

CHAPTER I

Introduction

1.1 Background

With the rapid advance of underwater robotics technology in the past years, underwater robots have and continues to receive a great deal of attention for application domain (such as marine exploration, structural inspection, environment monitoring, academic research etc.). Oceans, rivers and even man made bodies of water present challenging conditions to examine without special technology underwater robots provide researchers with helpful equipment to observe evaluate and perform tasks beneath the sea surface without depending on divers or expensive machinery. Screw propelled Remotely Operated Vehicle (Wang et al., 2024) or Autonomous Underwater Vehicle (AUVs) have been traditionally employed. Although these systems are successful, they are expensive, complicated and not suitable for small scale research or educational activities. This has led to an investigation of alternative designs, inspired by biology such as robots that mimic the swimming of fish.

These fish-inspired robots, also referred to as biomimetic robots replicate the anatomy and movements of real fish. (Mitin et al., n.d.) These robots can navigate smoothly and fluidly in three dimensions on account of red muscles, tendons and vertebrae. These allow them to be more involved physically with underwater environments and even with aquatic life in its natural environment. On the other hand, stability throughout maneuvering is a major challenge for underwater robotics (Berg et al., 2022). A robot platform feels forces like buoyancy drag and propeller thrust when submerged. If it is not balanced, the robot could also tip over on (accidentally. Effective control of these systems is the need of an hour. It is well-recognized that proportion-integration-differentiation (PID) control has the ability and experience in stabilizing movement through continuously eliminating the error between its desired and existing action (Du et al., 2021).

Fish-type robots, for example red fish muscles, tendons and vertebrae can often be imitated in the anatomical structure and swimming motion of real fish by so-called biomimetic robots (take) a co2py mode, moving smooth progress in three dimensions (Du et al., 2021). However, stability during operation is yet another critical challenge in underwater robotics especially when a robotic platform moves under water it experiences forces such as buoyancy drag and propeller thrust which can make the robot to roll, yaw or deviate from its planned path if the thrust is not well distributed. To deal with this, effective control strategies need to be implemented, the PID controller is for example famous for its ability to stabilize a motion by correcting error between desired and actual position. (Mitin et al., n.d.)

Besides physical design and control, computer-based tools are vital for the performance optimization of underwater robots. For instance, through the use of Computational Fluid Dynamics (CFD) (Du et al., 2021) simulations robot designers can look at water flow and drag forces as well as pressure distribution over the body (Wright et al., n.d.). This facilitates iterative design before fabrication and eliminates trial-and-error in experiments. By exploiting PID thrust control in combination with CFD-based investigation, low-cost and stable UA. Thus, the present work aims at obtaining a prototype which can provide with some controllability inspired on both classical control theory and on achievements from biomimetic design practice ("CFD-Based Hydrodynamic Analysis of Bionic Fish," 2023). Through combining stability-oriented control and simulation-supported analysis, the project seeks to enable a low-cost robotic platform that is suitable for underwater exploration and research.

However underwater robotics continues to develop in straight advancements in materials and electronics. The use of lightweight and strong materials such as PETG for 3D printing open the fabrication of customized robotic shapes at low cost (Berg et al., 2022). This makes experiment easier to students and researchers who may not have access to expensive industrial equipment. In addition the integration of controllers such as Arduino or ESP32 allows easy programming of motor control systems, (Chen et al., n.d.) opening the doors for adaptive control algorithms in the future. Another important consideration in underwater robotics

is energy efficiency. Since operating underwater requires higher power due to fluid drag optimizing motor performance and reducing unnecessary drag are important CFD simulations provide a beneficial tool in this regard open the robot's body to be shaped for minimum resistance and improved maneuverability (Hernández-Alvarado et al., 2016). By adding these aspects bio inspired design principles new techniques efficient control strategies and simulation based optimization researchers can create underwater robots that are not only cheap but also capable of stable precise and efficient operation (Chen et al., n.d.) . This makes them able for a several range of applications from environmental monitoring and underwater life observation to educational purposes and experimental research projects. (Journal of Materials Science, Geophysics, Instrumentation and Theoretical Physics 2021)

1.2 Problem Statement

Based on the background mentioned above the problems raised in this research are as follows. This propose the problem statement and the specific problems being discuss in this research.

1. How to develop and Design a remote control Underwater Robot cyber fish using PETG material able to moving underwater environments independently without direct human involvement?
2. How to analyze hydrodynamic performance through CFD Simulation and experimental validation velocity changes pressure variation, and turbulence around the robot body through CFD simulation across velocities of 1–5 m/s, validated against physical water tank experiments?
3. How to Develop a System able of Addressing Real Underwater Challenges develop a structurally strong and cheap robotic system that can withstand underwater conditions such as fluid drag and water pressure supporting applications in environmental monitoring and marine exploration?

1.3 Objectives and Benefits

1.3.1 Objectives

1. To develop and Design a remote control Underwater Robot aims to develop a bio inspired robotic cyber fish using PETG material able to moving underwater environments independently without direct human involvement.
2. To Analyze Hydrodynamic Performance through CFD Simulation and Experimental Validation This research analysis the velocity changes pressure variation, and turbulence around the robot body through CFD simulation across velocities of 1–5 m/s, validated against physical water tank experiments.
3. To Develop a System Capable of Addressing Real Underwater Challenges. This study aims to develop a structurally strong and cheap robotic system that can withstand underwater conditions such as fluid drag and water pressure, supporting applications in environmental monitoring and marine exploration.

1.3.2 Benefits of This Research

1. This study boost the researcher's technical and analytical knowledge and skills through the practical use of application of CFD simulation, design, and control system development. It also provide an important key stage toward academic and professional growth.
2. This study serves a reference for new research in underwater robotics and control systems, promoting interfiled collaboration between engineering and marine sciences. It provide a model for integrating new technology and simulation based learning in education.
3. This research promote to developing cheap efficient and capable underwater robotic systems. It can be applied in underwater industries and can use for military surveillance and marine security within ethical and legal boundaries it can reduce human risk in underwater tasks in deep sea.
4. The developed robot supports environmental monitoring pollution detection and marine ecosystem studies can use for military surveillance. It encourages us underwater exploration and helps protect marine biodiversity and resources.

1.4 Limitation of Problems

1. This research is limited to the mechanical design and control system development of the underwater robot, specifically designed to operate in deep-water environments.(size of robot 400mm)
2. The main focuses of our study on analysis the output and efficiency of the robot under specific underwater condition pressure, velocity, kinematic turbulence viscosity.
3. The study inspect the robot's energy efficiency and its ability to operate for certain time periods underwater.
4. The system testing of the robot will be regulate to underwater research, fish ponds, or controlled environments (e.g., laboratories or artificial aquariums)

1.4.1 Organization of the Thesis

1. Chapter 1 the background of our thesis, problem, objectives, and significance of the study. Chapter
2. Chapter 2 we explains the research methodology, including design, simulation, and control implementation.
3. Chapter 3 presents results and discussions.
4. Chapter 4 Conclusion of thesis

1.5 Theoretical Framework

1.5.1 Underwater Robot

Underwater robotics has improved rapidly in recent years helping applications in underwater exploration, underwater examination and environmental research. Old style underwater robots such as Remotely Operated Vehicles (ROVs) and self-controlled Underwater Vehicles (AUVs) depend on screw-type propellers that provide thrust and control but are often complex (Liu et al., 2023), costly, and less energy deliver. In response, research has move toward biomimetic and bio-inspired underwater robots, which replicate the motion of fish and marine animals to achieve make steady propulsion and enhanced control in underwater environments (Mitin et al., n.d.).

