Contents

Chemical Engineering Summer Internship Report at the University of Waterloo

Computational Fluid Dynamic (CFD) analysis
of an antibiotic drug eluting on orthopedic implants simulation

Romrawin Chumpu

Submitted in partial fulfilment of the requirements of
the course CHS301 Chemical Engineering Traning
the bachelor's degree of Chemical Engineering
August 14, 2018

School of Bio-Chemical Engineering and Technology
Sirindhorn International Institute of Technology
Thammasat University
Thailand

Abstract

One of the branches of medical treatments for arthritis symptoms is Osteoarthritis. Well-Known orthopedic surgery to recover these symptoms is hip arthroplasty or hip replacement. The development is getting the emphasize on when we insert the artificial joint (implants) into our body, bacteria in the surrounding can come along with even the instruments have been sterilized. To protect our body, many researched try to add or coat antibiotics drug in the implant parts. From previous works studied to extend the time for drug-eluting on orthopedic implant from 2 days to 30 days. By controlled diffusion release on the system. On the orthopedic implant, the appropriate time and concentration of an antibiotic drug would suggest during the surgery. The fluid flow of antibiotic prescription and its release profile are predicted correctly from the simulation to decrease the risk of a real test. Therefore, the work aims to simulate the antibiotic drug(vancomycin) release profile from the given mathematical model and to observe the flow of antibiotic drug from various geometry. The procedure and simulation have done in ANSYS software. The result shows that the faster velocity inlet, the less density of medicine, and no reaction of the blood-drug condition given the more correction by R^2 0.944(95%CI) to the KMUTT model. For the theoretical model the slower velocity inlet, the high density of the drug from $500 to 1000 kg/m^3$, and no reaction conditions resulted in R^2 0.9999 (95%CI) for the theoretical model.

Exceutive Summary

`Background`: From previous works, the center of interest of orthopedic surgery includes the development of drug-eluting from an orthopedic implant. One of the branch in medical treatments for arthritis symptoms is Osteoarthritis or the pain on the hip joint. By the increasing of the patients, a well-known orthopedic surgery to recover these symptoms is hip arthroplasty or hip replacement. The development is getting the emphasize on when we insert the artificial joint (implants) into our body, bacteria from the surrounding can come along with even the instruments have been sterilized. To protect our body, many types of research try to add or coat antibiotics drug to the implant parts. The simulation project aims to stimulate the controlled-release of antibiotics medications from the various geometry of implant parts.

`Method`: We used computational fluid dynamic (CFD) to analyze to the model of diffusion release on orthopedic implant model. The ANSYS software used in the calculation. The experiment divided into two part; 2D cylinder analysis and 3D sphere analysis. To observe the factors and conditions, the test varies on velocity inlet, density, reaction, and pseudo-layer conditions.

`Results`: The results give the released profile of each conditions in the form of

$$\frac{Mt}{M\infty} = 1 - e^{-ax-b}$$

Where $\frac{Mt}{M\infty}$ refers to mass release at the time point over the total mass release, a and b refers as a constant. The model velocity-2 and density-1 resulted in the best performance for the KMUTT model. Also, velocity-1 and density between 2-3 resulted in the nearest correlation coefficient to the theoretical model

`Conclusion`: Simulation on orthopedic implant vancomycin diffusion can adequately perform in any condition. However, the accurate simulation also needs the status of the drug that nontoxic to the organism. For the simple simulation, we also vary on any conditions which can summarize that the slowest velocity inlet and the highest density with no reaction on blood-drug condition resulted in the longest time of release profile.

Acknowledgements

In the internship opportunity, I had with the University of Waterloo was the best chance for learning and professional development for my undergraduate life at Sirindhorn International Institute of Technology, Thammasat University. I am honored as a student who was very lucky as I was provided with these opportunities. I am so grateful for having a chance to meet so many wonderful, expertise and professional people who led me through this internship period.

First of all, I would highly thanks to my supervisor during my internship period, Prof. Peter L. Douglas, dean of undergraduate in the faculty of Engineering, University of Waterloo. Although, Professor had so many works to do. Your time for teaching me was very precious in my life as a young chemical engineering.

Without IA&CR department of SIIT, I would not have this opportunity, I would like to state my thanks to Nishino's and NSTDA scholarship for giving support on the fund during an internship.

I would genuinely thank my family, Assoc. Prof. Dr. Nirattaya and Assoc. Prof. Dr. Cholwich (scientist mentors from NSTDA), and my friend for supporting me to choose this direction of an internship.

Bearing in mind previous I am using this opportunity to express my most profound gratitude and special thanks to all of BCET professors that have taught ever subjects to me for the whole three years of undergraduate. The knowledge entirely paid off during an internship.

This report is dedicated to the memory of my first extended period working aboard. Even though the time was only two months period, but I always feel at home because of Mr. Poompong Chaiwongkhot and Ms. Tanyakarn Treeratanaphitak, the other Thai Student Association of Waterloo, for always having meetings, helping and sharing experiences together.

As in the path of a good researcher and chemical engineering, I would genuinely appreciate the works of previous researchers who dedicated themselves totally to their works and for the next generation.

Contents

List of Figures

List of Tables

Part I

Introduction

1 The Universiry of Waterloo

The University of Waterloo was estabished in 1957 by the innovators and the entrepreneur of a group of business. The University of Waterloo is a public research university with a main campus in Waterloo, Ontario, Canada. The university offers academic programs administered by six faculties and eleven faculty-based schools. The University of Waterloo is most famous for its cooperative education (co-op) programs, which allow the students to integrate their education with applicable work experiences. The University claims to operate the largest post-secondary co-operative education program in the world, with over 17,000 undergraduate students in over 140 co-operative education programs.

Figure 1: Sign of the University of Waterloo

Faculties and schools, which operate a combined total of 11 schools and over 50 academic departments

Innovative solutions, innovative education at Waterloo

University of Waterloo is the place was built to teach people to think in new and innovative ways. That meant reaching out across disciplines and faculties, sharing resources, and sparking new directions in research. It meant working hand-in-hand with industry, letting people own their intellectual property and the success that came from commercialization.

Faculties and Schools of the University of Waterloo	
Faculty	**School**
Applied Health Sciences	School of Public Health and Health Systems
Arts	School of Accounting and Finance Balsillie School of International Affairs Renison University College
Engineering	School of Architecture
Environment	School of Environment, Enterprise and Development School of Environment, Resources and Sustainability School of Planning
Mathematics	David R. Cheriton School of Computer Science
Science	School of Optometry and Vision Science School of Pharmacy

Table 1: Listings of faculties and schools in the University of Waterloo

Constructed on a foundation of science, engineering and math, Waterloo has also become a leader in environmental education, architecture, the arts, psychology and human health.

Figure 2: Trainee at the University of Waterloo sign

1.1 Department of Chemical Engineering

A chemical engineering building was the first to rise in 1958, Also, as the first facauty on the university. Chemical engineers are the person at the forefront of technology and their role in modern society is becoming increasingly important. At Department of Chemical Engineering, thdy design, implement and supervise industrial processes where matter undergoes change. This could be led in the pharmaceutical, pulp and paper, food or plastics industries for example - anywhere a transformation of matter occurs. Chemical engineering here also play a major role in the emerging field of nanotechnology, with applications in the development of new materials and devices. They develop new processes to prevent pollutants from being released into our environment or to remove them after they are already there. The research decrease our energy use by increasing the efficiency in fossil fuel refining plants, or by experimenting with new forms of energy generation and storage. Increasingly chemical engineers are becoming involved in the control, manipulation and production of biological systems as well, which have many important applications in the area of health care and food production.

1.1.1 Chemical Engineering Programs

A large faculty complement, with a wide variety of expertise ranging from biotechnology, nanotechnology, polymers and materials science, to environmental and energy topics like global climate change and fuel cells. Being Chemical Engineering student at the University of Waterloo have access to this expertise for career suggestions, research experience, or guidance during projects.

Options available to chemical engineering students:

- Biomechanics
- Environmental Engineering
- International Studies in Engineering
- Life Sciences
- Management Sciences
- Mathematics
- Physical Sciences
- Statistics
- Water Resources

1.1.2 Research and Development

The spirit of Waterloo's chemical engineering research is both collaborative and interdisciplinary. Our faculty members hold cross appointments in Biology, Chemistry, Civil Engineering, Mechanical Engineering, Earth Sciences, Environmental Studies and Physics.

With more than 200 graduate students and 39 faculty members, our research covers all the major areas of chemical engineering - from core areas, such as; polymers, reaction

engineering and process systems engineering, to emerging areas, such as; fuel cell research, tissue engineering and nanotechnology.

Our research strengths include:

- Biochemical engineering, tissue engineering and industrial biotechnology
- Chemical kinetics, catalysis and reactor design, energy conversion and fuel cells
- Colloid and surface science, interfacial engineering
- Electrochemical process research and modeling
- Environmental and pollution control
- Flow in porous media and enhanced oil recovery
- Mathematical analysis, statistics and control (process systems)
- Nanotechnology
- Polymer reaction engineering, polymer processing and rheology
- Separation Processes
- Transport phenomena

Over the years we have developed an extensive network of industrial connections. The Institute for Polymer Research, for example, has the highest density of polymer reaction engineering researchers internationally in a single group - a distinction that helped earn the institute a global reputation for research initiatives in a wide variety of companies.

The Cell Factory Bioprocessing Research Network is a multi-university research network funded by Natural Sciences and Engineering Research Council of Canada (NSERC) and industrial partners. CellNet's research aims to develop biological cells as miniature factories for new and improved bio-manufacturing strategies.

Our recently updated laboratories, including a shared analytical instrumentation laboratory, provide the latest equipment. There are dedicated facilities and software for computational work as well, including access to Canada's largest supercomputing cluster.

In our labs, student and visitor safety is considered to be of prime importance. All researchers are required to complete online safety training from the University of Waterloo Safety Office. In addition, all researchers must read the Chemical Engineering Safety Manual and write a report addressing the safety aspects of their research project. The report is submitted to the Departmental Safety Manager, Ralph Dickhout, prior to working in any of our laboratories.

---

2 Safety Practice for Employee

Being a short term researcher at the Uninversity of Waterloo have been a lot of process including safety introduction. Eech employee must take an onlin learning about safetry in workplace and laboratory.

Three courses that I have taked and these all provides to the employee:

- Employee WHMIS 2015

- Employee Safety Orientation (SO1001)
- Workplace Violence Awareness

2.1 Employee WHMIS 2015

The Workplace Hazardous Materials Information System (WHMIS) has aligned with the Globally Harmonized System of Classification and Labelling of Chemicals (GHS). GHS is a worldwide system. Its goal is to have a common set of rules for classifying hazardous products, common rules for labels, and a standard format for safety data sheets (SDSs).

This online training module familiarizes employees with WHMIS 2015 and covers:

- Learn the hazards associated with WHMIS-regulated products.
- Understand the WHMIS 2015 hazard classes and symbols.
- Learn the purpose, content and function of Safety Data Sheets (SDS).
- Learn the legal rights and duties under WHMIS.

2.2 Employee Safety Orientation

This safety training module covers health, safety and environment information and services that apply to University of Waterloo workers, which not only includes paid workers, but also post-doctoral fellows, graduate students, unpaid learners, teaching assistants and research assistants. In this safety training module you will learn important and valuable information about your role and your rights and responsibilities in health and safety procedures at Waterloo. This training module outlines general health and safety information, while your faculty or department will have additional health, safety and environment requirements that are specific to your work area.

2.2.1 Internal Responsibility System (IRS)

Within an organization, such as the University of Waterloo, everyone is responsible for health and safety. This means that all faculty, staff, students, visitors and contractors share responsibility for health, safety and environmental care. The particular practices that make a workplace safe are specific to the work that each of us does.

This system of practices and procedures is called the Internal Responsibility System, which is in place to keep everyone at Waterloo healthy and safe. It is critical that we all do our part to work collaboratively towards a healthy and safe work environment. The figure here depicts the IRS and the flow of responsibility and accountability.

Authority and responsibility for health and safety are delegated downward, from senior levels ALL the way down to EACH individual worker.

Accountability for health and safety flows upward, where individual workers are held to lesser account legally, and rely on their superiors to correct hazards that they cannot correct themselves.

2.2.2 Invisible Hazards

Not all hazards can be seen. You also need to think about less visible hazards related to your work - things like chemicals, fumes, toxic dust, germs and viruses in labs and healthcare workplaces.

Some of these hazards can make you very sick. Sometimes they make you sick right away; other times you don't know that you are sick until months or even years later. That's why it's important to know about these hazards now.

The University and your supervisors must implement controls to reduce or eliminate hazards in the workplace. The simplest method of hazard control is to eliminate or remove the hazard. If this is not a feasible option other methods such as engineering or administrative controls need to be implemented. These types of controls include physical guarding and the use of Standard Operating Procedures and they need to be considered before relying solely upon Personal Protective Equipment.

The Employer's Duty Make sure that supervisors know enough and have enough experience and training to know where safety hazards may be and keep workers safe and healthy while they work.

The Supervisor's Duty Inform workers of health and safety hazards.

The Worker's Duty Report hazards they know of to the supervisor or employer as soon as possible so they can fix it.

Employers, supervisors and workers work together to make the workplace safer.

This training module will help you better understand the health and safety systems in place at Waterloo and your personal role and responsibility.

Dealing with Injuries and Health and Safety Incidents Do you know what to do if there is an injury or incident while you are at work?

If you are not sure, you are not alone. As a worker at Waterloo knowing what the proper procedure is in the event of an injury or incident at work is your responsibility. It is important that you are clear on the standard procedure.

In this final section of the Health and Safety Orientation Module you will learn what the proper incident procedure is and details on how and where to get help and how to properly report incidents.

If you are injured at work seek appropriate medical attention immediately.

Do not hesitate to call 911 if you feel the situation requires emergency medical assistance.

The following steps must be followed in the event of an injury or incident at work:

1. Obtain first aid or appropriate medical aid if necessary.
2. As soon as possible report the injury to your supervisor.
3. With your supervisor, complete the Incident and Investigation Report and send the report to the Safety Office. In addition to reporting injuries, everyone is expected to report to their supervisor near misses that have a potential for serious loss or high severity of injury. Incidents of this type could potentially have caused:

- Loss of life
- Fire or explosion
- Critical injuries (e.g., broken arms/legs, amputation, severe bleeding, blindness, severe burns)
- Equipment or property damage

4. If applicable, file a Workplace Safety and Insurance Board (WSIB) claim and participate in return to work.

2.3 Workplace Violence Awareness

Workplace Violence and Harassment Prevention - for employee Awareness Training

The Occupational Health & Safety Act includes legislation that protects workers from workplace violence and harassment.

Workplace Violence can effect people in any business or occupation. Legislation require employers to protect employees from hazards in the workplace including violence. In University is commited to be violence free. Awareness will help to reduce the risk of violence by knowing what todo in the violent situation

Workplace violence is

- the excercise and attempt of physical force by a person against a worker in a workplace, that causes physical injury to the other workers.
- a behavior that a worker could reasonably explain as a threat to exercise physical force against the worker in workplace.

Policy and Requirements Policy #34 - Health, Safety and Environment

address workplace violence by including the definitions and requirements of the legislation and applies to the entire campus communitry.

- The university is committed to maintaining a working and learning environment that is violence free, strives to assess and minimize the risk of violence to ensure the reasonable of safeguards are perfectly protecting the employees, students and visitors from incident of violence on Universiry propety and events.
- The University has implemented a Workplace Violence Prevention Program, which includes measures and precedures to protect employees, students and visitors, to ensure

that Universiry employees have the appropriate information and instruction to protect from violence in workplace.
- All employees, students and visitors ahve a responsibility in maintaining a violence free working and learning environment by not engaging in acts of the violence and is required to report the incidents.
- Any act of violence is unacceptable conduct that will not be tolerated. Anyone engaging in activities of violation can be explused and banned from the University premises.

Domestic Violence The perpetrator and victim are in a personal relationship, such as a spouse of former spouse.

Communication Under workplace violence legislation, empolyers and supervisors must also provide the information to a worker about risk of the workplace violence from a person with history if:

- the worker can expect to encounter that person in the course of work, and
- if the worker may be at risk of physical injury

In these cases, personal information may be disclosed.

Types and Sources Violence includes:

1. Threatening Behaviour - intimidate or threaten someone

- Physical intimidation - shaking fists, yelling in face
- Throwinf objects
- Destruction of property
- Verbal, written or electtonically transmitted threats to cause physical harm or damage to perperty

1. Physical Attack - actual exercise of physical force or assaulting someone include:

- hitting
- pushing
- kicking
- punching
- use of a weapon A physical assault is important that behaviors associated with violence are identified and addressed to prevent them from escalating into physical assault.

Risk Groups People who work in particular job

- Work with the general public
- Handle money
- Carry enforcement duties
- Provide counselling services
- Work with people under the influence of drugs or alcohol

Risk Assessment is the measurement for preparing the incidents

Preventive Measure measurement to reduce the likelihood of someone engaging in violent behaviour. 1) workplace design - consider physical factors such as building security to reduce

the opportunity for the theft and violence. 2) administrative practices - the procedures that reduce the risks involved in handling cash or valuables. 3) work practices - individual groups practices that protect your personal safety on a day-to-day basis.

---

3 Simulation fields

3.1 Compulational fluid dynamics (CFD) analysis

CFD is a branch field of fluid mechanics that combines numerical analysis and data structures to solve and analyze problems that result on fluid flows scheme. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

3.1.1 Background and history of CFD

The fundamental basis of almost all CFD problems is the *Navier-Stokes equations*, which define many single-phase (gas or liquid, but not both) fluid flows. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations.

Historically, methods were first developed to solve the linearized potential equations. Two-dimensional (2D) methods, using conformal transformations of the flow about a cylinder to the flow about an airfoil were developed in the 1930s.

One of the earliest type of calculations resembling modern CFD are those by Lewis Fry Richardson, in the sense that these calculations used finite differences and divided the physical space in cells. Although they failed dramatically, these calculations, together with Richardson's book "Weather prediction by numerical process", set the basis for modern CFD and numerical meteorology. In fact, early CFD calculations during the 1940s using ENIAC used methods close to those in Richardson's 1922 book.

The computer power available paced development of three-dimensional methods. Probably the first work using computers to model fluid flow, as governed by the Navier-Stokes equations, was performed at Los Alamos National Lab, in the T3 group. This group was led by Francis H. Harlow, who is widely considered as one of the pioneers of CFD. From 1957 to late 1960s, this group developed a variety of numerical methods to simulate transient two-dimensional fluid

flows, such as Particle-in-cell method (Harlow, 1957), Fluid-in-cell method (Gentry, Martin and Daly, 1966), Vorticity stream function method (Jake Fromm, 1963), and Marker-and-cell method (Harlow and Welch, 1965). Fromm's vorticity-stream-function method for 2D, transient, incompressible flow was the first treatment of strongly contorting incompressible flows in the world.

The first paper with three-dimensional model was published by John Hess and A.M.O. Smith of Douglas Aircraft in 1967. This method discretized the surface of the geometry with panels, giving rise to this class of programs being called Panel Methods. Their method itself was simplified, in that it did not include lifting flows and hence was mainly applied to ship hulls and aircraft fuselages. The first lifting Panel Code (A230) was described in a paper written by Paul Rubbert and Gary Saaris of Boeing Aircraft in 1968. In time, more advanced three-dimensional Panel Codes were developed at Boeing (PANAIR, A502), Lockheed (Quadpan), Douglas (HESS), McDonnell Aircraft (MACAERO), NASA (PMARC)and Analytical Methods (WBAERO, USAERO and VSAERO). Some (PANAIR, HESS and MACAERO) were higher order codes, using higher order distributions of surface singularities, while others (Quadpan, PMARC, USAERO and VSAERO) used single singularities on each surface panel. The advantage of the lower order codes was that they ran much faster on the computers of the time. Today, VSAERO has grown to be a multi-order code and is the most widely used program of this class. It has been used in the development of many submarines, surface ships, automobiles, helicopters, aircraft, and more recently wind turbines. Its sister code, USAERO is an unsteady panel method that has also been used for modeling such things as high speed trains and racing yachts. The NASA PMARC code from an early version of VSAERO and a derivative of PMARC, named CMARC, is also commercially available.

In the two-dimensional realm, a number of Panel Codes have been developed for airfoil analysis and design. The codes typically have a boundary layer analysis included, so that viscous effects can be modeled. Professor Richard Eppler of the University of Stuttgart developed the PROFILE code, partly with NASA funding, which became available in the early 1980s. This was soon followed by MIT Professor Mark Drela's XFOIL code. Both PROFILE and XFOIL incorporate two-dimensional panel codes, with coupled boundary layer codes for airfoil analysis work. PROFILE uses a conformal transformation method for inverse airfoil design, while XFOIL has both a conformal transformation and an inverse panel method for airfoil design.

An intermediate step between Panel Codes and Full Potential codes were codes that used the Transonic Small Disturbance equations. In particular, the three-dimensional WIBCO code, developed by Charlie Boppe of Grumman Aircraft in the early 1980s has seen heavy use.

Developers turned to Full Potential codes, as panel methods could not calculate the non-linear flow present at transonic speeds. The first description of a means of using the Full Potential equations was published by Earll Murman and Julian Cole of Boeing in 1970. Frances Bauer, Paul Garabedian and David Korn of the Courant Institute at New York University (NYU) wrote a series of two-dimensional Full Potential airfoil codes that were widely used, the most important being named Program H. A further growth of Program H was developed by Bob Melnik and his group at Grumman Aerospace as Grumfoil. Antony Jameson, originally at Grumman Aircraft and the Courant Institute of NYU, worked with David Caughey to

develop the important three-dimensional Full Potential code FLO22 in 1975. Many Full Potential codes emerged after this, culminating in Boeing's Tranair (A633) code, which still sees heavy use.

The next step was the Euler equations, which promised to provide more accurate solutions of transonic flows. The methodology used by Jameson in his three-dimensional FLO57 code (1981) was used by others to produce such programs as Lockheed's TEAM program and IAI/Analytical Methods' MGAERO program. MGAERO is unique in being a structured cartesian mesh code, while most other such codes use structured body-fitted grids (with the exception of NASA's highly successful CART3D code, Lockheed's SPLITFLOW code and Georgia Tech's NASCART-GT). Antony Jameson also developed the three-dimensional AIRPLANE code which made use of unstructured tetrahedral grids.

In the two-dimensional realm, Mark Drela and Michael Giles, then graduate students at MIT, developed the ISES Euler program (actually a suite of programs) for airfoil design and analysis. This code first became available in 1986 and has been further developed to design, analyze and optimize single or multi-element airfoils, as the MSES program. MSES sees wide use throughout the world. A derivative of MSES, for the design and analysis of airfoils in a cascade, is MISES, developed by Harold "Guppy" Youngren while he was a graduate student at MIT.

The Navier-Stokes equations were the ultimate target of development. Two-dimensional codes, such as NASA Ames' ARC2D code first emerged. A number of three-dimensional codes were developed (ARC3D, OVERFLOW, CFL3D are three successful NASA contributions), leading to numerous commercial packages.

3.1.2 Methodology

In all of these approaches the same basic procedure is followed.

During preprocessing

- The geometry and physical bounds of the problem can be defined using computer aided design (CAD). From there, data can be suitably processed (cleaned-up) and the fluid volume (or fluid domain) is extracted.
- The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform, structured or unstructured, consisting of a combination of hexahedral, tetrahedral, prismatic, pyramidal or polyhedral elements.
- The physical modeling is defined - for example, the equations of fluid motion + enthalpy + radiation + species conservation
- Boundary conditions are defined. This involves specifying the fluid behaviour and properties at all bounding surfaces of the fluid domain. For transient problems, the initial conditions are also defined.

The simulation is started and the equations are solved iteratively as a steady-state or transient.

Finally a postprocessor is used for the analysis and visualization of the resulting solution.

Discretization methods

The stability of the selected discretisation is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretisation handles discontinuous solutions gracefully. The Euler equations and Navier-Stokes equations both admit shocks, and contact surfaces.

Some of the discretization methods being used are:

Finite volume method

The finite volume method (FVM) is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large problems, high Reynolds number turbulent flows, and source term dominated flows (like combustion).

In the finite volume method, the governing partial differential equations (typically the Navier-Stokes equations, the mass and energy conservation equations, and the turbulence equations) are recast in a conservative form, and then solved over discrete control volumes. This discretization guarantees the conservation of fluxes through a particular control volume. The finite volume equation yields governing equations in the form,

$$\frac{\partial}{\partial t}\iiint Q\,dV + \iint F\,d\mathbf{A} = 0$$

, where $ Q$ is the vector of conserved variables, F is the vector of fluxes, $ V$ is the volume of the control volume element, and $ $\mathbf{A}$ is the surface area of the control volume element.

Finite element method

The finite element method (FEM) is used in structural analysis of solids, but is also applicable to fluids. However, the FEM formulation requires special care to ensure a conservative solution. The FEM formulation has been adapted for use with fluid dynamics governing equations. Although FEM must be carefully formulated to be conservative, it is much more stable than the finite volume approach. However, FEM can require more memory and has slower solution times than the FVM.

In this method, a weighted residual equation is formed:

$$R_i = \iiint W_i Q\,dV^e$$

where $ R_{i}$ is the equation residual at an element vertex $ i, Q $ is the conservation equation expressed on an element basis, $ W_{i}$ is the weight factor, and V^e is the volume of the element.

Finite difference method

Main article: Finite difference method The finite difference method (FDM) has historical importance and is simple to program. It is currently only used in few specialized codes, which handle complex geometry with high accuracy and efficiency by using embedded boundaries or overlapping grids (with the solution interpolated across each grid).

$$\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0$$

where Q is the vector of conserved variables, and FF, GG, and HH are the fluxes in the xx, yy, and zz directions respectively.

Spectral element method

Spectral element method is a finite element type method. It requires the mathematical problem (the partial differential equation) to be cast in a weak formulation. This is typically done by multiplying the differential equation by an arbitrary test function and integrating over the whole domain. Purely mathematically, the test functions are completely arbitrary - they belong to an infinite-dimensional function space. Clearly an infinite-dimensional function space cannot be represented on a discrete spectral element mesh; this is where the spectral element discretization begins. The most crucial thing is the choice of interpolating and testing functions. In a standard, low order FEM in 2D, for quadrilateral elements the most typical choice is the bilinear test or interpolating function of the form $v(x, y) = ax + by + cxy + d$. In a spectral element method however, the interpolating and test functions are chosen to be polynomials of a very high order (typically e.g. of the 10th order in CFD applications). This guarantees the rapid convergence of the method. Furthermore, very efficient integration procedures must be used, since the number of integrations to be performed in numerical codes is big. Thus, high order Gauss integration quadratures are employed, since they achieve the highest accuracy with the smallest number of computations to be carried out. At the time there are some academic CFD codes based on the spectral element method and some more are currently under development, since the new time-stepping schemes arise in the scientific world.

Boundary element method

In the boundary element method, the boundary occupied by the fluid is divided into a surface mesh.

High-resolution discretization schemes

High-resolution schemes are used where shocks or discontinuities are present. Capturing sharp changes in the solution requires the use of second or higher-order numerical schemes that do not introduce spurious oscillations. This usually necessitates the application of flux limiters to ensure that the solution is total variation diminishing.

Turbulence models

In computational modeling of turbulent flows, one common objective is to obtain a model that can predict quantities of interest, such as fluid velocity, for use in engineering designs of the system being modeled. For turbulent flows, the range of length scales and complexity of phenomena involved in turbulence make most modeling approaches prohibitively expensive; the resolution required to resolve all scales involved in turbulence is beyond what is computationally possible. The primary approach in such cases is to create numerical models to approximate unresolved phenomena. This section lists some commonly used computational models for turbulent flows.

Turbulence models can be classified based on computational expense, which corresponds to the range of scales that are modeled versus resolved (the more turbulent scales that are resolved, the finer the resolution of the simulation, and therefore the higher the computational cost). If a majority or all of the turbulent scales are not modeled, the computational cost is very low, but the tradeoff comes in the form of decreased accuracy.

In addition to the wide range of length and time scales and the associated computational cost, the governing equations of fluid dynamics contain a non-linear convection term and a non-linear and non-local pressure gradient term. These nonlinear equations must be solved numerically with the appropriate boundary and initial conditions.

Reynolds-averaged Navier-Stokes

Reynolds-averaged Navier-Stokes (RANS) equations are the oldest approach to turbulence modeling. An ensemble version of the governing equations is solved, which introduces new apparent stresses known as Reynolds stresses. This adds a second order tensor of unknowns for which various models can provide different levels of closure. It is a common misconception that the RANS equations do not apply to flows with a time-varying mean flow because these equations are 'time-averaged'. In fact, statistically unsteady (or non-stationary) flows can equally be treated. This is sometimes referred to as URANS. There is nothing inherent in Reynolds averaging to preclude this, but the turbulence models used to close the equations are valid only as long as the time over which these changes in the mean occur is large compared to the time scales of the turbulent motion containing most of the energy.

RANS models can be divided into two broad approaches:

Boussinesq hypothesis

This method involves using an algebraic equation for the Reynolds stresses which include determining the turbulent viscosity, and depending on the level of sophistication of the model, solving transport equations for determining the turbulent kinetic energy and dissipation. Models include k-?? (Launder and Spalding), Mixing Length Model (Prandtl), and Zero Equation Model (Cebeci and Smith). The models available in this approach are often referred to by the number of transport equations associated with the method. For example, the Mixing Length model is a "Zero Equation" model because no transport equations are solved; the $k - \epsilon$ is a "Two Equation" model because two transport equations (one for k and one for ϵ) are solved.

Reynolds stress model (RSM)

This approach attempts to actually solve transport equations for the Reynolds stresses. This means introduction of several transport equations for all the Reynolds stresses and hence this approach is much more costly in CPU effort.

Large eddy simulation

Volume rendering of a non-premixed swirl flame as simulated by LES.

Large eddy simulation (LES) is a technique in which the smallest scales of the flow are removed through a filtering operation, and their effect modeled using subgrid scale models.

This allows the largest and most important scales of the turbulence to be resolved, while greatly reducing the computational cost incurred by the smallest scales. This method requires greater computational resources than RANS methods, but is far cheaper than DNS.

Detached eddy simulation

Detached eddy simulations (DES) is a modification of a RANS model in which the model switches to a subgrid scale formulation in regions fine enough for LES calculations. Regions near solid boundaries and where the turbulent length scale is less than the maximum grid dimension are assigned the RANS mode of solution. As the turbulent length scale exceeds the grid dimension, the regions are solved using the LES mode. Therefore, the grid resolution for DES is not as demanding as pure LES, thereby considerably cutting down the cost of the computation. Though DES was initially formulated for the Spalart-Allmaras model (Spalart et al., 1997), it can be implemented with other RANS models (Strelets, 2001), by appropriately modifying the length scale which is explicitly or implicitly involved in the RANS model. So while Spalart-Allmaras model based DES acts as LES with a wall model, DES based on other models (like two equation models) behave as a hybrid RANS-LES model. Grid generation is more complicated than for a simple RANS or LES case due to the RANS-LES switch. DES is a non-zonal approach and provides a single smooth velocity field across the RANS and the LES regions of the solutions.

Direct numerical simulation

Direct numerical simulation (DNS) resolves the entire range of turbulent length scales. This marginalizes the effect of models, but is extremely expensive. The computational cost is proportional to Re^3. DNS is intractable for flows with complex geometries or flow configurations.

