DESIGN, ANALYSIS AND DYNAMICS OF MECHANICAL MICROSWIMMERS

A project report submitted in partial fulfillment of the requirements for the award of the degree of

BACHELOR OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

B.BHUPATHI	**(315126520264)**
V.V.V.SAI KRISHNA	**(315126520223)**
T.JAGADEESH	**(315126520270)**
R.NITISH KUMAR	**(315126520285)**
R.USHA	**(315126520252)**

Under the guidance of

ROOPSANDEEP BAMMIDI, M.Tech (Ph.D)

Assistant professor

Department of Mechanical Engineering

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES (A)

(Approved by AICTE, permanently affiliated to ANDHRA UNIVERSITY)

Accredited by NBA and NAAC with 'A' Grade

Sangivalasa -531162, Bheemunipatnam, Visakhapatnam.

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES
(Autonomous)

(Affiliated to Andhra University, Accredited by NBA)

Sangivalasa531162, Bheemunipatnam Mandal, Visakhapatnam

PRAGNANAM BRAHMA
ANITS

DEPARTMENT OF MECHANICAL ENGINEERING

CERTIFICATE

This is to certify that the project entitled ***"DESIGN, ANALYSIS AND DYNAMICS OF MECHANICAL MICROSWIMMERS"*** *describes the Bona-fide work done by* ***B.BHUPATHI(315126520264), V.V.V.SAI KRISHNA(315126520223), T.JAGADEESH (315126520270), R.NITISH KUMAR(315126520285), R.USHA (315126520252)*** *in partial fulfillment for the award of Degree in* ***BACHELOR OF TECHNOLOGY*** *in* ***MECHANICAL ENGINEERING*** *under my supervision and guidance of* ***B ROOPSANDEEP*** *during the academic year* ***2018-2019***.

PROJECT GUIDE	**HEAD OF THE DEPARTMENT**
B.ROOPSANDEEP	**Dr. B. NAGARAJU**
Assistant professor	**Dept. of Mechanical Engineering**
Dept. of Mechanical Engineering	

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
Sangivalasa-531 162 VISAKHAPATNAM Dist. A.P.

THIS PROJECT APPROVED BY THE BOARD OF EXAMINERS

INTERNAL EXAMINER

15/4/19

15/4/19

EXTERNAL EXAMINER

ACKNOWLEDGEMENT

We present this report on **"DESIGN, ANALYSIS AND DYNAMICS OF MECHANICAL MICROSWIMMERS"** in the partial fulfillment of the requirement for the award of BACHELOR OF TECHNOLOGY in MECHANICAL ENGINEERING.

We intend to express our thanks with sincere obedience to **Prof. T. Subrahmanyam**, Principal, ANITS and **Prof. B. Naga Raju**, Head of the department, Mechanical Engineering, ANITS for providing this opportunity.

We take this opportunity to express our deep and sincere thanks to our esteemed guide **Mr. B. RoopSandeep**, Assistant Professor, Mechanical Engineering Department, ANITS, a source of constant motivation and best critic, for his inspiring and infusing ideals in getting our project done successfully.

Lastly we are grateful to one and all who have contributed either directly or indirectly in the completion of the project.

B.BHUPATHI	**(315126520264)**
V.V.V. SAI KRISHNA	**(315126520223)**
T.JAGADEESH	**(315126520270)**
R.NITISH KUMAR	**(315126520285)**
R.USHA	**(315126520252)**

ABSTRACT

The fascinating field in present civilization scenario at the edge of science is micro-swimmers, which is a combination of bio physics with self-propulsion mechanisms involving swimming strategies at low Reynolds number. These microswimming robots offer many advantages in biomedical applications such as drug delivery to some specific locations in our human body and also conducting some surgical operations like opening of blocked arteries etc.

In recent times, blocked arteries becomes a major case in the medical world. This is can be diagnosed by AngioPlasty(is a minimally invasive, endovascular procedure to widen narrowed or obstructed arteries or veins, typically to treat arterial atherosclerosis) method. So this is the main reason to choose the aorta as our domain for analysis purpose.

In this we present a micro-swimmer with three different heads they are spherical head, Capsule type head and Tapered cylindrical or elliptical head. We model them using SOLIDWORKS and analysis in ANSYS FLUENT.

TABLE OF CONTENTS

CHAPTER 1

1. INTRODUCTION

Micro-swimmers are the bio-hybrid micro-robots consisting of artificial microstructures. It is designed mechanically with self-propulsion. This is the application of engineering principles and technology to medicine and biology for healthcare purposes. In the present situation of medical life, it is necessary to close the gap between engineering and medicine for advanced health care treatment. Micro-swimmer is the next step for that, having wide range of applications. It is able to travel in the human body through the Arteries, then it can treat the entire body internally and can give the information and report of every part of the body. As arteries of a human body looks like a thin wire, then the micro-swimmer which will be induced in it should be in micro level. So here the design of micro-swimmer matters very important and it is very complicated. Hence our main agenda is design, analysis and dynamics of mechanical micro-swimmer. Tiny swimmers, be they micro-organisms or micro-bots, live in a world dominated by friction. In this world, technically, the world of low Reynolds numbers, motion is associated with energy dissipation. In the absence of external energy supply objects rapidly come to rest. It is both conceptually interesting, and technologically important, for example in applications to nano-medicine , to try and understand what classes of strategies lead to effective swimming in a setting dominated by dissipation. A particularly promising class of strategies is where the motion is, in a sense, only apparent; where a shape moves with little or no motion of material particles.

Artificial micro- nano-robots have attracted lots of researchers to carry on extensive study due to their considerable promise for diverse biomedical tasks such as targeted therapy, tissue removal and micro-manipulation. Purcell found the advantages of the nonreciprocal motion and demonstrated two efficient swimming modes at low Reynolds number the flexibleoar and the corkscrew. Although Qiuetal presented a micro-swimmer that moves with reciprocal periodic body-shape changes in non-Newtonian fluids, its propulsion performance depended on the fluid viscosity upon varying the shear rate.

Recently, more attention has been focus on helical magnetic micro-swimmers which can convert the rotation energy along their helical axis into the translation energy actuated by the uniform magnetic field. Generally speaking, there are two major physical mechanisms to generate a propulsion force in the small scale to overcome the drag force exerted by the surrounding liquid: "inertia propulsion" and "viscous propulsion". The inertia propulsion generates displacements of mass to exchange momentum between the swimming body and the surrounding fluid. The viscous propulsion uses viscous stresses with the surrounding fluid to generate a propulsion force. To classify these two propulsion mechanisms, the Reynolds number (Re). which stands for the ratio between the inertia force and the viscosity force, is commonly used. The Reynolds number is defined by UL/v, where U is the characteristic velocity (can be the propulsion speed or actuator speed), L the characteristic physical dimension (swimming body size or actuator size) and v the kinematic viscosity of the fluid. For the large Re, the inertia force is dominantly responsible for propulsion while for the small Re the viscous force is responsible. As the physical dimension critically affects Re, a suitable propulsion mechanism should be selected and used based on the size of the propulsion object or actuator. The locomotion of microscopic organisms, such as bacteria, has been a topic of fascination and mathematical analysis for well over sixty years.

1.1 ORIGIN OF MICRO-SWIMMERS:

Apart from seeking fundamental knowledge about how these evolutionarily simple organisms move and interact with their environment, there has recently been great interest in mimicking or using bacteria as micro-robots with biomedical and environmental monitoring applications. Of particular interest are magnetotactic bacteria, which can readily be steered by applying magnetic fields. Organisms that are not naturally magnetotactic can also be controlled with magnetic fields after incorporation of magnetic particles; this has recently been demonstrated with the alga Chlamydomonas reinhardtii. Commonly studied magnetotactic bacteria include the strains MO-1 and MC-1 (Magneto coccus marinus), which are similar in morphology and differ from the species most widely studied in other contexts, such as Escherichia coli, Bacillus subtilis, or Vibrio alginolyticus. For in vivo biomedical applications, micro-swimmers need to find an obstacle-free path and make themselves along the planned path, to accomplish the tasks of targeted therapy and targeted drug delivery. For manual operation, this would allow for considerable time and skilled operators. The manual control method is an open-loop tele-operation without any feedback. Related studies have been reported in the literature for different operation spaces, such as operation in a micro-fluidic chip in 3D space, and in vivo operation and for different driving methods, such as bacteria-driven operation and chemically driven operation. The manual control method is not appropriate for tasks that require high repetition with high precision.

1.2 MOTION OF MICRO-SWIMMER:

Micro swimming robots offer many advantages in biomedical applications, such as delivering potent drugs to specific locations in targeted tissues and organs with limited side effects, conducting surgical operations with minimal damage to healthy tissues, treatment of clogged arteries, and collecting biological samples for diagnostic purposes. Reliable navigation techniques for micro swimmers need to be developed to improve the localization of robots inside the human body in future biomedical applications. In order to estimate the dynamic trajectory of magnetically propelled micro swimmers in channels, that mimic blood vessels and other conduits, fluid-micro robot interaction and the effect of the channel wall must be understood well. In this study, swimming of one-link robots with helical tails is modeled with Stokes equations and solved numerically with the finite element method. Forces acting on the robot are set to zero to enforce the force-free swimming and obtain forward, lateral and angular velocities that satisfy the constraints. Effects of the number of helical waves, wave amplitude, relative size of the cylindrical head of micro swimmer and the radial position on angular and linear velocity vectors of micro swimmer are presented.