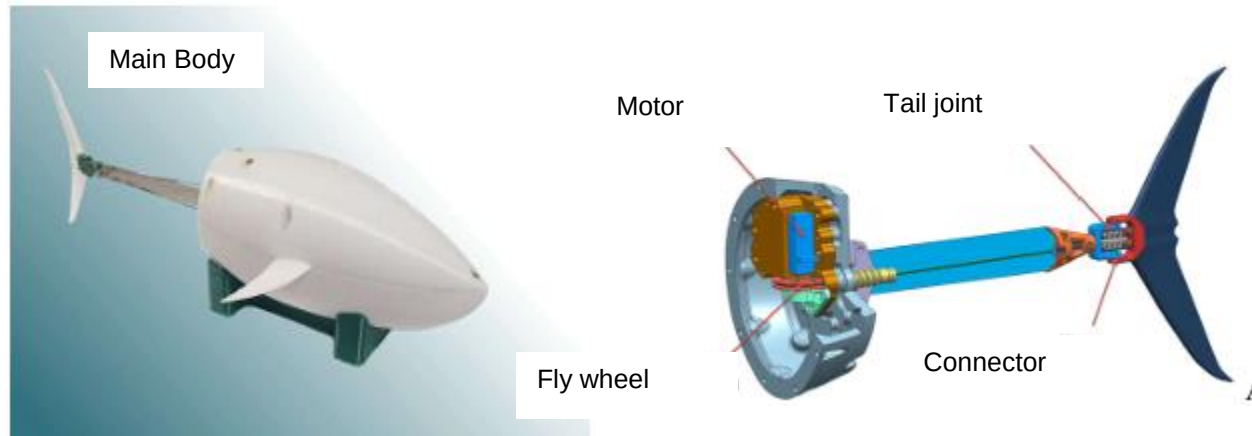


Figure 1. Bio-inspired underwater robots (Mitin et al., 2021)

These robots typically imitate natural movement strategies such as the real fish swimming depending on their body flexibility. The use of bio inspired propulsion not only enhanced hydrodynamic energy but also decreased environmental interference making them suitable for long term underwater working process and ecosystem related monitoring (Liu et al., 2023). The biomorphic propulsion system is central to the design of fish-like underwater robots. As it is presented by a thunniform robotic fish uses an oscillatory caudal fin controlled by servomotors that simulate muscle contractions through elastic chords. This design allows for adjustable frequency and amplitude of fin oscillation, directly influencing swimming speed and energy efficiency. Further studies highlight that the flexibility and stiffness of the fish body and tail significantly affect thrust generation and propulsion performance. Haoshen Ou (2023) conducted a CFD-FSI (Computational Fluid Dynamics Fluid Structure Interaction) analysis on a bio-inspired AUV and confirmed that optimized caudal fin rigidity enhances thrust production while maintaining power efficiency. These findings support the concept that combining soft (Berg et al., 2022), flexible materials with intelligent control mechanisms can lead to more efficient and stable underwater movement, a principle widely used in current bio inspired robot designs (Mimetics et al., n.d.; Rasheed et al., 2024).

1.5.2 Control System Approach

Effective motion control is important for the stability and efficiency of underwater robots. Traditional control methods such as Proportional Integral Derivative controllers remain a very important due to their simplicity, robustness, and ability to reduce errors easily (*PID*, n.d.) State the integration of PID-based reviews control within simulations to reach stable swimming response in nature inspired robots balancing water resistance and thrust forces in real time. Recent progress also explore flexible and sliding mode controllers which increase stability under external forces and irregular dynamics typical in underwater environments (Katzschmann et al., 2018). In case the study on Adaptive Integral Sliding Mode Control highlights its accuracy to maintain precise attitude and trajectory control under uncertain in real time conditions surpassing traditional PID systems in terms of response and stability (Liu et al., 2023). These advanced methods are important for making self control underwater robots capable of maintaining position depth and trajectory in real-world underwater conditions.

1.5.3 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics plays an important role in new underwater robot advancement. It allows researchers to simulate and examine the complex interactions between fluid flow and robotic surfaces before creating model slightly decrease the cost and experimental iterations. According to Wright et al. (2020), combining CFD with FSI analysis enables engineers to predict hydrodynamic forces, assess drag reduction strategies, and evaluate propulsion performance under various material and motion parameters Additionally("CFD-Based Hydrodynamic Analysis of Bionic Fish," 2023), CFD studies assist in optimizing fin geometry, body contours and motion kinematics to improve energy efficiency and maneuverability In conjunction with PID control CFD simulations can simulate real-time behavior helping refine control laws and mechanical designs This integrated approach CFD based optimization coupled with bioinspired propulsion and adaptive control forms the foundation of current research trends in underwater robotics. ("CFD-Based Hydrodynamic Analysis of Bionic Fish," 2023)



Figure 2. Pressure cloud map Haoshen Ou (2023)

1.5.4 Research Gap

The reviewed literature state notable advancements in the development of bio inspired underwater robots, particularly in CFD structural design, and propulsion modeling. Studies such as Mitin et al. (2021) and Wright et al. (2020) demonstrate that bio inspire propulsion using fish-like tail motion can effectively reduce drag and improve maneuverability compared to traditional propeller systems. Similarly, the application of Computational Fluid Dynamics (CFD) has become a key tool for predicting water flow behavior and optimizing body shapes before fabrication, however, several research gaps still exist (Dai et al., 2021; Hernández-Alvarado et al., 2016)

1. Most previous studies focused only on CFD simulation, without integrating FEA to analyze structural strength, pressure, and deformation under real underwater conditions.
2. Some researchers performed physical model tests but did not validate results with CFD or FEA, reducing the reliability and accuracy of findings.
3. Most existing fish-like robots rely solely on tail motion, resulting in slow forward–backward movement and poor maneuverability.
4. The robots cannot maintain the required underwater depth or perform multi-directional motion (side or rotational control).
5. There is no fully operational system that combines CFD, FEA, and PID control to attain both real time efficiency and stable motion in low cost, small-scale underwater robots.

This chapter discuss previous research's related to underwater robotics focusing on bio inspired propulsion systems CFD based analysis and control system. The discussion highlighted the progress made in real time optimization and stability control, while also notice different gaps in existing studies. Earlier works were bounded to CFD simulation or physical model testing, deficient an integrated approach that includes CFD, FEA,(Katzschmann et al., 2018; Syam et al., 2025) and PID control for efficient and stable underwater operation. Based on these findings, the next chapter explains the research methodology used in this study including the design process, simulation setup, and control system implementation.

CHAPTER II

RESEARCH METHODOLOGY

2.1 Research Location

This research has been conducted starting in August 2025 at the Control and Robotics Laboratory, Department of Mechanical Engineering, Faculty of Engineering, Hasanuddin University the research schedule is presented in Table.

Table 1. Research Description

NO	ACTIVITIES	RESEARCH TIME TABLE (2025-2026)					
		August	September	October	November	December	May
1	Literature review						
2	Proposal Preparation						
3	Seminar Proposal						
4	Research a. Simulation b.Experimental Method						
5	Thesis writing						
6	3D Printing						
7	Experiment						

2.2 Underwater Robot Design

This is a compact and fully waterproof underwater robot designed and modeled using Autodesk Fusion 360. The total body length measures 210 mm, with the main body opening at 316 mm and a side fin radius of 40 mm (80 mm diameter). The central body height is 160 mm, while the side mounted wings extend up to 124 mm. The tail part we adds an additional 212 mm in length. To improve buoyancy and body strength, the robot keep a partial hollow inside structure and a uniform wall thickness of approximately 3 mm. A distinct dome-shaped housing is incorporated for installing internal components such as electronics or sensors. The propulsion system includes two vertical side attached thrusters at the angle 45° degree for controlled up and down beside with a bottom thruster for forward and reverse motion.

Our robot is 3D printed using by PETG (Polyethylene Terephthalate Glycol), a strong and flexible material that is also naturally waterproof making it able for long term underwater operations. Compared to PLA, PETG offers superior strength and sealing output(Syam et al., 2025), to keep safe that the internal components remain fully protected. A key design function of this robot is its flexible tail part, which is made from a soft rubber material. This flexible tail enhance the robot's mobility in water, absorbs small impacts and increase directional stability during movement.

2.2.1 Design Highlights:

- Compact, hydrodynamic, and waterproof structure
- Multi-axis control with high maneuverability
- Durable PETG body and soft rubber tail for enhanced flexibility
- This robot is ideal for exploration, research, and environmental monitoring Our design balances reliability with responsiveness, making it a practical and low cost solution for underwater robotics applications.