Coherent vortex simulation

The coherent vortex simulation approach decomposes the turbulent flow field into a coherent part, consisting of organized vortical motion, and the incoherent part, which is the random background flow. This decomposition is done using wavelet filtering. The approach has much in common with LES, since it uses decomposition and resolves only the filtered portion, but different in that it does not use a linear, low-pass filter. Instead, the filtering operation is based on wavelets, and the filter can be adapted as the flow field evolves. Farge and Schneider tested the CVS method with two flow configurations and showed that the coherent portion of the flow exhibited the $-\frac{40}{39}$ energy spectrum exhibited by the total flow, and corresponded to coherent structures (vortex tubes), while the incoherent parts of the flow composed homogeneous background noise, which exhibited no organized structures. Goldstein and Vasilyev applied the FDV model to large eddy simulation, but did not assume that the wavelet filter completely eliminated all coherent motions from the subfilter scales. By employing both LES and CVS filtering, they showed that the SFS dissipation was dominated by the SFS flow field's coherent portion.

PDF methods

Probability density function (PDF) methods for turbulence, first introduced by Lundgren, are based on tracking the one-point PDF of the velocity, $f_V(\boldsymbol{v}; \boldsymbol{x}, t)d\boldsymbol{v}$, which gives the probability

of the velocity at point $\boldsymbol{x}$ being between $\boldsymbol{v}$ and $\boldsymbol{v} + d\boldsymbol{v}$. This approach is analogous to the kinetic theory of gases, in which the macroscopic properties of a gas are described by a large number of particles. PDF methods are unique in that they can be applied in the framework of a number of different turbulence models; the main differences occur in the form of the PDF transport equation. For example, in the context of large eddy simulation, the PDF becomes the filtered PDF. PDF methods can also be used to describe chemical reactions. and are particularly useful for simulating chemically reacting flows because the chemical source term is closed and does not require a model. The PDF is commonly tracked by using Lagrangian particle methods; when combined with large eddy simulation, this leads to a Langevin equation for subfilter particle evolution.

Vortex method

The vortex method is a grid-free technique for the simulation of turbulent flows. It uses vortices as the computational elements, mimicking the physical structures in turbulence. Vortex methods were developed as a grid-free methodology that would not be limited by the fundamental smoothing effects associated with grid-based methods. To be practical, however, vortex methods require means for rapidly computing velocities from the vortex elements - in other words they require the solution to a particular form of the N-body problem (in which the motion of N objects is tied to their mutual influences). A breakthrough came in the late 1980s with the development of the fast multipole method (FMM), an algorithm by V. Rokhlin (Yale) and L. Greengard (Courant Institute). This breakthrough paved the way to practical computation of the velocities from the vortex elements and is the basis of successful algorithms. They are especially well-suited to simulating filamentary motion, such as wisps of smoke, in real-time simulations such as video games, because of the fine detail achieved using minimal computation.

Software based on the vortex method offer a new means for solving tough fluid dynamics problems with minimal user intervention. All that is required is specification of problem geometry and setting of boundary and initial conditions. Among the significant advantages of this modern technology;

It is practically grid-free, thus eliminating numerous iterations associated with RANS and LES. All problems are treated identically. No modeling or calibration inputs are required. Time-series simulations, which are crucial for correct analysis of acoustics, are possible. The small scale and large scale are accurately simulated at the same time.

Vorticity confinement method

The vorticity confinement (VC) method is an Eulerian technique used in the simulation of turbulent wakes. It uses a solitary-wave like approach to produce a stable solution with no numerical spreading. VC can capture the small-scale features to within as few as 2 grid cells. Within these features, a nonlinear difference equation is solved as opposed to the finite difference equation. VC is similar to shock capturing methods, where conservation laws are satisfied, so that the essential integral quantities are accurately computed.

Linear eddy model

The Linear eddy model is a technique used to simulate the convective mixing that takes place

in turbulent flow. Specifically, it provides a mathematical way to describe the interactions of a scalar variable within the vector flow field. It is primarily used in one-dimensional representations of turbulent flow, since it can be applied across a wide range of length scales and Reynolds numbers. This model is generally used as a building block for more complicated flow representations, as it provides high resolution predictions that hold across a large range of flow conditions.

Two-phase flow

Simulation of bubble swarm using volume of fluid method The modeling of two-phase flow is still under development. Different methods have been proposed, including the Volume of fluid method, the Level set method and front tracking. These methods often involve a tradeoff between maintaining a sharp interface or conserving mass. This is crucial since the evaluation of the density, viscosity and surface tension is based on the values averaged over the interface. Lagrangian multiphase models, which are used for dispersed media, are based on solving the Lagrangian equation of motion for the dispersed phase.

Solution algorithms Discretization in the space produces a system of ordinary differential equations for unsteady problems and algebraic equations for steady problems. Implicit or semi-implicit methods are generally used to integrate the ordinary differential equations, producing a system of (usually) nonlinear algebraic equations. Applying a Newton or Picard iteration produces a system of linear equations which is nonsymmetric in the presence of advection and indefinite in the presence of incompressibility. Such systems, particularly in 3D, are frequently too large for direct solvers, so iterative methods are used, either stationary methods such as successive overrelaxation or Krylov subspace methods. Krylov methods such as GMRES, typically used with preconditioning, operate by minimizing the residual over successive subspaces generated by the preconditioned operator.

Multigrid has the advantage of asymptotically optimal performance on many problems. Traditional solvers and preconditioners are effective at reducing high-frequency components of the residual, but low-frequency components typically require many iterations to reduce. By operating on multiple scales, multigrid reduces all components of the residual by similar factors, leading to a mesh-independent number of iterations.

For indefinite systems, preconditioners such as incomplete LU factorization, additive Schwarz, and multigrid perform poorly or fail entirely, so the problem structure must be used for effective preconditioning. Methods commonly used in CFD are the SIMPLE and Uzawa algorithms which exhibit mesh-dependent convergence rates, but recent advances based on block LU factorization combined with multigrid for the resulting definite systems have led to preconditioners that deliver mesh-independent convergence rates.

Unsteady aerodynamics

CFD made a major break through in late 70s with the introduction of LTRAN2, a 2-D code to model oscillating airfoils based on transonic small perturbation theory by Ballhaus and associates. It uses a Murman-Cole switch algorithm for modeling the moving shock-waves. Later it was extended to 3-D with use of a rotated difference scheme by AFWAL/Boeing that resulted in LTRAN3.

Biomedical engineering

Simulation of blood flow in a human aorta

CFD investigations are used to clarify the characteristics of aortic flow in detail that are otherwise invisible to experimental measurements. To analyze these conditions, CAD models of the human vascular system are extracted employing modern imaging techniques. A 3D model is reconstructed from this data and the fluid flow can be computed. Blood properties like Non-Newtonian behavior and realistic boundary conditions (e.g. systemic pressure) have to be taken into consideration. Therefore, making it possible to analyze and optimize the flow in the cardiovascular system for different applications.

CPU versus GPU Traditionally, CFD simulations are performed on CPU's. In a more recent trend, simulations are also performed on GPU's. These typically contain slower but more processors. For CFD algorithms that feature good parallellisation performance (i.e. good speed-up by adding more cores) this can greatly reduce simulation times. Lattice-Boltzman methods are a typical example of codes that scale well on GPU's.

---

4 Project background

My work involves the simulation of fluid from the object which uses computational fluid dynamic (CFD) analysis to simulate an antibiotic drug release profile by the given mathematical model. To consistly simplify and comapare the release profiles, the mathematical model will present in this form,

$$\frac{Mt}{M\infty} = 1 - e^{-ax-b}$$

Where $\frac{Mt}{M\infty}$ refers to mass release at the time point over the total mass release, a and b refers constant. This equation will be compared in the Chapter 2.

Part II

Training Activities

5 Background

5.1 Project background

From previous works, the center of interest of orthopedic surgery includes the development of drug eluting from orthopedic implant. One of the branch in medical treatments for athritis symptoms is also Osteoarthritis. By the increaseing of the patients, a well-known orthopedic surgery to recover this symptoms is hip arthroplasty or hip replacement. The development is getting the emphasize on when we insert the the artificial joint (implants) to our body, bacteria from the surrounding can come along with even the instrtments have been sterilized. To protect our body, many researches try to add or coat antibiotics drug to the implant parts. The simulation project aims to stimulate the controlled-release of antibiotics drugs from the implant parts.

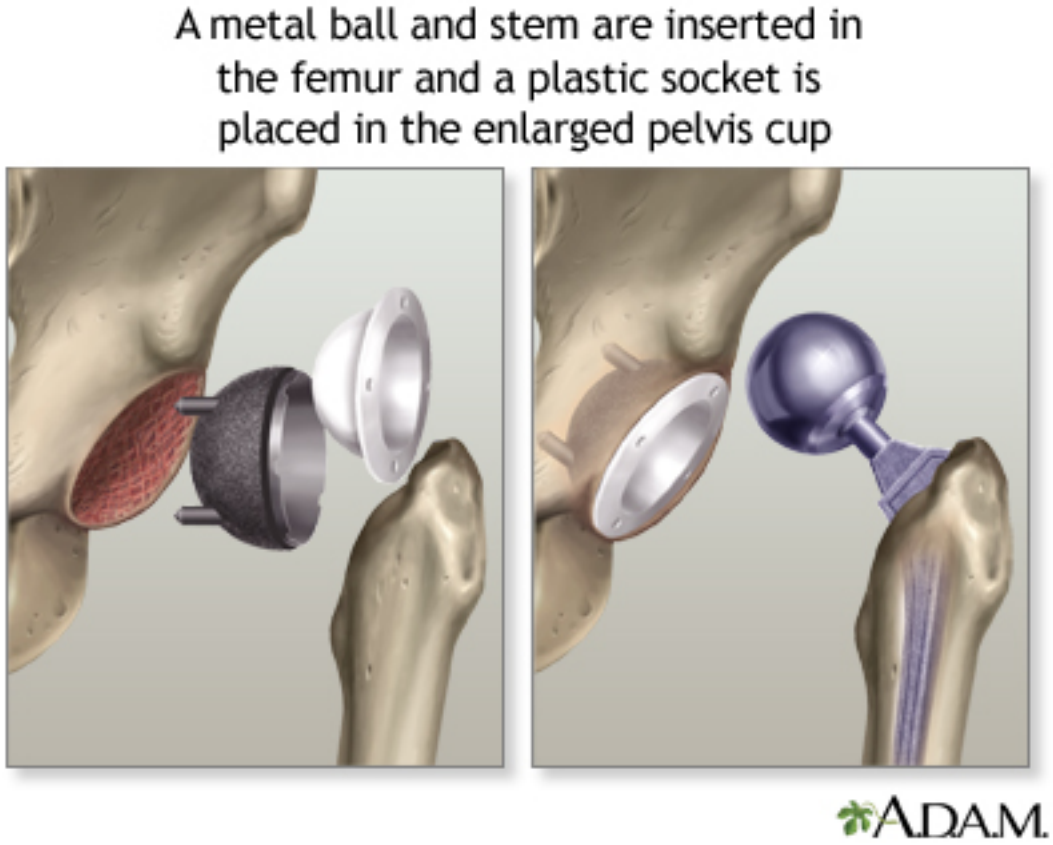


Figure 3: Hip joint replacement, A.D.A.M. Health Encyclopedia

The new hip that replaces the old one is made up of these parts:

- Acetabular Cup or a socket, which is usually made of strong metal.
- Polyethylene Insert, which fits inside the socket. It is most often polymer form.
- Femoral Head, this part is artificial part of your thigh bone.
- Neck and femoral stem are attached to the thigh bone to anchor the joint.

Figure 4: Hip replacement parts

The previous work used Vancomycin as an antibiotics drug stores in reservior inside the coated plastic surface geometry. The plastic is formed by poly-L-lactic acid (PLLA) polymer. On this project, we will study release profile of vancomycin through PLLA layer by Compulational Fluid Dynamics (CFD) analysis based on the properties of PLLA and the types of orthopedic implants.

From the objective of the studies, to find the vancomysin release profile from various surface object. I translate the complicated shape as in Figure 4 to the simple forms. The simple geometry for the controlled-release are sphere, half-sphere and cylinder respectively as shown in Figure 5.

Figure 5: Model geometry: shpere, half-sphere and cylinder surface models

We will use these principle shapes to simulate the fluid flow by using ANSYS software. Then, we can design the vancomysin delivery from various types of orthopedic implants. Surface of the implants will be coated by containing thin surface of vancomysin reservior.

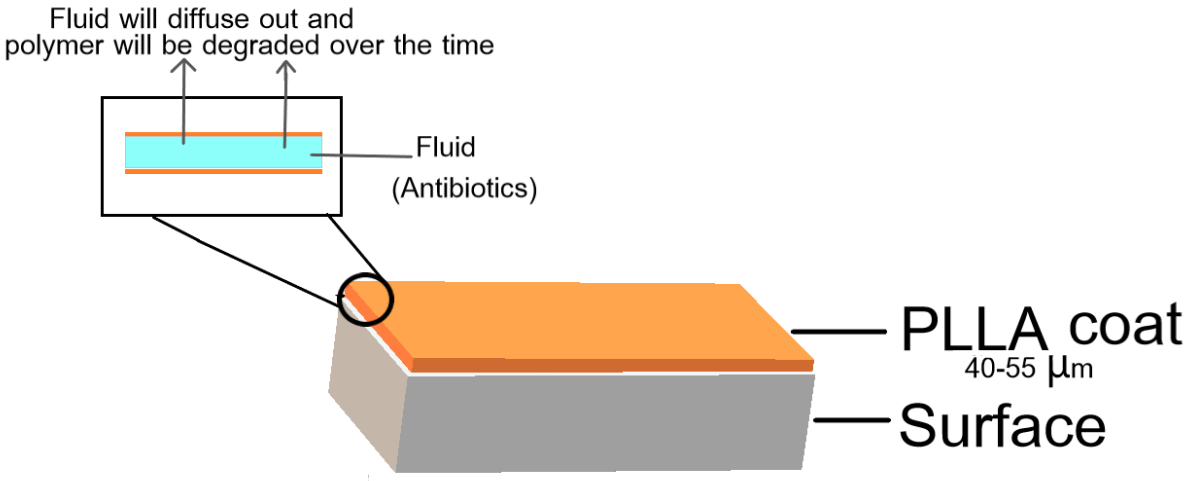
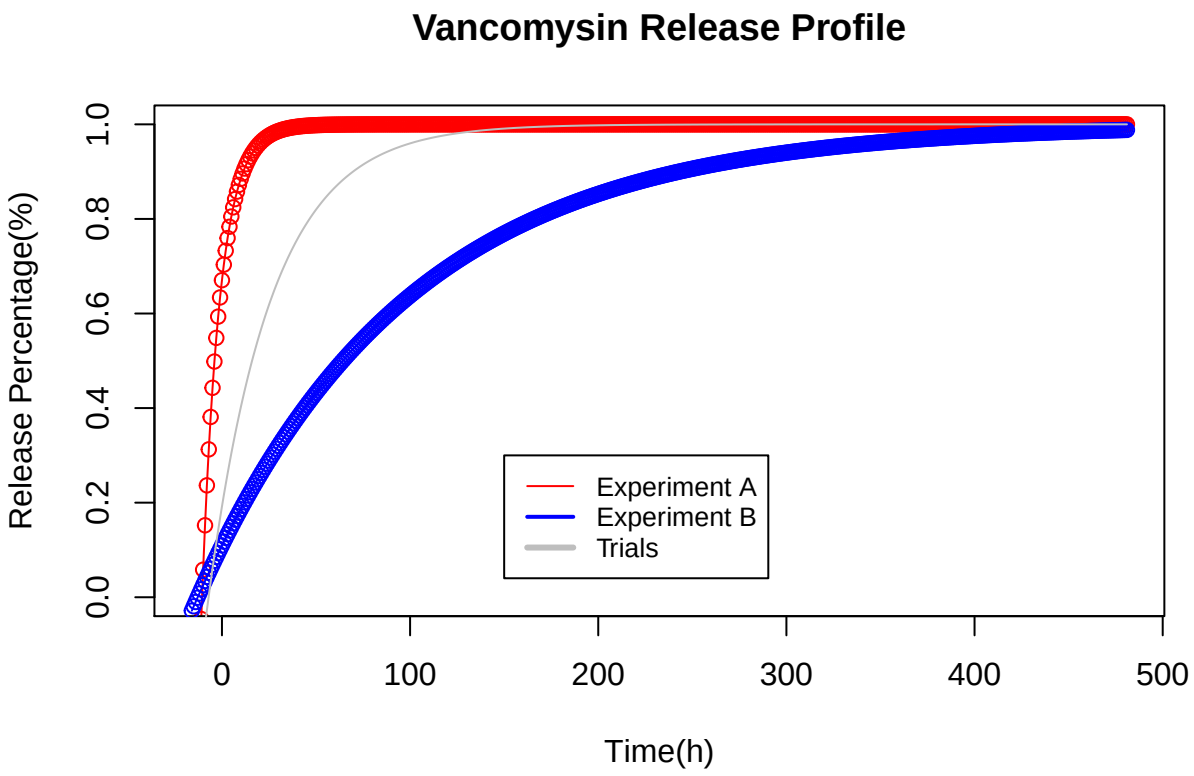


Figure 6: Surface geometry is coated by PLLA

From the previous studies, they conducted *in vitro* study of release profile, And, the graph between time(h) and release percentage(%) are shown below.



For experiment A (red line), the mathematical model shown as,

$$\frac{Mt}{M\infty} = 1 - e^{-0.105x - 1.110}$$

.

From experiment B (blue line), the mathematical model shown as,

$$\frac{Mt}{M\infty} = 1 - e^{-0.009x - 0.116}$$

. This is the desirable mathematical model for drug diffusion

The result from experiment A defined as undesirable brust relase that only release on 3 days. They would like to extend the release profile up to 20 days as experiment B.

Solving brust release problem, previous works incresed the release time by:

- adding multi-layer of polymer coat. [1]
- modify polymer composition, or trying other materials, like ceramic or metal. [2]

We will find the best vancomysin release profile from various orthopedic implants and the properties of polymer cost by computational fluid dynamic (CFD) simulation.

5.2 Fundamental Mechanisms

Assume we have a metal sphere, it is coated by the polymer that contain antibiotics drug (A) at the head joint of the bone.

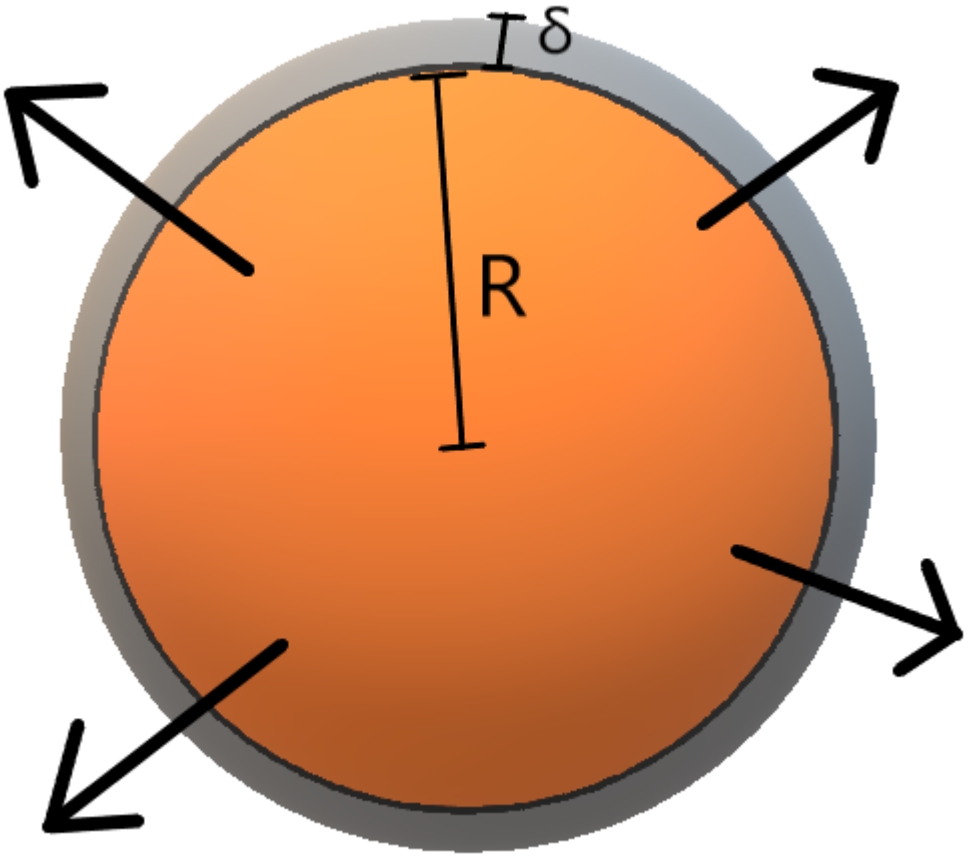


Figure 7: drug diffusion out of the spherical surface

The drug-release through the polymer to environment liquid. Assume we have the uniform concentration as C_{Ao}. The system assume to be steady state condition which is $in = out$. The coat is infiniteimal thin.

Chemical formula of the system, as shown when the drug diffusion from inside A to outside and then the compound react with the blood formed to be B

$A \rightarrow B$

5.2.1 Shell balance

From the system, the drug only diffusion to outside direction of the sphere. Assume the thickness of the coat is constant.

Material balance: $N_{Ar}(A\pi r^2)\Big|_{r+\delta} = 0$

When $\delta \to 0$

$$\frac{d}{dr}(N_{Ar}r^2) = 0$$

From Fick's first law of diffustion, the equimolar counter-diffustion states the flux as $N_{Ar} = -D_{ab}\frac{dc}{dr}$

Boundary conditions are stated as

- at $r = R$, concentration $C_A = 0$
- at $r = R + \delta$, concentration $C_A = C_{Ao}$

$\frac{d}{dr}(-D_{ab}\frac{dc}{dr}r^2) = 0$

$\int_0^r \frac{d}{dr}(-D_{ab}\frac{dc}{dr}r^2)dr = \int_0^r 0dr$

$-D_{ab}\frac{dc}{dr} = C_1$

$-D_{ab}\frac{dc}{dr} = \frac{C_1}{r^2}$

$-D_{ab}\frac{1}{C_1}d_{C_A} = \frac{1}{r^2}dr$

$\int_0^{C_A} -D_{ab}\frac{1}{C_1}d_{C_A} = \int_0^r \frac{1}{r^2}dr$

$\frac{C_A}{C_1} = -\frac{1}{r} + C_2$

$$C_A = -\frac{C_1}{r} + C_2$$

Applies to boundary conditions:

- at
$$r = R \to 0 = -\frac{C_1}{r} + C_2 \tag{1}$$
- at
$$r = R + \delta \to 0 = -\frac{C_1}{R+\delta} + C_2 \tag{2}$$

(1) − (2) :

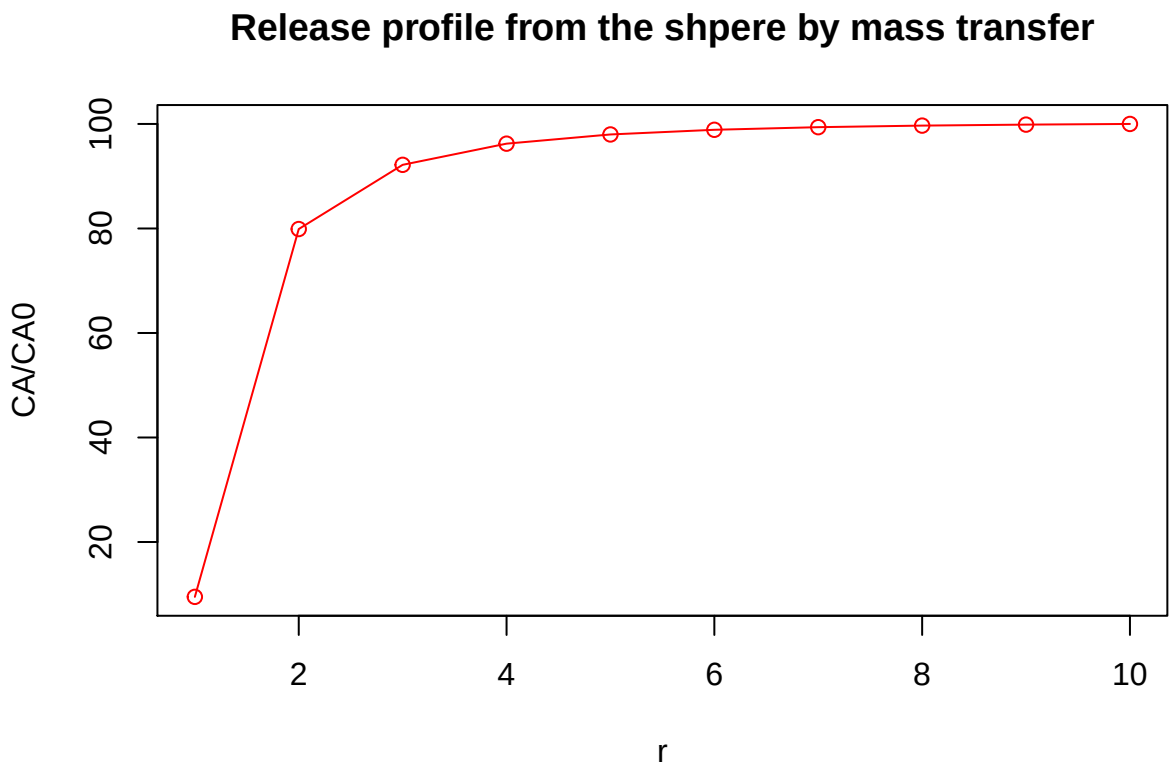
$$C_1 = \frac{R}{\delta}(R+\delta)C_{Ao}$$

$$C_2 = \frac{1}{\delta}(R+\delta)C_{Ao}$$

$$C_A = -\frac{R(R+\delta)C_{Ao}}{r\delta} + \frac{1}{\delta}(R+\delta)C_{Ao}$$

$$\frac{C_A}{C_{Ao}} = -\frac{1}{r}(R+\delta)(1-\frac{R}{r})$$

```
R = 10
delta = 0.055
C_A0 = 1
```



5.2.2 Diffustion eluting out of the sphere

By the process diffusion limited drug delivery

A spherical hip joint that contains antibiotics is controlled drug-release form the coat which can be assumed by the uniform concentration of the drug around the sphere.

By combing continuity equation with Fick's law of diffusion

$\frac{\partial C_A}{\partial t} + \nabla N_A = R_A$ (reaction)

Assume antibiotics drug does not change the form by diffusion, the equation has no reaction.

$\nabla N_A = \frac{1}{r^2}\frac{\partial}{\partial r}(r^2 N_{A,r}) + \frac{1}{r\sin\theta}\frac{\partial}{\partial\theta}(N_{A,\theta}\sin\theta) + \frac{1}{r\sin\phi}\frac{\partial N_{A,\phi}}{\partial\phi}(N_{A,\phi})$

the concentrations over θ and ϕ coordinates are constant over the time.