The hydrodynamic characteristics of micro-swimmers can be achieved by the step wise analysis in the Ansys-fluent by adding corresponding values of flow medium(blood).Blood is a body fluid in humans and other animals that delivers necessary substances such as nutrients and oxygen to the cells and transports metabolic waste products away from those cells. It has some standard values like velocity, pressure, density, etc., by providing these data, the velocity vector curves and boundary layer analysis of micro-swimmers can be developed. These swimmers actually a size of microscopic level and therefore the body may not feel the movement of this micro-swimmer in the arteries. Many of the researches showing that micro-swimmers can have the self propulsion capability when fabricated with a combination of mechanical and electrical devices to give them self actuation and then motion, but this will be very difficult to fabricate such complex structures in the micro size swimmer. So these micro-swimmers are preferable to be worked by an external agent. The most preferable media to make them move is the magnetic field. Magnetic properties can be given to the swimmer and in the presence of a magnetic field it will move. The head of micro-swimmer should be made with a magnetic material. We have different ferromagnetic materials available on our earth. We select the best material from these different materials with the in-toxic and high magnetic strength. The helical tail also has to be made with some magnetic materials. GaAs is the material mostly used to fabricate the tail of the micro-swimmer. This tail is to be attached to a soft nickel spherical body on one side, authors demonstrated the forward motion of the structure in the direction of the helical axis by applying a rotational magnetic field in that direction and pointed out that linear swimming velocity was affected not only by the size of the magnetic head, but also the strength of the applied magnetic field. Cobalt is another ferromagnetic material used for the fabrication of the micro-swimmer head deposited on one side of the micro-swimmer. Varying the magnetic field strength can give the movement to the micro-swimmer. The blood properties should not change with the magnetic material used. The research journals have been showing that these micro-swimmers first experimented in the water and then by correlating the properties of blood with water.

1.3 CLASSIFICATION OF MICRO-SWIMMERS:

There are different types of heads of micro-swimmers but all of these may not show same hydrodynamic characteristics in the fluid media. The properties may change due to the shape of head and tail as they are the only reason for the propulsion. Universally accepted models are solid spherical head and the sperm head. According to the flow and velocity of blood, we should concern the design of head. The head should be structurally tough enough and have a suitable shape for the propulsion. So we considered three types of heads for analysis and simulation. They are spherical head, sperm head and cylindrical head.

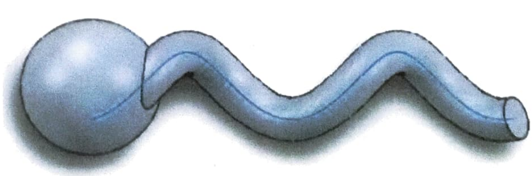


Fig 1.1 Spherical Head Microswimmer

The spherical head micro-swimmer design is not a complicated task compared to other designs. Geometry of this micro-swimmer can make ease of swimming through any fluid media. This is the foremost design used in any application. As its head is spherical, the propulsion in the fluid is very smooth. When the fluid strikes the head on the front part, boundary layer forms around the sphere and pushes the swimmer forward. Hence the swimmer moves easily with this technique.

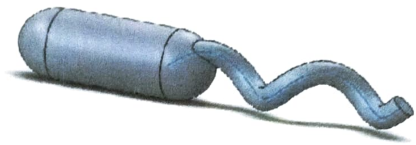


Fig1.2 Capsule Head Microswimmer

The second design has a band head with helical tail. It looks like a capsule, so named after it. It is a basic three dimensional geometric shape consisting of a cylinder with hemispherical ends. Another name for this shape is spherocylinder. This shape is usually used for containers for pressurized gases and for pharmaceutical capsules. Comparing with other designs, it has more space for accommodation of extra payload. This design also is somewhat better for swimming technique, as its ends are semispherical. Hard spherocylinders provide a good model for colloidal particles with short ranged repulsive interactions.

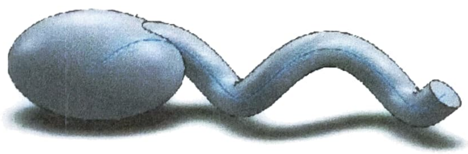


Fig1.3 Elliptical Head Microswimmer

The third design is sperm like head micro-swimmer with a helical tail. This is the most efficient design in any fluid dynamics having exceptional hydrodynamic characteristics compared to other two designs. This design used in every application like, the cockpit of an aeroplane, design of bullet shape, foremost part of a rocket. This design is also can be named as aerofoil shape as it have less drag forces compared to other designs. This is the basic ideal design taken from the sperms of living beings.

Apart from the individual characteristics, these are the mostly used designs almost in all research journals, so we considered these as the reference models for our analysis of hydrodynamic characteristics.

1.4 RELATION OF PARAMETERS:

In physics and engineering, fluid dynamics is a sub discipline of fluid mechanics that describes the flow of fluids- liquids and gases. It has several sub disciplines, including aerodynamics and hydrodynamics. Fluid dynamics has a wide range of applications, including calculating forces and moments on aircrafts, determining the mass flow rate of petroleum through pipelines, predicting weather patterns. The fluid dynamic analysis can give the information about how the hydrodynamic forces and other parameters act on the bodies when they tend to move or when placed at a position in a fluid flow. In fluid dynamics, drag (sometimes called fluid resistance) is a force, acting opposite to the relative motion of any object moving with respect to a surrounding fluid. This can exists between two fluid layers or between a solid and a fluid layer. Unlike other resistive forces, such as dry friction, which are nearly independent on velocity, drag force depends on velocity.

Drag force is proportional to the velocity for a laminar flow and the squared velocity for a turbulent flow. There are different types of drag 1.parastic drag 2.form drag 3.skin friction 4.interference drag 5.lift induced drag 6.wave drag. Drag mainly depends on the shape, size, and speed of the object.

One way to express the drag force is

$$F_D = \frac{1}{2} \rho v^2 C_D A$$

where

F - is the **drag force**,

D - is the density of the fluid

V - is the speed of the object relative to the fluid,

A - is the cross sectional area and

Symbol rho - is the drag coefficient – a dimensionless number.

The drag coefficient depends on the shape of the object and on the Reynolds number.

$$R_e = \frac{vD}{\nu},$$

Where D is some characteristic diameter or linear dimension and ν is the kinematic viscosity of the fluid. At high Reynolds number the drag cofficient is more or less constant and drag will vary as the square of the speed. These are the parameters that effect the motion of a microswimmer in the fluid medium.

CHAPTER 2

LITERATURE REVIEW

Swimming micro-robots have great potential in biomedical applications such as targeted drug delivery, medical diagnosis, and destroying blood clots in arteries. Inspired by swimming microorganisms, micro-robots can move in bio-fluids with helical tails attached to their bodies. In order to design and navigate micro-robots, hydrodynamic characteristics of the flow field must be understood well. This work presents computational fluid dynamics modeling and analysis of the flow due to the motion of micro-robots that consist of magnetic heads and helical tails inside fluid-filled channels akin to bodily conduits; special emphasis is on the effects of the radial position of the robot. Time-averaged velocities, forces, torques, and efficiency of the micro-robots placed in the channels are analyzed as functions of rotation frequency, helical pitch (wavelength) and helical radius (amplitude) of the tail. Results indicate that robots move faster and more efficiently near the wall than at the center of the channel. Forces acting on micro-robots are asymmetrical due to the chirality of the robot's tail and its motion. Moreover, robots placed near the wall have a different flow pattern around the head when compared to in-center and unbounded swimmers. According to simulation results, time-averaged forward velocity of the robot agrees well with the experimental values measured previously for a robot with almost the same dimensions.

Some of the references taken for the future study and analysis of hydrodynamic characteristics of micro-swimmers are as follows:

Gray and Hancock et al. [1] (1955) modeled swimming of a sea-urchin spermatozoa based on the fluid forces calculated by the resistive force theory, which offers a general framework for the calculation of the resultant propulsion and drag forces from the integration of local forces in normal and tangential directions that are proportional to the velocity components in those directions over the tail.

Berke et al. [2](2008) investigated hydrodynamic interactions of swimming organisms with solid surfaces by measuring the distribution of E. coli swimming between glass plates and compared their results with a hydrodynamic model.

Najafi et al. [3] proposed another model with linearly linked three spheres for a micro robot that is operated based on the periodical internal motion. According to the numerical study, this swimmer could also work in the low Reynolds number range.

Howse et al. [4] coated half of the polystyrene sphere (diameter 1.62 μm) with platinum. When this particle was placed in a hydrogen peroxide solution, it was propelled by breaking hydrogen peroxide into oxygen and water. In a short time period, the swimming was controlled by the hydrogen peroxide concentration at the speed of several μm/s. However, in a long time period, the propelling motion was random. This propulsion mechanism is not clearly understood yet as of today, though there have been two explanations introduced: the osmotic propulsion model and the bubble detachment propulsion model.

Dr. dieter breitschwerdt et al. [5] From a physical point of view, in particular the motion of microorganisms has attracted a lot of attention [12]. Their motion at small scales is mainly governed by viscous forces which dominate over inertial forces, quantified by a small Reynolds number. At low Reynolds numbers microswimmers have to perform non-reciprocal deformations of their cell body in order to swim in the absence of external forces and torques.

Bradely j.Nelson et al. [6] —Wireless magnetic micro-robots show great potential for targeted drug delivery or as minimally invasive surgical tools in the human body. In order to swim through bodily fluids, such as the vitreous humor in the eye, they must be equipped to successfully move through visco-elastic fluids, where they are obstructed by fibrous networks or microparticles. Prior researchers have shown an increased propulsion efficiency with increasing viscoelastic properties for artificial helical swimmers and bacteria with helical flagella.

Zhang et al. and Peyer et al. [7] proposed Magnetic helical microrobots, powered by external rotating magnetic fields. These microscopic helical swimmers are inspired by the corkscrew motion of bacteria flagella, such as Escherichia coli or Borrelia burgdorferi. They can perform 3-dimensional (3D) navigation in various liquids under low-strength rotating magnetic fields (<10 mT).