2.3 Underwater robot fish design

2.3.1 Robot fish and its dimensions

1. Robot side view

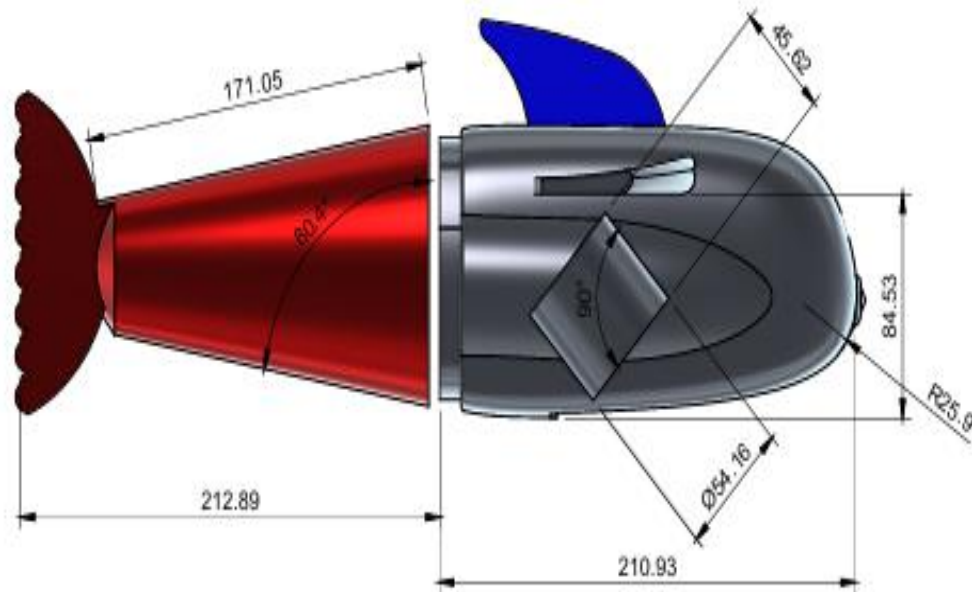


Figure 3. Robot side view

2. Robot main body

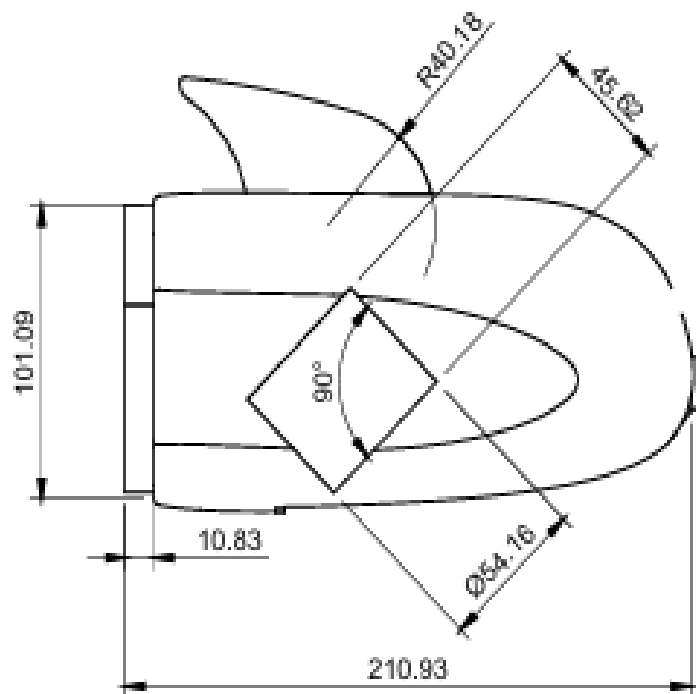


Figure 4. Robot main body

3. Robot fish tail

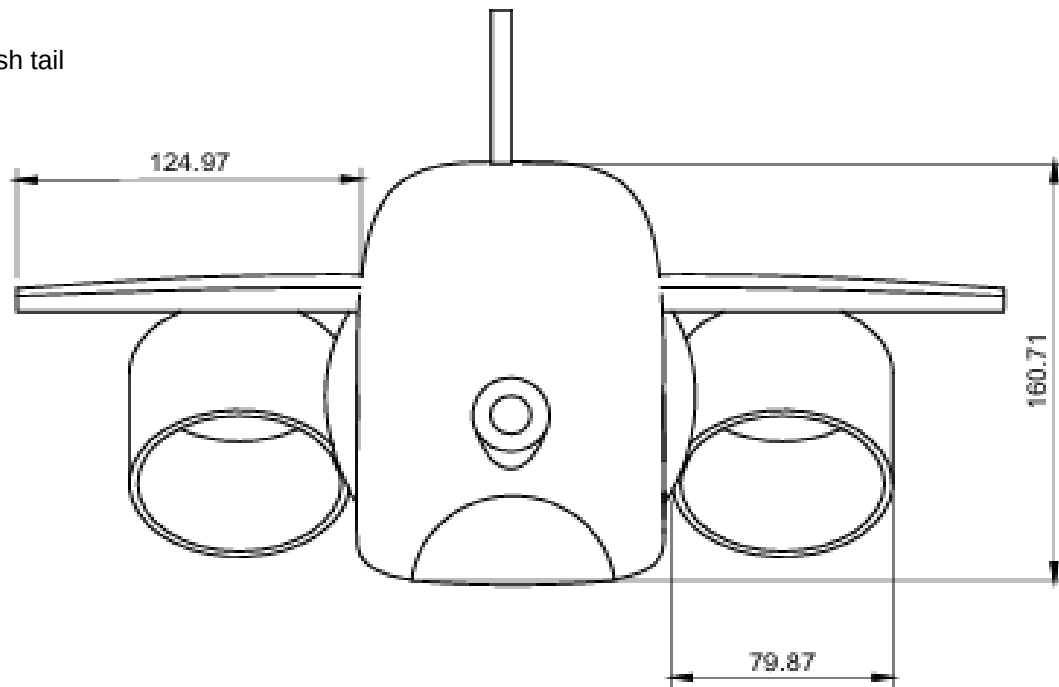


Figure 5. Robot fish tail

4. Sketch and dimensions

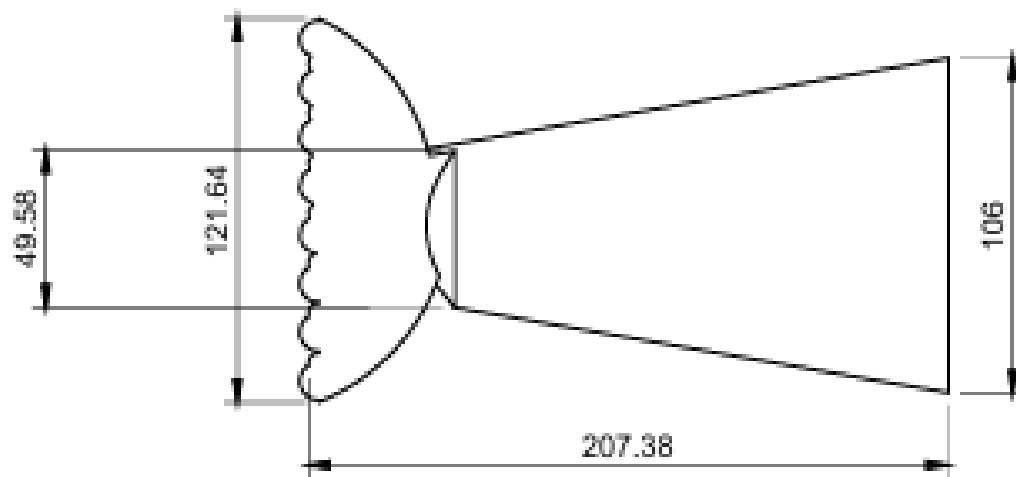


Figure 6. Sketch and dimensions

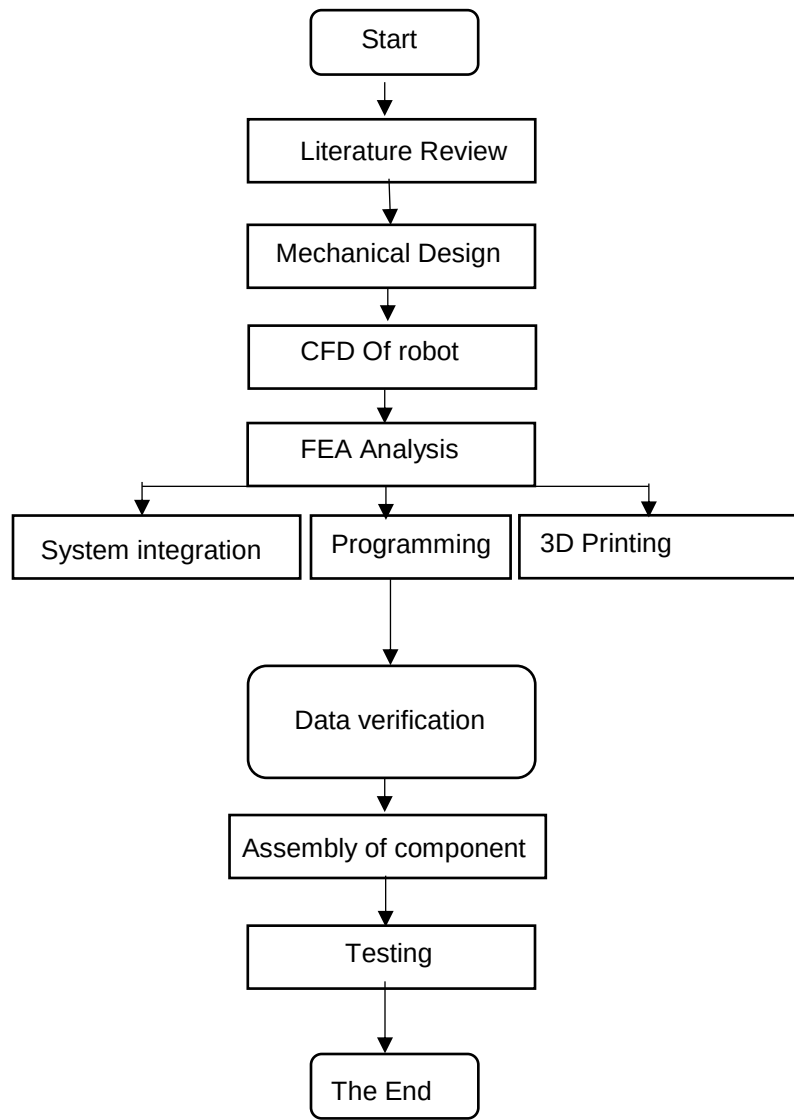
2.4 Research Design

The research applies a simulation based and experimental method combining both computational and practical techniques. The design steps involves 3D modeling of the underwater robot inspired by fish mobility, focus on body curvature and tail shape for efficient flow interaction. CFD simulations are apply to evaluate real time behavior like drag, pressure distribution, and velocity contours. In parallel FEA is applied to identify the body response under static pressure and hydrodynamic forces, to make sure the robot's body can endure underwater loads.(Dai et al., 2021)

Following the simulation steps, remote control system is developed and calibrated to balance the robot's motion and maintain a particular underwater depth. This well integrated design methodology aims to achieve an specific balance between hydrodynamic efficiency, body reliability and motion stability. The overall research process consists of four main steps. (Krishnan & Simon Sagi, n.d.).

2.4.1 Methodology Flowchart

Table 2. Flowchart



2.5 System Design

The system design focuses on developing a compact, efficient, and stable underwater robot inspired by fish locomotion. The robot's structure is designed using Fusion 360 and SolidWorks based on the streamlined profile of thunniform fish species to minimize drag and improve flow smoothness. The robot's total length is approximately 40 cm, with a uniform body to ensure balance in underwater conditions.(Aldhaheeri et al., 2022)

We have used Lightweight and durable materials such as PETG are selected for 3D printing, providing a strong yet buoyant body suitable for underwater environments. The propulsion mechanism consists of a rear-mounted tail fin driven by a DC or servo motor to replicate oscillatory fish motion. The internal hollow space the microcontroller (Arduino or ESP32). We also have used waterproof sealing components to protect electronic systems during underwater operation time.(Wright et al., n.d.)

The system's components arranged allows easy unification of sensors, power units, and control components, allowing future advancement or modification. The robot is refine for low-cost fabrication, ease of maintenance, and balancing during underwater operation, which makes it ideal for both academic research and small scale environmental monitoring tasks.(Pedoman_Tugas_Akhir_Des23_FIX_Kirim (1), n.d.)