So, $\nabla N_A = \frac{1}{r^2}\frac{\partial}{\partial r}(r^2 N_{A,r})$

$\frac{\partial C_A}{\partial t} + \frac{1}{r^2}\frac{\partial}{\partial r}(r^2 N_{A,r}) = 0$

From Fick's second law of diffusion

$N_{A,r} = -cD_{y_A}\nabla y_A + y_A(N_A + N_B)$

where equimolar concentration of A and B, $N_A + N_B = 0$

$N_{A,r} = -CD_{y_A}\frac{\partial y_A}{\partial r}$

$y_A = \frac{C_A}{C} \rightarrow \frac{dy_A}{dr} = \frac{1}{C}\frac{dC_A}{dr}$

$N_{A,r} = -CD_{y_A}\frac{1}{C}\frac{dC_A}{dr}$

$\frac{\partial C_A}{\partial t} - \frac{1}{r^2}\frac{\partial}{\partial r}(r^2 D_{y_A}\frac{dC_A}{dr}) = 0$

$$\frac{\partial C_A}{\partial t} = D_{y_A}\frac{\partial^2 C_A}{dr^2}$$

By initial condition is $t = 0$ for all r, $C_A = C_{A_o}$

And boundary conditions are,

When

$$t > 0, r = R, C_A = C_{A_o}$$
$$t > 0, r = 0, \frac{dC_A}{dr} = 0$$

Solving diffustion equation,

Assume that:

$C_A(r,t) = R(r)T(t)$

Differentiating, $\frac{\partial C_A}{\partial t} = R\frac{\partial T}{\partial t}$

$\frac{\partial^s C_A}{\partial r^2} = T\frac{\partial^2 R}{\partial r^2}$

So, $R\frac{\partial T}{\partial t} = T\frac{\partial^2 R}{\partial r^2}$

$$\frac{1}{T}\frac{\partial T}{\partial t} = \frac{1}{R}\frac{\partial^2 R}{\partial r^2} = k$$

In this case, k prefers to be $-\lambda^2$

$\frac{1}{T}\frac{\partial T}{\partial t} = -\lambda^2$

$T = Ae^{-\lambda^2 t}$———-(1)

$\frac{1}{R}\frac{\partial^2 R}{\partial r^2} = -\lambda^2$

$R = B\cos(\lambda r) + C\sin(\lambda r)$———(2)

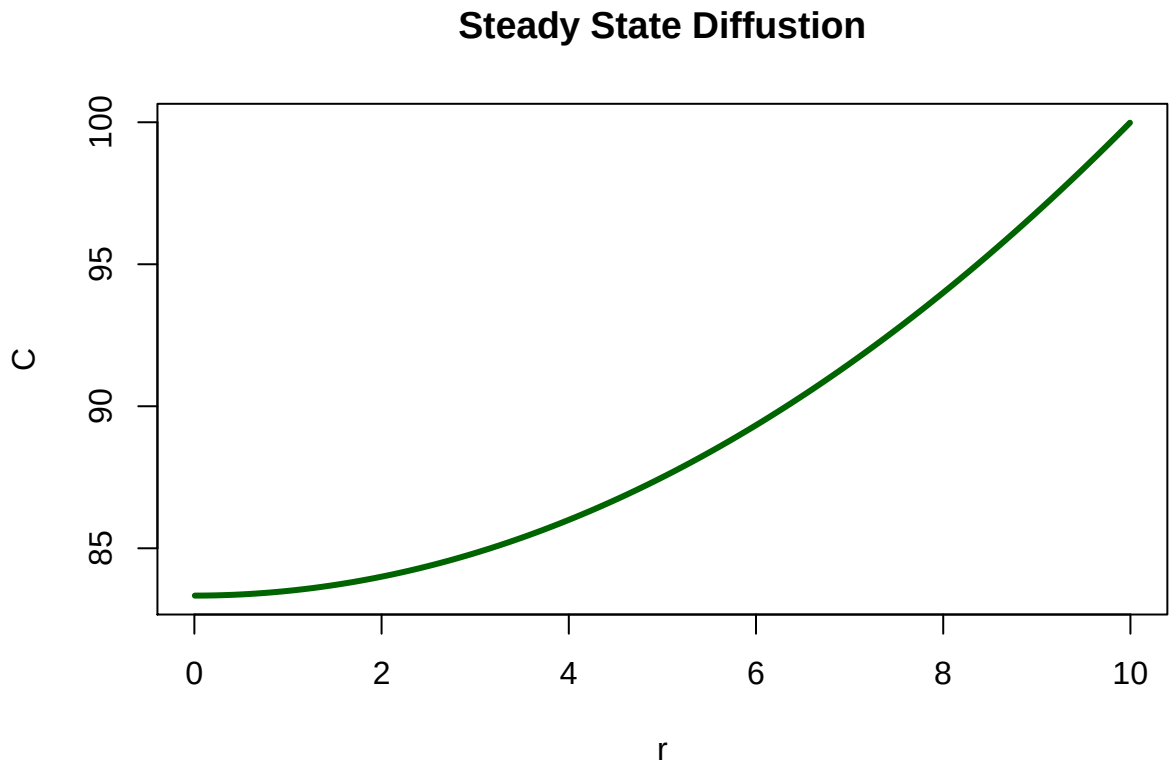
From, $C_A(r,t) = R(r)T(t) = (1) \times (2)$

So, $C_A(r,t) = Ae^{-\lambda^2 t}[B\cos(\lambda r) + C\sin(\lambda r)]$

$$C_A(r,t) = e^{-\lambda^2 t}[C_1\cos(\lambda r) + C_2\sin(\lambda r)]$$

```
D <- 1
Q <- 1
Cext <- C
```

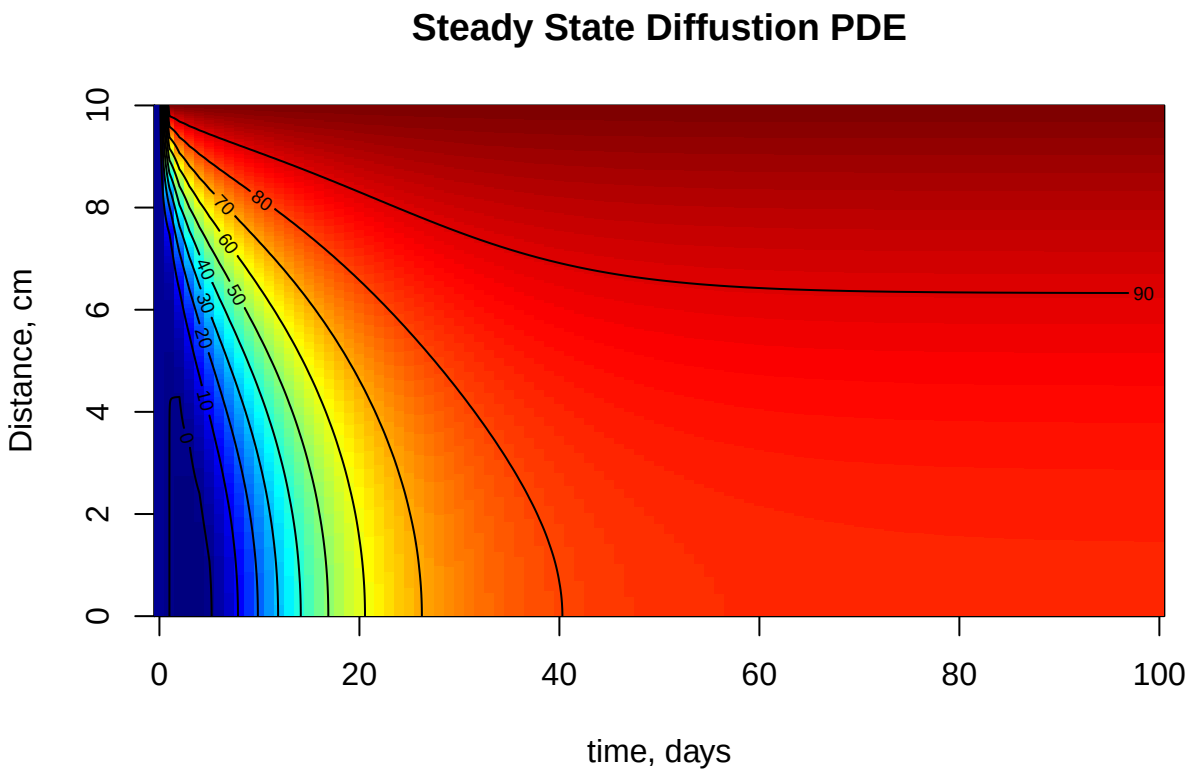
```
plot (Grid$x.mid, Spherical$y, type = "l", main = "Steady State Diffustion", lwd = 3, xl
```




```
##    user  system elapsed
##    0.22    0.03    0.25
```

```
##        time        1        2        3
##  [99,]   98 83.32220 83.32222 83.32228
## [100,]   99 83.32325 83.32327 83.32334
## [101,]  100 83.32420 83.32423 83.32429
```

```
image(out, grid = Grid$x.mid, xlab = "time, days",
      ylab = "Distance, cm", main = "Steady State Diffustion PDE", add.contour = TRUE)
```



Therefore, from this plot we can conclude that our antibiotics drug will last for 30 days. That's all for one dimension analysis.

5.2.3 Spherical diffusion steady state and non-steady state diffusion

Steady state From

When $\frac{dC_A}{dr} = 0$,

$C_s = ax + b$

---

6 Literature Review

6.1 Modeling of diffusion controlled drug delivery [Review] [3]

Main work of this paper summarizes mathematical model for slab, cylindrical, spherical shape.

This papers introduces the realistic mathematical description of drug release kinetics based on controlled parameters. From particular type of dosage forms which are slab, sphere and cylinder, the kinetic simulation allow for safety of treatment, quantitative prediction of the drug by its eluting effects.

The rate limiting step of the drug eluting is the slowest rate in mass transport processes. This work emphasized the art of mathematical modeling of predominently diffusion controlled drug delivery system, which can referred as the term "diffustion controlled" in the system that *limited drug controlled*.

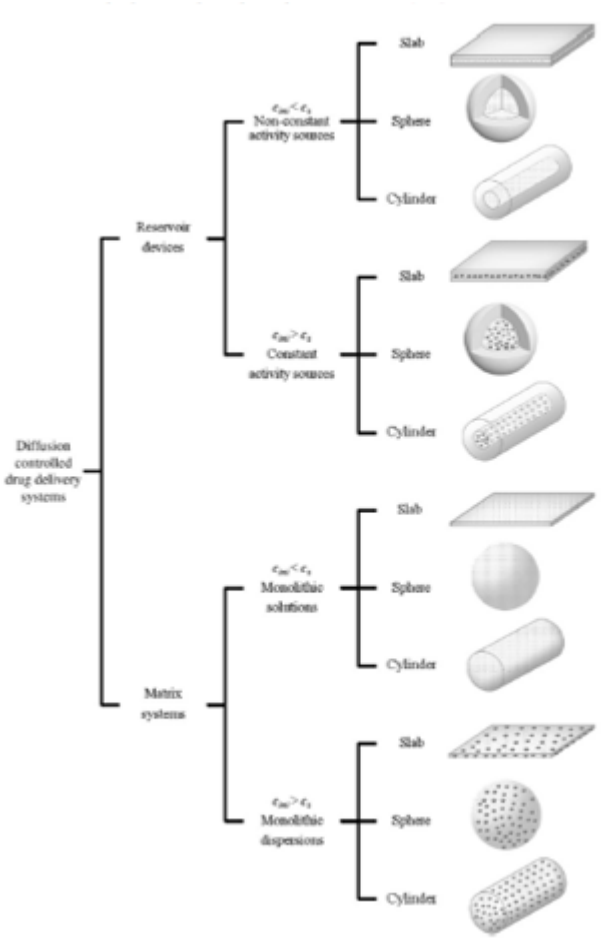


Figure 8: Siepmann J and Siepmann F models

They reviewd the moddels of diffusion controlled-drug from the different reservior devices and activity sources.

From the desired meodel that related to our simulation, core-shell structure model suit to the work. Core-shell structure includes the condition constant

activity source and non-constant activity source. This model will be the standard reference of the resulted model.

---

7 ANSYS Engineering Software for Simulation

Figure 9: ANSYS, Inc. logo

ANSYS is engineering simulaiton and 3D design software which to simulate modeling solution for any engineering problems. The software has capability to create general computer models of structures, electronics and machine components. ANSYS gives the output in many form of calculations, which are toughness, elasticciry, temperature distribution, strength of materials, pressure distribution, fluid flow, etc.

Generally, ANSYS is used to prior calculation with the propose of determining how the system will fuction with different specifications. The advantage of ANSYS simulaton is the experiment will not conduct the crash tests or real test product, which will beneficially decrease the cost of production.

---

8 Setting up the experiments

I set up the experiment on ANSYS by two big conditions;

- 2D cylinder model analysis
- 3D sphere model analysis

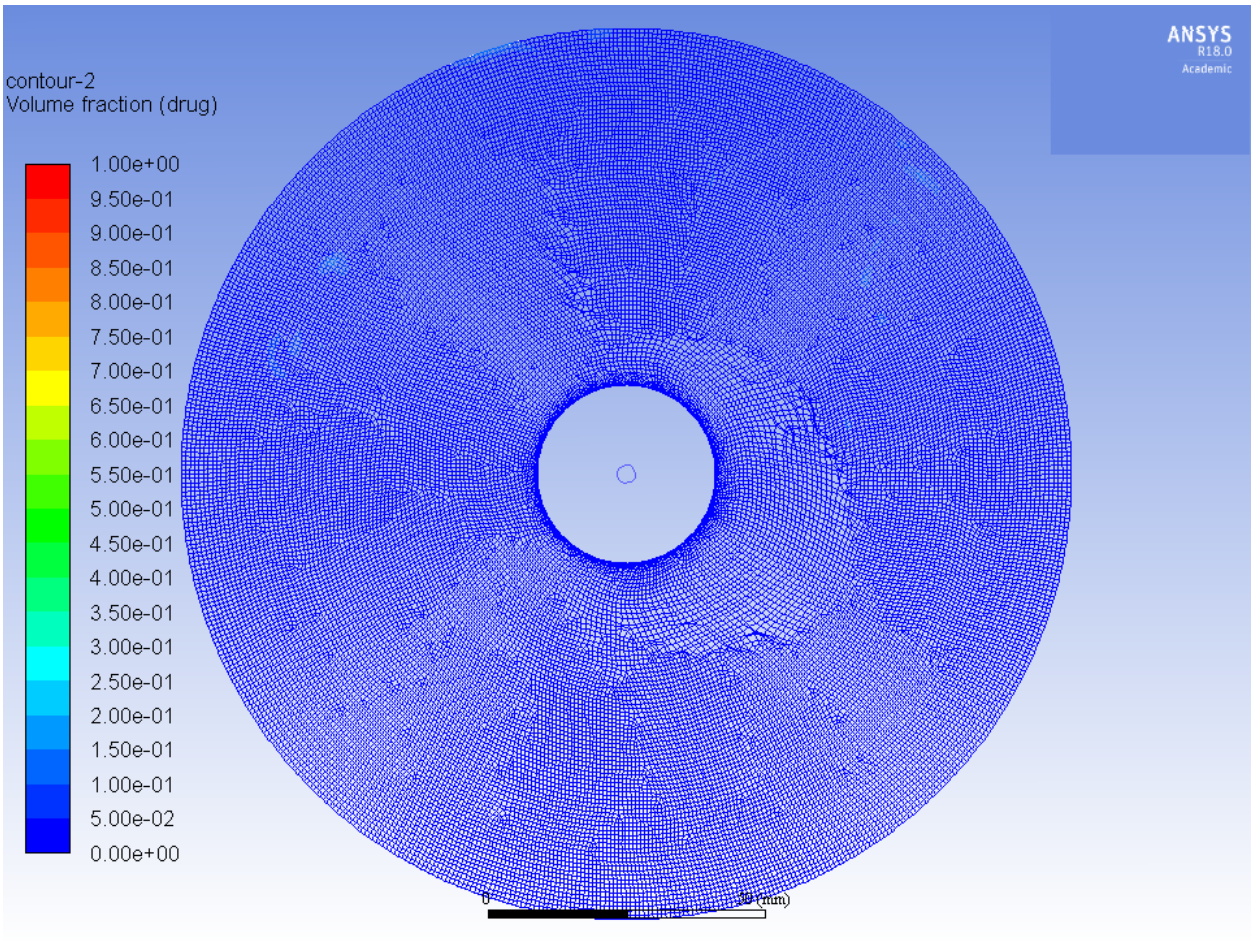


Figure 10: 2D model

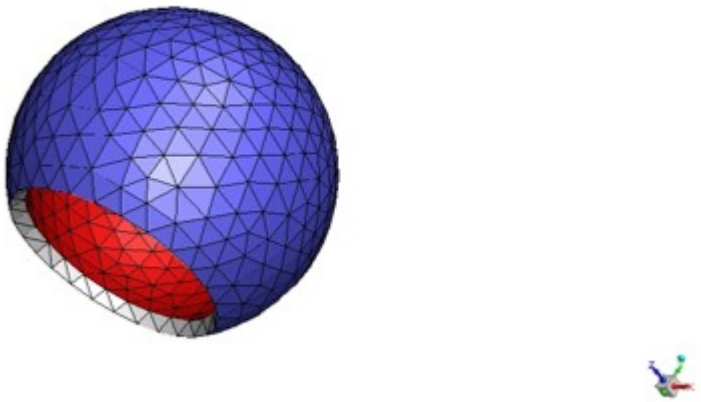


Figure 11: Hip head CAD

8.1 Set up experiments on 2D analysis

8.1.1 General set up for all experiment

In ANSYS, I set up these conditions as for every simulation

Pressure = 101,325 Pa

Gravity = -9.81 m/s^2 in y-direction

Time = transient

8.2 Method of calculation

PISO method will be used in the calcultion

PISO of Pressure-Implicit with Splitting of Operators method was proposed by Issa in 1986 without iterations and with large time steps and a lesser computing effort. It is an extension of the SIMPLE algorithm used in computational fluid dynamics (CFD) to solve the Navier-Stokes equations. PISO is a pressure-velocity calculation procedure for the Navier-Stokes equations developed originally for non-iterative computation of unsteady compressible flow, but it has been adapted successfully to steady-state problems.

Algorithm Steps

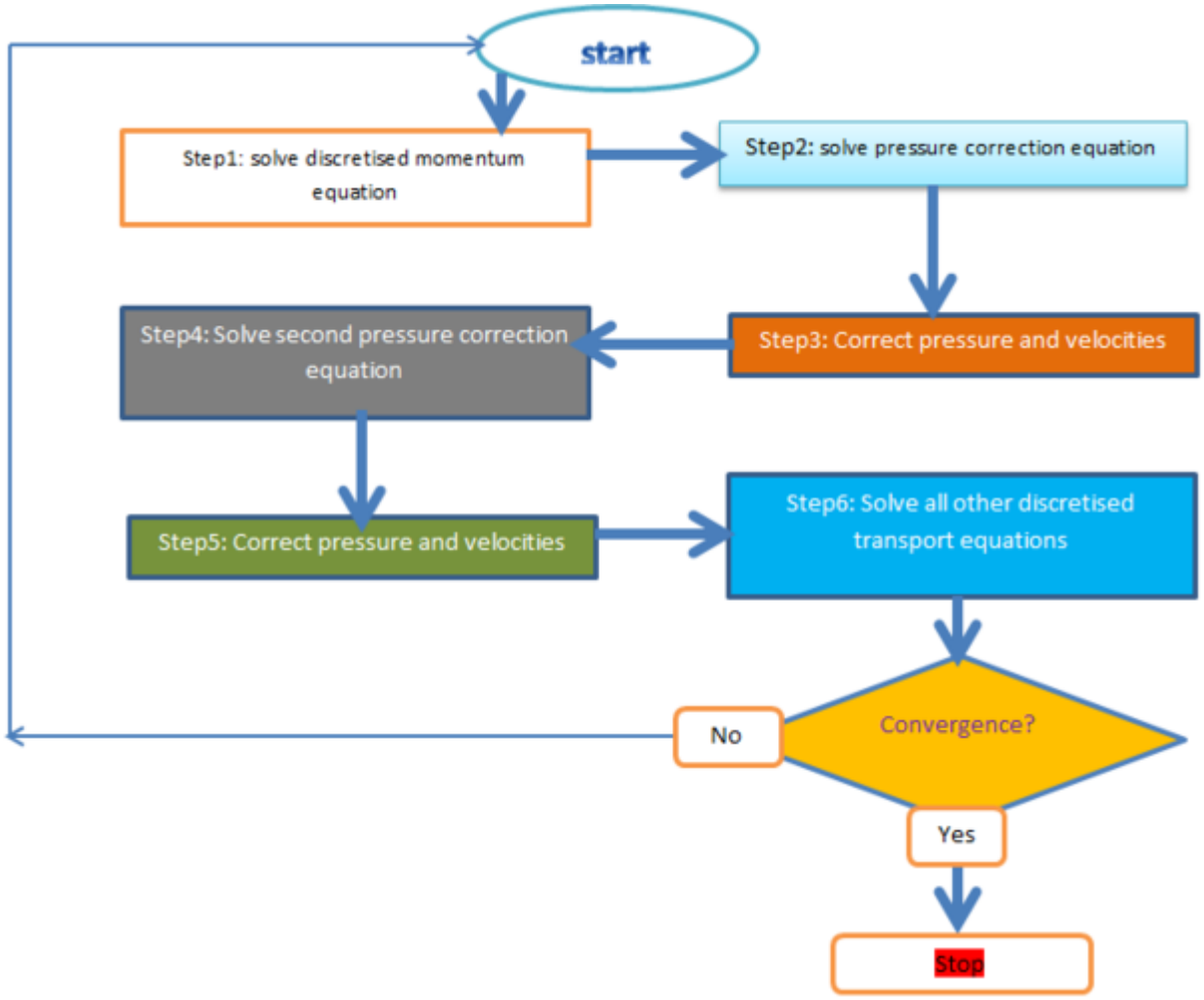


Figure 12: Flow chart of PISO algorithm

The algorithm can be summed up as follows:

1. Set the boundary conditions.
2. Solve the discretized momentum equation to compute an intermediate velocity field.
3. Compute the mass fluxes at the cells faces.
4. Solve the pressure equation.
5. Correct the mass fluxes at the cell faces.
6. Correct the velocities on the basis of the new pressure field.
7. Update the boundary conditions.
8. Repeat from 3 for the prescribed number of times.
9. Increase the time step and repeat from 1.

As already seen for the SIMPLE algorithm, the steps 4 and 5 can be repeated

for a prescribed number of times to correct for non-orthogonality.

8.2.1 Advantages and disadvantages of PISO method

1. Generally gives more stable results and takes less CPU time but not suitable for all processes.
2. Suitable numerical schemes for solving the pressure-velocity linked equation.
3. For laminar backward facing step PISO is faster than SIMPLE but it is slower concerning flow through heated fin.
4. If momentum and scalar equation have weak or no coupling then PISO is better than SIMPLEC.

8.3 Models

I setting up the models of the simulation as

8.3.1 Multi-phase - 2 phases; blood, drug, with no interaction.

`The VOF Model`

The VOF model is a surface-tracking technique applied to a fixed Eulerian mesh. It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. In the VOF model, a single set of momentum equations is shared by the fluids, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. Applications of the VOF model include stratified flows, free-surface flows, filling, sloshing, the motion of large bubbles in a liquid, the motion of liquid after a dam break, the prediction of jet breakup (surface tension), and the steady or transient tracking of any liquid-gas interface.

The experiment was used VOF model for CFD analysis.

Step for setting up multiphase model

1. Enable the multiphase model you want to use (VOF, mixture, or Eulerian) and specify the number of phases. For the VOF and Eulerian models, specify the volume fraction scheme as well.

On trees tap `Model` → `Muliphase` → `Edit...`.

8.3.2 Species transport - investigating drug flow

`Mass Diffusion in Laminar Flows`

When you choose to solve conservation equations for chemical species, ANSYS Fluent predicts the local mass fraction of each species, , through the solution of a convection-diffusion equation for the th species. This conservation equation takes the following general form:

$$J_i = -\rho D_{i,m} \nabla Y_i - D_{T,i} \frac{\nabla T}{T}$$

Where Y_i refers to local mass fraction of each species, $D_{i,m}$ refers to the mass diffusion coefficient for species i in the mixture, and $D_{T,i}$ is the thermal diffusion coefficient.

`Diffusion at Inlets`

For the pressure-based solver in ANSYS Fluent, the net transport of species at inlets consists of both convection and diffusion components. For the density-based solvers, only the convection component is included. The convection component is fixed by the inlet species mass fraction specified by you. The diffusion component, however, depends on the gradient of the computed species field at the inlet. Thus the diffusion component (and therefore the net inlet transport) is not specified a priori.

---

9 Results

9.1 Intial trials simulation

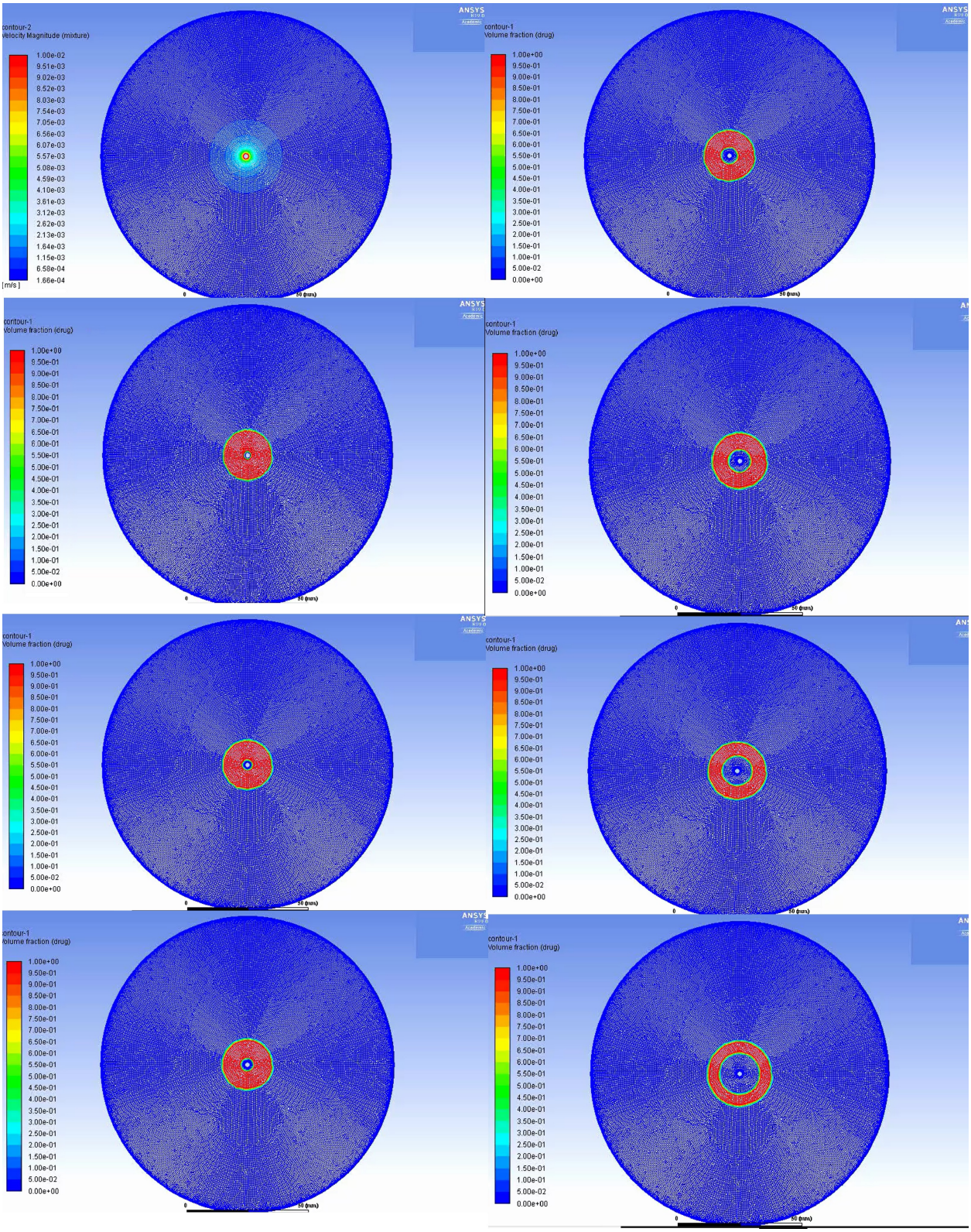


Figure 13: Initial simulation from the source

9.2 Controlling velocity inlets

By the result of solving burst effect consequence to the velocity of the diffusion inlet.

`Control condtions`

Control drug velocity from

- $2.778x10^{-7}m/s$ or $1mm/hr$
- $5.555x10^{-8}m/s$

and fix density at $100m/s$ [from the standard of antibiotic drug density]

`Results`

Velocity control results		
Conditions	**Value**	**Equations**
Velocity-1	$2.778x10^{-7}m/s$	$\frac{M_t}{M_\infty} = 1 - e^{-0.0021x-0.00000096}$
Velocity-2	$5.555x10^{-7}m/s$	$\frac{M_t}{M_\infty} = 1 - e^{-0.015x-0.003587}$

Table 2: Velocity control results

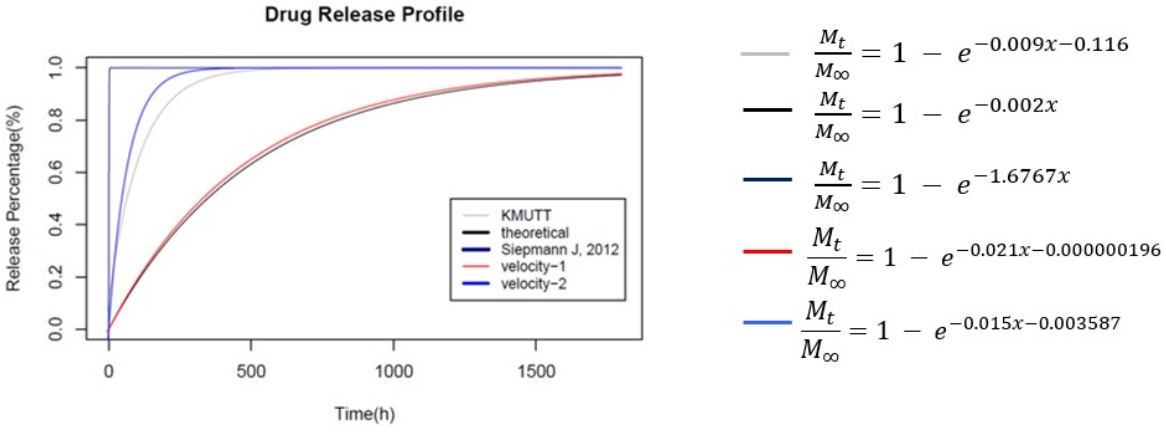


Figure 14: Graph of velocity control result

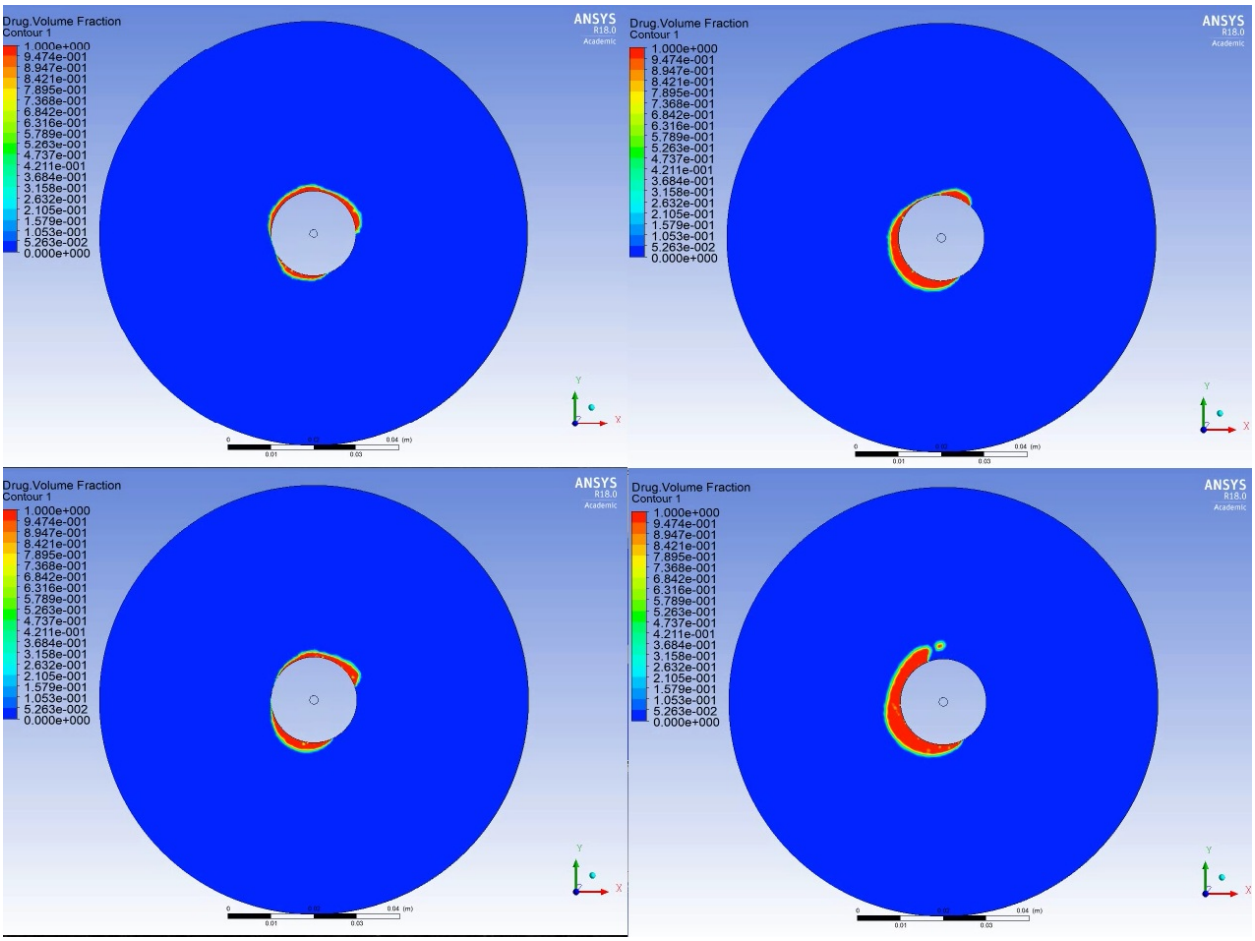


Figure 15: Velocity-1 simulation

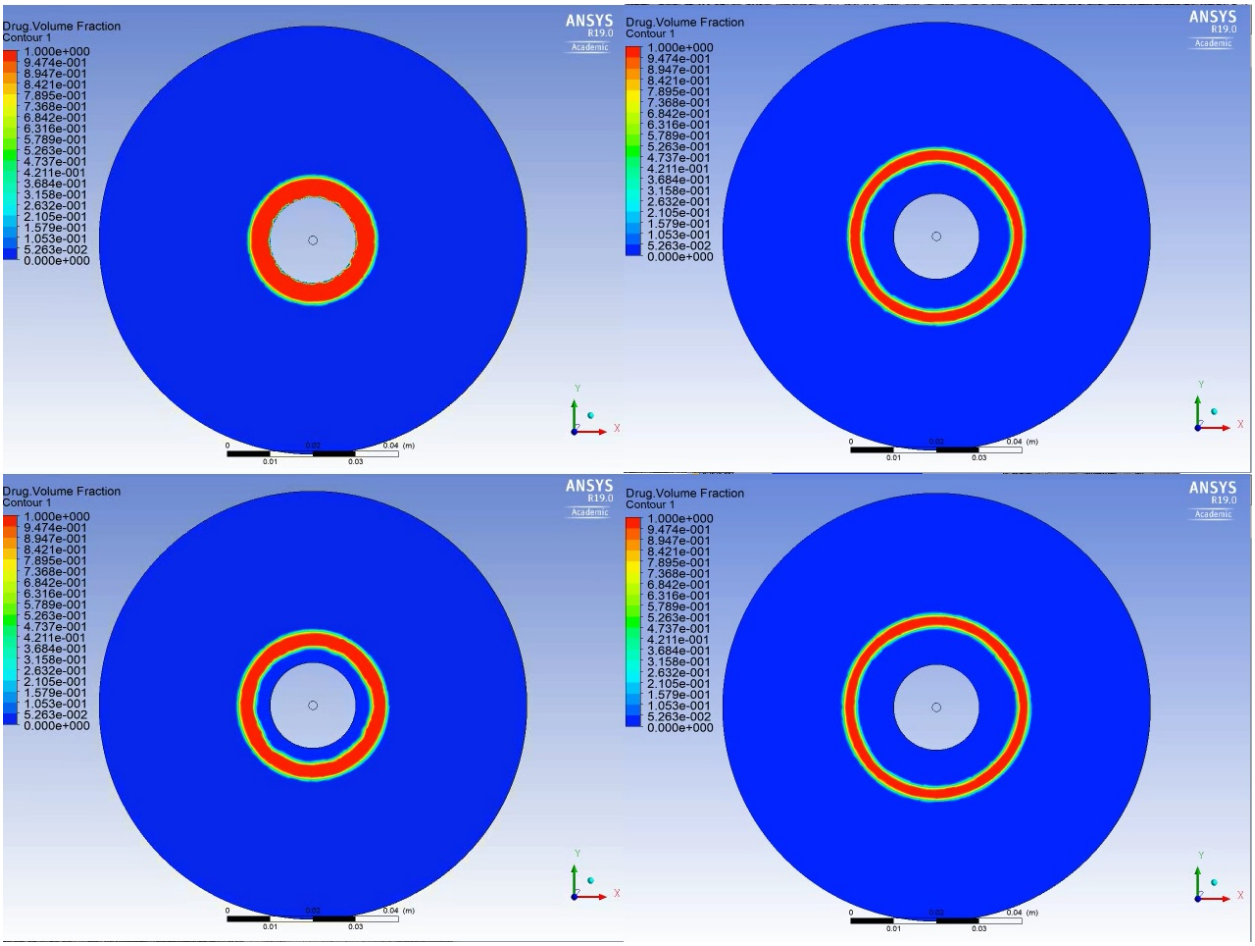


Figure 16: Velocity-2 simulation

9.3 Controlling density of fluids

By the result of solving burst effect consequence to the velocity of the diffusion inlet.