A. D. Gilbert etal. [8] One of the simplest magnetically based swimming devices realised in the laboratory consists of two ferromagnetic beads joined by an elastic element which could be a vesicle, polymer or protein fibre. The elastic element could also be the component to be transported in a possible medical application. Rotational motion is driven by the torque from an external field. At the same time the direction of the external field and the magnetic anisotropy of the particles influences the directions of the individual ferromagnetic or 'soft' magnets and the dipole force between them, driving a radial component of motion.

Lauga et al. [9] Many microorganisms swim close to boundaries, and as a result the effect of boundaries on fluid-based locomotion has been extensively studied. E. coli bacteria display circular trajectories near boundaries. clockwise when the wall is rigid and anti-clockwise near a free surface.

Bernard Bonnard et al. [10] Micro-swimmers were popularized by the seminal presentation and were analyzed very recently using optimization techniques in a series of articles, motivated in particular by industrial applications in robotics to design micro-robots whose control is based on the swimming mechanisms of biological micro-swimmers. In particular are cent model was proposed into analyze the observed motions of an abundant variety of zooplanktons called copepods.

Francisco Alarcon et al. [11] Micro-swimmers in 3D can generate coordinated response, such as a tendency to swim along each other, or create giant density fluctuations induced by the emergence of a percolating dynamic cluster. We have found that the key factor to produce these collective motions is the hydrodynamic signature of the micro-swimmers. Since the set-up of many experiments is a suspension where particles can move in a quasi 2D geometry, we developed a systematic numerical study such that experimental parameters are simulated.

Joost de Graaf et al. [12] Computationally, hydrodynamic aspects of micro-swimmer suspensions have been studied using a variety of fluid dynamical solvers, including Stokesian dynamics, multi particle collision dynamics (MPCD) boundary element methods and lattice-Boltzmann (LB) simulation. Treating Stokes flow has great advantages from a theoretical point of view but is often difficult to achieve in simulations. Methods such as LB and MPCD are constructed to solve the full Navier-Stokes equation, including the inertial term.

Adam Wysocki et al. [13] Both in a natural environment and in micro-fluidic devices, micro-swimmers usually do not swim in straight but rather in curved or branching micro-channels. Therefore, the influence of surface curvature on accumulation of micro-swimmers is of great interest. For example, a polar micro-swimmer close to a flat wall behaves similarly to a polar particle near a concave surface (e.g., a cavity). This is of high relevance for the design micro-swimmers with controlled wall-adhesion properties.

U Kei Cheang et al. [14] The rotating flagellum forms a helical traveling wave so it is not a reciprocal motion. Various artificial helical swimmers have achieved controlled propulsion under magnetic rotation. The archetypal flexible strategy is the sperm flagellum.

Nonreciprocal traveling waves propagate down the flexible flagellum to produce propulsion even for in-plane (hence achiral) beating patterns.

Kathrin E. Peyer et al. [15] In nature, micro-organisms have found numerous ways to propel themselves, including beating flexible flagella and cilia. Bacterial swimmers, such as the extensively researched E.coli bacterium, use a molecular motor to rotate helically shaped flagella. The continuous rotation of a helix is a non-reciprocal motion and, therefore, perfectly suited for low Reynolds number navigation.

CHAPTER 3

EXPERIMENTAL SETUP

3.1 TOOLS OR SOFTWARES USED :

- SOLID WORKS
- ANSYS-FLUENT

3.2 SOLIDWORKS :

We used solid works software for the modeling procedure, that is of 2016 version. This version is released on October 1st 2015 and developed by Dassault systems. Solid works is a solid modeling computer aided design (CAD) and computer aided engineering (CAE) computer program that runs on Microsoft windows. However our model is very complicate in design, but solid works done that easily as it is an user friendly software and utilizes a parametric feature-based approach. Parametric refers to constraints whose values determine the shape or geometry of the model or assembly. Design intent is how the creator of the part wants it to respond to the changes and updates. Features refer to the building blocks of the part, are the shapes and operations that construct the part. Building a model in solid works usually starts with a 2D sketch. Solid works files use the Microsoft structured storage file format.

3.2.1 TERMINOLOGY

These terms appear throughout the SOLIDWORKS software and documentation.

ORIGIN:

Appears as two blue arrows and represents the (0,0,0) coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the (0,0,0) coordinate of the sketch. You can add dimensions and relations to a model origin, but not to a sketch origin

PLANE:

Flat construction geometry. You can use planes for adding a 2D sketch,section view of a model, or a neutral plane in a draft feature, for example.

AXIS:

Straight line used to create model geometry, features, or patterns. Youcan create an axis in different ways, including intersecting two planes. TheSOLIDWORKS application creates temporary axes implicitly for every conicalor cylindrical face in a model.

FACE:

Boundaries that help define the shape of a model or a surface. A face is aselectable area (planar or non-planar) of a model or surface. For example,a rectangular solid has six faces.

EDGE:

Location where two or more faces intersect and are joined together. Youcan select edges for sketching and dimensioning, for example.

VERTEX:

Point at which two or more lines or edges intersect. You can select verticesfor sketching and dimensioning, for example.

3.2.2 USER INTERFACE

The SOLIDWORKS application includes user interface tools and capabilities to help you create and edit models efficiently, including:

WINDOWS FUNCTIONS

The SOLIDWORKS application includes familiar Windows functions, such as dragging and resizing windows. Many of the same icons, such as print, open, save, cut, and paste are also part of the SOLIDWORKS application.

SOLIDWORKS DOCUMENT WINDOWS

SOLIDWORKS document windows have two panels. The left panel, or Manager Pane, contains:

1.FEATURE MANAGER DESIGN TREE :

Displays the structure of the part, assembly, or drawing. Select an item from the feature manager design tree to edit the underlying sketch, edit the feature, and suppress and unsuppress the feature or component, for example.

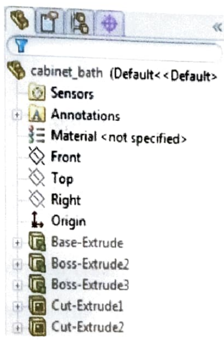


Fig 3.1 Feature Manager Design Tree

2. PROPERTY MANAGER :

This provides settings for many functions such as sketches, fillet features, and assembly mates.

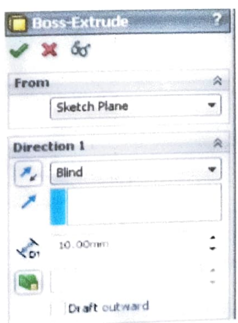


Fig 3.2 Property Manager

3. CONFIGURATION MANAGER

Lets you create, select, and view multiple configurations of parts and assemblies in a document. Configurations are variations of a part or assembly within a single document. For example, you can use configurations of a bolt to specify different lengths and diameters.

Fig3.3 Configuration Manager

3.2.3 FEATURES :

Once you complete the sketch, you can create a 3D model using features such as an extrude (the base of the faucet) or a revolve (the faucet handle). Some sketch-based features are shapes such as bosses, cuts, and holes. Other sketch-based features such as lofts and sweeps use a profile along a path. Another type of feature is called an applied feature, which does not require a sketch. Applied features include fillets, chamfers, or shells. They are called "applied" because they are applied to existing geometry using dimensions and other characteristics to create the feature. Typically, you create parts by including sketch-based features such as bosses and holes. Then you add applied features.

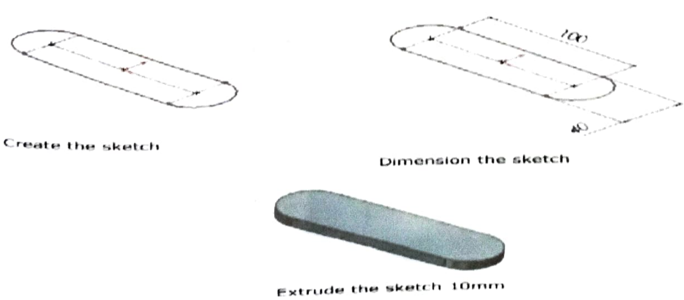


Fig 3.4 Extrude Feature

3.2.4 ASSEMBLIES :

You can combine multiple parts that fit together to create assemblies. You integrate the parts in an assembly using Mates, such as Concentric and Coincident. Mates define the allowable direction of movement of the components. In the faucet assembly, the faucet base and handles have concentric and coincident mates. With tools such as Move Component or Rotate Component, you can see how the parts in an assembly function in a 3D context. To ensure that the assembly functions correctly, you can use assembly tools such as Collision Detection. Collision Detection lets you find collisions with other components when moving or rotating a component.

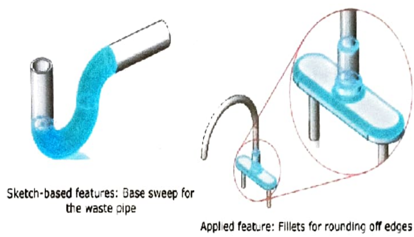


Fig 3.5 Assembly of parts

3.2.5 DRAWINGS :

You create drawings from part or assembly models. Drawings are available in multiple views such as standard 3 views and isometric views (3D). You can import the dimensions from the model document and add annotations such as datum target symbols

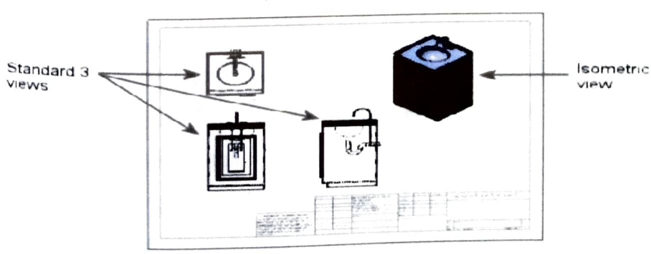


Fig3.6 2-D Drawing Sheet

Fig 3.8 Concentric Mate

3.3 ANSYS:

Ansys-Fluent is used for the simulation of the model designed in the solid works. Fluent software contains the broad, physical modeling capabilities needed to model flow, turbulence, heat transfer and reactions for industrial applications. These range from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing and from clean room design to wastewater treatment plants. Fluent spans an expansive range, including special models, with capabilities to model in-cylinder combustion, aero-acoustics, turbo machinery and multiphase systems.