2.6 CFD Simulation Environment Description

To analyze the hydrodynamic and structural performance of the designed underwater robot, Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA) were conducted using simulation software such as Fusion 360 CFD and Simscale Workbench. These examine were conducted to improve robot's geometry review drag and pressure distribution and ensure body reinforce under underwater conditions. Simulation was performed in a controlled water tank environment, designed to copy real underwater conditions. We took tank dimensions were as follows:

2.6.2 Water tank measurement

- Length of our water tank 5 meters
- Width of our water tank 2 meters
- Depth of our water tank 3 meters
- This rectangular domain provided sufficient space for fluid flow analysis and realistic interaction with the robot geometry.

2.7 Simulation Objectives and Conditions

Within this test tank, the underwater robot was placed in a stationary position while water was simulated to flow from left to right across the domain. The simulation setup focused on analyzing the following physical parameters:

2.7.1 Simulation Goals

- Hydrostatic Pressure Distribution
- Flow Velocity Field (Magnitude and Direction)
- Temperature Distribution
- Turbulent Pressure and Velocity Effects

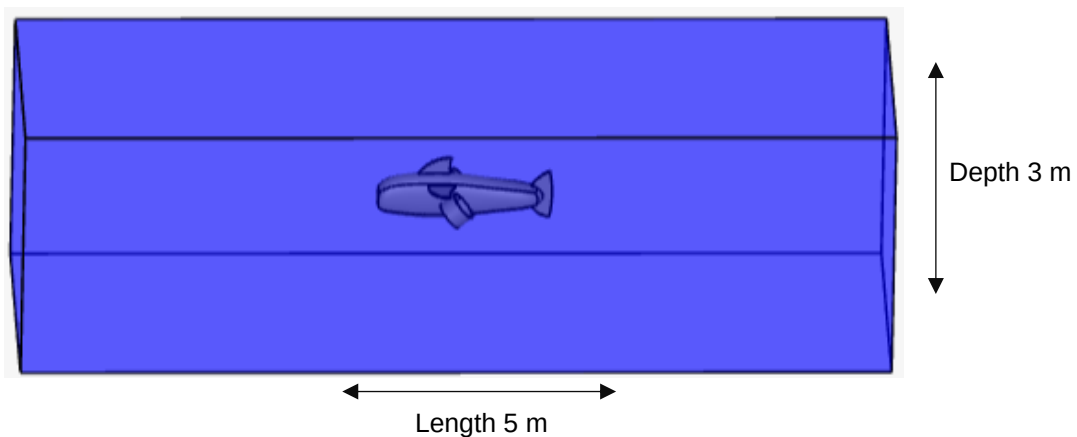


Figure 7. CFD simulation Environment Description

2.8 Simulation setup conditions:

The CFD simulation of the underwater robotic fish figure 8 was conducted on the SimScale platform. The robot body was assigned PETG material with a density of 1270 kg/m^3 while water was defined as the working fluid throughout the domain. Initial conditions included a modified gauge pressure a fluid temperature held constant at standard ambient underwater conditions and turbulence parameters turbulent kinetic energy (k) and specific dissipation rate (ω) initialized using the k - ω SST turbulence model to accurately capture near wall and free stream flow behavior. The inlet velocity was varied from 1 m/s to 5 m/s to evaluate the hydrodynamic response across different flow regimes while a pressure outlet condition was applied at the domain exit. The tank walls were treated as no-slip boundaries and the robot surface was defined as a solid interface to enable momentum between the fluid and solid regions. Mesh refinement was applied around the robot body and wake region to resolve boundary layer effects and vortex formation, with residuals monitored throughout to ensure convergence at each velocity condition. Prior to simulation, the tail section was converted from silicone rubber to a solid body to ensure compatibility with the finite volume solver.