`Control condtions`

Control drug velocity from

- $100 kg/m^3$
- $500 kg/m^3$
- $1000 kg/m^3$

and fix density at $1449.2 kg/m^3$ [from the standard of antibiotic drug density]

Results

Density control results			
Conditions	**Value**	**Concentration**	**Equations**
Density-1	$100kg/m^3$	$0.069mol/m^3$	$\frac{M_t}{M_\infty} = 1 - e^{-0.015x-0.003587}$
Density-2	$500kg/m^3$	$0.345mol/m^3$	$\frac{M_t}{M_\infty} = 1 - e^{-0.0045x-0.000105}$
Density-3	$1000kg/m^3$	$0.690mol/m^3$	$\frac{M_t}{M_\infty} = 1 - e^{-0.001x-0.00000015}$

Table 3: Velocity control results

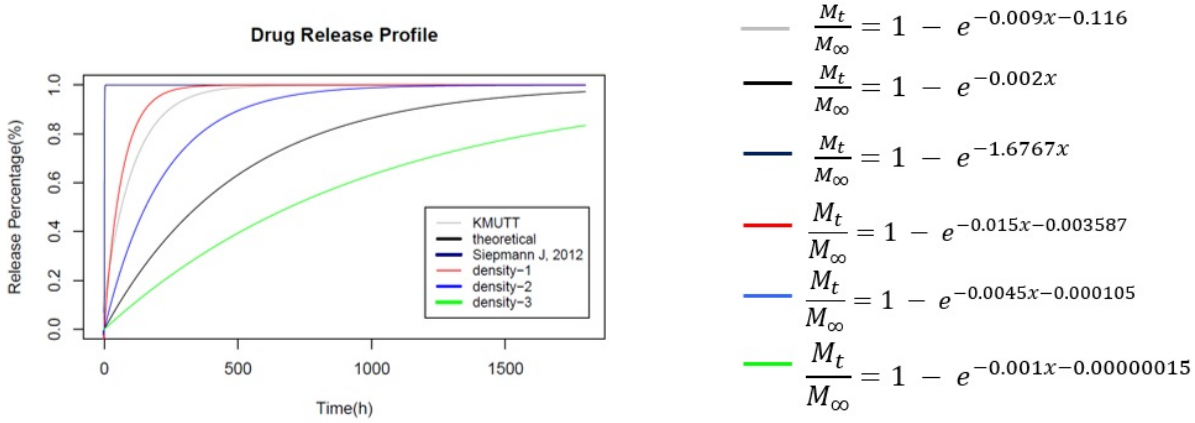


Figure 17: Graph of velocity control result

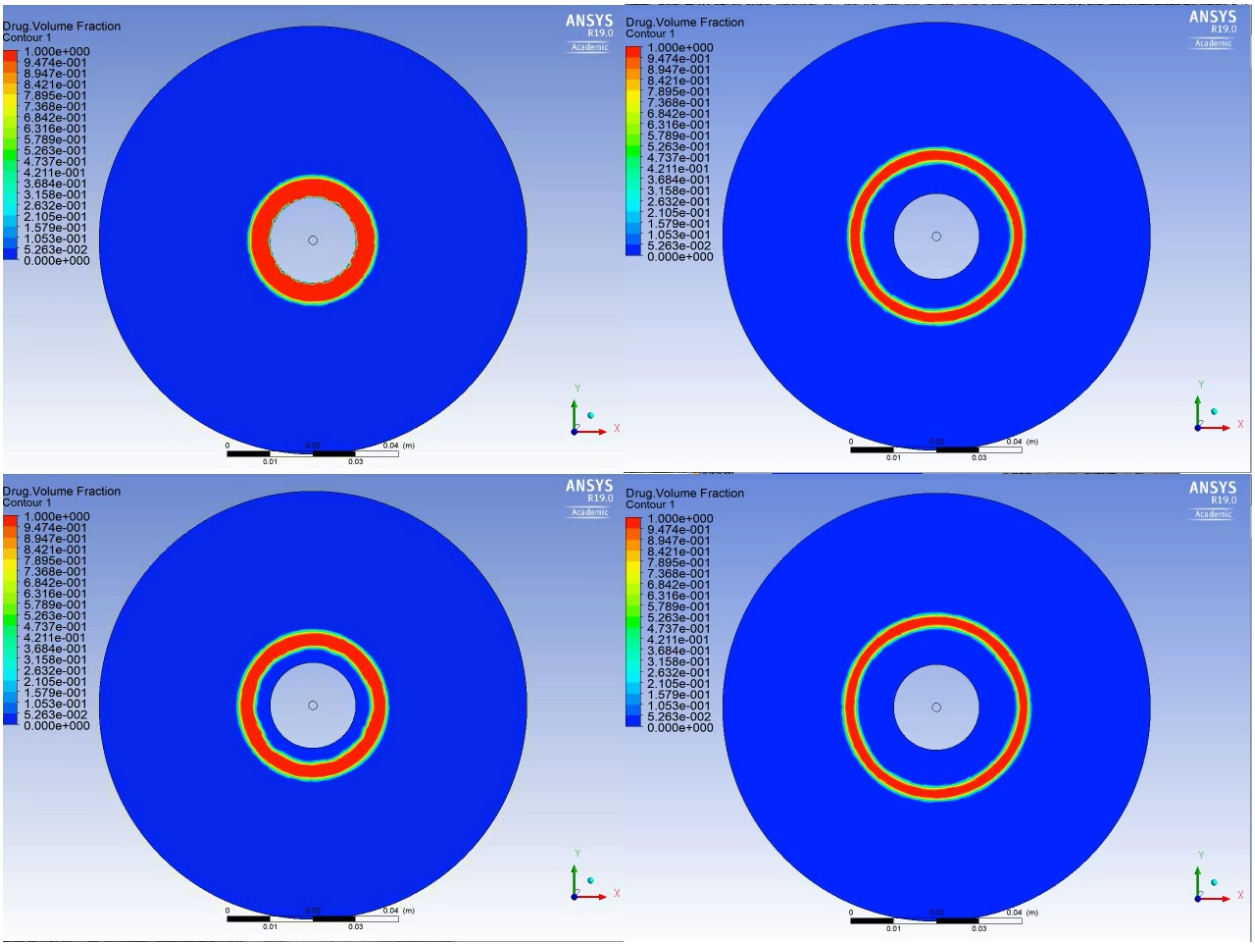


Figure 18: Density-1 simulation

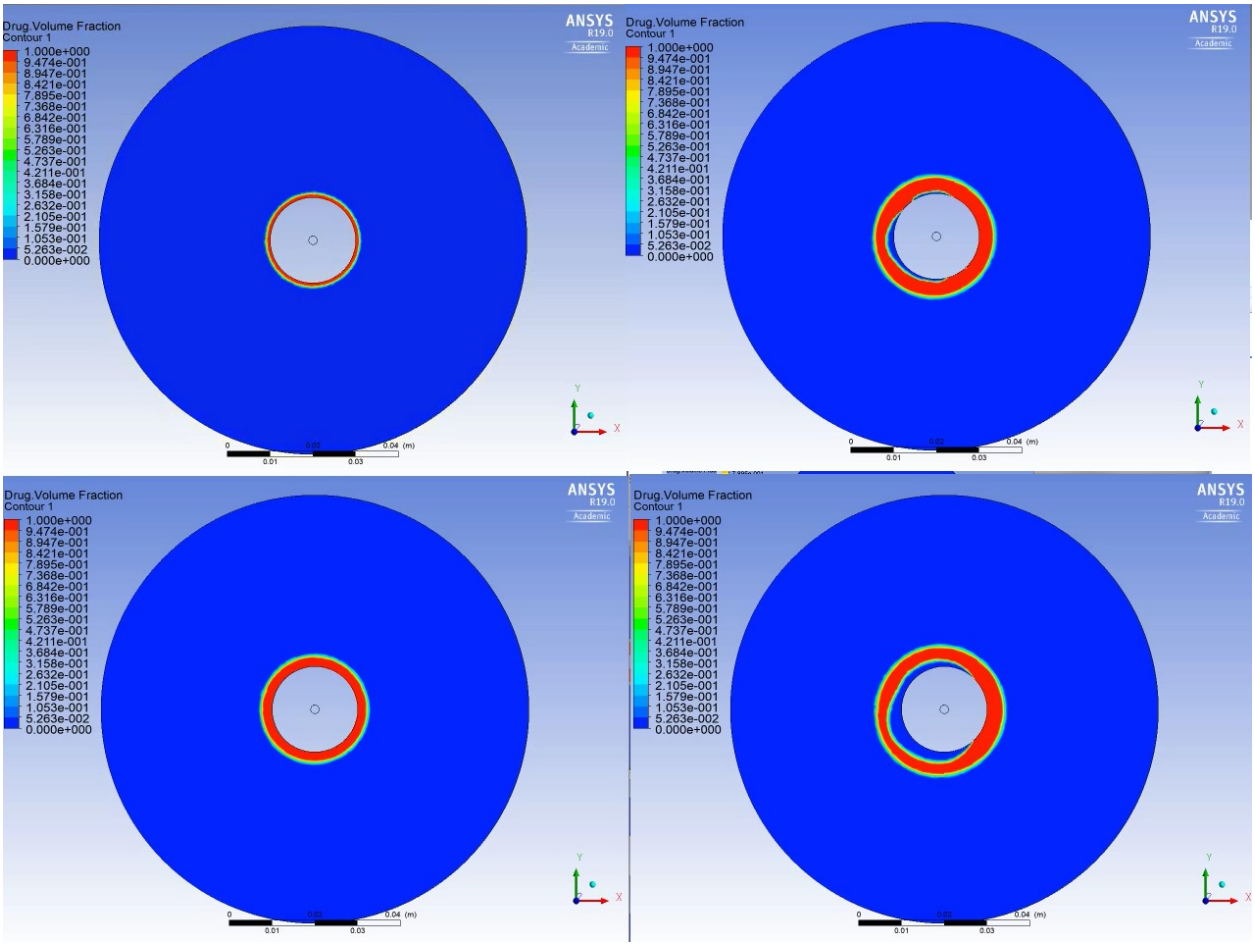


Figure 19: Density-2 simulation

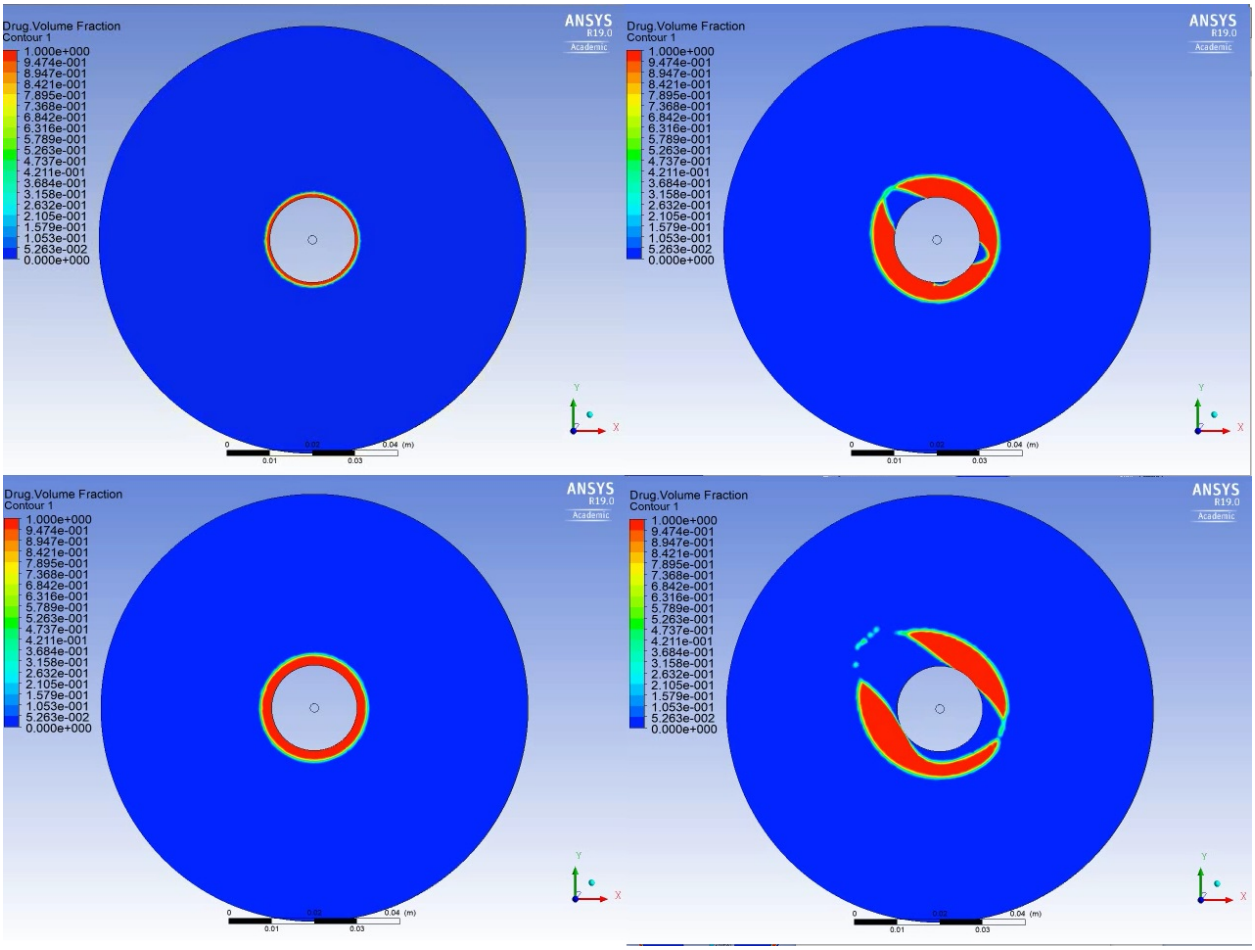


Figure 20: Density-3 simulation

9.4 Vancomycin properties

The density of vancomycin in $1mol$ is $1449.2kg/m^3$

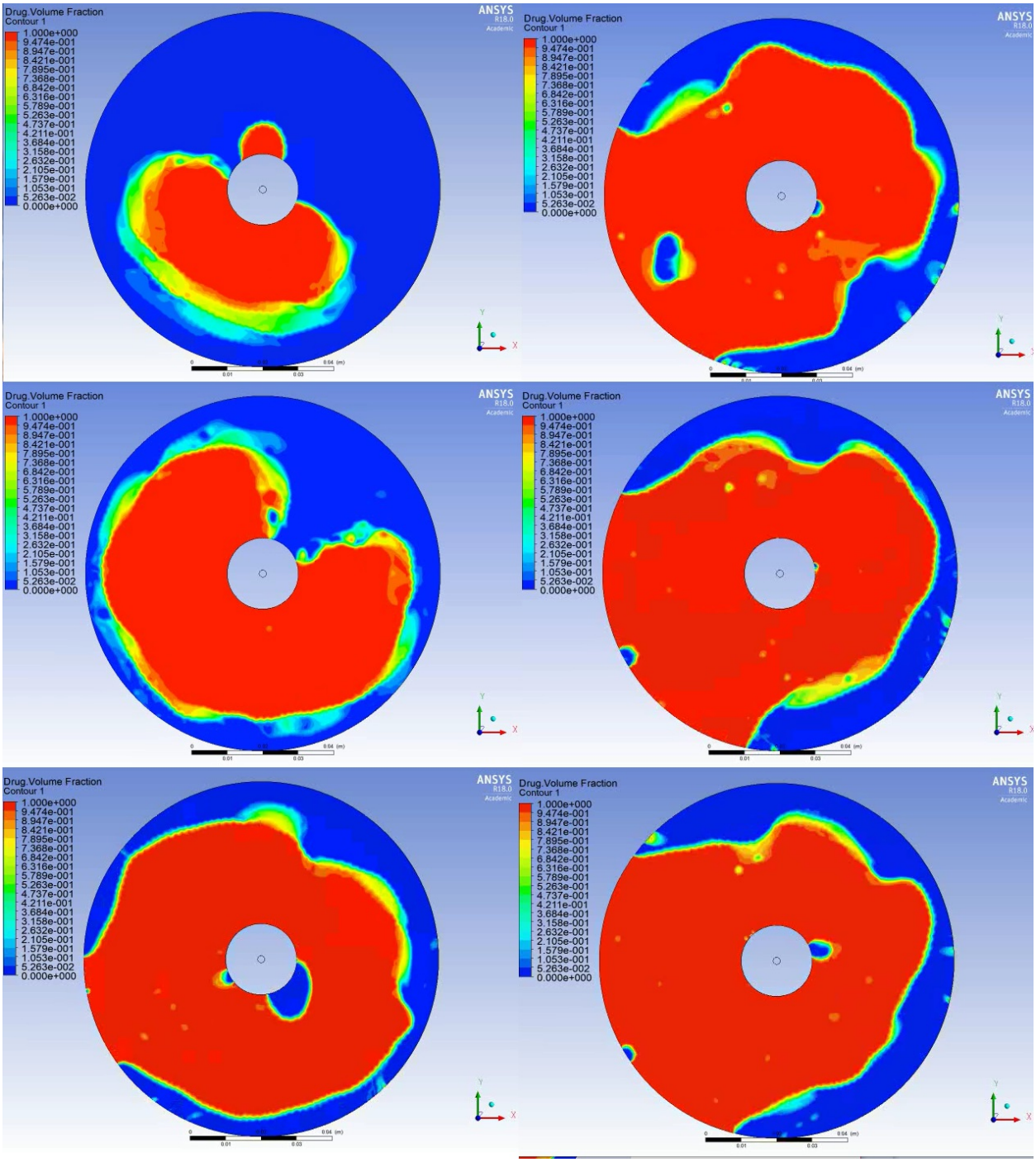


Figure 21: Vancomycin properties simulation

9.5 Reaction on drug and blood

Controls by mass transfer between blood to drug is 0.005 per hour. Equation of reaction 1 is

$$\frac{M_t}{M_\infty} = 1 - e^{-0.009x - 0.0.000002387}$$

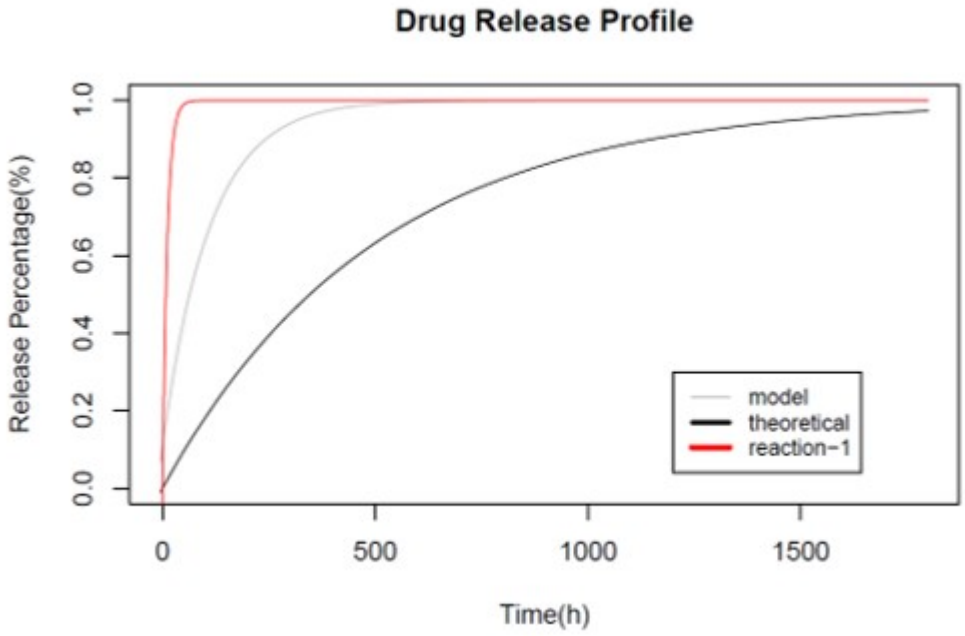


Figure 22: Reaction blood-drug simulation

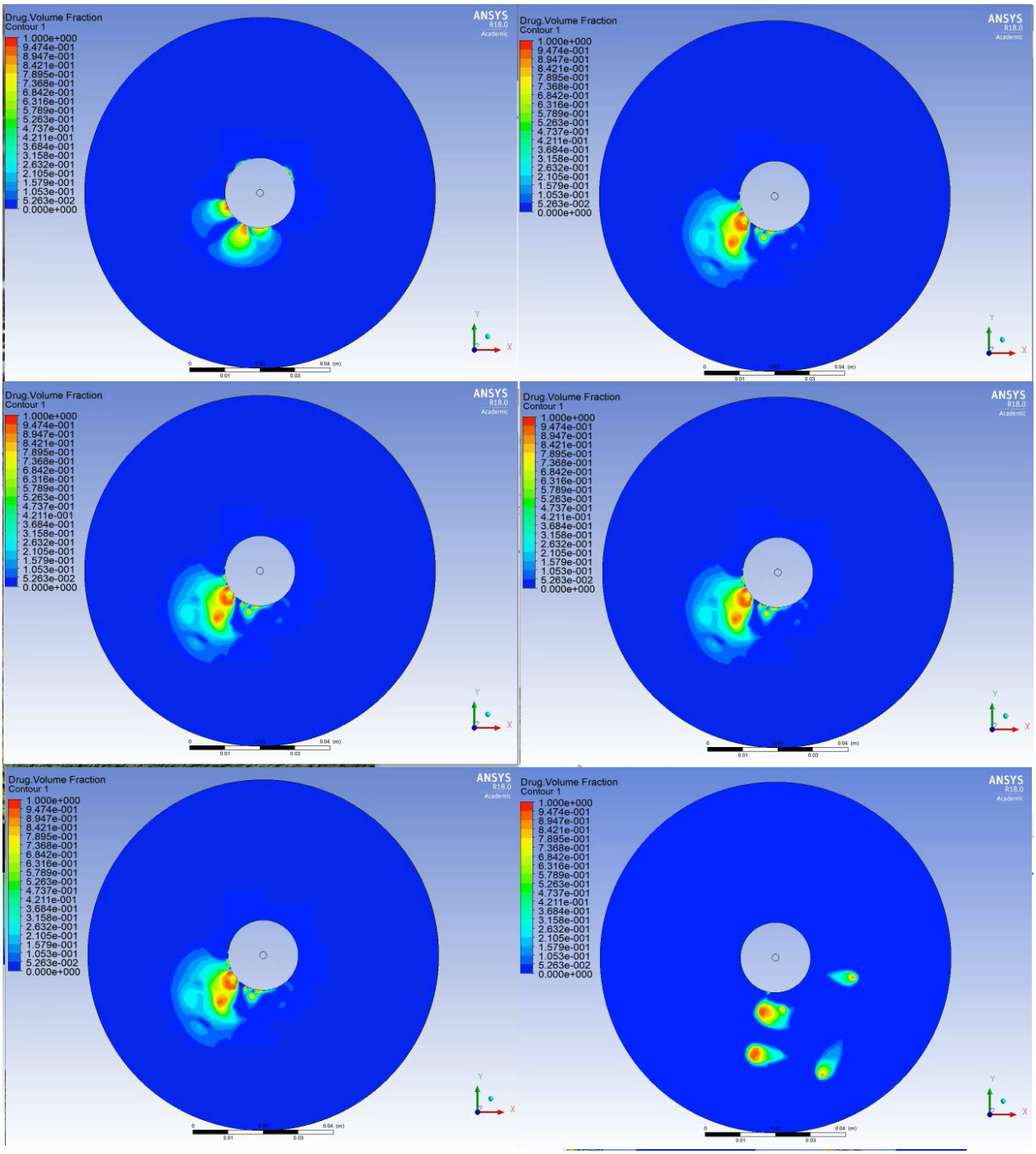


Figure 23: Reaction blood-drug simulation

9.6 Add pseudo-layer of polymer

I try the experiment on adding polymer layers.

`Control condtions`

Try to add layer thickness from

- $0.001mm$
- $1mm$

`Results`

control results		
Conditions	**Value**	**Equations**
Layer-1	$0.001mm$	$\frac{M_t}{M_\infty} = 1 - e^{-0.58x - 0.000023}$
Layer-2	$1mm$	$\frac{M_t}{M_\infty} = 1 - e^{-0.038x - 0.000012}$