Fluent also offers highly scalable, high performance computing (HPC) to help solve complex, large-model computational fluid dynamics (CFD) simulations quickly and cost-effectively. Fluent already solves the toughest design challenges with well-validated results across the widest range of CFD and multi-physics applications. But today, engineers need to accomplish more, in less time and with less training than ever before introducing the new way to enjoy CFD simulations.

3.4 ANSYS WORKBENCH:

The ANSYS Workbench interface consists primarily of a toolbox region, theProject schematic, the toolbar, and the Menu bar. Depending on the analysis type and/or application or workspace, you may also other windows, tables, charts, etc. One way to work in ANSYS workbench is to drag an item such as a component or analysis system from the toolbox to the project schematic or to double-click on an item to initiate the default action. You can also use the context menus, accessible from a right mouse click, for additional options. You will view your analysis systems in the components that make up your analysis in the project schematic, including all connection and links between the systems. The individual applications in which your work will display separately from the ANSYS Workbench GUI, but the results of the actions you take in the application may be reflected in the project schematic. The ANSYS Workbench interface is arranged into two primary areas: the toolbox and the project schematic. The toolbox contains the system templates where you will manage your project. In addition, you will see a toolbar and a menu bar with frequently used functions. To start a new analysis, select a system from the Toolbox and double-click or drag it onto the Project Schematic. An analysis system block appears in the Project Schematic. It will contain the components, called cells, required to complete the analysis, as well as optional components such as Resources. Items that must be completed before you can continue are indicated with an Attention required state indicator.

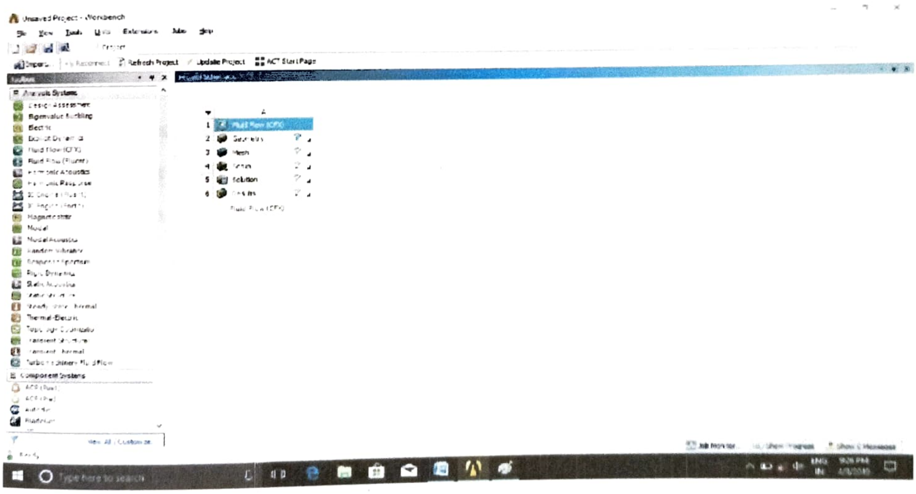


Fig 3.9 ANSYS Workbench

After you add an analysis system, you can load an existing geometry by right-clicking on the Geometry cell of the analysis system and choosing Import Geometry from the context menu. After you import geometry, you can edit it in Design Modeler by selecting Edit from the context menu. Once your geometry is complete to your satisfaction, right-click on the Setup cell and select Edit. Your model will open the appropriate application (CFX, ANSYS, FLUENT, etc.). From there, you can set up your model and solve your analysis using the tools and features of that application. To open Design Modeler and create a new geometry, right click on the Geometry cell and choose New Geometry. You can then build your new geometry in the Design Modeler window that opens. Alternatively, you can add a Design Modeler system to the Project Schematic. The use of Finite Element Analysis for the structural engineering is an important advantage. The design of the structural not only benefits, but may require the use of this advanced analysis approach. The nature of structural components involves several concerns and requirements. Safety, reliability, strength, stiffness, and low cost all come into the picture. Coupled with this, are the additional design requirements of interference, manufacturability, and over all function. By using finite element modeling as a primary analysis tool, the constraints of creativity are removed. It provides a powerful procedure to mathematically model physical phenomenon

3.5 THE BASIC PROCEDURE IN ANSYS WORK BENCH

1. Select your desired analysis system from the Toolbox (at left), drag it into the Project Schematic (at right), and drop it inside the highlighted rectangle.
2. Right-click on the Geometry cell to create a new geometry or import existing geometry.
3. Continue working through the system from top to bottom. Right-click and selected ton a cell to start the appropriate application and define the details for that part of the analysis.

As you complete each task, a green check mark appears in the cell, indicating that you can proceed to the next cell. ANSYS Workbench automatically transfers your data between cells. When you select Save (either from the ANSYS Workbench window Orin an application), the entire project is saved. You can connect systems to build more complex projects.

Access Engineering In the Data.	1. Double-click the Engineering Data cell or right-click the Engineering Data cell and select **Edit...**

Import data into a Data source.	1. Access Engineering Data. 2. Select a data source in the **Outline Filter** pane. 3. Check the **Edit library** box to the right of the data source Title. 4. Choose **File> Import Engineering Data....** 5. Select a file and choose **Open.** **Note:** Only recognized data will be imported into the data Source.
Export a Data Source	1. Access Engineering Data. 2. Select a data source in the Outline Filter pane. 3. Choose File> Export Engineering Data.... 4.In the Save As dialog, select the folder, provide a Filename, and choose Save.
Export Individual Data	1. Access Engineering Data. 2. Select a data source in the Outline Filter pane. 3. Select one or more items in the Outline pane. 4. Choose File> Export Engineering Data.... 5. In the Save As dialog, select the folder, provide a Filename and choose Save.

3.6 FLUENT:

FLUENT is a commercially sold CFD program that is widely accepted for its high fidelity. The program has many different capabilities and functions. Both 2-dimensional and 3-dimensional cases can be tested with laminar, inviscid, or turbulent flow. Depending on the model and flow type, the designer chooses which flow field type is best. For large models, over 200,000 cells and a coupled solver, a standard PC with single memory cannot process the model. Super computers with multiple processors must be used to solve these larger models. The following are the steps followed to run a FLUENT model.

Step 1: Create a CATIA model and import it in ICEM CFD. Create a domain around the model and do the structured meshing of the same. It is meshed with medium sized elements as the size of the mesh will dramatically affect the accuracy of the results and the processing time.

Step 2: Export the mesh into FLUENT.

Step 3: With FLUENT opened, select either 2-D or 3-D, depending on the mesh created in CATIA. There is also 2ddp and 3ddp. The "dp" stands for double precision, and is more accurate than just the standard 2d or 3d. Case run time is larger for this method, but the accuracy of the results is important to the validity of the model.

Step 4: Import the mesh and run a check to see if there are errors in the mesh. The size of the mesh can also be seen to verify that there are not too many cells to run the test on a standard PC.

Step 5: Choose the type of solver: segregated or coupled. Coupled solver requires much more computer memory. The solver is the method that the program uses to solve the equations.

Segregated solver- solves the equations one by one e.g. equation of momentum, continuity, energy (if compressible flow) and turbulent factor. Then check for convergence.

Coupled solver- solves the same equations as the segregated solver, but does this simultaneously instead of one by one.

Step 6: Choose the type of flow: laminar, invisid, or turbulent. If choosing turbulent flow select the turbulent model to use. It is difficult to determine the most accurate turbulent model without testing several different ones. The accuracy depends on the mesh shape and flow pattern.

Step 7: Set the fluid properties and boundary conditions. The fluid properties include but are not limited to density, viscosity, velocity, wall friction, and much more.

Step 8: Select the solution controls and discretization methods. This depends on the shape of the body and type of flow. A description of each solution and discretization method should be read before the designer chooses which one is best for their model. The FLUENT online help menu should be used for these questions.

- SIMPLE and SIMPLEC are good choices for non-complicated flow problems (such as laminar flow).
- PISO is used for transient flows.

Step 9: Set the residuals and turn on what parameters are to be monitored. For example, the lift and drag of the body can be monitored.

Step 10: Iterate until the solution converges. The convergence criterion is determined by setting the residuals.

These steps can slightly vary from mesh to mesh, but in general these ten steps allow the user to run a FLUENT model.

CHAPTER 4

DESIGN PROCEDURE AND ANALYSIS

4.1 DESIGN IN SOLIDWORKS:

The design of all microswimmers is done by using the SolidWorks software. Here we have three different geometric models that have to be designed in the SolidWorks.

4.1.1 Design of heads:

Design of spherical head:

To create a solid sphere in SolidWorks, use the Revolve Boss/Base feature. The steps are described below

Create a new sketch.

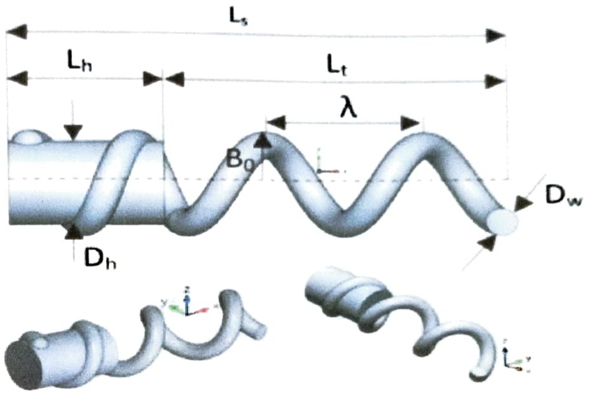
1. Draw a circle with a line intersecting it directly through the centre point with the required radius of
2. Trim one side of the circle away, leaving the central sketch line as solid.
3. Create a Revolve Boss/Base.
4. Direction Angle should be set to 360°.
5. Accept the feature.