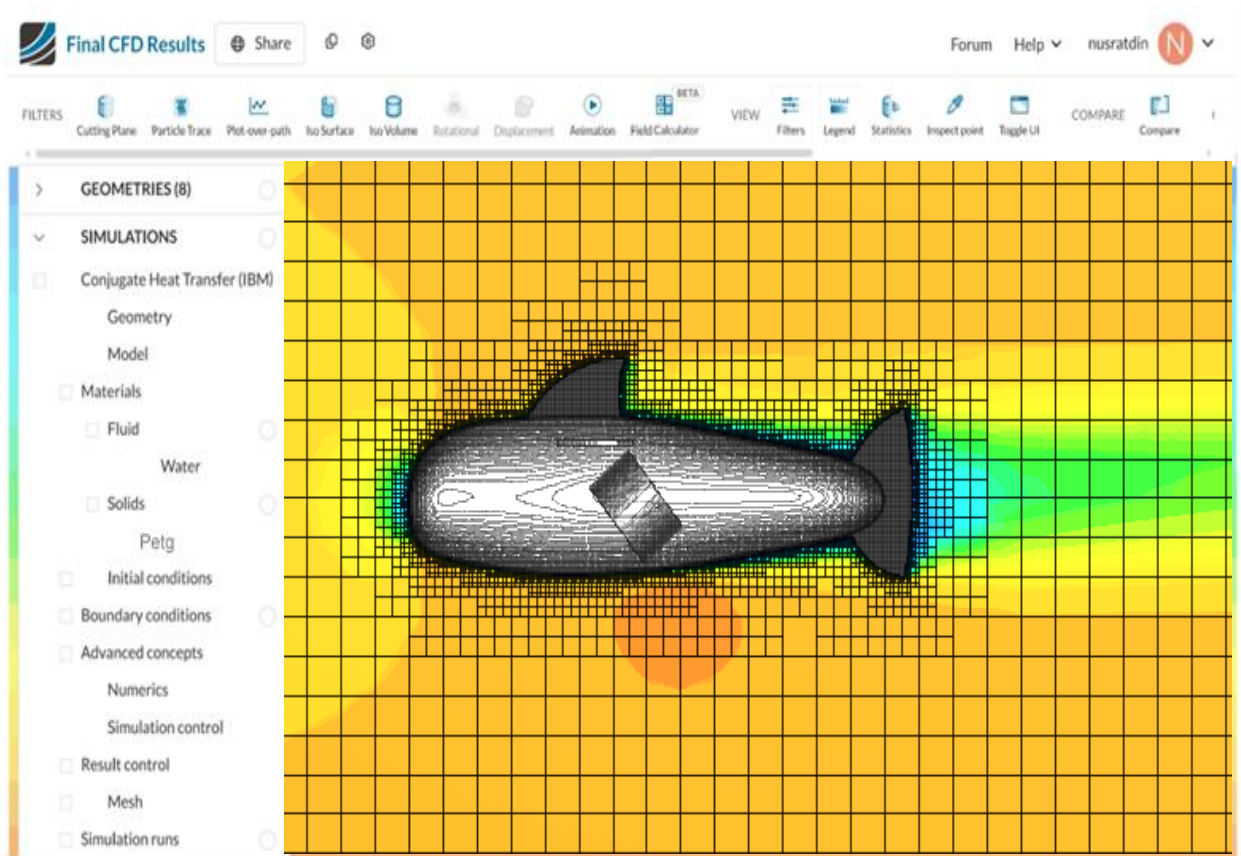


Figure 8. General CFD Simulation setup conditions.

CFD SIMULATION SETUP CONDITIONS

Underwater Robot | Hydrodynamic | SimScale

Table 3. CFD Simulation Setup Conditions

2.8.1. General Setup

Parameter	Value
Platform	SimScale (Cloud-Based CFD)
Simulation Type	Hydrodynamic
Fluid	Water
Robot Material	PETG — Density: 1270 kg/m³
Note on Material	Material is PETG
Tail Modification	Silicone rubber → converted to solid body for solver compatibility

Table 4. Boundary Conditions

Boundary	Condition Type	Details
Inlet	Velocity Inlet	V = 1–5 m/s, normal to face
Outlet	Pressure Outlet	Zero gauge pressure
Tank Walls	No-Slip Wall	Zero velocity at wall surface
Robot Surface	Solid Interface	Fluid-solid momentum

2.9 CFD Pressure and Flow Analysis of the Robot Fish

We performed a simulation to observe how water flows around my underwater robot fish and the how pressure is distributed on its body. The figures 8 illustrate the Computational Fluid Dynamics (CFD) results of an underwater robot shows laminar water flow conditions. The results include pressure range and velocity vectors, presented from front, top, and side views to understand how water react with the robot body. The pressure scale is shown in kilopascals (kPa), ranging approximately from 0.29 kPa dark blue to 6.22 kPa red.

2.10 Pressure analysis under 1 m/s velocity

2.10.1 Pressure distribution front view

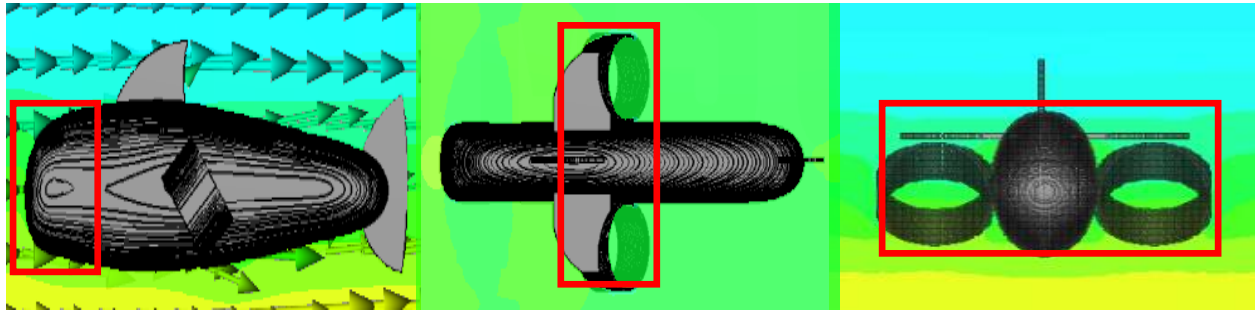
The pressure distribution around the nose of our underwater robot at the velocity of 1 m/s Figure 9. (a) can be clearly observed. The central portion of the nose shows a blue region indicating low pressure values around 3.8-4.0 kPa. In contrast the outer sides of the front face show green to light yellow colors corresponding to pressure values of approximately 2.7-3.8 kPa. This pressure pattern show that the incoming water flow splits smoothly around the robot rather than interact directly at the center. The slightly higher pressure on the sides indicates laminar flow observed which helps to decrease frontal drag and improves hydrodynamic efficiency.

2.10.2 Pressure Distribution Top View

From the top Figure 9. (b) View the pressure distribution around the robot at the velocity of 1 m/s appears mostly uniform and green showing pressure values close to 2.7 kPa. Both the left and right sides experience nearly identical pressure levels showing that the flow is symmetrical. This laminar flow indicates that no significant turbulence pressure indicate meaning the robot is unlikely to experience unwanted yaw or rotational motion due to flow forces. However balanced pressure distribution is an important indicator of directional stability and straight-line motion during underwater operation.

2.10.3 Pressure distribution side view

In the figure 9. (C) side view the water flow direction is from left to right as indicated by the black velocity arrows at the velocity of 1 m/s . Far from the robot the arrows are longer showing higher flow velocity. And as the flow approaches the robot surface the arrows become shorter which indicating a decrease in velocity due to interaction with the solid body. This behavior confirms the presence of boundary layer effects and energy loss near the underwater robot surface. At the nose region the pressure changes between low blue to green shades 1.5-2.7 kPa showing a streamlined shape that allows water to accelerate around the front of robot nose. Along the middle section of the robot pressure increases slightly and appears as green to yellow colors 3.8-4.5 kPa. This increase changes occurs because the water flow slows down and interacts more directly with the body. Toward the rear and tail region, the pressure decreases again changes back toward blue and light green colors below 3.2 kPa indicating flow recovery and reduced drag in the wake region. Overall Flow and Pressure Interpretation across all three views, the pressure distribution remains linear and well balanced with no high pressure approaching the maximum value of 6.22 kPa. The lowest pressure regions remain close to 0.29 kPa, mainly around streamlined areas where the flow accelerates. The combination of low front pressure symmetric side pressure and gradual pressure reduction toward the tail describe that the robot has a hydro dynamically optimized shape. Overall the CFD results show that the underwater robot design promotes linear motion reduced drag and efficient flow interaction making it suitable for controlled and energy efficient underwater navigation.



- a.** Pressure analysis front of robot the Pressure Range in Nose 3.8-4.0 kPa
b. Pressure analysis middle of robot the Pressure Range in middle section 3.8-4.5 kPa.
c. Pressure analysis X direction of robot the Pressure Range at the tail 3.2 kPa.



Figure 9. CFD Pressure and flow analysis of the robot fish ,(a) Pressure analysis front of robot the Pressure Range in Nose 3.8-4.0 kPa **(b).** Pressure analysis middle of robot the Pressure Range in middle section 3.8-4.5 kPa. **(C).** Pressure analysis X direction of robot the Pressure Range at the tail 3.2 kPa.

2.10.4 Pressure Distribution Graph

This simulation result figure 10 shows the pressure distribution around the robot fish at the velocity of 1 m/s. At the front section nose point of robot the pressure is relatively high because the incoming water directly strikes the nose of the robot. As the flow moves along the body, the pressure decreases due to drag and the streamlining effect. In the middle part slight pressure change occur as a result of the thrusters and body geometry. Towards the back the flow begins to decrease and the pressure gradually stabilizes as the water exits the wake region. Overall this pressure profile shows that the robot fish remains stable and the observed changes are normal effects of flow interaction with the body and thrusters. The important point is that direction of our robot is moving right to left side.