Table 4: Peudolayer results

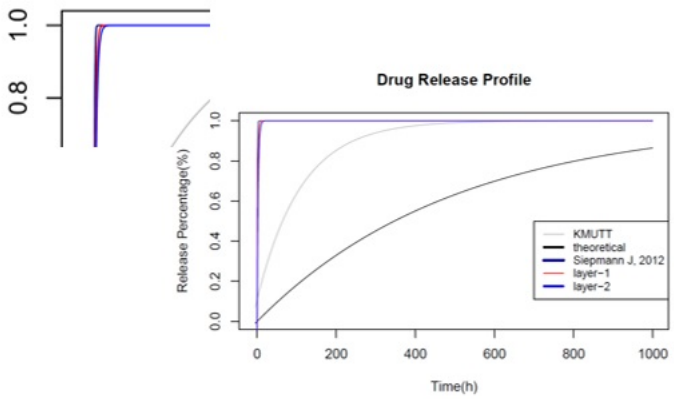


Figure 24: Graph of layer result

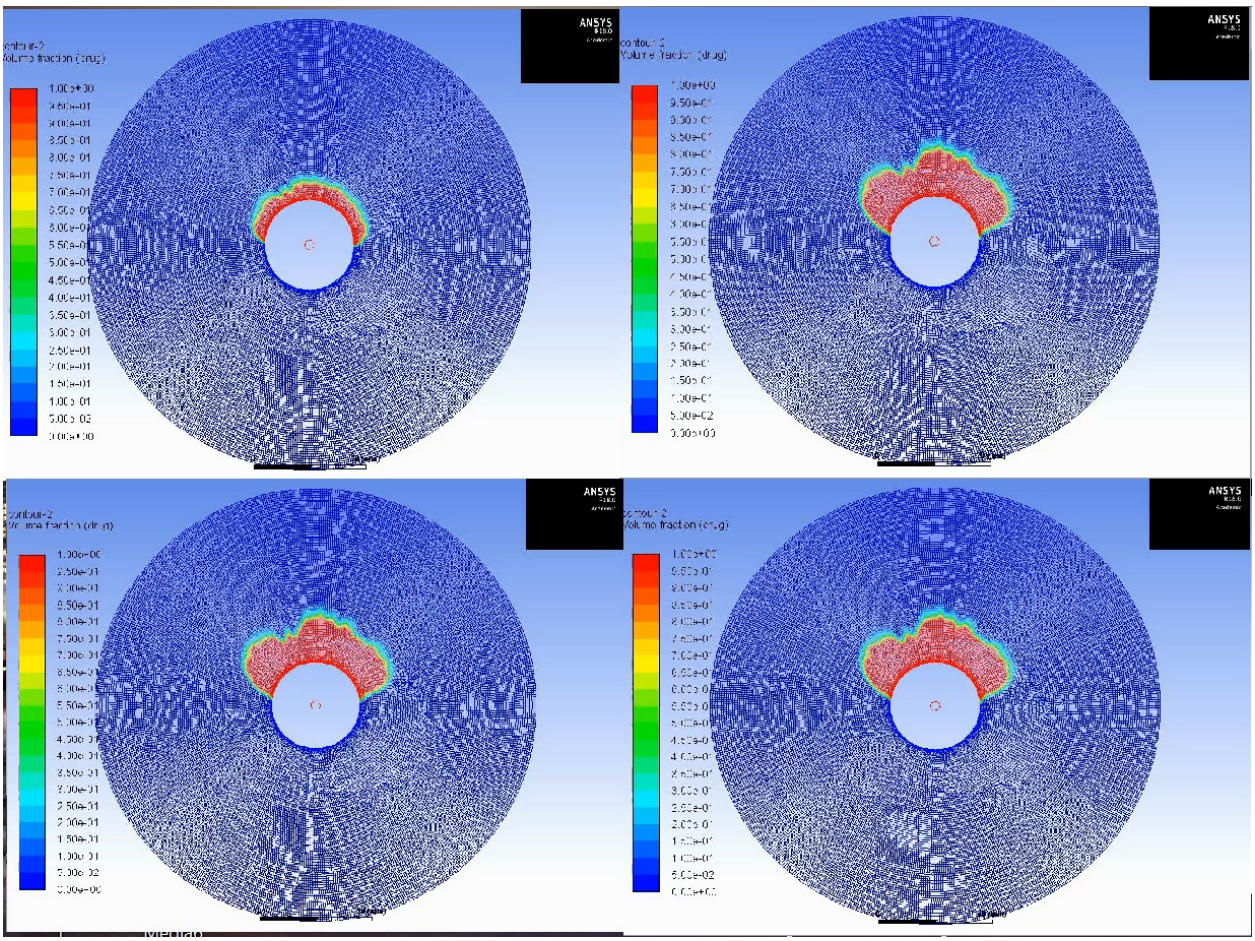


Figure 25: Layer-1 simulation

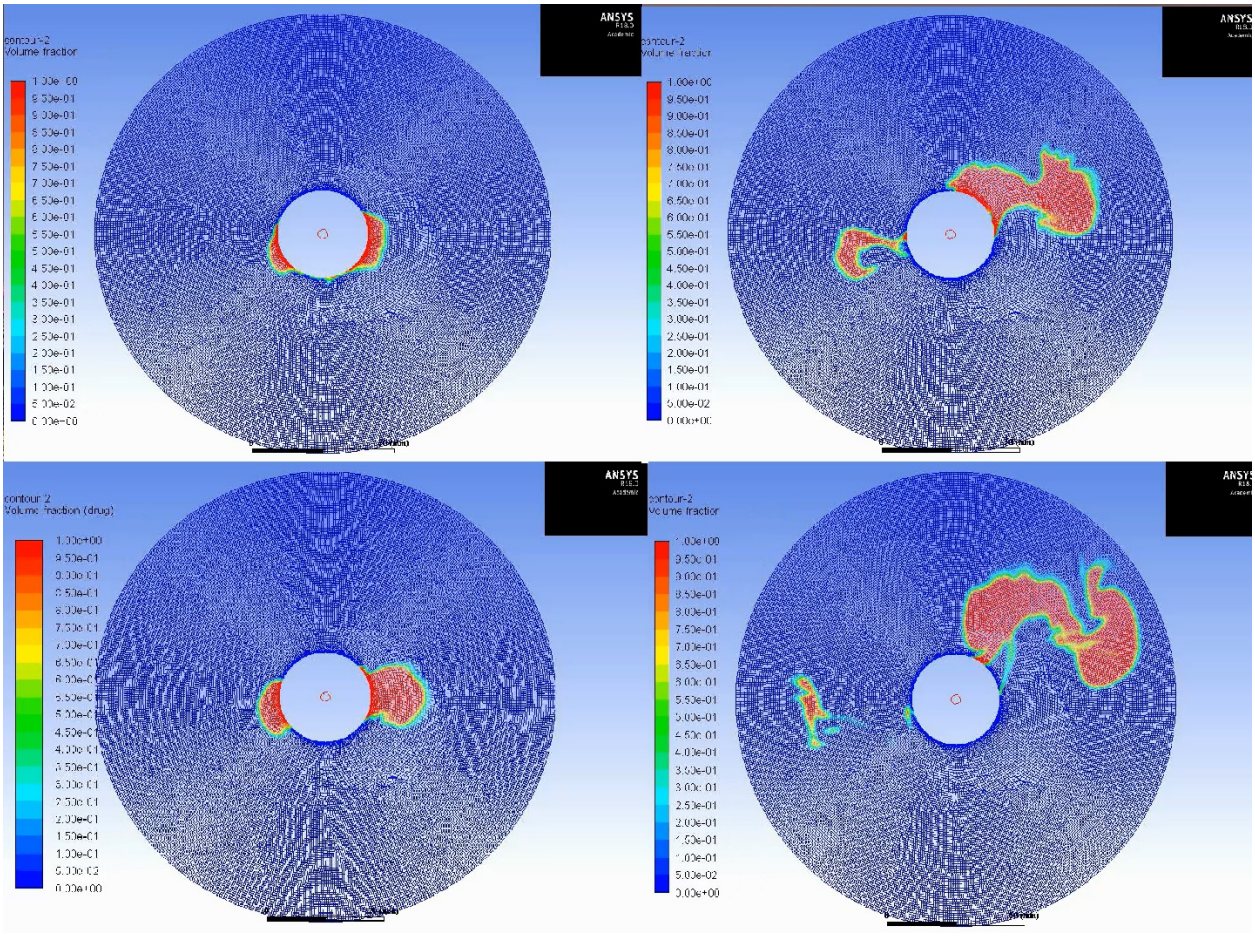


Figure 26: Layer-2 simulation

9.7 3D analysis

9.7.1 Shpere

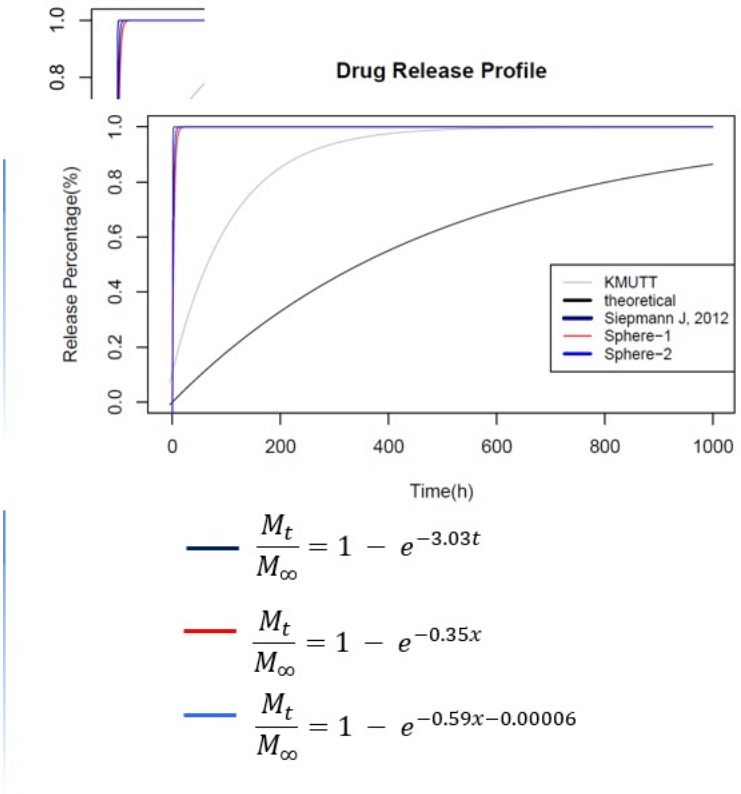


Figure 27: 3D sphere equation

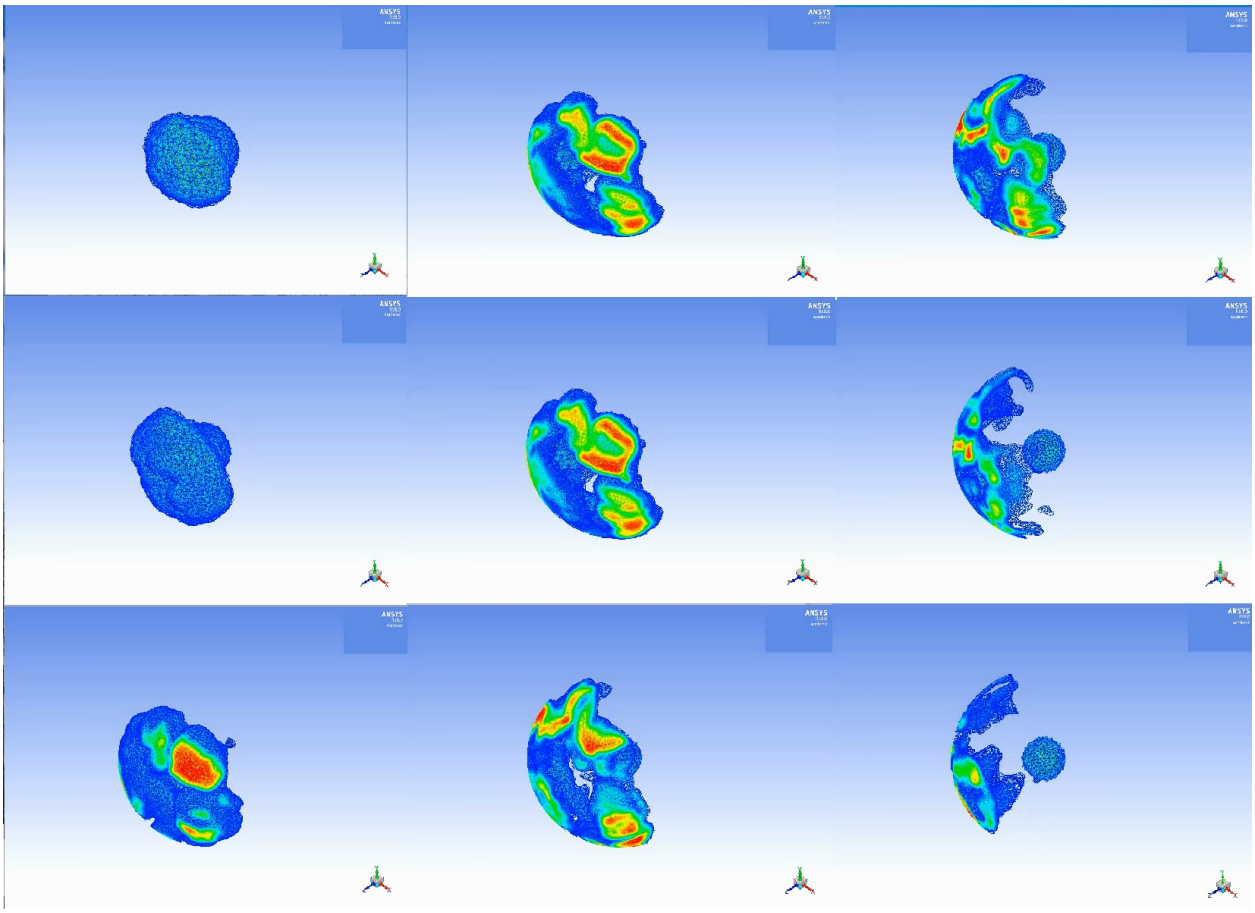


Figure 28: sphere-1 simulation

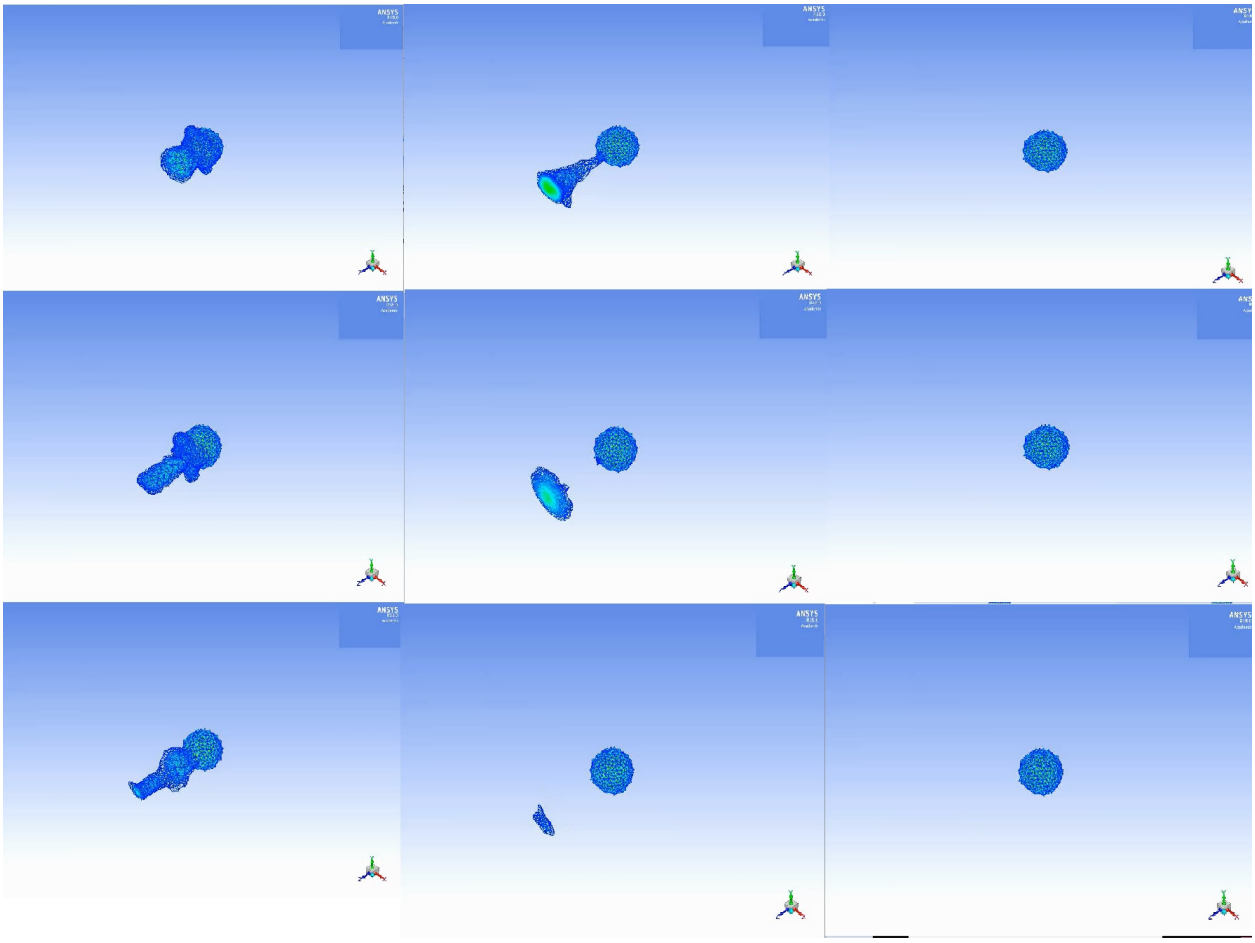
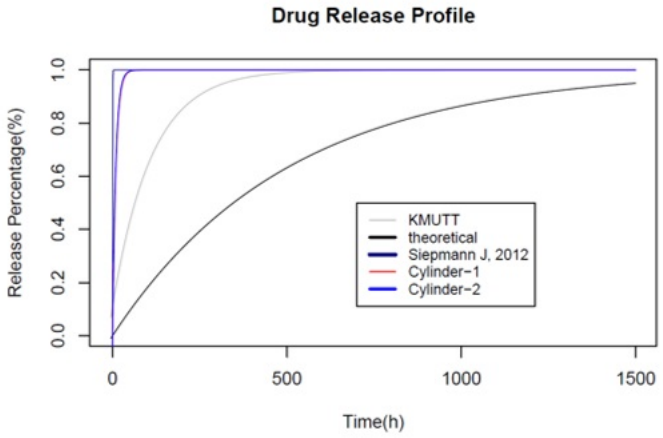


Figure 29: sphere-2 simulation

9.7.2 Cylinder



$$\frac{M_t}{M_\infty} = 1 - e^{-2.515t}$$

$$\frac{M_t}{M_\infty} = 1 - e^{-0.105x-0.000000808}$$

$$\frac{M_t}{M_\infty} = 1 - e^{-0.11x-.00000014}$$

Figure 30: cylinder equation

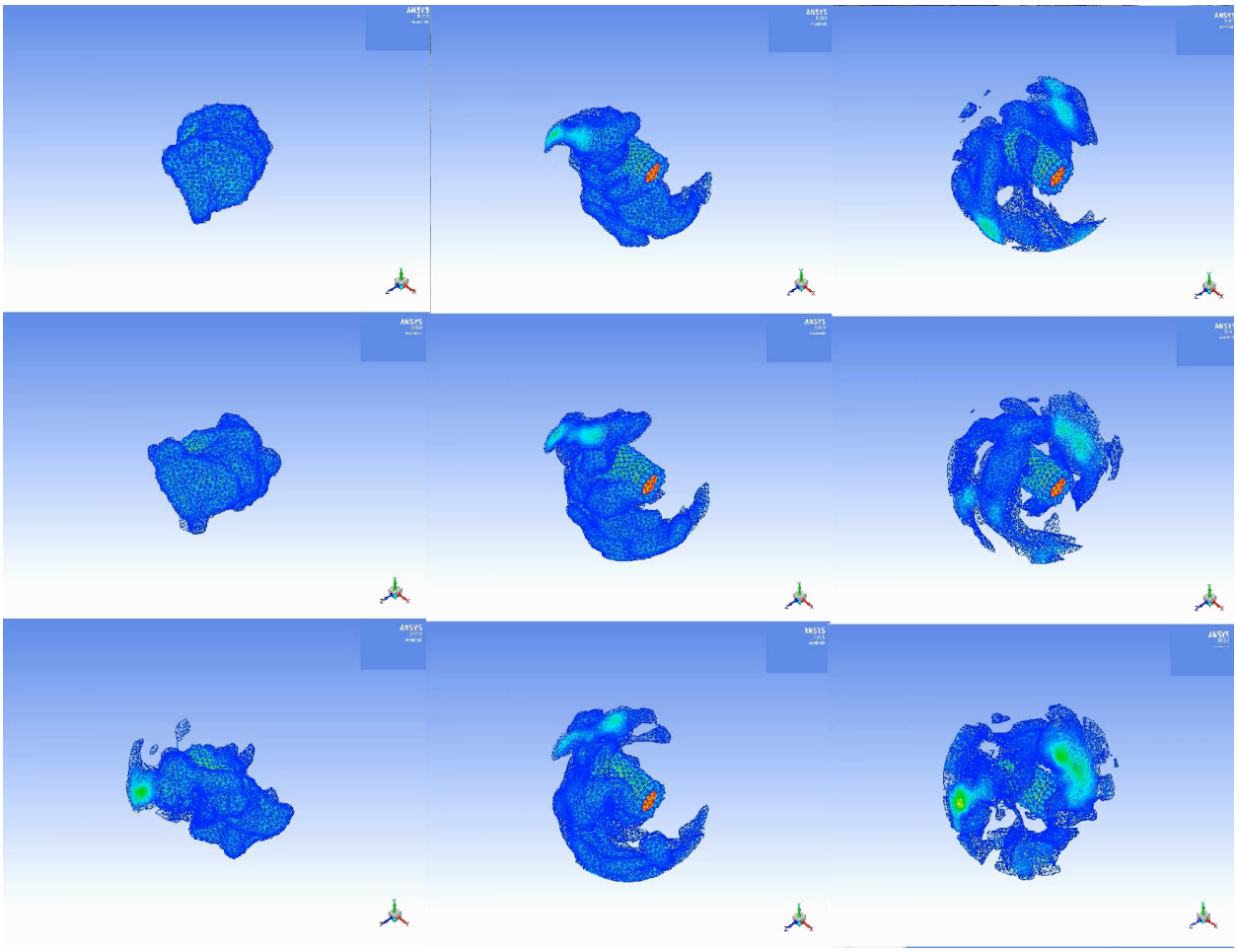
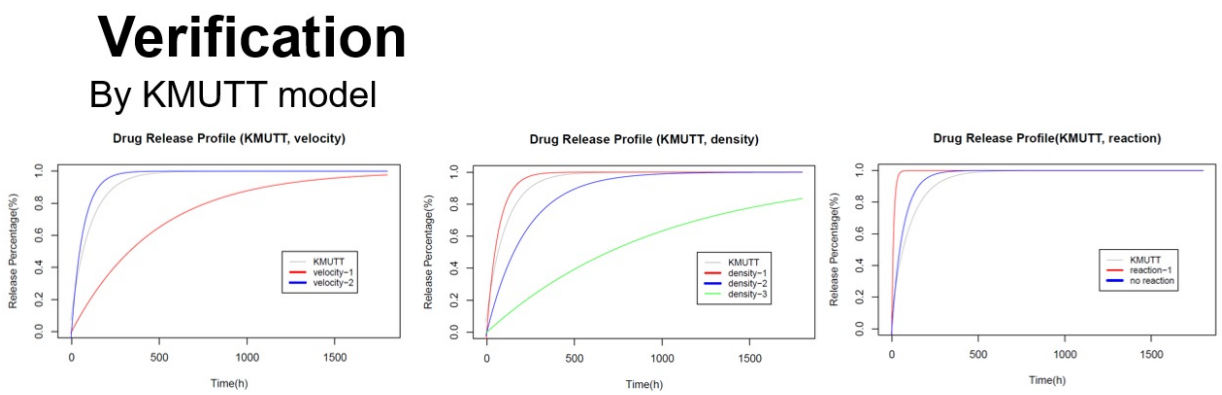


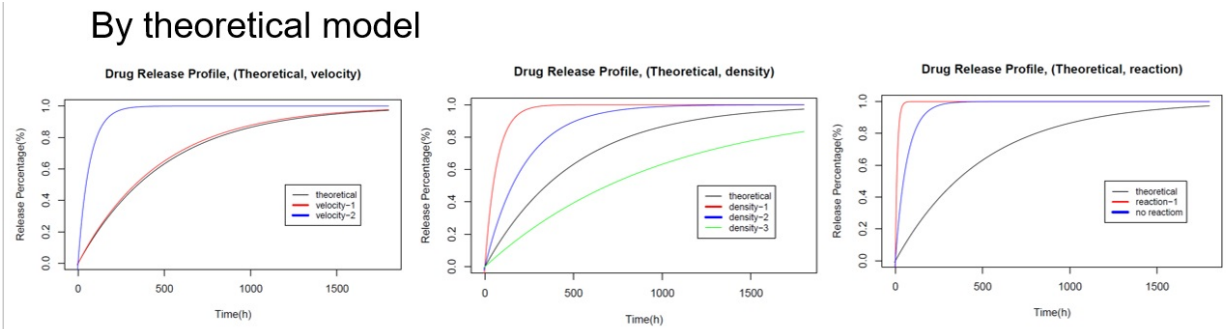
Figure 31: cylinder-1 simulation

10 Verification



Velocity – close to velocity-2 (faster) – 0.944 (95%CI) = 5.00 x $10^{-7} \mp$ 0.164m/s
Density - close to density- (low) – 0.944 (95%CI) = 250 $\mp$ 6.5 kg/m^3
Reaction – no reaction – 0.944 (95%CI)

Figure 32: Based on KMUTT model



Velocity – close to velocity-1(slower) – 0.999 (95%CI) = 2.8 x $10^{-7} \mp$ 0.06 m/s
Density – from 500 to 1000 kg/m^3 = 640 $\mp$ 12.5 kg/m^3
Reaction – no reaction

Figure 33: Based on theoretical model

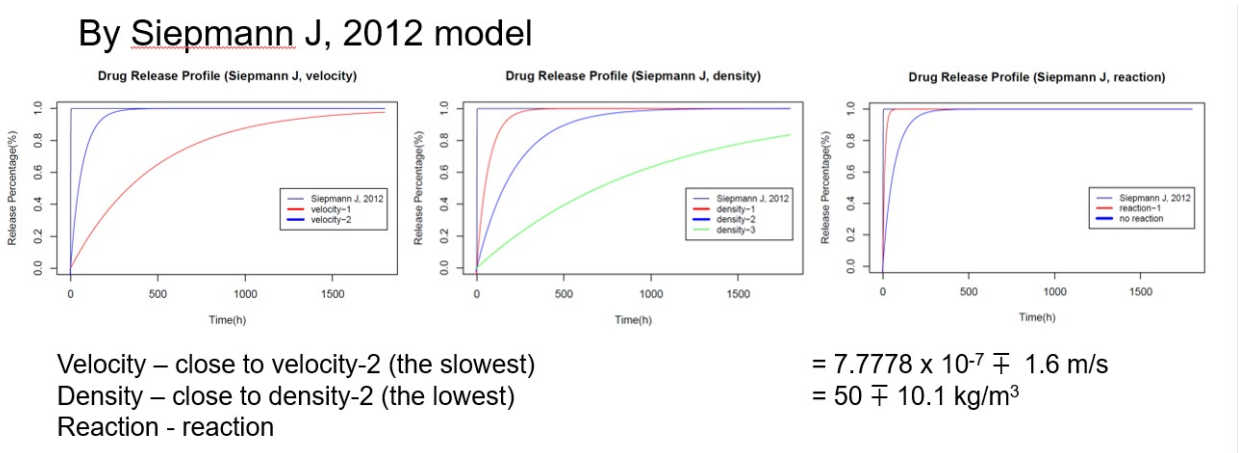


Figure 34: Based on Siepmann J model

Part III

Knowledge and gained from training

11 Research skills

I have gained the high-level of research from the training at the University of Waterloo. Research environment there induces me to learn how to effectively uses the knowledge for the whole three years that I have studied from SIIT. Department of Chemical Engineering of UW opens my mind about the research field that I have ever seen before. I have familiar with the people who is an expert only specific area such as CFD in chemical engineering on process analysis. Moreover, I have known that I can do the challenging project by learning hardly by myself to recognize genuinely in the project.

12 Simulations

I increase my skills in drawing 3D CAD. For the software, I have used ANSYS program for analyzing the process. In the University of Waterloo, they emphasize the field of the simulation process. Because of computer-based research can predict and estimate the risk before taking a test. To research this field, correctly simulation is critical. Moreover, from the internship there I have tried to reasonable link between the theoretical assumption and the simulation.

Part IV

Comments and recommendations

I would really like to recommend the training in the University of Waterloo. The work is very appropriate for chemical Engineering proved by I would state that the knowledge from SIIT was totally paying off during an internship by design experiment on the research and explain deeply on the results meaning.

References

[1] Elliot J. Carr and Giuseppe Pontrelli. *Modelling mass diffusion for a multi-layer sphere immersed in a semi-infinite medium: application to drug delivery*. Applied Mathematical Modelling, 40, 15-16, 2018.

[2] Say Chye Joachim Loo, Zhi Ying serlin Tan, Yi Jun Chow and Siew Ling Ivy Lin. *Drug Release From Irradiated PLGA and PLLA Multi-Layered Films*. Wiley InterScience (www.interscience.wiley.com), 99(7), 2009. DOI: 10.1002/jps.22079

[3] Juergen Siepmann and Florence Siepmann. *Modeling of diffusion controlled drug delivery*. Journal of Controlled Release, 161, 2012, 351-362, DOI: 10.1016/j.jconrel.2011.10.006

Appendix

A Appendices

A.1 2D diffusion modeling on drug eluting out of sphere

A.1.1 Introduction

In this modeling, the mixture of blood-drug model in ANSYS fluent is the main fluid used in this system. First, the flows are set up as continuous phase model. I use the lastest version of ANSYS product which is `ANSYS 19.0`.

Step 1 Start the project in `ANSYS workbench`. Open the window by click on the workbench icon as shown in Figure 35.

Figure 35: ANSYS workbench

The window of the ANSYS workbench will show as in Figure 36.

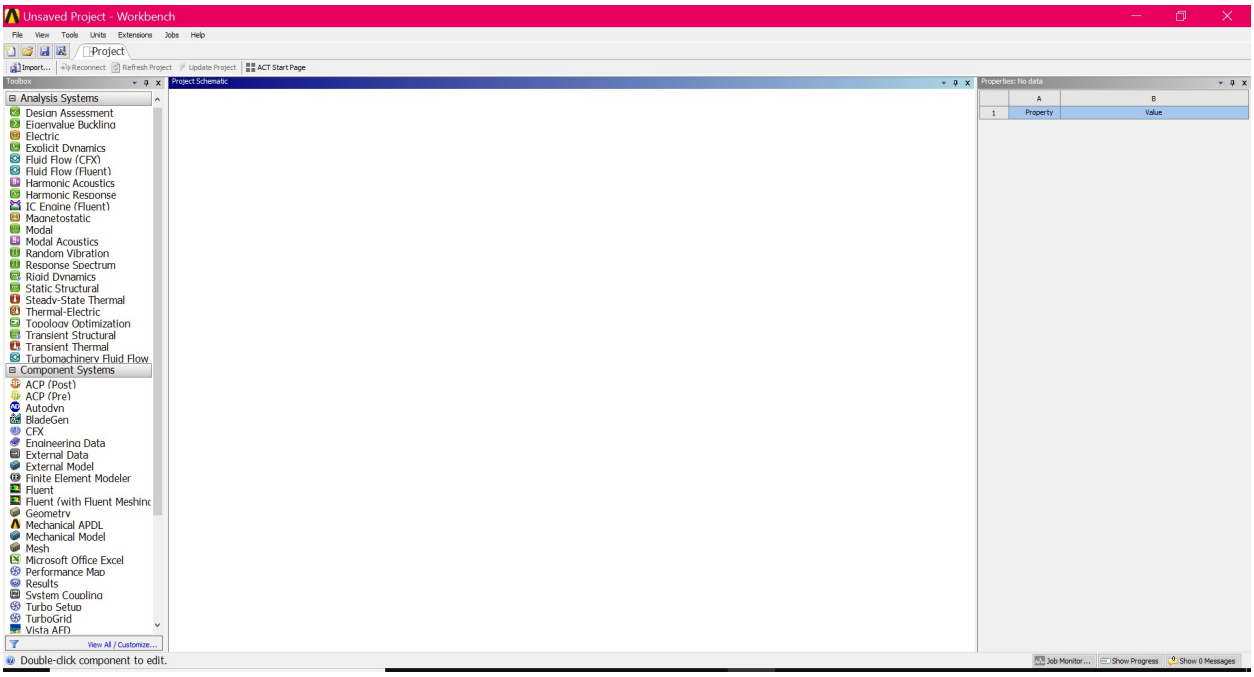


Figure 36: Step 1

On `Analysis System` Select Fluid Flow (Fluent) on the left tap bar, drag the icon to the Project Schematic area(Figure A.1.1)

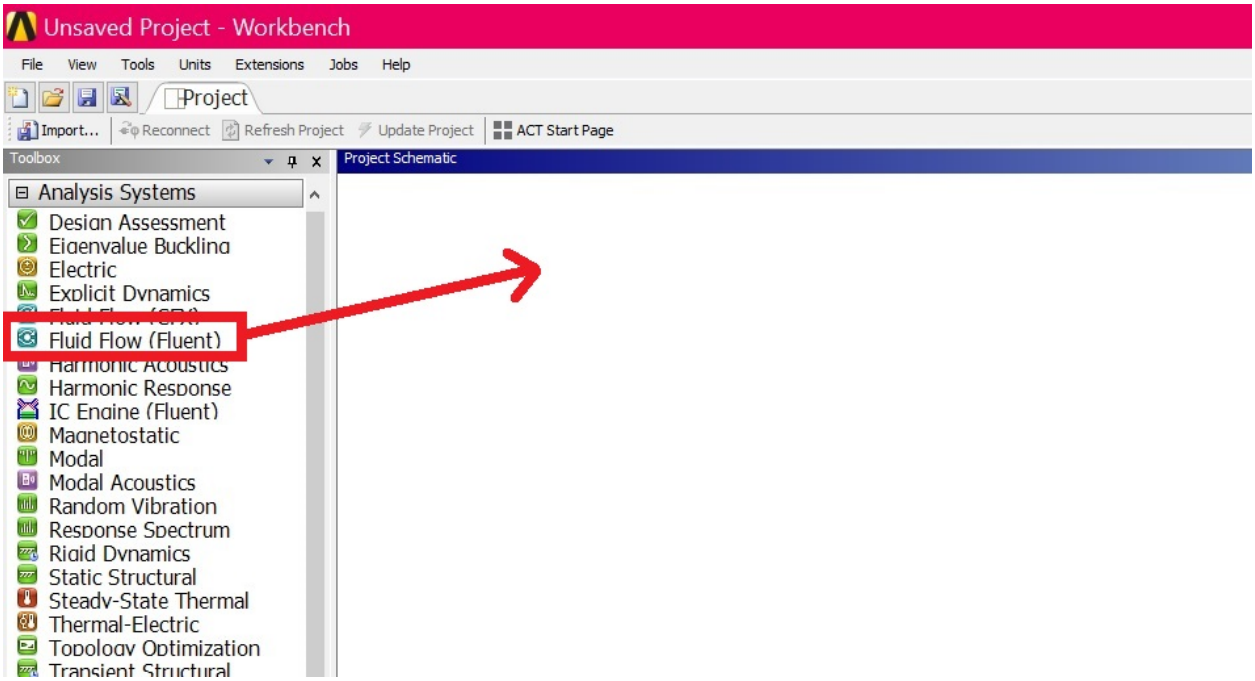


Figure 37: Hold on the left click and pull in to the Project Schematic area

When you hold the left click and drag the Fluent ion to the right hand side, the red rectange with the sentence,`Create Standalone s`, will show on the project schematic area as shown in Figure 38

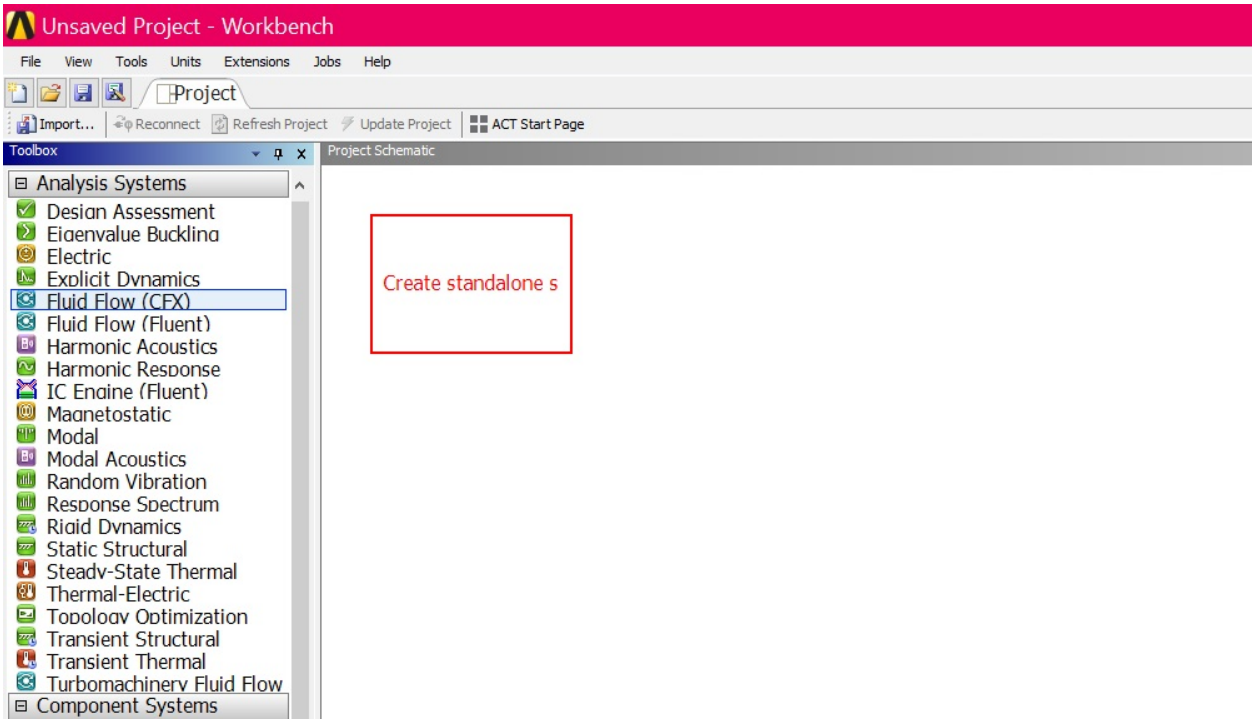


Figure 38: the red rectange with the sentence, 'Create Standalone s' show the area that the work would be done in series on the workbench

Push the left click up. The rectangle of Fluent part will pop up as shown this Figure 39.