Similarly the other two heads also can be drawn using the standard values from the research journals. The standard values for a cylindrical head is given by one of the research fellows. They are as follows:

Table 1. Geometric parameters of models

Symbol	Description	Values(μm)	Non-Dimensional values
D_h	Diameter of the cylindrical head	400	1
L_h	Length of the cylindrical head	600	1.500
Λ	Wave length of the tail	625	1.5625
B_o	Wave amplitude	200	0.5
L_t	Length of the tail	1250	3.125
D_w	Wire diameter	130	0.325
L_s	Total length of the micro swimmer	1850	4.625
D_{ch}	Length of the channel	1000	2.5
N_λ	Number of waves	2	2

Here the values are for a cylindrical head, by taking the diameter of the cylinder we designed the geometry of sphere. The dimensions are in micro order because of such micro design. These values are taken from the reference paper where they experimented the desired design practically in fluid medium



4.1 Cylindrical head with a helical tail

4.1.2 Design of helical tail :

After completing design of all the three heads, we have to design a helical shaped tail with one end attached to the head of the micro-swimmer. For the design of that helical tail the following steps should be followed.

You can create a helix or spiral curve in a part.

1. In a part, do one of the following:

   a. Open a sketch and sketch a circle.

   b. Select a sketch that contains a circle.

   The diameter of the circle controls the starting diameter of the helix or spiral.

2. Click Helix and Spiral (Curves toolbar) or Insert > Curve > Helix/Spiral .
3. Set values in the Helix/Spiral PropertyManager.
4. Click .

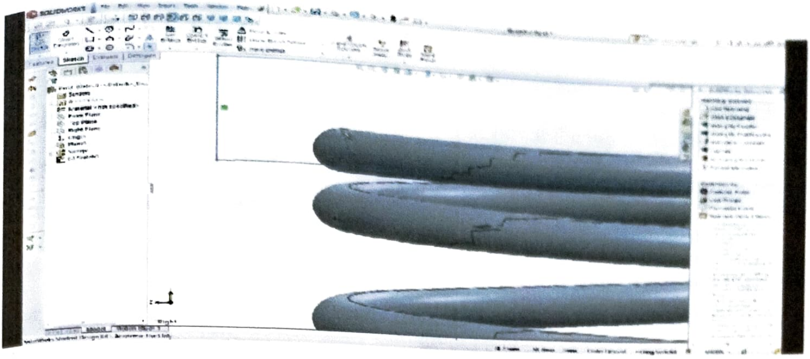


Fig 4.2 Spiral Curve

This helical tail has to be attached to those three different heads by using the assembly option. The following are the steps to assemble the helical tail and the head

1. click on the assembly option.
2. Then go to the mate command and give the required surfaces to be mate.
3. Then adjust the position by giving alignment option.

After assembling the tail to these heads

Spherical Head

Capsule Head

Elliptical Head

4.1.3 Design of domain:

Then we have to create a domain to send these bodies into it. As we are analyzing the hydrodynamic characteristics of bodies when flow through the blood, we must consider the domain into which the body has to be send. This domain may takes the geometry of the largest of the arteries. Its name is Aorta. The domain is also designed by using the solidworks software. The design dimensions of the domain are

Ascending aorta- length -160mm

diameter- 2.1cm/m^2

Descending aorta- length-290mm

diameter- 1.6cm/^2

The aorta is the largest artery in the body. The aorta begins at the top of the left ventricle. the heart's muscular pumping chamber. The heart pumps blood from the left ventricle into the aorta through the aorticvalve.

One way of classifying a part of the aorta is by anatomical compartment. where the thoracic aorta (or thoracic portion of the aorta) runs from the heart to the diaphragm. The aorta then continues downward as the abdominal aorta (or abdominal portion of the aorta) diaphragm to the aortic bifurcation.

Another system divides the aorta with respect to its course and the direction of blood flow. In this system, the aorta starts as the ascending aorta then travels superiorly from the heart and then makes a hairpin turn known as the aortic arch. Following the aortic arch, the aorta then travels inferiorly as the descending aorta. The descending aorta has two parts. The aorta begins to descend in the thoracic cavity, and consequently is known as the thoracic aorta. After the aorta passes through the diaphragm, it is known as the abdominal aorta. The aorta ends by dividing into two major blood vessels, the common iliac arteries and a smaller midline vessel, the median sacral artery.

Ascending aorta:

The ascending aorta begins at the opening of the aortic valve in the left ventricle of the heart. It runs through a common pericardial sheath with the pulmonary trunk. These two blood vessels twist around each other, causing the aorta to start out posterior to the pulmonary trunk, but end by twisting to its right and anterior side. The transition from ascending aorta to aortic arch is at the pericardial reflection on the aorta.

Descending aorta :

The descending aorta is part of the aorta, the largest artery in the body. The descending aorta begins at the aortic arch and runs down through the chest and abdomen. The descending aorta anatomically consists of two portions or segments, the thoracic and the abdominal aorta, in correspondence with the two great cavities of the trunk in which it is situated. Within the abdomen, the descending aorta branches into the two common iliac arteries which serve the pelvis and eventually legs.

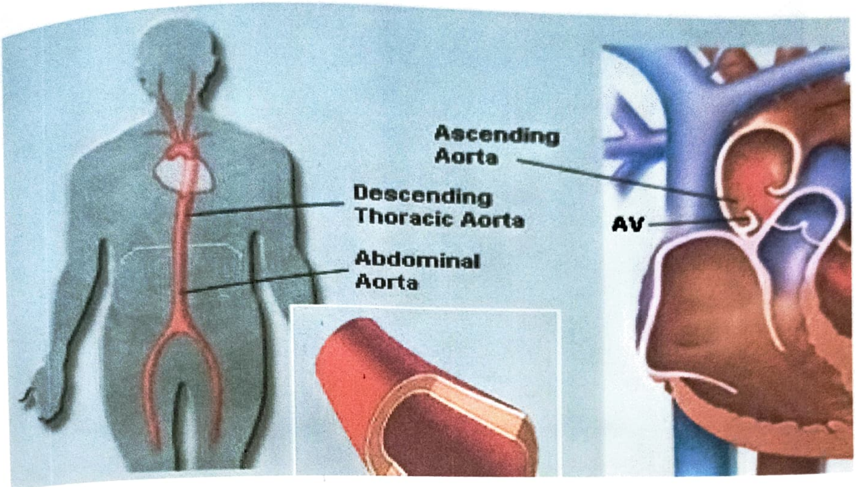


Fig 4.3 Aorta

Assembly of domain and microswimmer:

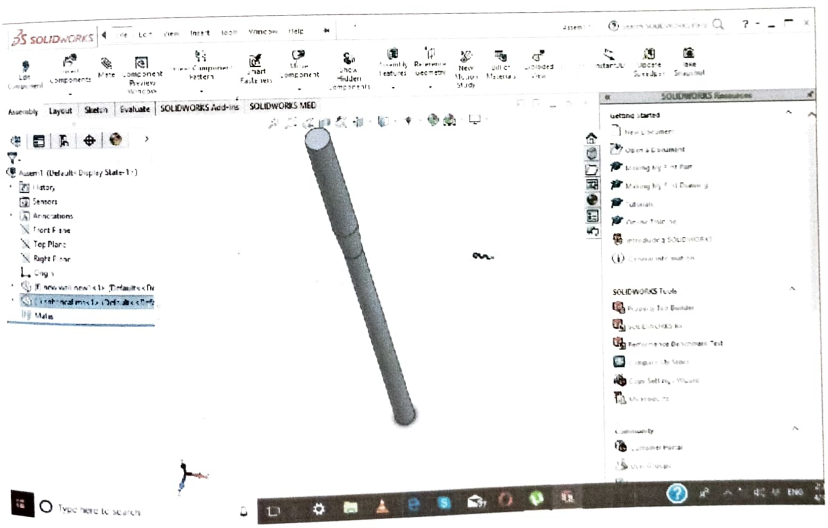


Fig 4.4 Assembly of domain and microswimmer

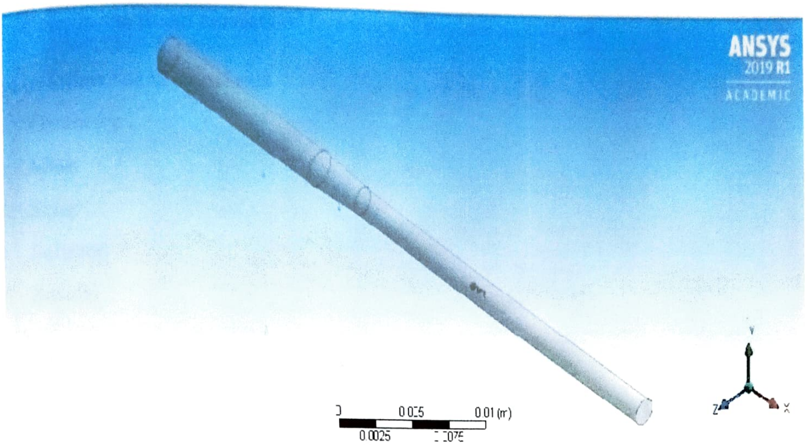


Fig 4.5 Domain with the spherical head microswimmer

The assembly that obtained is to be in IGS file format so as to move or can import for further analysis procedure.

4.2 Analysis in ANSYS FLUENT:

Before starting the analysis in the AnsysWorkbench we have to select the fluent fluid flow from the analysis systems on the left side of the workbench.