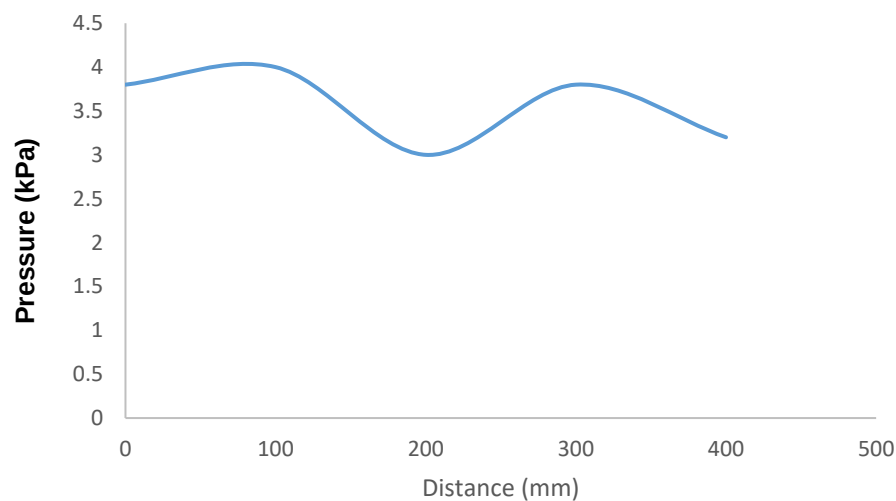


Figure 10. Pressure distributing on underwater robot body graph

2.11 Velocity analysis under 1 m/s

2.11.1 Velocity Behavior at the Nose Region

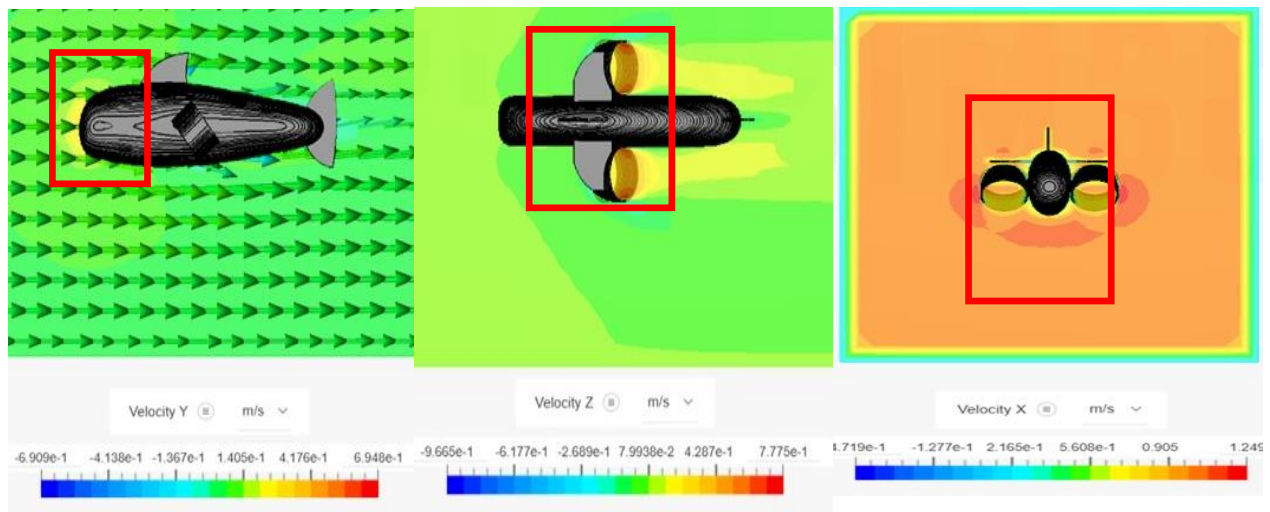
The velocity of 1 m/s at the nose of the robot, in the figure 11. (a) The velocity contours show blue to light green colors in the Velocity-Y and vector views. According to the color scale, the velocity at the nose lies approximately between -3.37×10^{-1} to 4.17×10^{-1} m/s. This is normal velocity region indicates that the incoming water flow significantly slows down when it reaches the front surface of the robot. This reduction in velocity is caused by the resistance created by the robot nose leading to decrease water flow effects. The smooth color transition around the nose also shows that the water is redirected laminar flow rather than forming strong turbulence.

2.11.2 Velocity Behavior at the Middle Section

The velocity of 1 m/s Figure 11. (b) Velocity-Z view, the middle section of the robot is mostly surrounded by green to yellow colors. According to the scale this corresponds to velocity values ranging from approximately 2.80×10^{-1} m/s to 4.29×10^{-1} m/s. These values indicate a relatively stable and moderate velocity region along the central body of the robot. The flow in this region remains attached to the surface and moves smoothly along the body, which is a sign of good hydrodynamic design. The absence of dark blue or red zones in the middle section shows that there are no sudden velocity drops or excessive acceleration.

2.11.3 Velocity Behavior at the Tail Region

The velocity of 1 m/s At the tail of the robot, figure 11 c the velocity increases again, which is clearly visible in both the Velocity x view and the vector velocity view. The tail region shows yellow shades, corresponding to velocity values of approximately 6.95×10^{-1} m/s. This increase in velocity show as the water accelerates after passing over the robot body and enters the wake region. The slowly acceleration at the tail indicates smooth flow recovery and reduced energy loss, which is important for minimizing drag and improving propulsion efficiency.



- a. Velocity analysis front of robot 4.17×10^{-1} m/s. b. Velocity analysis Middle Section 4.17×10^{-1} m/s. a. Velocity analysis at tail of robot 6.95×10^{-1} m/s

Figure 11. Velocity and flow analysis of the robot fish ,(a) Velocity analysis front of robot 4.17×10^{-1} m/s,(b) Velocity analysis Middle Section 4.17×10^{-1} m/s, (c) Velocity analysis at tail of robot 6.95×10^{-1} m/s

The velocity of the flow is initially higher which is 0.41 m/s but when the water strikes the front of the robot fish, it starts to decrease due to drag force and touch 0.14 m/s. In the middle section, slight disturbances are observed because of the thrusters. After this region, the flow becomes smoother and the velocity gradually increases again towards the rear and velocity approach 0.695 m/s. This overall velocity profile shows that the robot remains stable, and the observed variations are mainly the result of drag and thruster effects rather than instability.

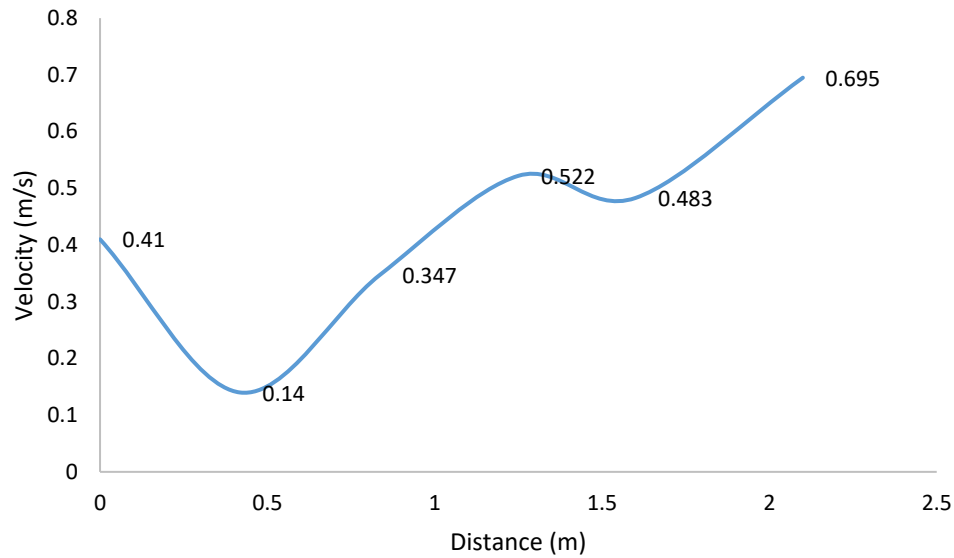


Figure 12. Velocity distance change graph

2.11.5 Simulation Acceleration Analysis

The acceleration behavior of the underwater robot analyzed by using CFD simulation to observe how the velocity of the robot varies with time during the starting of motion. In this analysis the robot motion observed over a total time interval of 3 seconds. The simulation results show that the robot slowly increases its velocity as it moves forward in the water. At the starting of the motion the velocity is relatively low because the propulsion system first needs to overcome the initial hydrodynamic resistance. As time increase the robot continues to accelerate and the velocity increases smoothly. According to the simulation the robot reaches approximately 0.2 m/s within the first 0.5 seconds. The velocity then increases to about 0.4 m/s after 1 second 0.6 m/s after 2 seconds and finally around 0.8 m/s at 3 seconds this slowly increase in velocity shows balanced acceleration of the underwater robot linear interaction between the robot body and the water the simulation therefore shows that the robot is able of achieving consistent acceleration under the given flow conditions.