(Optional) Change the project name, in my case, I change the project name to be `2D_sphere`.

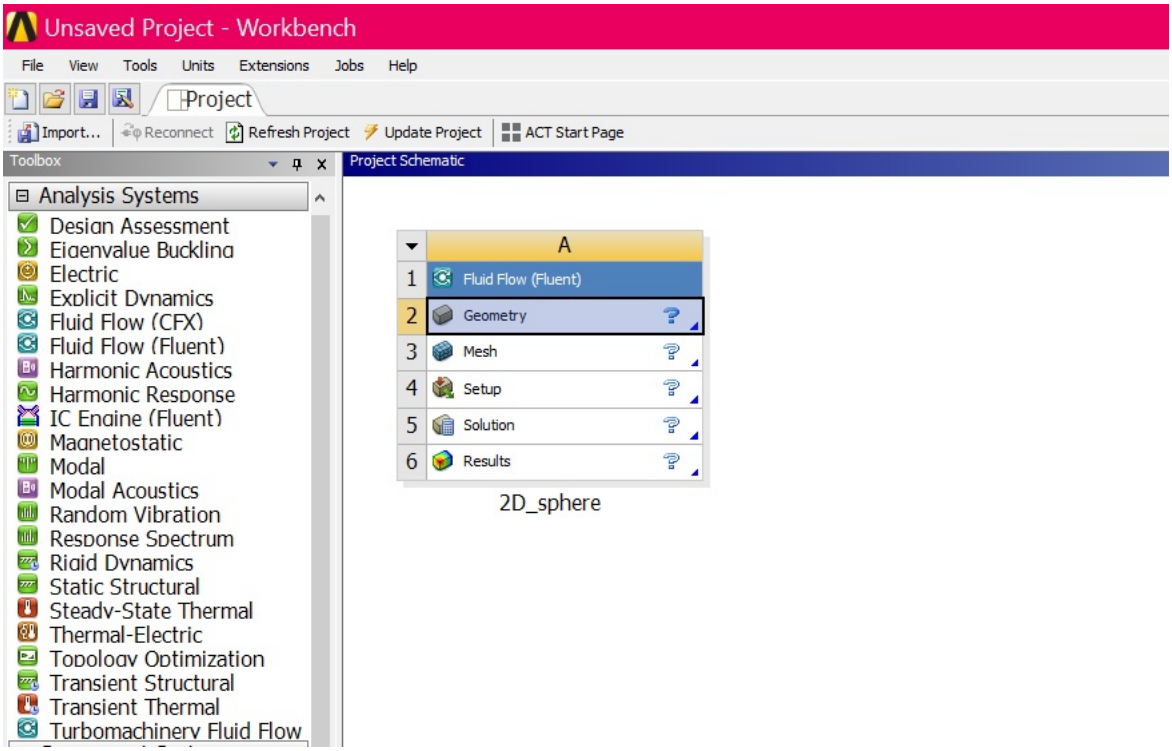


Figure 39: The Fluent options and controls are orderly shown as this rectangular box

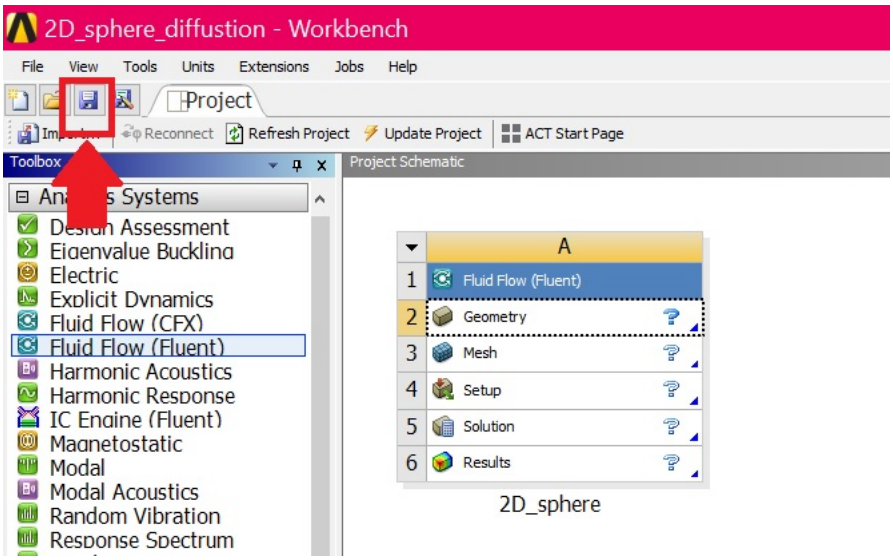


Figure 40: Save a project by click on a save symbol as indicated

Save the project Click on the `savesymbol` as point in Figure 40 or click `File` → `Save As...` → `Name.wbpj` → `Save` button.

In my case, I change the project to `2D_sphere_diffusion`

Step 2 Create geometry of the system fluid flow.

Double cick on the tap of `Geometry`.

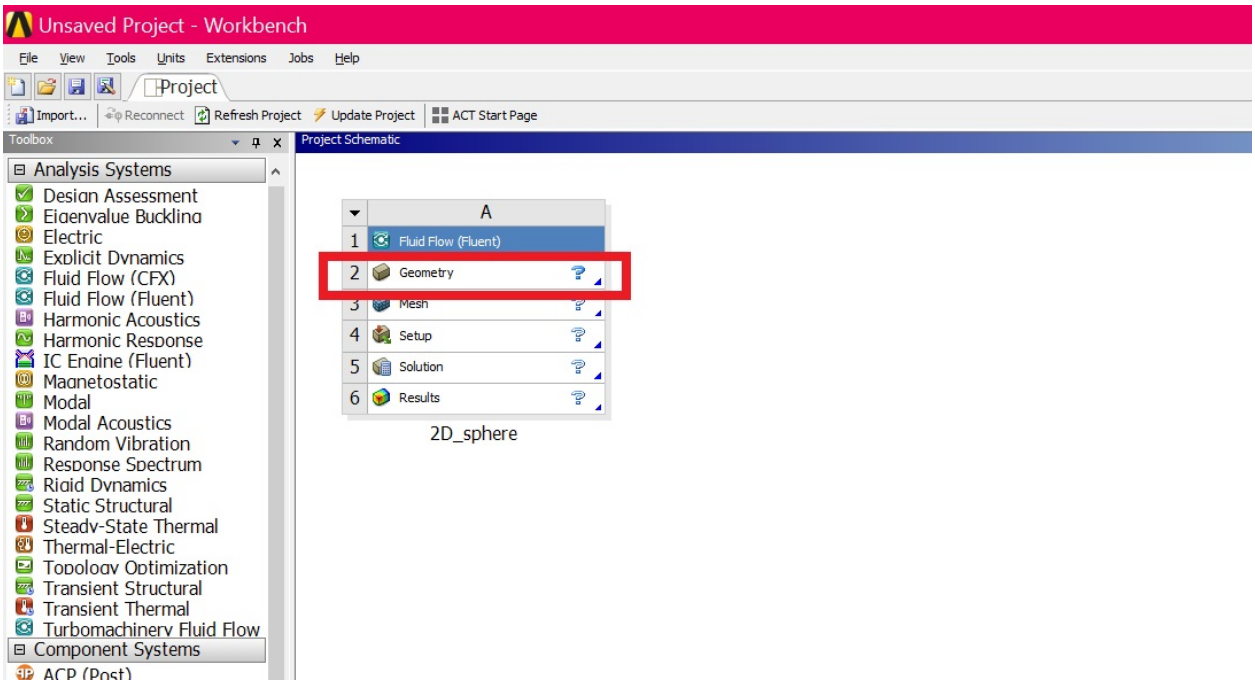


Figure 41: Next step input geometry of the object, click on the geometry tap on the Fluent control bar

The window of ANSYS program names `SpaceClaim 19.0` or `DesignModeler 19.0` will pop up for drwing an object.

For this simulation, I used `SpaceClaim 19.0`. (I will explain how to use both `SpaceClaim 19.0` and `DesignModeler 19.0` in sequence in `Appendix` part)

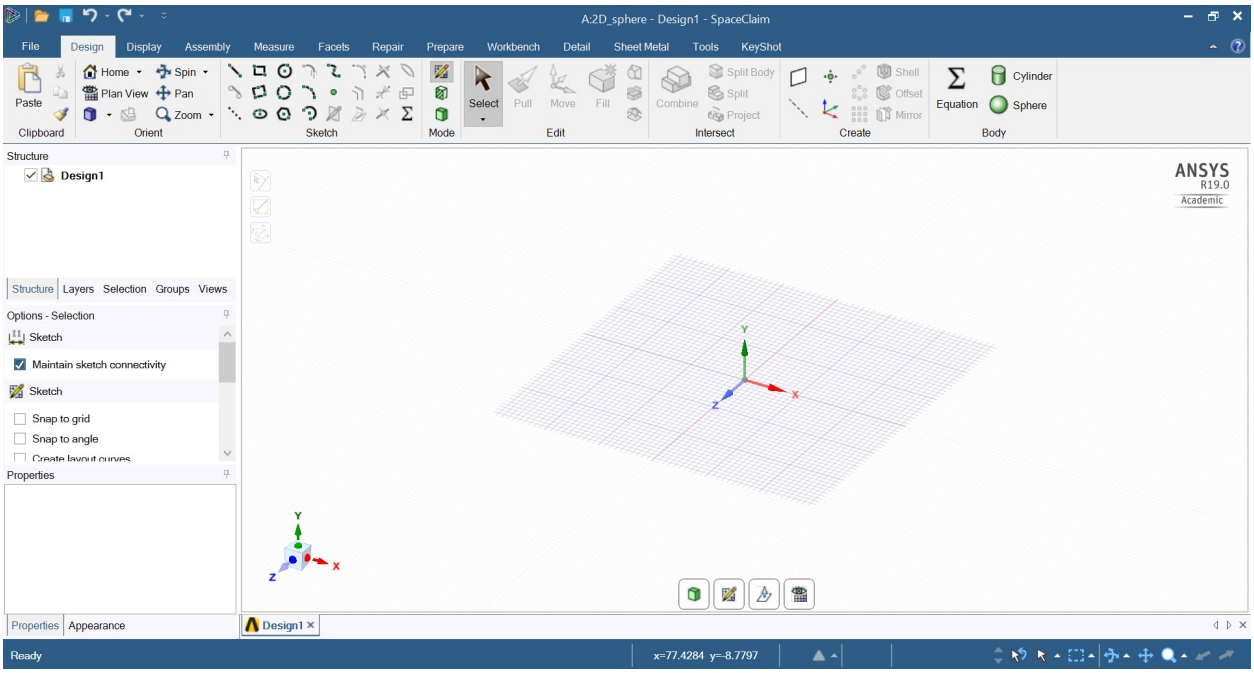


Figure 42: **SpaceClaim 19.0** window

Click `Close` on ANSYS SpaceClaim direct modeler

Start to draw the desired geometry. In my case, I start to draw the circle as the hip joint metal by the diameter of $20cm$, the circle of the polymer coated the metal joint by diameter $23cm$ and the rectangular as the blood flow area enclose the the hip joint and polymer by the 40×100 scale.

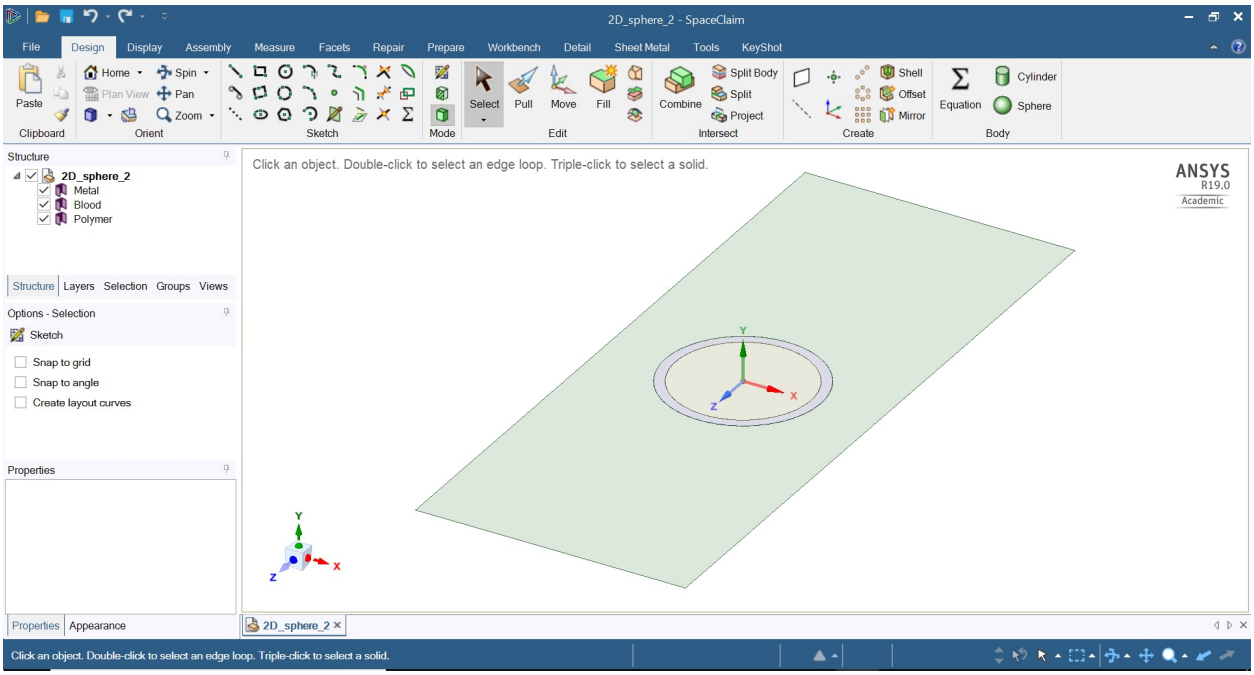


Figure 43: Sketeh the the desired geometry in SpaceClaim 19.0

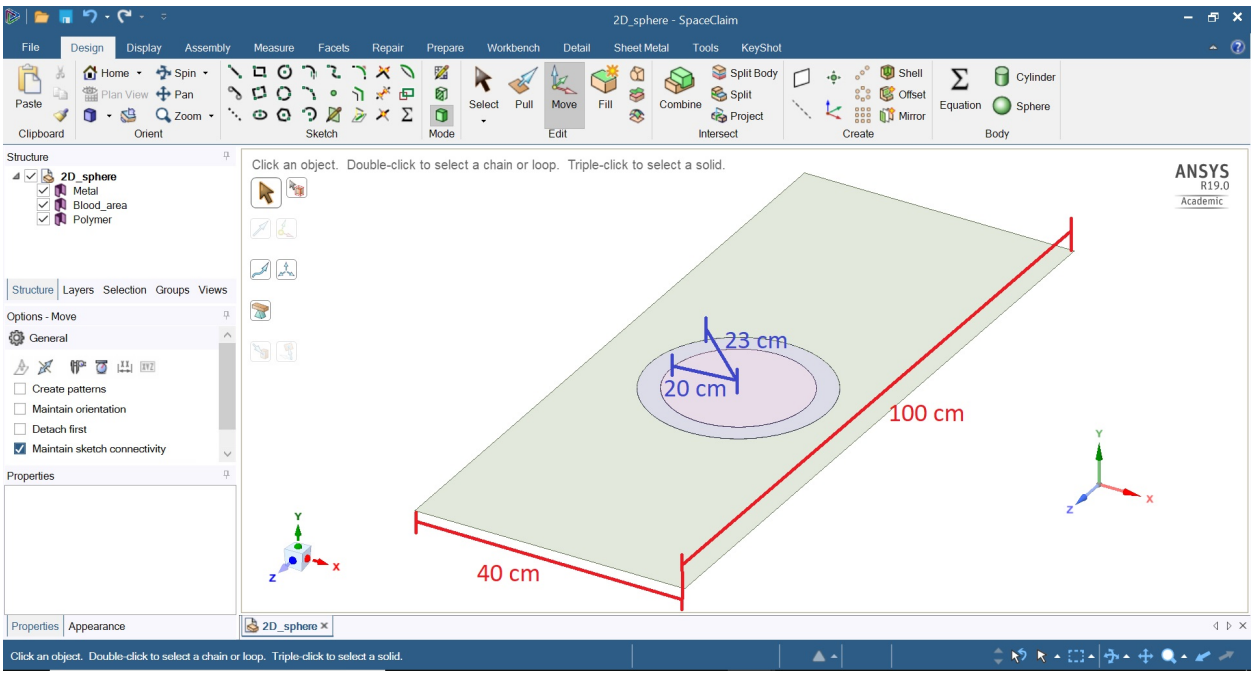


Figure 44: Details on geometry

`Saving the SCDM project` by Click on the `savesymbol` or click `File` →

`Save As...` → `Name.scdoc` → `Save` button.

Close `SpaceClaim` window and reopen your project `Workbench` window.

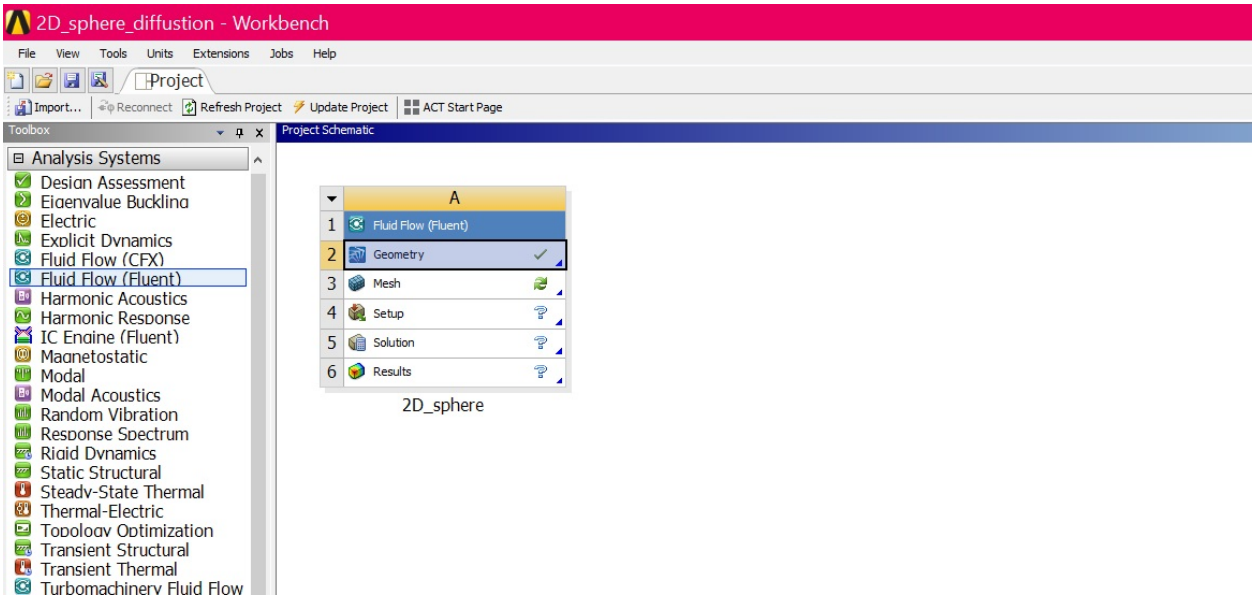


Figure 45: Project control in ANSYS Workbench

Now, we can notice the `Check` sign on geometry tab Figure 45, which mean that this geometry is appropriate.

In case of the change in geometry configurations, the tab of geometry will show the `thunder symbol`, click on the left click and → `Update` as in Figure 46

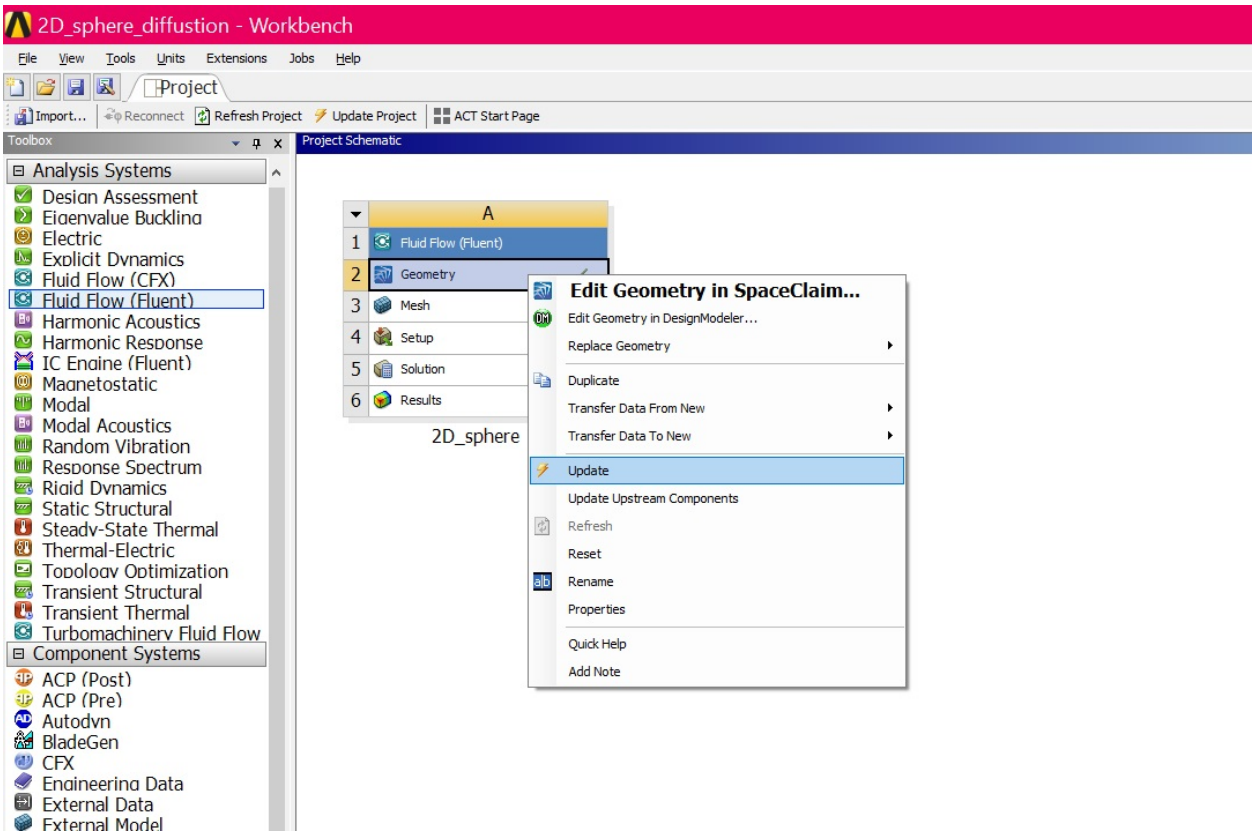


Figure 46: Click on update geometry

if the geometry is appropriate, the workbench will change the sign to Figure 47.

Figure 47: Change of the sign notify that the program can continue the process

Next, On the right hand side in the options of properties of schematic A2: Geometry, change the `Analysis type` on the row 12 into `2D` (in case of 2 dimension).

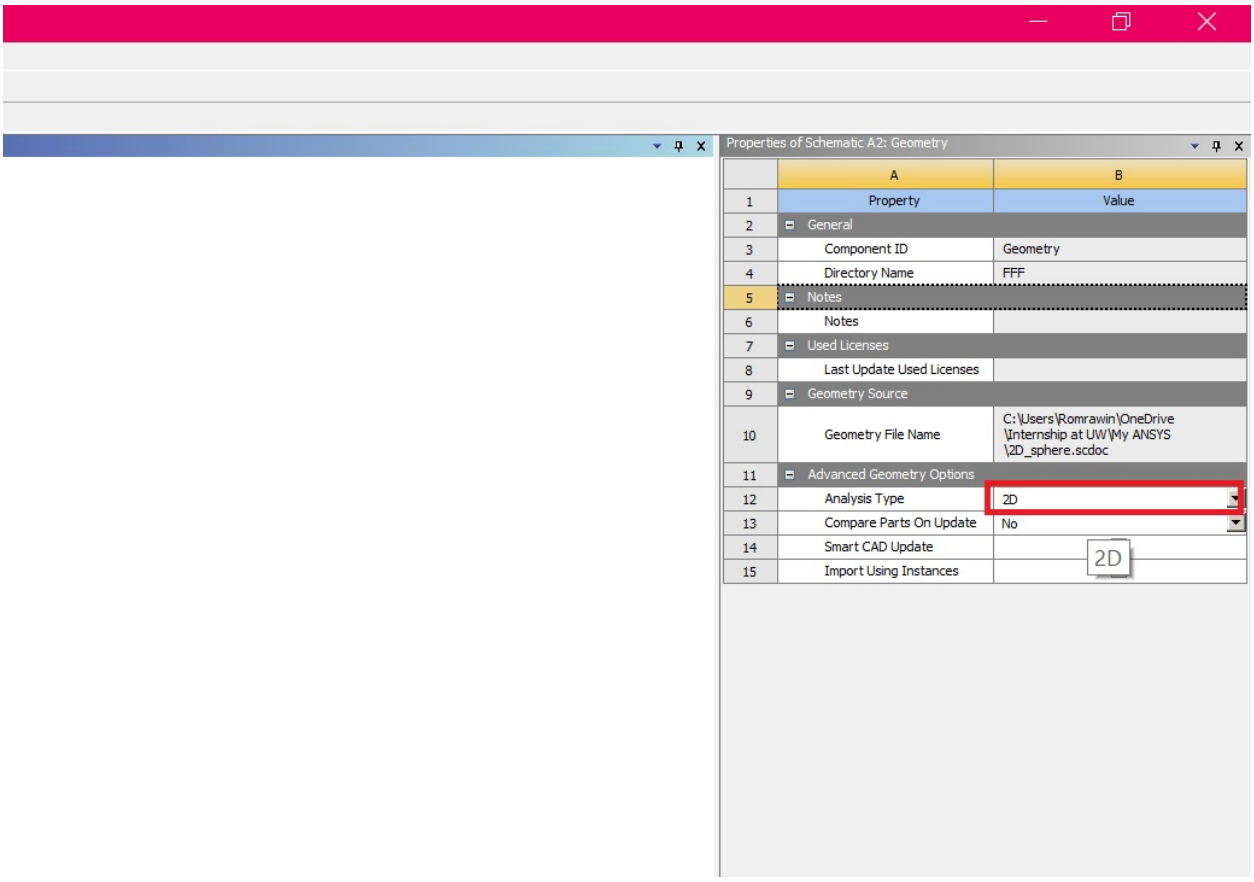


Figure 48: Change Analysis type

Step 3 Create mesh in the object.

Meshing

Meshing is the process that generates the small parts of the object by dividing the whole component into a number of element (finite element analysis). The more mesh influences the more accuracy, convergence, and speed of the solution. Creating the appropriate number of mesh is necessary in engineering simulation. Therefore, The better and more automated the meshing tools, the better the solution.

Double click on mesh tab on the below of geometry tab as in Figure 45.