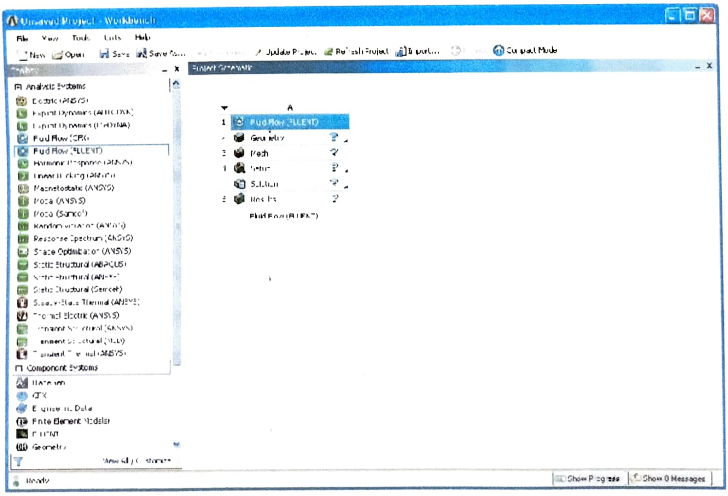


Fig 4.6 Ansys fluent

The analysis in the fluent consists of five steps. They are as follows

Steps to follow for the analysis

1. Geometry
2. Mesh
3. Setup
4. Solution
5. Results

GEOMETRY

1. In this, the IGS file created in the solid works should to be imported here.
2. Now we have to perform Boolean operations to edit the imported geometry.

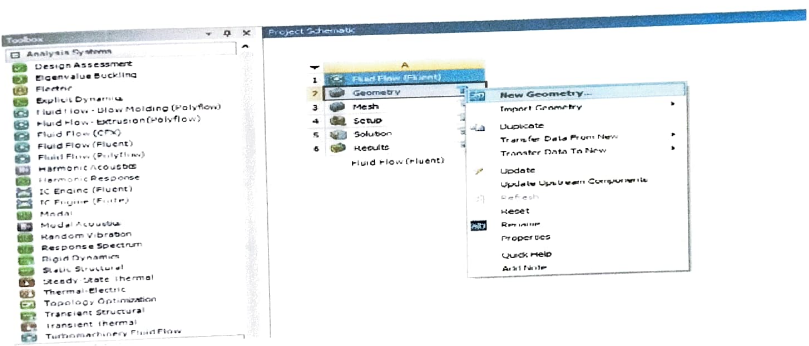


Fig 4.7 Geometry view

3. After performing the Boolean operations, we should update the file to ensure that all the corresponding parameters are fixed as per the requirement.

MESH

1. In this, the updated file is used for meshing.
2. First, we should create names for the design updated. The names can be given by the left clicking the options and select the named selection option.
3. After giving the names we should select interfaces to perform the meshing.

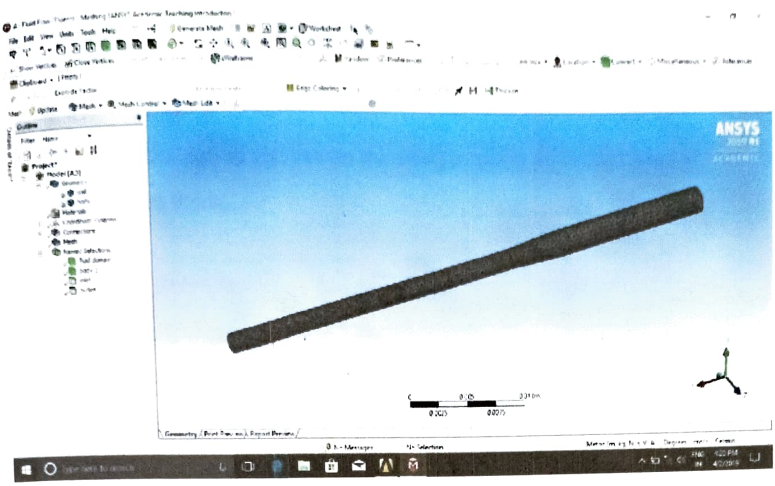


Fig 4.8 Meshing view

4. Then the design is meshed and then click on update option.

SETUP:

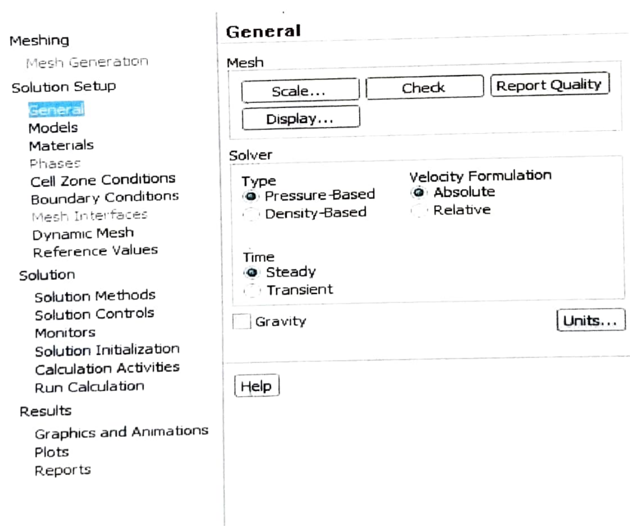


Fig 4.9 Setup view

Models:

1. The models, materials, cell zone conditions, and the boundary conditions are given in this option
2. Select models option and put on the energy option and select viscous, laminar, and k-epsilon from them.

Materials:

1. The materials should be given here.
2. Here we taken blood as the fluid and solid iron oxide as body.
3. Blood properties are given as follows.

Table 2

PROPERTIES	STANDARD VALUES
Density	1060 Kg/cubic-metre
Specific gravity	3513 J/Kg.K
Thermal conductivity	0.52 W/m.K
Viscosity	0.003 Kg/m-s

Cell zone conditions:

After inserting the materials, the body has given iron oxide element based on literature survey and domain is filled with blood and have given the standard values for the blood from the references.

Boundary conditions:

In this, the inlet velocity and the outlet pressure are given to the domain.

Inlet velocity- 0.66 m/sec

Outlet pressure- 11000 Pa

SOLUTION:

Solution methods:

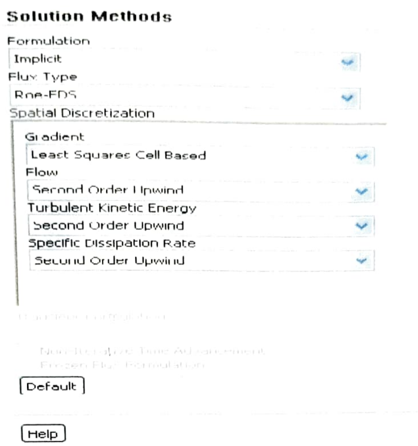


Fig 4.10 Solution Method View

Solution initialization:

- hybrid initialization
- standard initialization

ANSYS Fluent will compute and update the Initial Values based on the conditions defined at all boundary zones. If you want to change one or more of the values, you can enter new values manually in the fields next to the appropriate variables.

Run calculations:

Here we have given 100 iterations to calculate the convergence of the solution, but we got the convergent solution at 50 iterations.

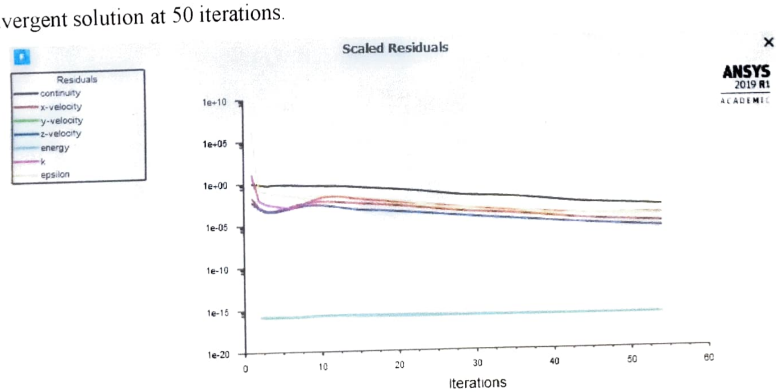


Fig 4.11 iterations view

PLOT:

By taking velocity on y-axis and position which is default on x-axis, we generated the graph.

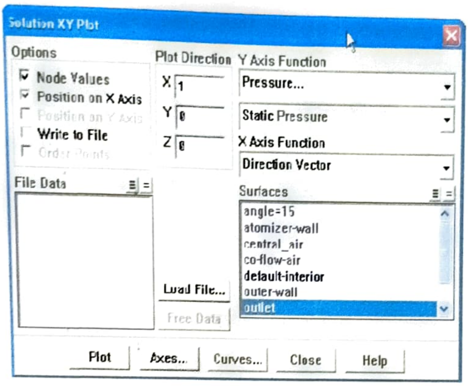


Fig4.12 Plot view

CHAPTER 5

RESULTS & DISCUSSIONS

From this analysis, we plotted the graphs of velocity and positions. As we study different head microswimmers, the trends of velocity magnitude differs a lot. Blood is the fluid used for the analysis of these variations as our main concern is all about the Biomedical applications like drug delivery.

Here we have a domain and blood is flowing in it and the micro-swimmer moves against the flow. At that instance the blood try to push the swimmer along with its flow. There creates some Reaction force. Due to this the velocity differ at each position of the body in the domain. The graphs are plotted, taking position on x-axis and velocity magnitude on y axis. Various graphs are shown below.