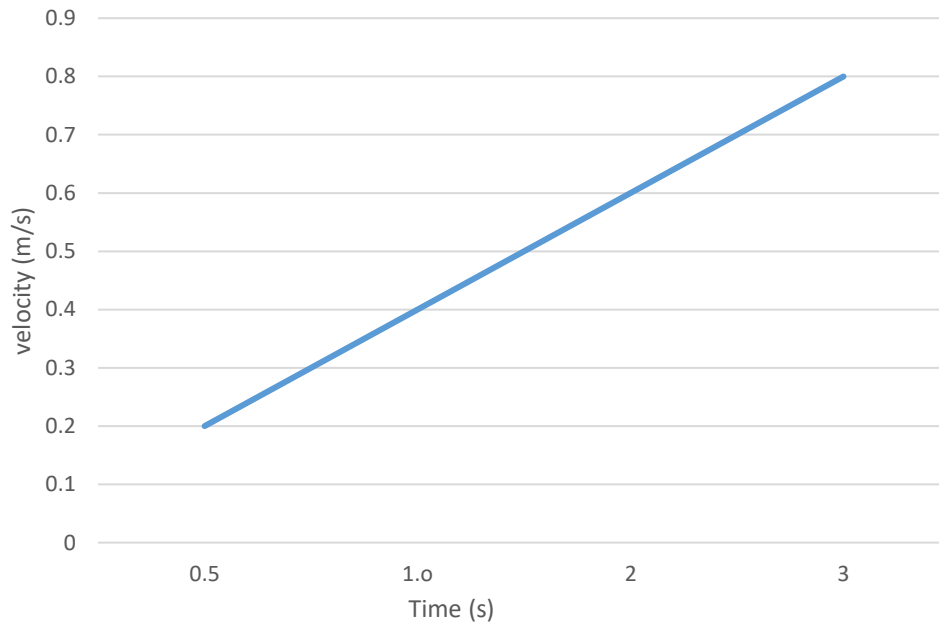


Figure 13. The acceleration behavior of

2.12 Turbulence analysis under 1 m/s of velocity

2.12.1 Turbulent Kinematic Viscosity V_t

The analysis of turbulent kinematic viscosity (V_t) around the underwater robotic fish provides valuable insight into how fluid motion behaves in the presence of the model. This simulation helps identify regions of turbulent flow development, which directly affects the robot's stability and efficiency.

2.12.2 Turbulence Analysis side view:

The robot is placed centrally within a flowing domain the velocity of 1 m/s, and it is clear that the turbulence intensity is minimal near the tail blue zone figure 14. (a), where the turbulent kinematic viscosity remains very low, approximately in the range of $2.64 \times 10^{-11} \text{ m}^2/\text{s}$ to $2.82 \times 10^{-4} \text{ m}^2/\text{s}$, and gradually increases along the body. Maximum kinematic viscosity is detected just around the nose of our robot and middle section $2.64 \times 10^{-11} \text{ m}^2/\text{s}$ to $2.8 \times 10^{-4} \text{ m}^2/\text{s}$. where turbulent kinematic viscosity reaches values close to $2.8 \times 10^{-4} \text{ m}^2/\text{s}$. suggesting that the frontal area experiences the most fluid drag force resulting in higher turbulent mixing. The color change from bit blue shows that as the water hits the front of the robot, turbulence is generated and expands backward as the flow separates and interacts with the fish-like structure.

2.12.3 Turbulence Analysis front view:

The front view figure 14 (a) provides a direct visualization of how turbulence develops around the nose and lateral surfaces of the underwater robot at the velocity of 1 m/s. Key observations include. The high turbulence zones the red and orange areas surrounding the robot nose and side fins shows regions of intense turbulent mixing where incoming water flow impacts the body head on. These areas are with turbulent kinematic viscosity values approaching approximately $2.64 \times 10^{-11} \text{ m}^2/\text{s}$ to $2.82 \times 10^{-4} \text{ m}^2/\text{s}$ symmetrical distribution the symmetry of the turbulent zones on both sides shows that the model and the boundary conditions are well balanced. This shows that the robot's body induces uniform turbulence on both sides, enhancing simulation reliability.

2.12.4 Turbulence Analysis Top View

At the velocity of 1 m/sv the top down perspective allows insight into how turbulence trails and evolves longitudinally along the body. Key points include frontal Peak Turbulence as in the front view, the highest turbulent viscosity appears at the front of the robot, confirming that the nose is the first and most impacted area of fluid interaction, with (V_t) values rising up to approximately $2.64 \times 10^{-11} \text{ m}^2/\text{s}$ to 2.82×10^{-4} . The red and yellow areas extend backward along the body towards tail indicating how turbulence show downstream due to flow separation and interaction with robot surface. With turbulent kinematic viscosity slowly decreasing to values around $2.81 \times 10^{-4} \text{ m}^2/\text{s}$ to $5.64 \times 10^{-4} \text{ m}^2/\text{s}$. noticeably the tail section shows a decrease in turbulent viscosity transitioning into green and blue hues. This indicates a more laminar or stabilized wake, where (V_t) further decreases to approximately $2.82 \times 10^{-4} \text{ m}^2/\text{s}$ and below. This graph shows the turbulent kinematic viscosity around the robot fish during the CFD simulation.

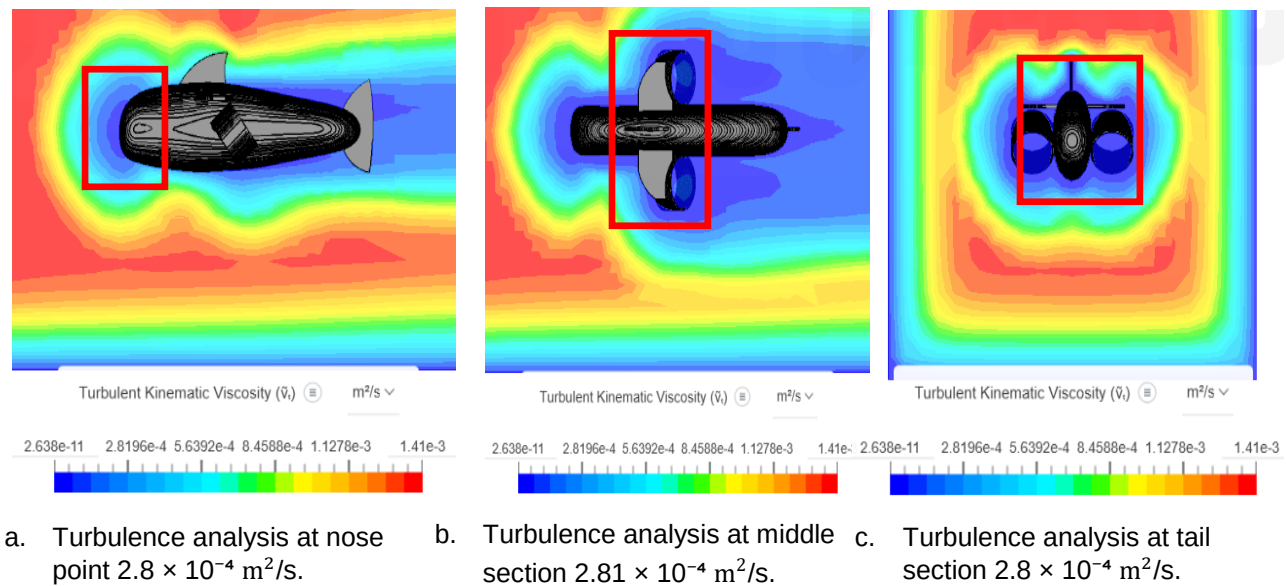


Figure 14.. Turbulence distribution around robot body ,(a) Turbulence analysis at nose point $2.8 \times 10^{-4} \text{ m}^2/\text{s}$, (b) Turbulence analysis at middle section $2.81 \times 10^{-4} \text{ m}^2/\text{s}$. (c) Turbulence analysis at tail section $2.8 \times 10^{-4} \text{ m}^2/\text{s}$.