The Meshing Window will pop up as in Figure 49

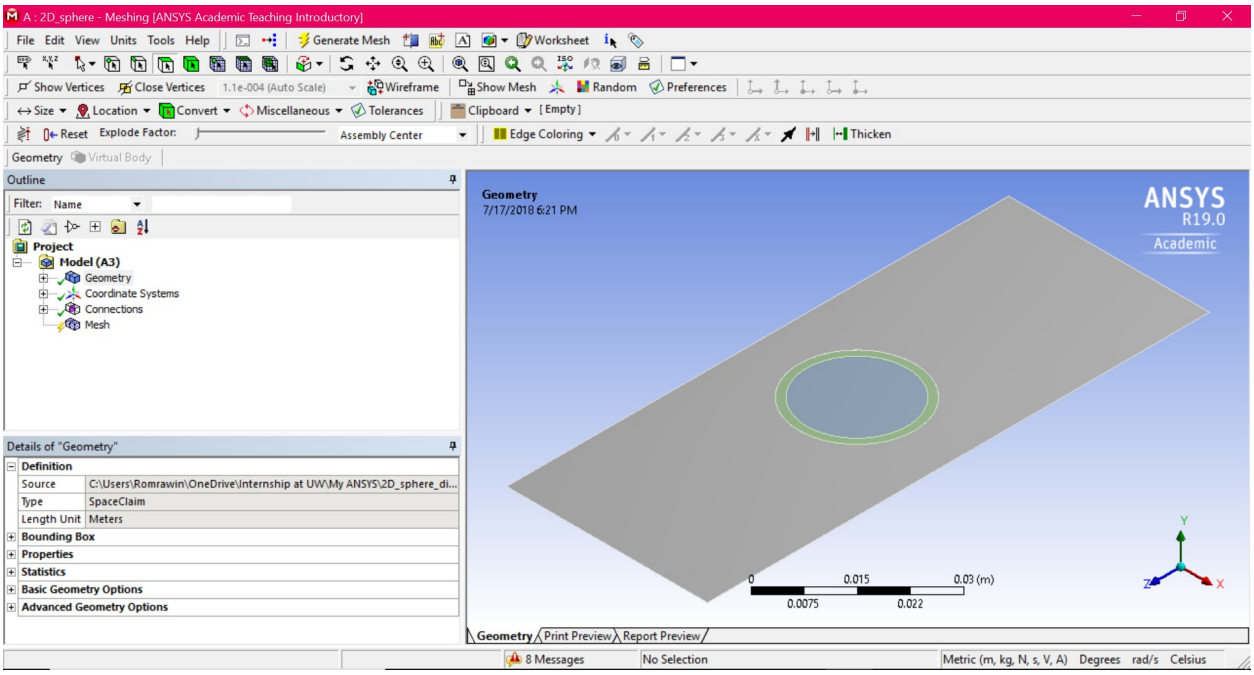


Figure 49: Change Analysis type

Generate a mesh on the object by right click on `thunder symbol` Mesh → click `Generate Mesh`

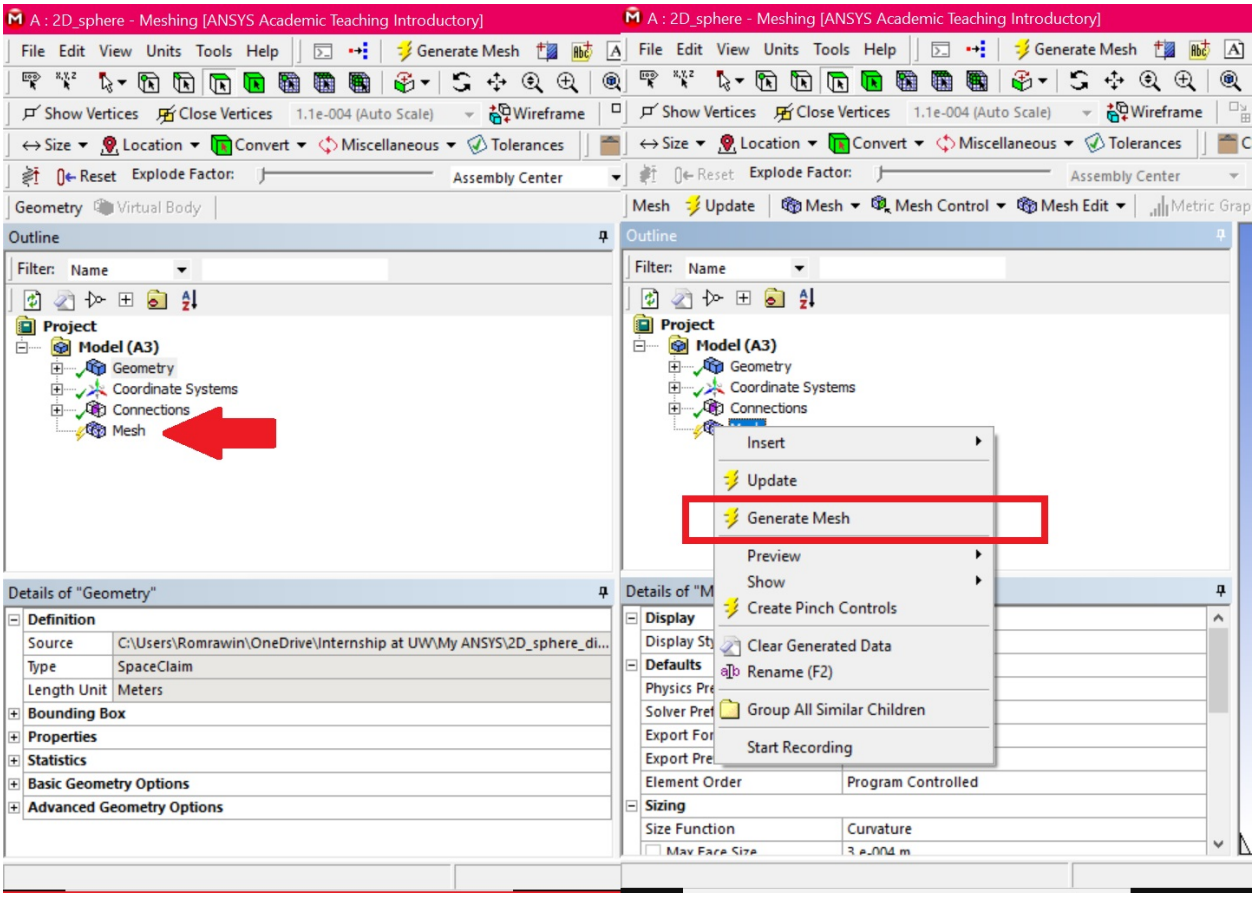


Figure 50: Generate a mesh on the object

After that the program will run to generate the mesh as in Figure 51

Control the size of the mesh by the tab `Sizing` on the `Details of Mesh`, click on `Max Face Size` change it to the smallest number as you want. I change to `0.0003`. Next, right click on `Mesh` → click `Update`

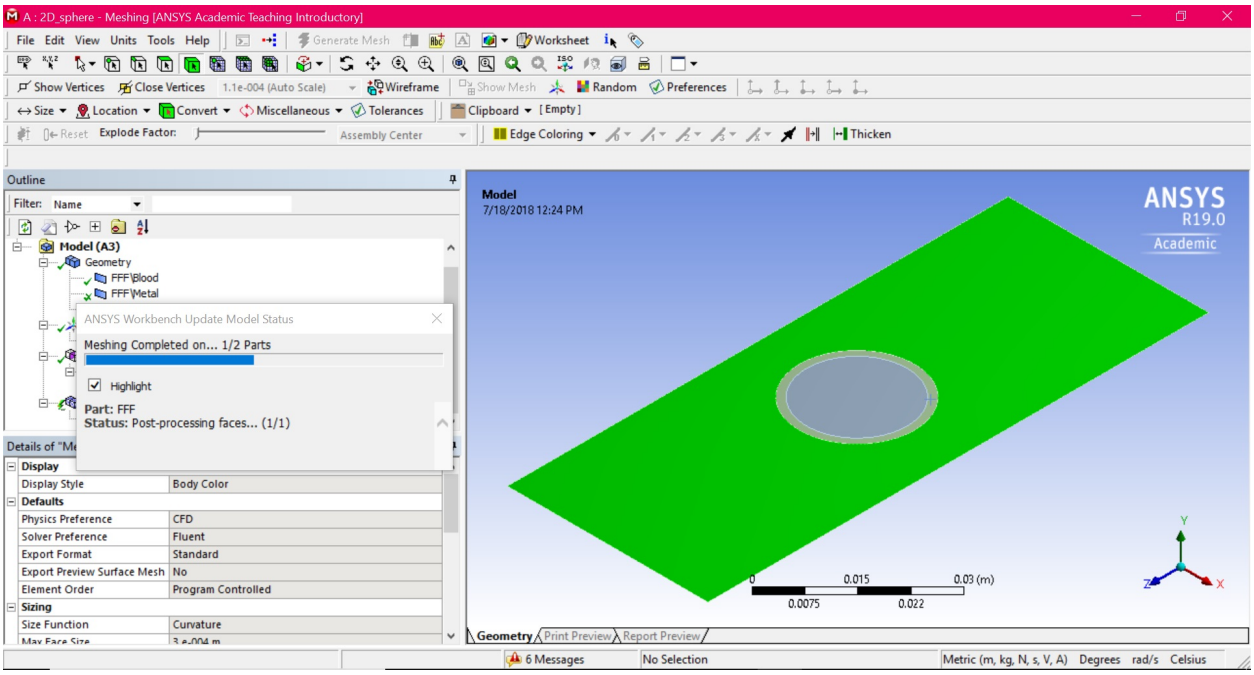


Figure 51: Generate a mesh on the object

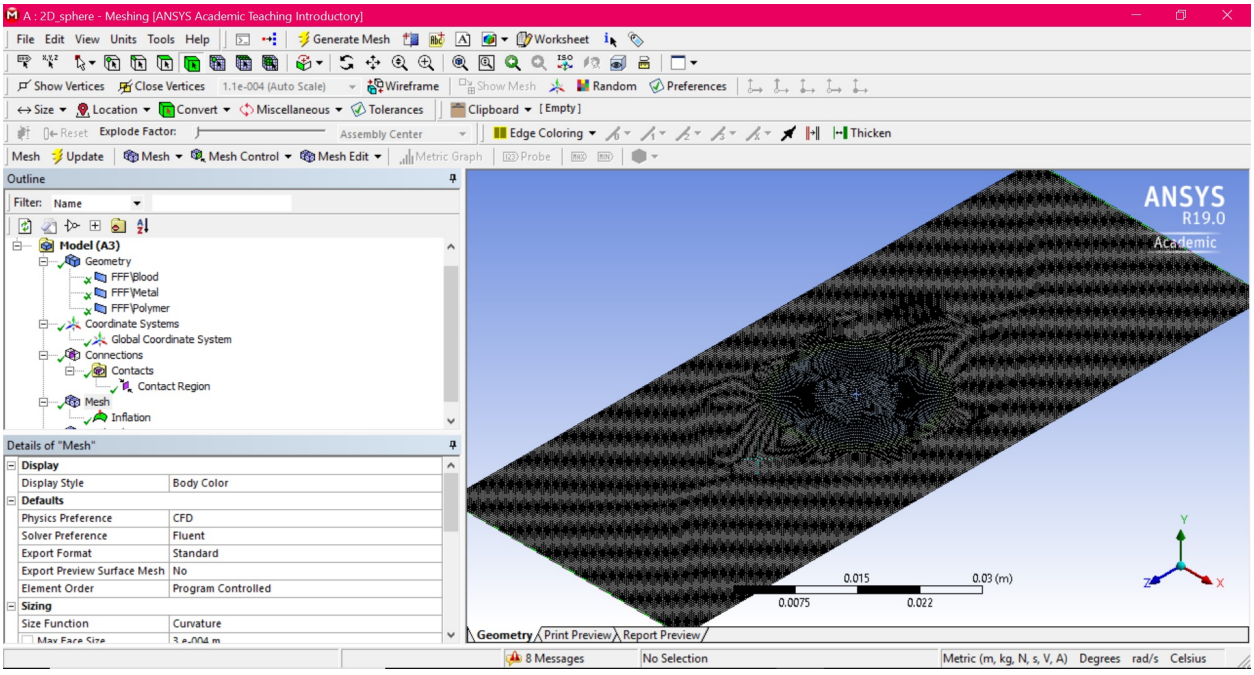


Figure 52: The mesh is generated

Now, we finish meshing process. Close meshing window and reopen the

workbench. In workbench, right click on `thunder symbol mesh` and → click `Update`

The `thunder symbol` will change to `check symbol` as shown in Figure 53.

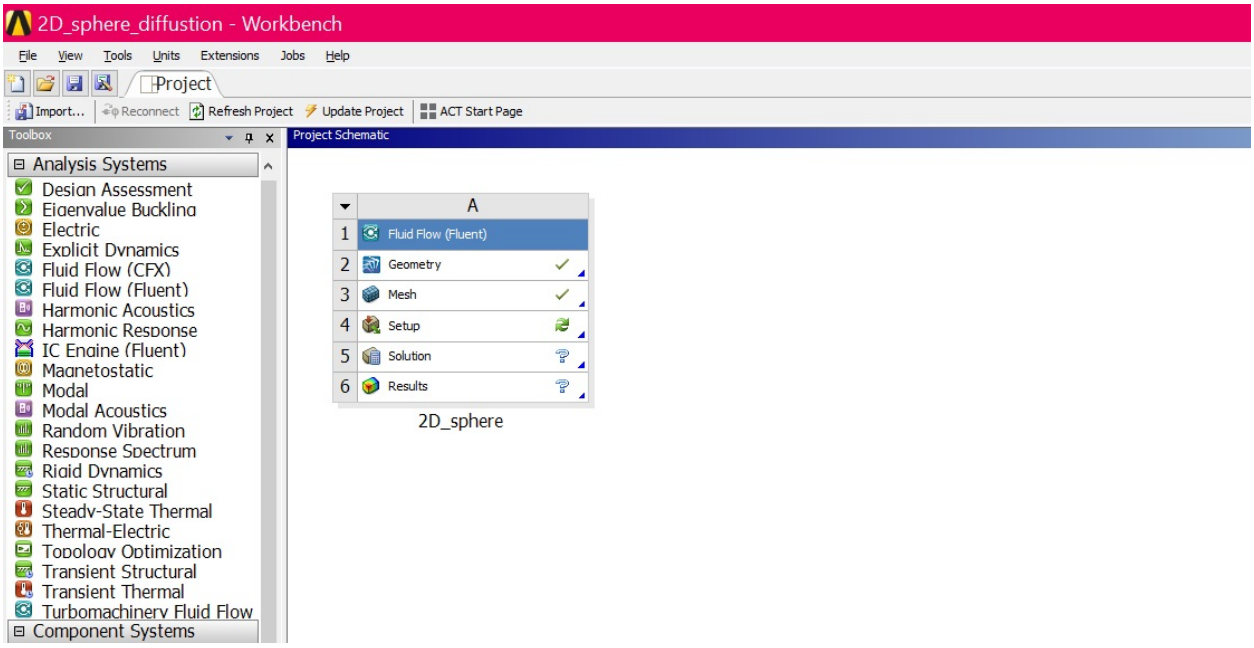


Figure 53: The mesh is appropriate and the process can continue

Named sections

I will name the section by Figure 54 zones.

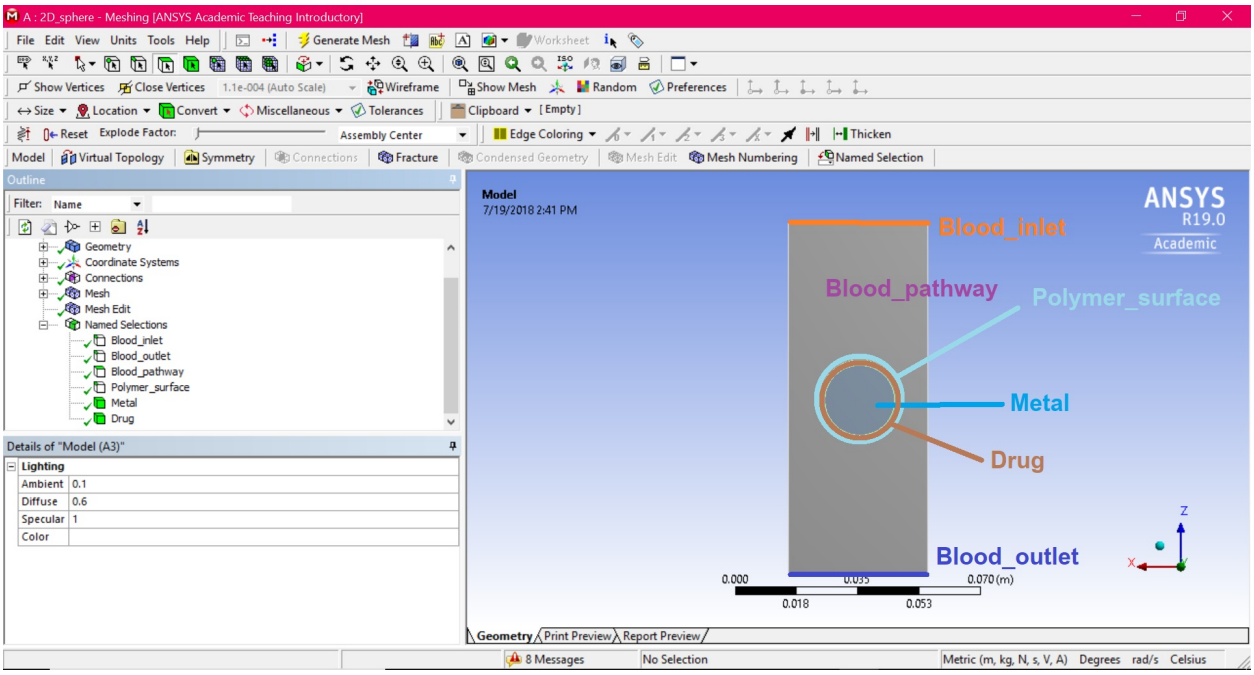


Figure 54: Name the section

For name each sections, Choose the selection tool on the above tab.

Figure 55: Selection region tools

After you select the region on the object, click right → `Create Named Selection` as in Figure 56

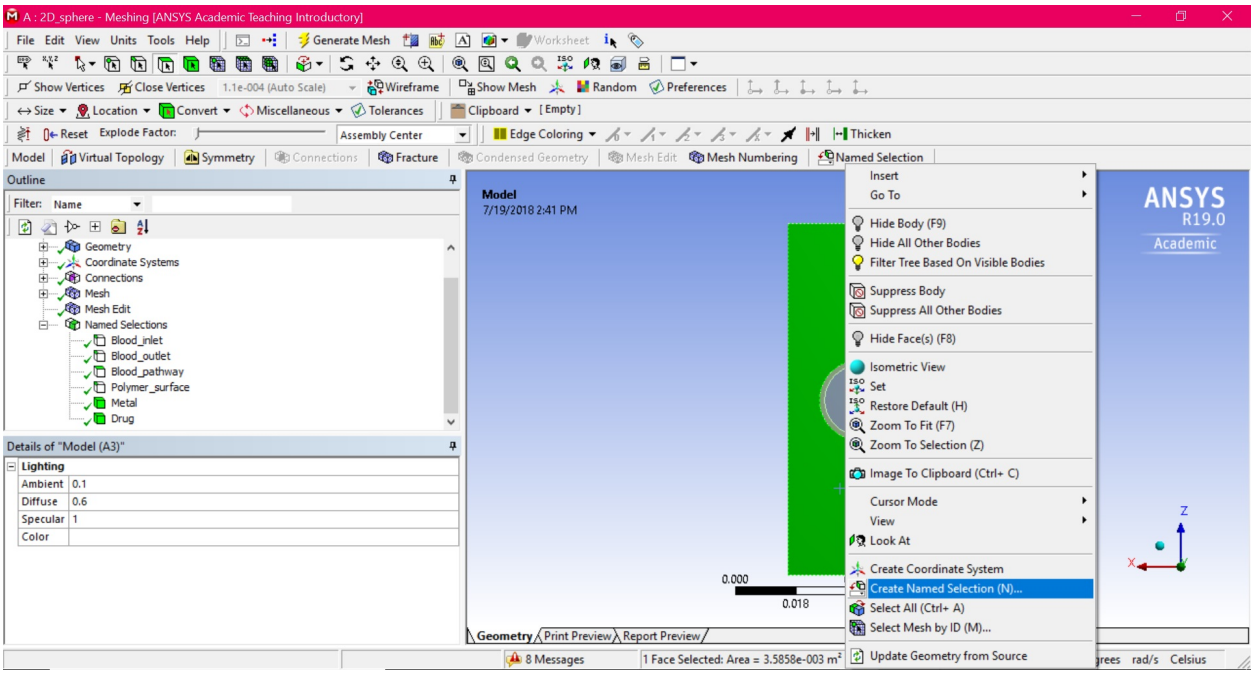


Figure 56: Create name selection

Create the region name.

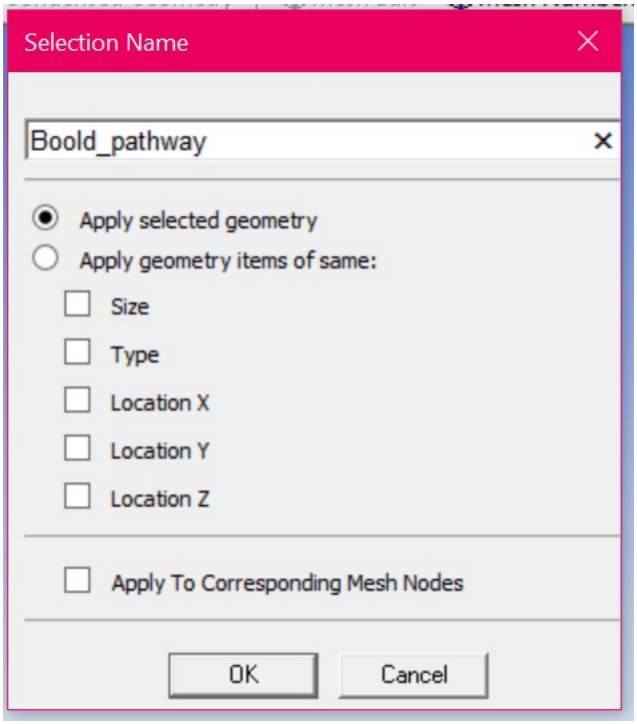


Figure 57: Create name selection

Finish name sections, close the Meshing window, click on `thunder symbol` `Mesh` *rightarrow* `Update`.

Step 4 Set up `ANSYS Fluent`

Double click on `Set up` in Figure 53. ANSYS Fluent option will pop up as in **??**.

Select:

`Dimension` → 2D

`Processing Options` → serial

`Options` → `Double Precision`,

Leave other as the same, Click `OK`

ANSYS software will pop up as in the window

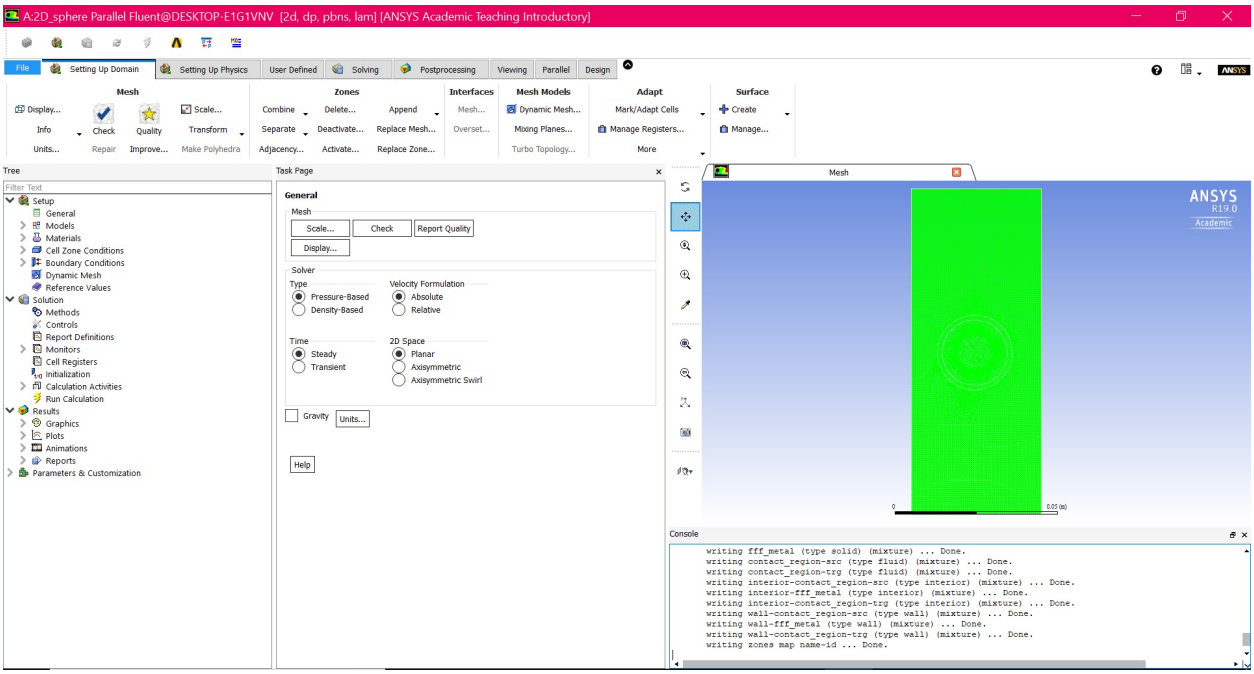


Figure 58: ANSYS Fluent window

In `Fluent` window (Figure 59), the program set up to have two control spaces, the first zone is the above of window which qualify the type of control such

as `Setting Up Domain`, `Setting Up Physics`, `User defied`, so on. Second zone controls can be accessed in the middle of the window, which correspond to the lists on the left and in the workbench such as `Set up`, `Solution`, `Result`, so on.

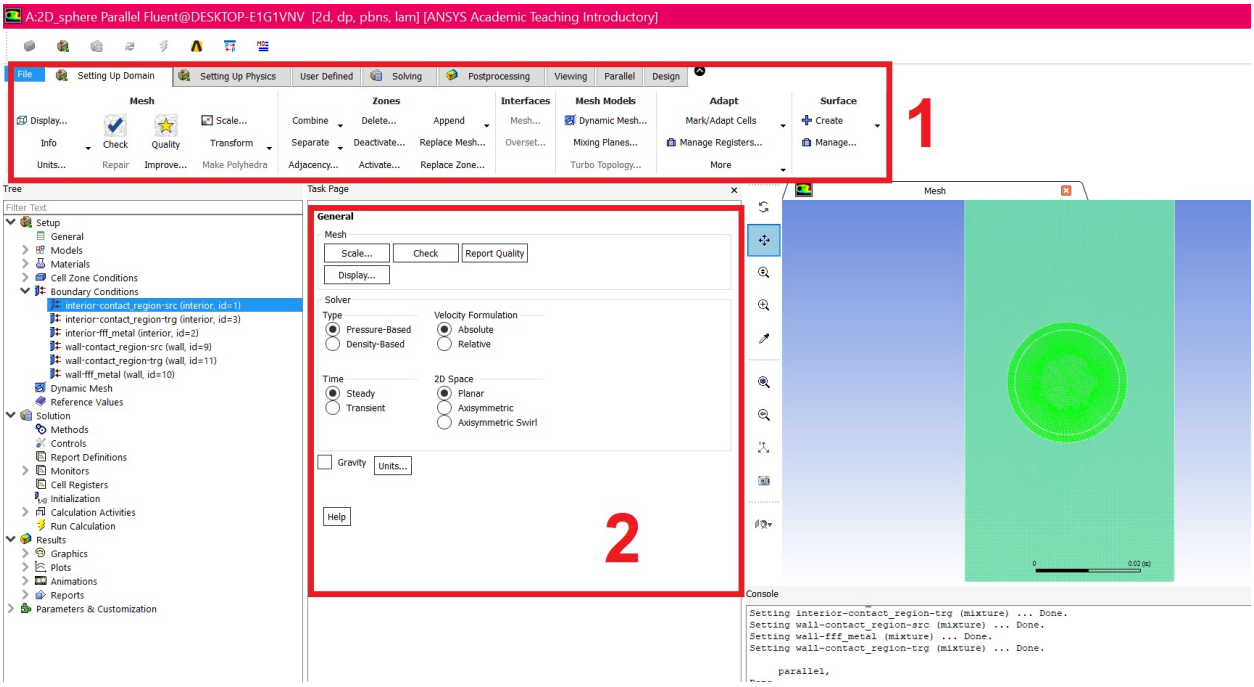


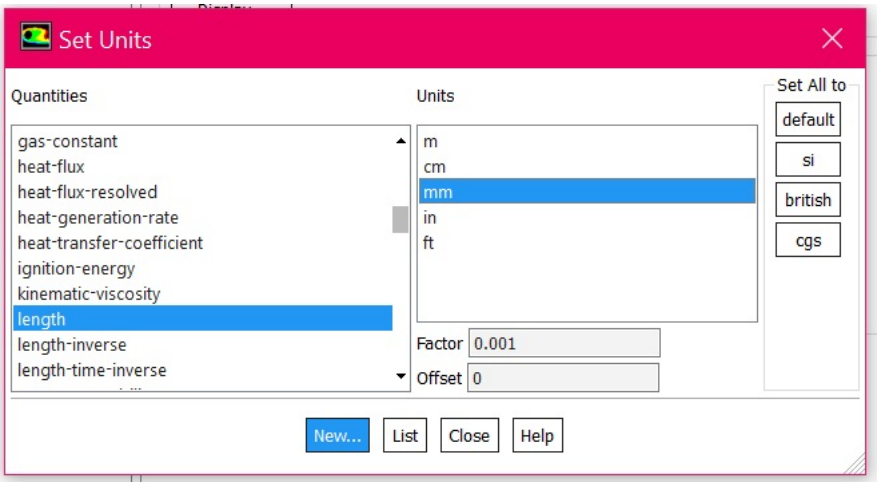
Figure 59: ANSYS Fluent window and control area

Begin Set up

1. Setting up domain In setting up the domain, you will check the mesh-related properties

Change the units

Clicking `Setting Up Domain` → `Mesh` → `Units`. In my case, I will change the units **length** from m to mm



Check the mesh

Clicking `Setting up Domain` → `Mesh` → `Check`

The values of the mesh will report in the console.

```
## Domain Extents:
##    x-coordinate: min (m) = -1.951386e-02, max (m) = 2.048614e-02
##    y-coordinate: min (m) = -4.973950e-02, max (m) = 5.026050e-02
##  Volume statistics:
##    minimum volume (m3): 3.356570e-09
##    maximum volume (m3): 8.400654e-08
##      total volume (m3): 4.000000e-03
##  Face area statistics:
##    minimum face area (m2): 9.167062e-06
##    maximum face area (m2): 4.132829e-04
##  Checking mesh.........................
## Done.
```

If the errors on the mesh occur, it will be reported in this step. And, ensure that minimum volume is not negative because ANSYS Fluent cannot begin the calculation.

Review the mesh quality

ANSYS Fluent will report the mesh quality in the console.

```
## Minimum Orthogonal Quality =  9.46787e-01 cell 314 on zone 4
## (ID: 90748 on partition: 0) at location (-2.89720e-03 -4.34247e-03)
```

```
## (To improve Orthogonal quality , use "Inverse Orthogonal Quality"
## in Fluent Meshing, where Inverse Orthogonal
## Quality = 1 - Orthogonal Quality)
##
## Maximum Aspect Ratio =  1.62824e+01 cell 318 on zone 4
## (ID: 90752 on partition: 0) at location (-2.83154e-03 -4.51415e-03)
```

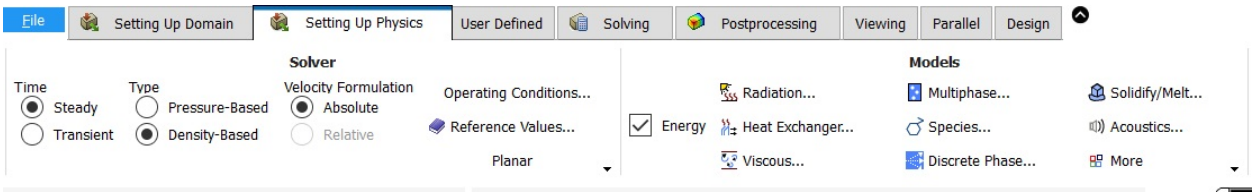
Minimum cell orthogonality is an important indicator of mesh quality (vary from 0 to 1), which the lower values indicating the poorer quality cells. In our case, Minimum Orthogonal Quality = 0.946787, which can refer to a good quality. Generally the minimum orthogonality should not be below 0.01 on the average. For maximum aspect ratio, the higher ratio in cell refer to a result of boundary layer inflation in the meshing step.