Spherical type head with micro-swimmer:

1. **Streamlines :**

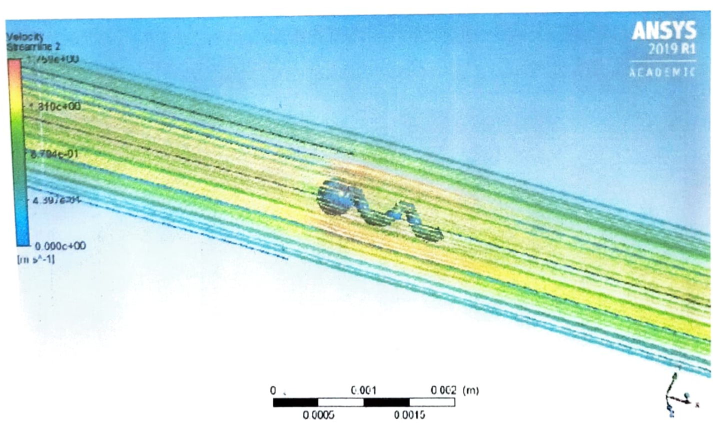


Fig 5.1 streamlines

2. Vector :

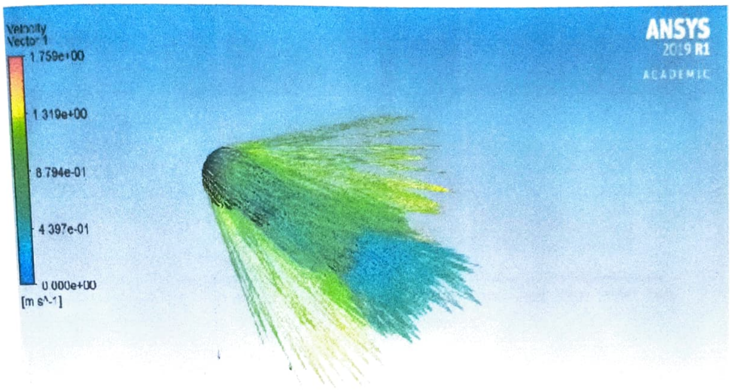


Fig 5.2 velocity vector

3. Contour :

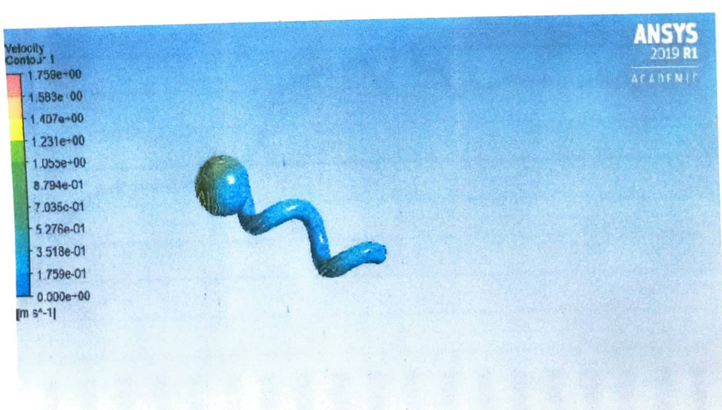


Fig 5.3 velocity contour

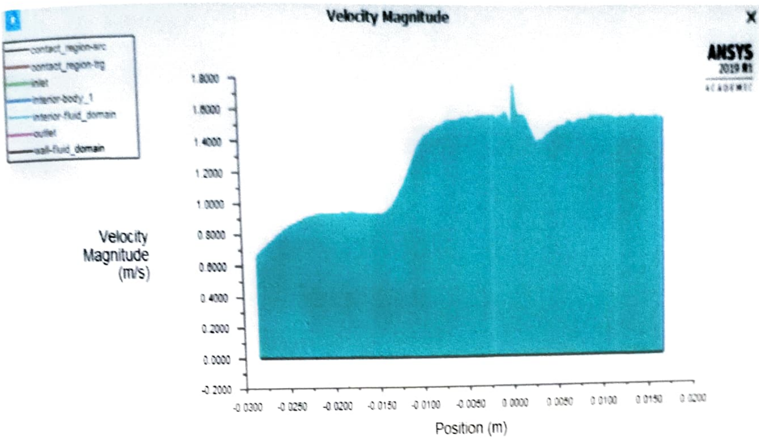
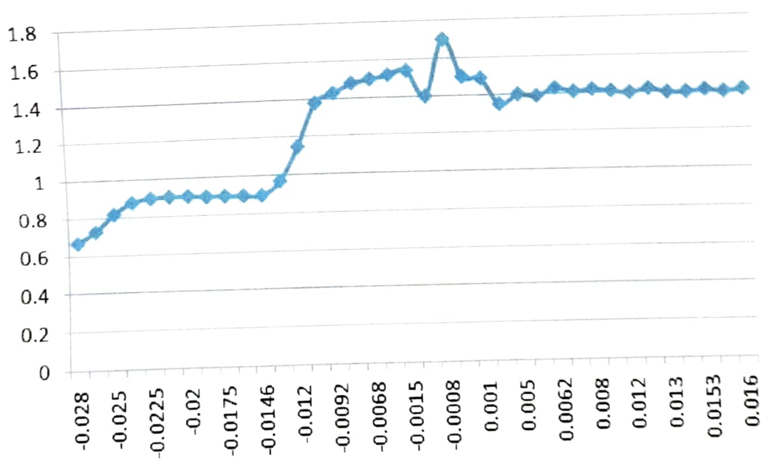


Fig5.4 x-y plot in Ansys

4.Graph:

X -axis – position (m)

Y -axis – velocity magnitude (m/s)



Capsule type head with microswimmer:

1.Streamlines :

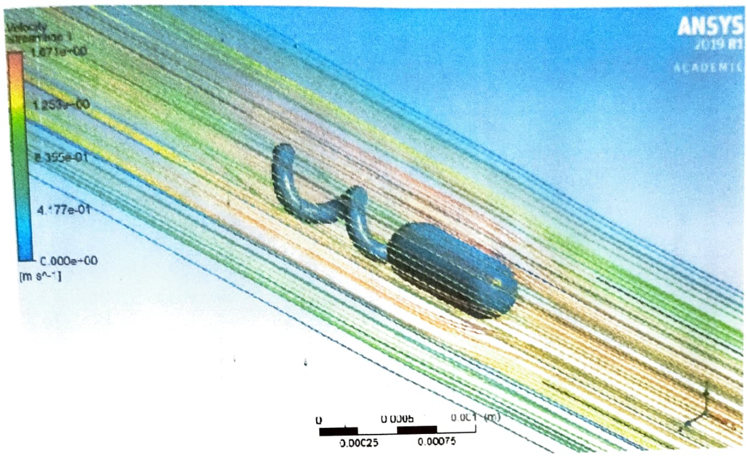


Fig 5.4 stream lines

2.Vector :

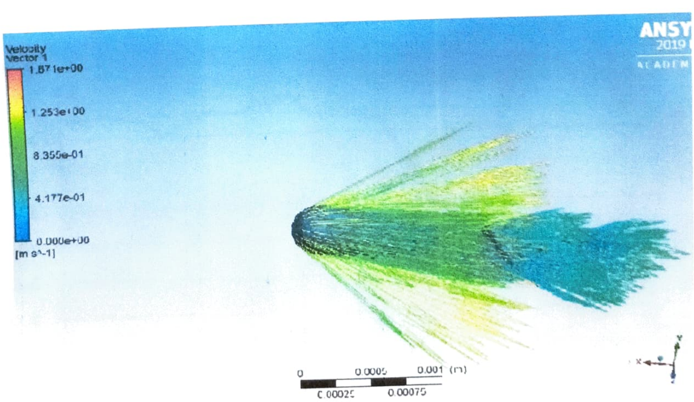


Fig 5.5 velocity vector

3.Contour :

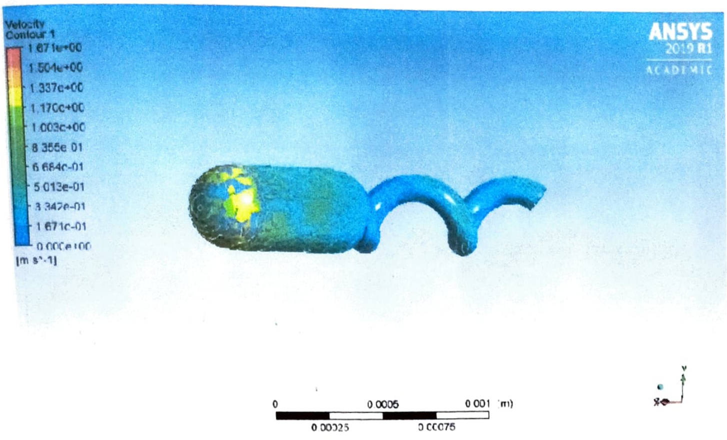
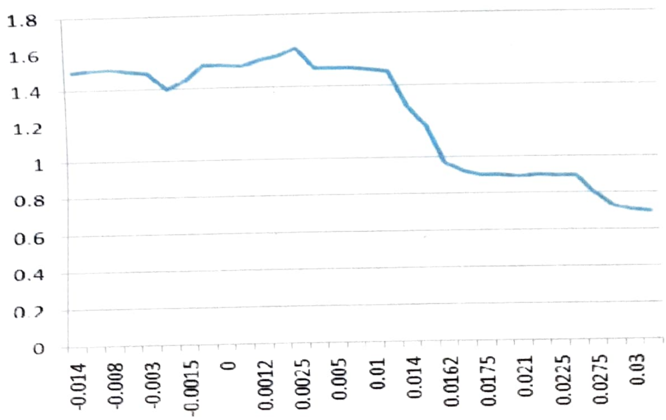


Fig 5.6 velocity contour

4.Graph :

X -axis – position (m)

Y -axis – velocity magnitude (m/s)



Ellipse type head with microswimmer:

1.Streamlines:

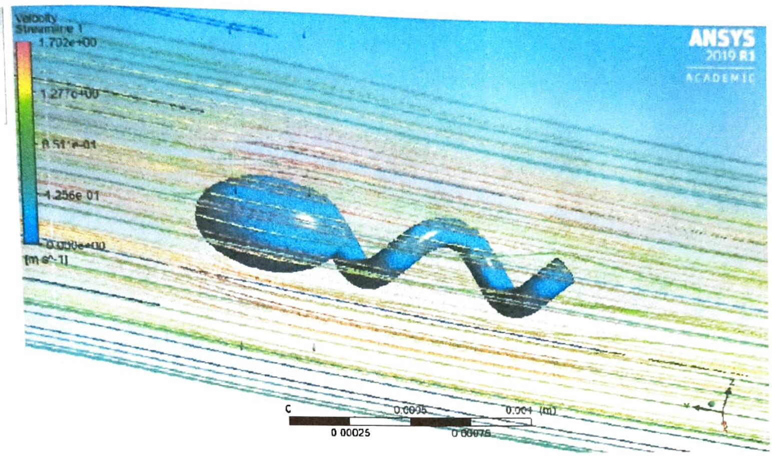


Fig 5.7 streamlines

2. Vector:

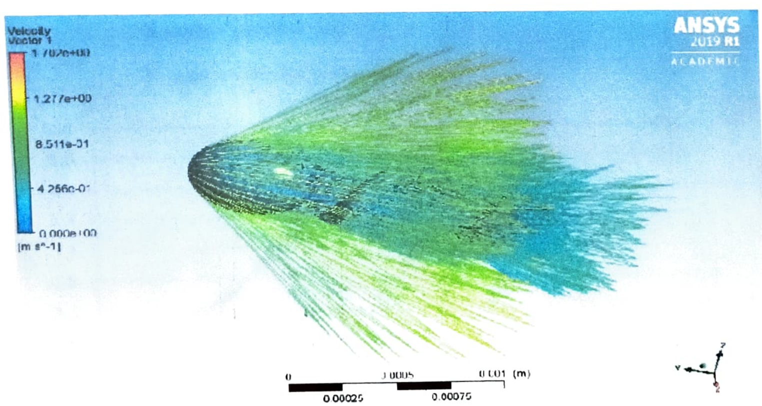


Fig 5.8 velocity vector

3.Contour:

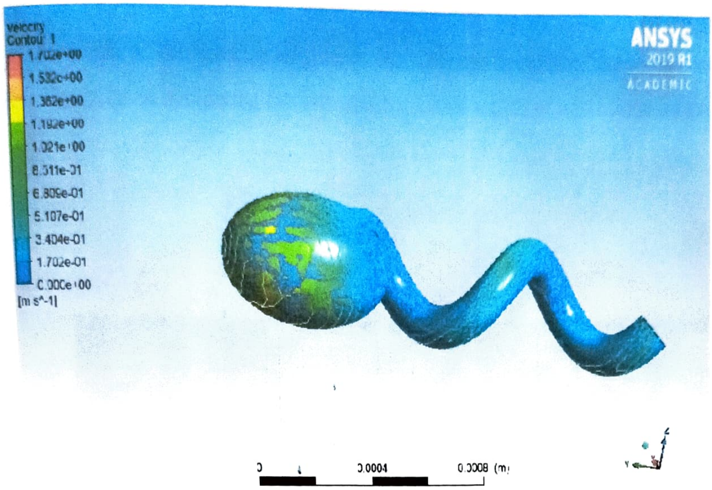
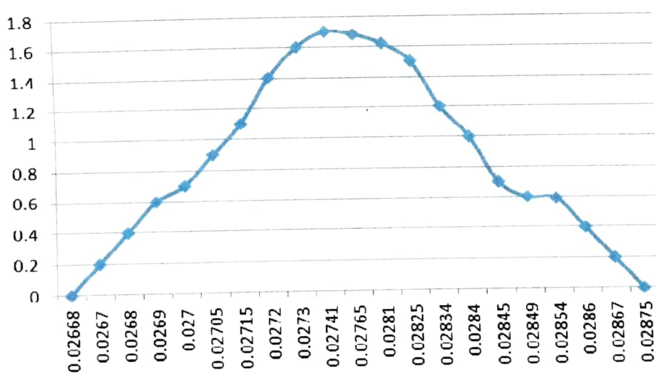


Fig 5.9 velocity contour

4. Graph :

X -axis – position (m)

Y -axis – velocity magnitude (m/s)



Wall or Domain

There should be a reference to compare among three models. So we taken an empty domain with blood flowing in it. The velocity magnitude may changes because of the cross-section of the domain. This helps in comparing the velocity magnitude trends of other three models.

1.Streamlines:

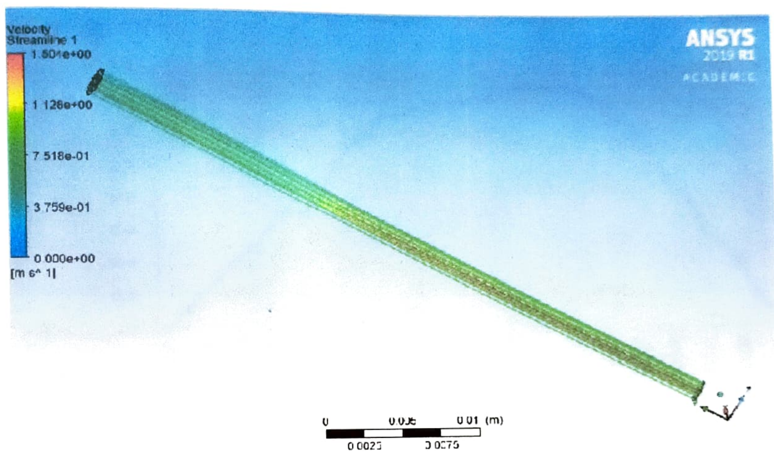


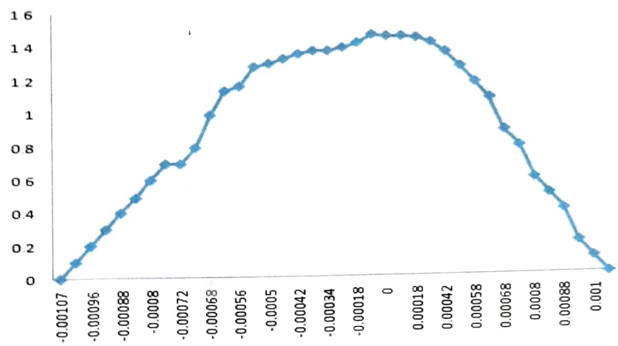
Fig 5.10 streamlines

The streamlines are shown above, as there is no body in the flow, the streamlines are following a straight path. This would be the optimized condition for the blood velocity.

2.Graph:

X-axis- position (m)

Y-axis- velocity magnitude (m/s)



The above graphs showing the streamlines, velocity vectors and velocity gradients of three different heads. But we don't know whether which model is best for the application,compared to others. For this, we have taken domain with no body. Then we compared these reference graphs with the graphs of three models. The graphs of elliptical head are somewhat close to the reference graphs. So we conclude that it would be the best.

CHAPTER 6

CONCLUSIONS

1. The considered three models of micro-swimmers were taken from the standard journals and designed and developed in SOLIDWORKS. The head and tails are created as a single component based on the required dimensions. Then the part file is converted into IGS file to open the file in ANSYS software.
2. Analysis and flow simulation is done in ANSYS FLUENT under necessary conditions, by creating a domain for the micro-swimmer. The graphs are presented between velocity and positioning of micro-swimmer.
3. Then these graphs are considered as results, whose are the reactions between the blood and micro-swimmer.
4. When the blood is continuously flowing in the aorta, the swimmer will be injected into it. The swimmer initially is in rest before the activation of its mechanism for few milliseconds. In this small time lapse, the blood flow strikes the swimmer head. In this instance, the velocity magnitude with respect to position are plotted.
5. Finally we got the comparision between the three different models on the basis of their velocity variations. The elliptical head type microswimmer is some what having close values with the reference values of velocity. This would be the best among three models.

CHAPTER 7
REFERENCES

[1] Gray and Hancock et al. "The propulsion by large amplitude waves of uniflagellar micro-organisms of finite length"

[2] Berke AP, Turner L, Berg HC, Lauga E. "Hydrodynamic attraction of swimming microorganisms by surfaces".

[3] Najafi et al. " Mini and Micro Propulsion for Medical Swimmers".

[4] Jonathan R. Howse. "Self-motile colloidal particles: from directed propulsion to random walk".

[5] Martel S., Mohammadi M., Felfoul O., Lu Z., and Pouponneau P., "Flagellated magnetotactic bacteria as controlled MRI-trackable propulsion and steering systems for medical nanorobots operating in the human microvasculature", Int. J. Robot. Res., vol. 28, pp. 571–582, 2009

[6]Bradely j.Nelson et al. Wireless Intraocular Microrobots: Opportunities and Challenges

[7]Li Zhang,1 Jake J. Abbott. "Artificial bacterial flagella: Fabrication and magnetic control".

[8] Andrew D. Gilbert "Theory of ferromagnetic materials".

[9] Eric Lauga, Willow R. DiLuzio. "Swimming in Circles: Motion of Bacteria near Solid Boundaries".

[10]Bernard Bonnard et al. "Sub-Riemannian geometry, Hamiltonian dynamics, micro-swimmers, copepod nauplii and copepod robot".

[11] Francisco Alarc´on and Ignacio Pagonabarraga. "Micro-swimmer suspensions using high performance computing"..

[12] Joost de Graaf et al. " Lattice-Boltzmann Simulations of Microswimmer-Tracer Interactions".

[13] Adam Wysocki et al. " Dynamics of swimming bacteria at complex interfaces".

[14] Hancock, G.J., "The self-propulsion of microscopic organisms through liquids", Proceedings of the Royal Society of London, Series A, Vol. 217, No. 1128, pp. 96-121, 1953.

[15] Kathrin E. Peyer et al. " Microbiorobotics: Biologically Inspired Microscale Robotic Systems".