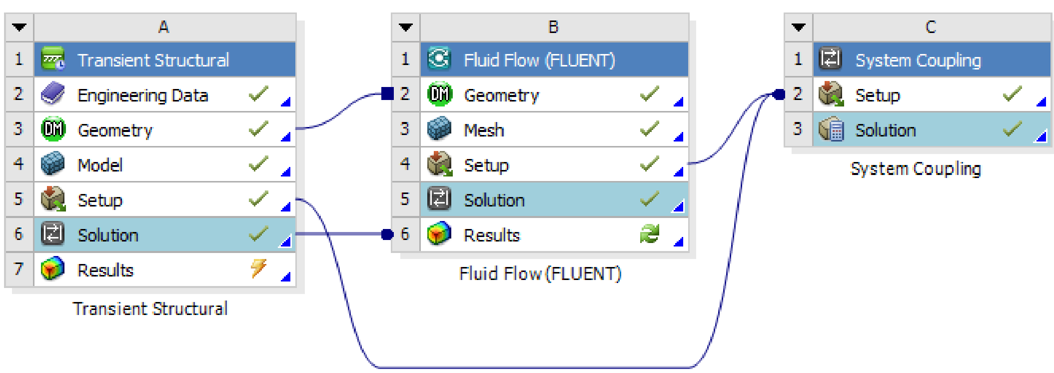


Figure 3.21. Two way FSI setup in ANSYS

In the system coupling, the end time is set to 0.5s with step size of 0.01s. Minimum iteration is set to 1 whereas maximum iterations are set to 30. Next, data transfers are established between Fluid Solid Interface in Transient Structural and fsi in FLUENT. Here, the surface of the fluid system around the leaflets transfers force to the surface of the