2. Setting up physics

In this step, we will control the properties in a solver.

Clicking on `Setting Up Physics` → `Solver`

we will set the properties as



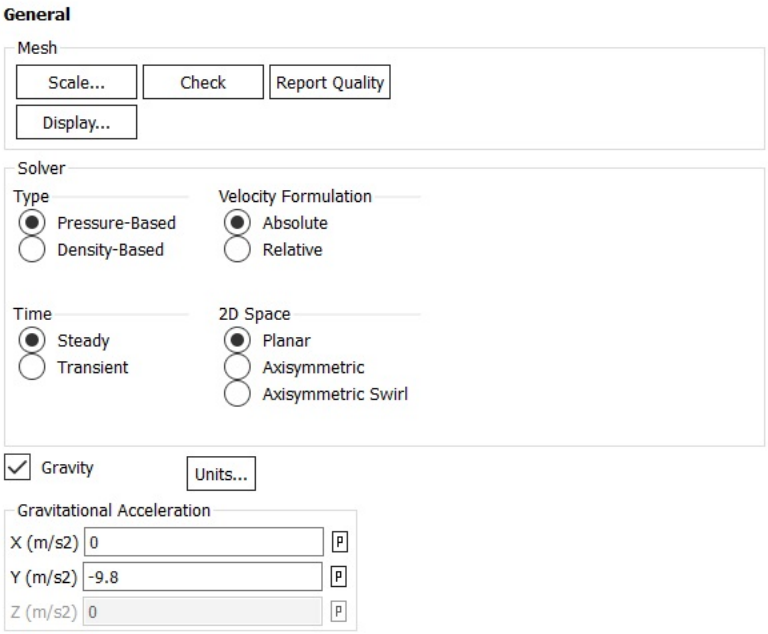
Type → Density-based coupled solver (DBCS) ; because major aim of our process is performed by quality of mass transfer. In this case, DBCS method initializes the equation from continuity, momentum, enery, and species and then obtains the pressure from the equaition of state.

Time → Steady

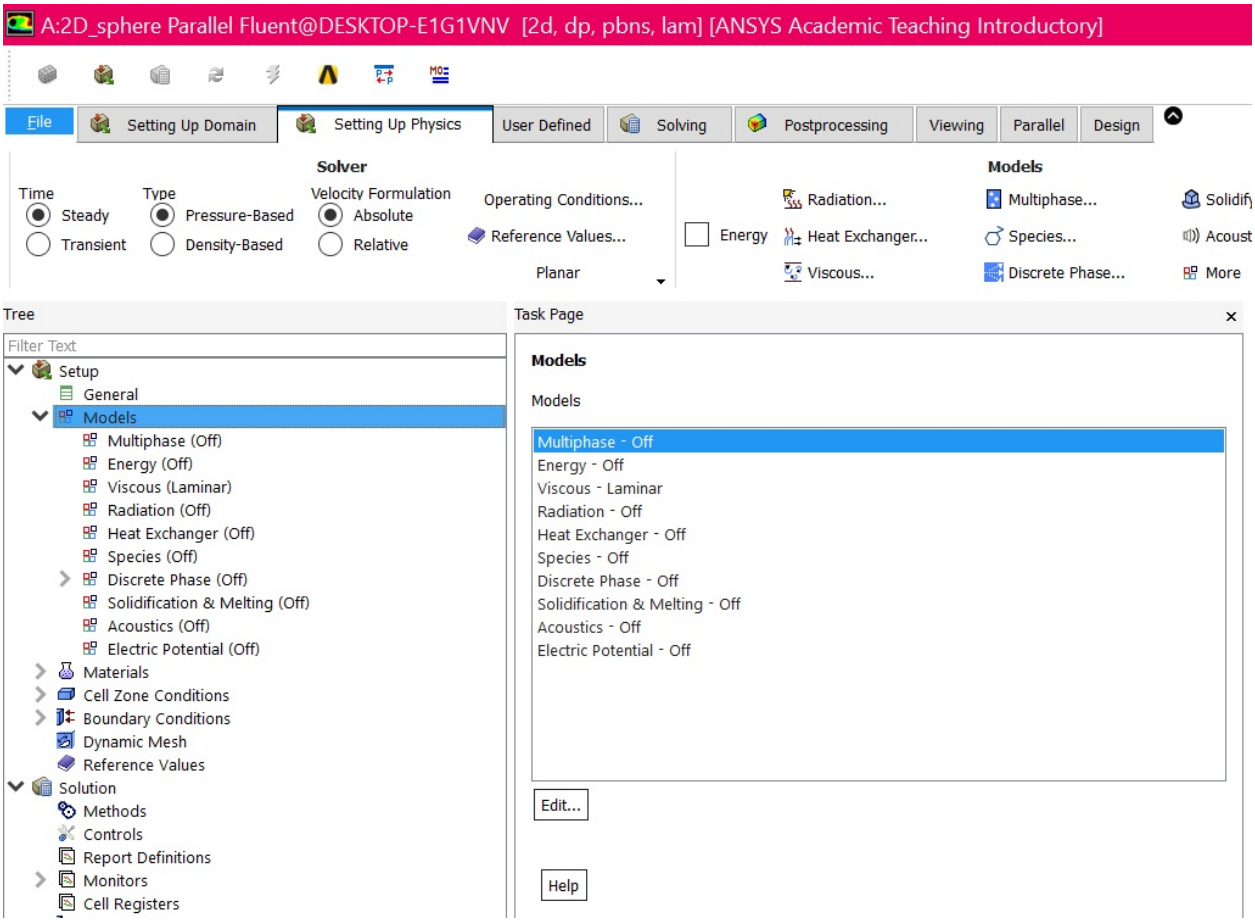
Velocity-formulation → Absolute

2D Space → Planar

Also, setting up the gravity force in y-direction = -9.8 m/s^2



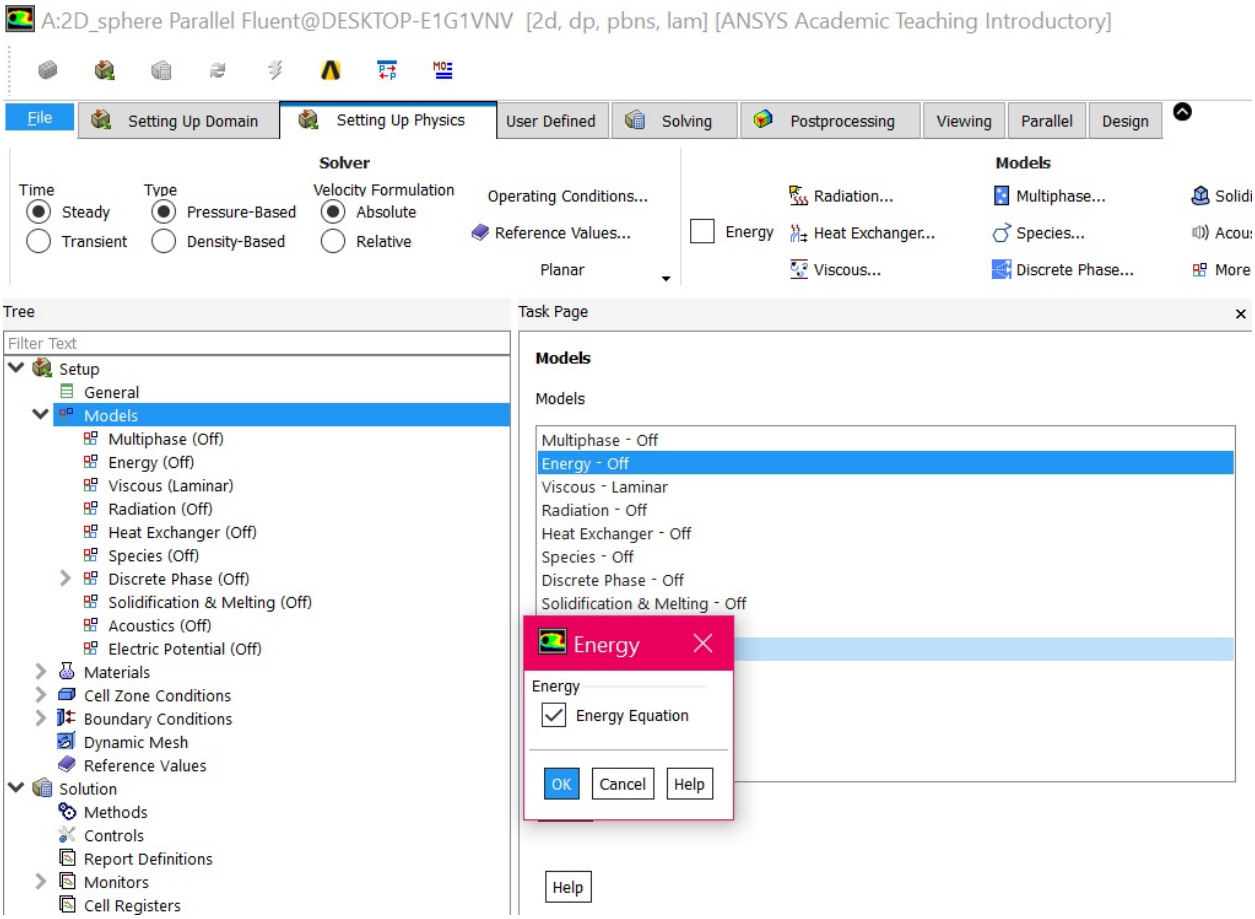
Next, set up the CFD simulation using the `Models` group in the `Setting Up Physics` in the tab or clicking on the `Tree`



Setting Up Model

1. Enable heat transfer by activating energy mode

Clicking on `Setting Up Physics` → Select `Energy` → `Energy Equation` → OK



2. Select Viscous Model → `Laminar`

Clicking on `Setting Up Physics` → Select `Viscous..` → `Laminar` → OK

Select **Laminar** because the system of blood flow is laminar flow with the average mass flow is 0.0235 kg/s

3. Set up material

This step is for set up material for your CFD simulation by clicking on the `Flask symbol` above or clicking or the `Materials` on the left tree and click → `Create/Edit...`

Blood Properties	
Property	**Value**
Density	1060 $[kg/m^3]$
C_p (Specific Heat)	3513 $[J/kg-K]$
Thermal Conductivity	0.44 $[W/m-K]$
Viscosity	0.003 $[kg/m-s]$

Table 5: Blood Properties

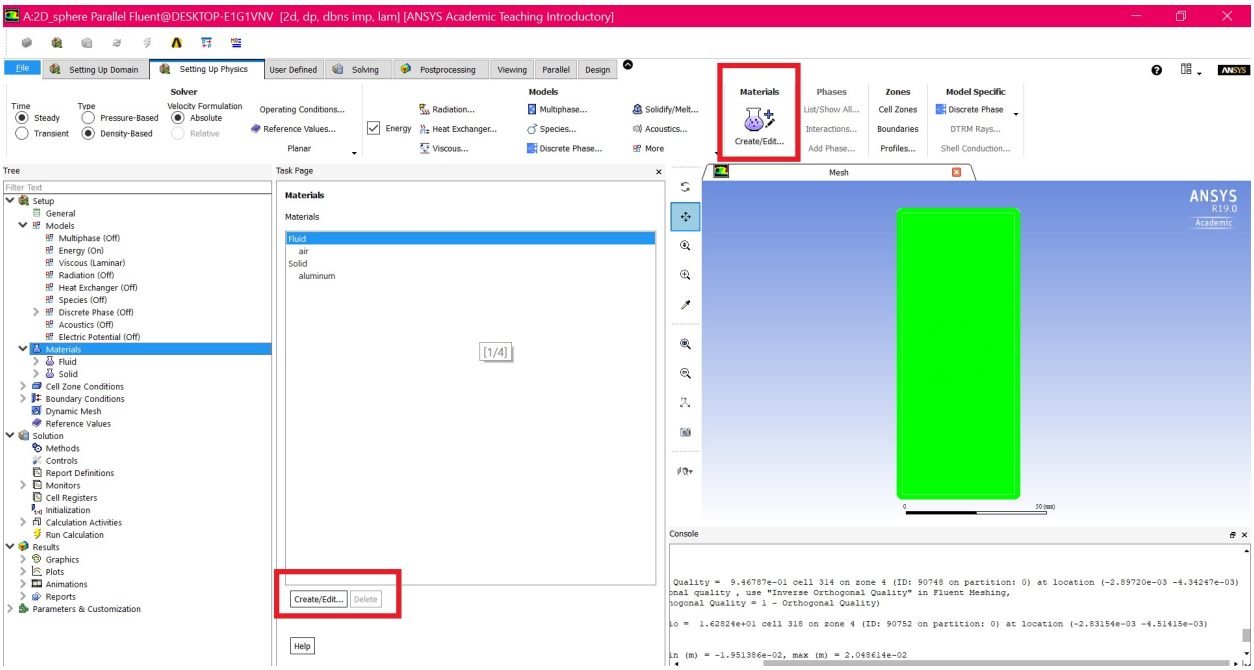
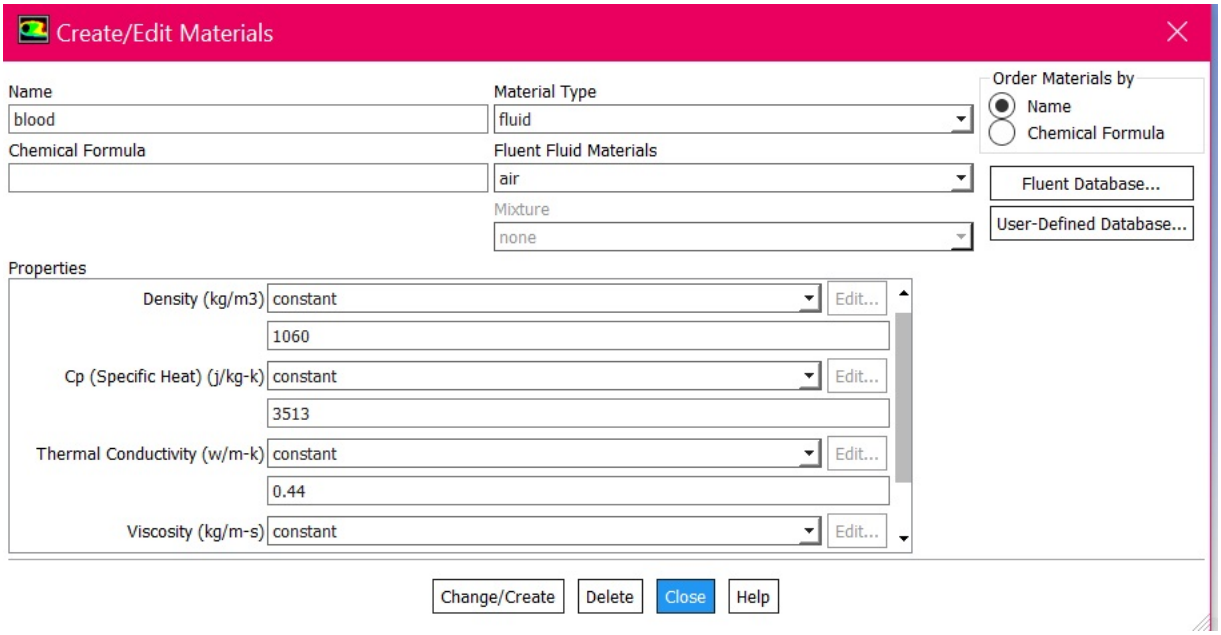


Figure 60: Set up material; blood and drug

a. Create Fluid: Blood

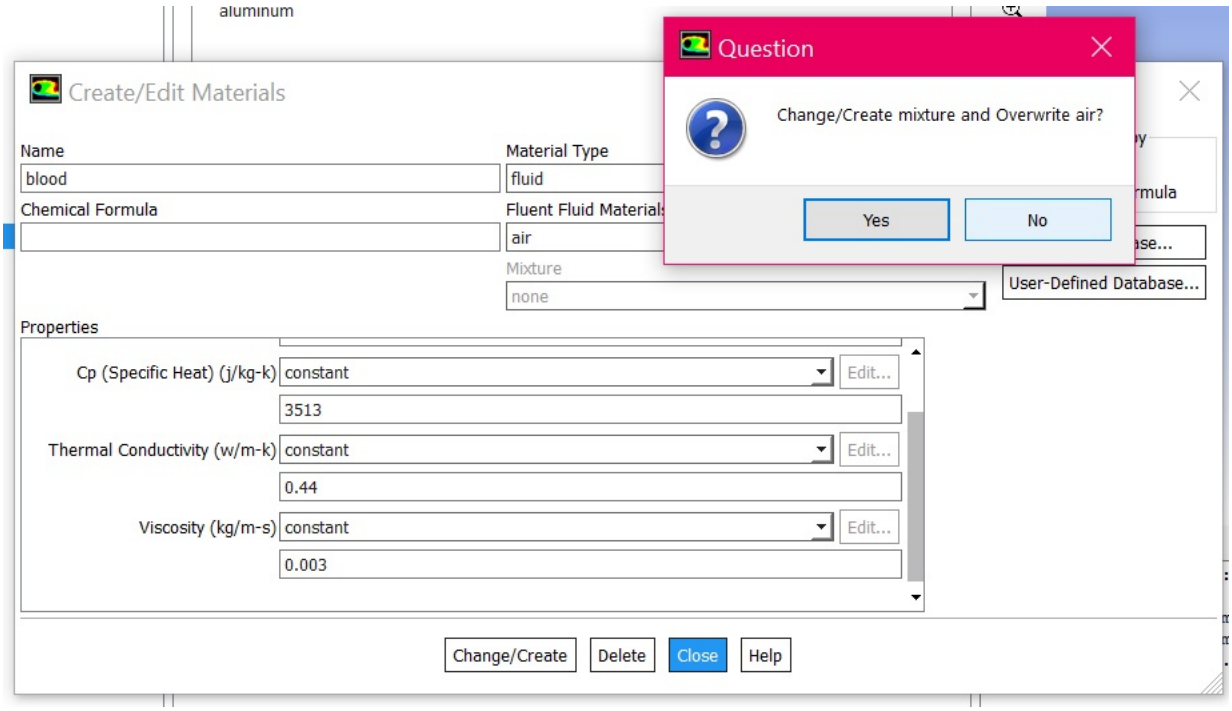
Create/Edit Materials in Fluid dialog box, change the **Name** to `blood`

Change Properties



Then, click **Change/Create**.

A **Question** in dialog box will pop up → click **No**. So, the new material is created in Fluid Material tab.



b. Create Fluid: Drug

Drug Properties	
Property	**Value**
Density	1449.265 $[kg/m^3]$
C_p (Specific Heat)	4000 $[J/kg - K]$
Thermal Conductivity	0.5 $[W/m - K]$
Viscosity	0.00001 $[kg/m - s]$

Table 6: Drug Properties

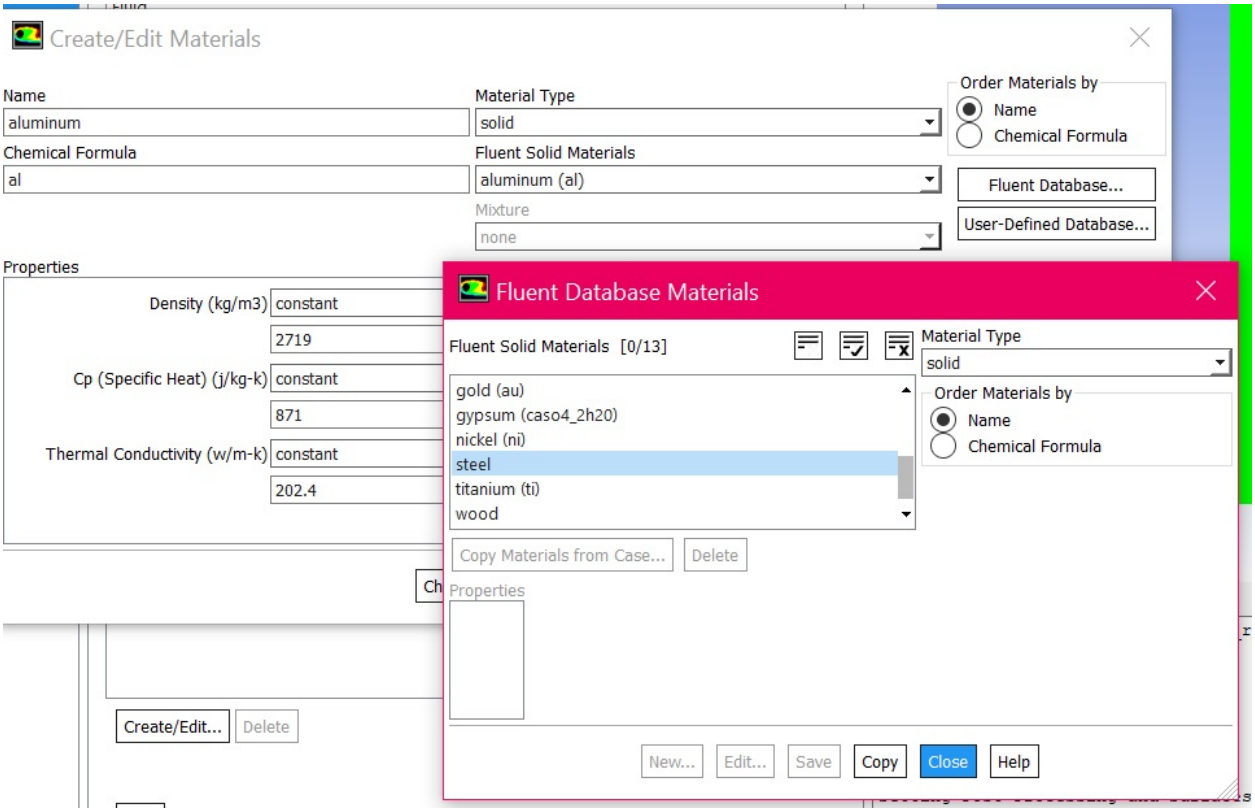
From the previous study, they use vancomycin as an antibiotic drug. So, I will add the properties of vancomycin in drug fluid

Create the same step as in blood

c. Create Solid: Steel Normally, steel is the major component of a artificial hips because its resistance.

Create solid is the same in Fluid, data of steel is already stored in **Fluent database**.

Click on **Fluent Database** $\rightarrow$ select **Steel** $\rightarrow$ click **copy** to create a new solid material. Then, $\rightarrow$ click **Change/Create**. Steel material will be stored in set up material.



On the models,

Radiation → Off

Heat Exchager → Off

Discrete Phase → Off

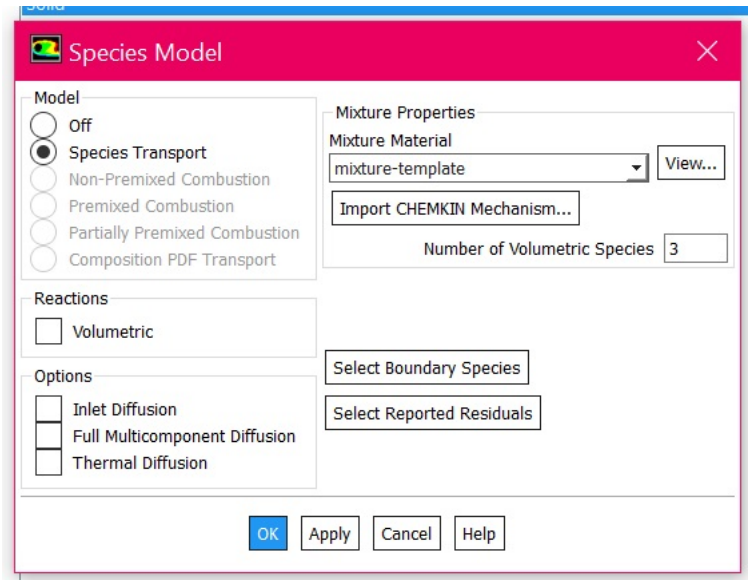
Acoustics → Off

Electric Potential → Off

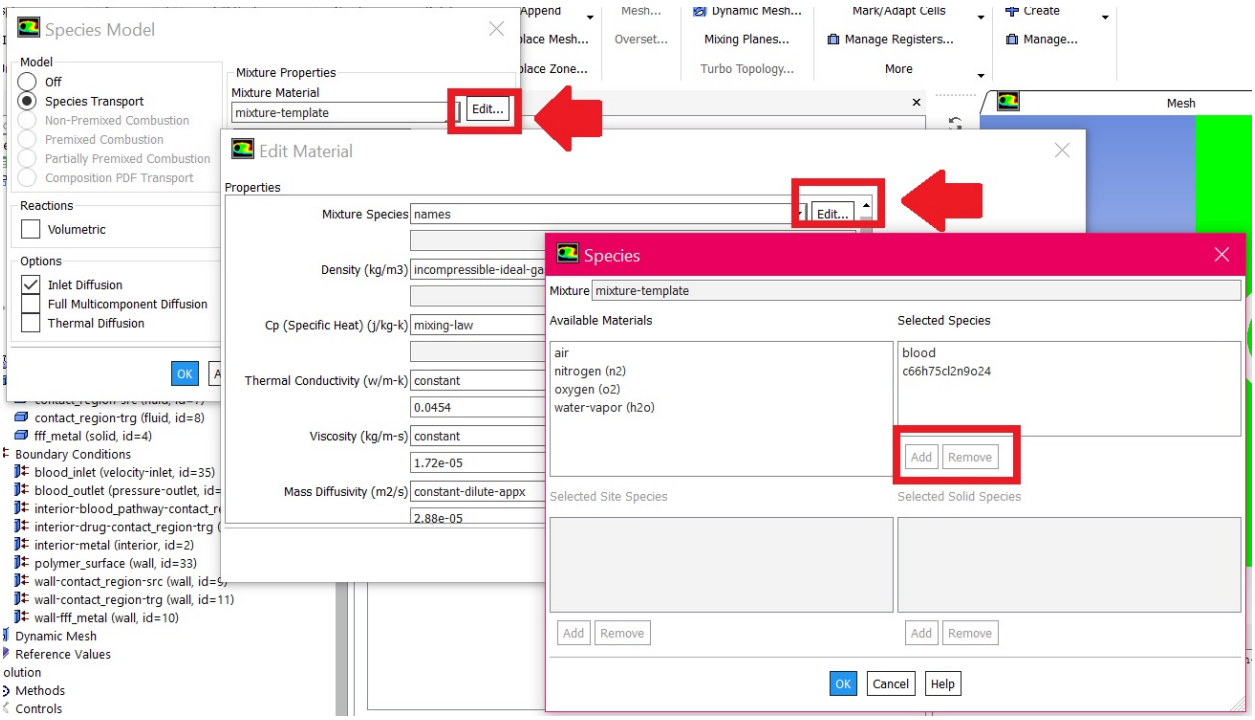
4. Enable **Chemical species** transport and reaction

Clicking on **Setting Up Physics** → **Models** → **Species**

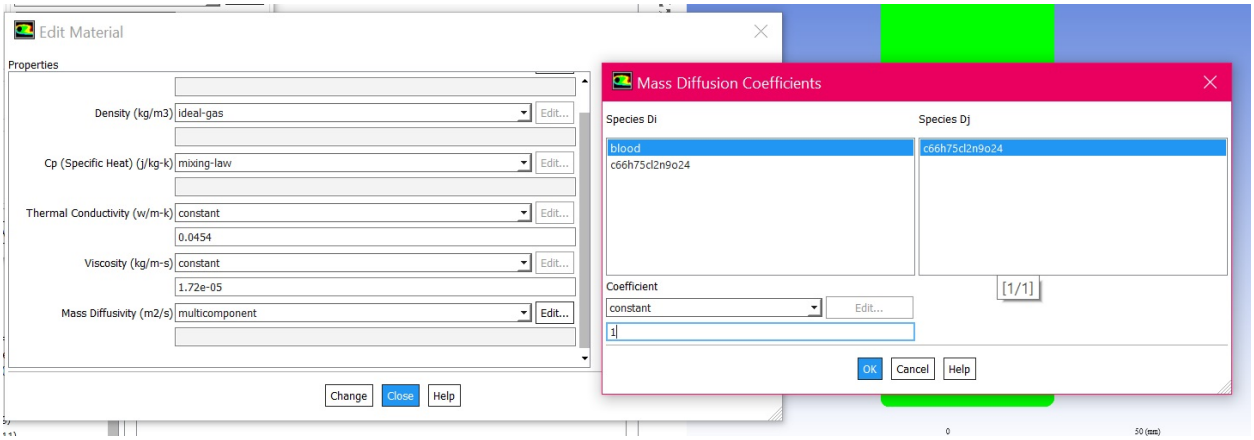
Specied model dialog box will show up, → **Species Transport** in the model list

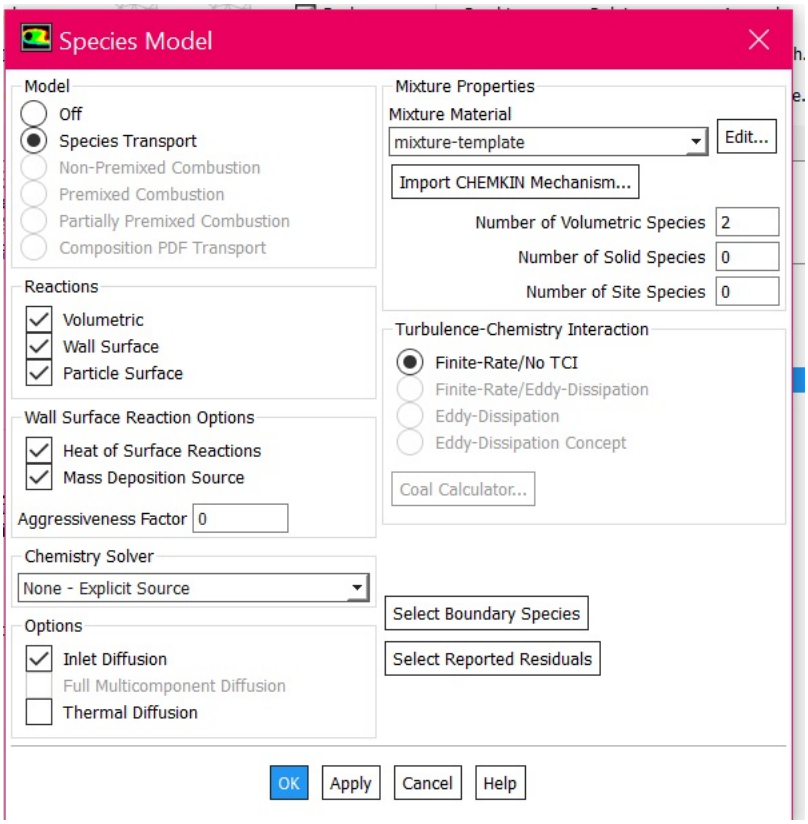


Select **mixture-template** → **Edit** → click **Edit...** on Mixture Species again



Then, Click adding **Blood** and **Drug** in selected species box. And, remove irrelevant species out. → Click **OK**





5. Setting **Cell zone condition**
6. Setting **Boundary condition**
7. **Initialize model**
8. **Solving model**