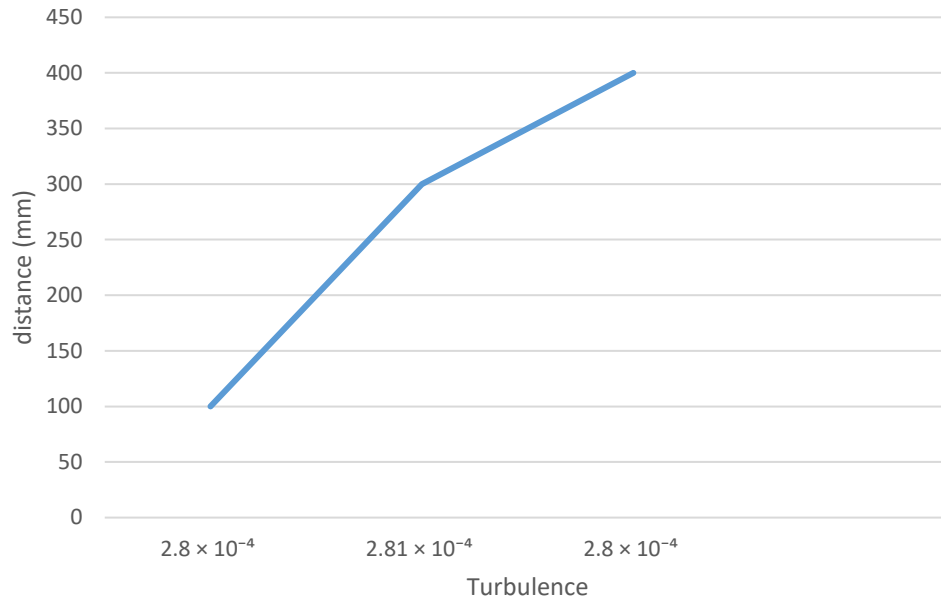


Figure 15. Turbulence distribution around robot body

2.13 Residual Convergence

The residual slopes graph 16 at the velocity of 2 m/s shows stable convergence for all main variables velocity components for example (U_x, U_y, U_z), turbulence quantities (k, ω) slowly decrease to acceptable levels the acceptable level is between 10^{-3} and 10^{-6} . If the slope above this range robot will not stable the pressure (P_{rgh}) residual decreases moderately but remains within a reasonable range. The temperature (T) slope stays nearly constant around 10^0 which shows that the temperature field is already stable and does not require further adjustment. So our solution is numerically stable and well-converged.

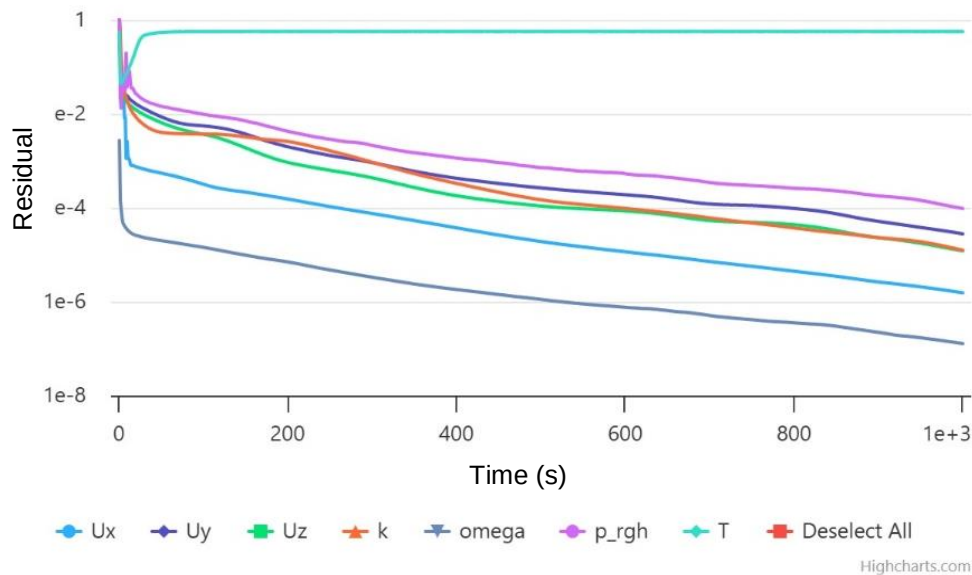


Figure 16. Residual Convergence

2.14 Finite Element Analysis FEA of Underwater Robot

2.14.1 Objective of the Test

Our main goal to perform this finite element analysis was to examine the body strength and water pressure resistance of the robot fish when dive at different water depths. Our robot is designed to operate underwater operation it is important to ensure that its body and fins can safely withstand under the hydrostatic pressure and maintain stability without permanent changes.

Finite Element Analysis (FEA) was conducted using Autodesk CFD software. The top body were assigned as fixed supports, and uniform external pressure was applied on the robot's top surface to simulate the hydrostatic pressure of surrounding water. The material used for the robot body is PETG plastic, with an average yield strength of 47.49 MPa.

Two tests were performed corresponding to different water depths:

- FEA analysis at 3 m water depth = 30 kPa
- FEA analysis at 5 m water depth = 50 kPa

2.14.2 Observation

Under the 20 kPa pressure 2 m depth the simulation results see Figure show that the maximum Von Mises stress is approximately 21.89 MPa, concentrated near the tail fin region. The other body parts experiences much lower stress levels the rang from 4 MPa to 13 MPa, as represented by the blue-to-green slope zones. The body remains within the elastic limit and no deformation is expected. Hence the prototype is stable and safe for operation at this depth. When the pressure was increased to 50 kPa (5 m depth), the stress level rose sharply near the fins and motor thrust area, where visible bending was observed in the physical prototype. This indicates that the stress in these regions exceeded the material's yield limit, causing localized deformation. So the conclusion is that robot is not structurally safe at 5 m depth without reinforcement.

2.14.3 Safety Factor (SF) of our robot

The safety factor which shows the gap between yield strength and the maximum applied stress and it calculated as follows:

$$SF = \frac{\text{Yield strength}}{\text{Maximum stress}} \quad (1)$$

FEA analysis at 3 m depth:

$$SF = \frac{47.49}{29.36} = 1.6 \quad (2)$$

FEA analysis at 5 m depth:

$$SF = \frac{47.49}{74.36} = 0.9 \quad (3)$$

If the safety factor value of anybody structure is equal to or greater than 1 the prototype is considered safe this means that the material can withstand the applied stress without deformation or any change in body. If the safety factor is less than 1 we consider tha the prototype is not safe, as the stress has greater then material's yield limit and may cause permanent deformation, body collapse may occur.

So our simulation results confirms that the design is safe at shallow depths (3 m) but fails beyond 5 m due to insufficient strength in the fin section. The FEA simulation and prototype testing confirm that the PETG body performs safely under 20 kPa pressure (2 m depth) but becomes unstable under 50 kPa (5 m depth). The tail fin and motor thrust zone of our prototype are the important areas experiencing the maximum stress. For improved performance in deep water reinforcing the fins or using a stronger material such as ABS, carbon-fiber PETG, or aluminum for structural parts can be used for deep water.

2.15 Von Mises Stress

FEA under 3 meters water depth and 30 KPa pressure. This figure 17 the Von Mises Stress result of the underwater robot body under applied water pressure 30 kPa in SimScale. The color distribution on the model represents the stress intensity on different parts of the structure. The blue regions indicate areas of low stress which is around 1.5×10^{-6} kPa to 5.8×10^{-1} kPa meaning those parts are experiencing minimal deformation or load. The green to yellow regions show normal stress which is approximately 1.76 kPa to 2.0 kPa, and while the red zone near the tail shows the maximum stress points. In the stress scale below the model the maximum stress value appears to be relatively low (in the range of a few kilopascals) that suggesting us that the body structure is safe and stable under the applied water pressure of approximately 0.03 MPa (3 meters underwater) so the model seems structurally safe and withstands with no signs of material failure or excessive deformation.

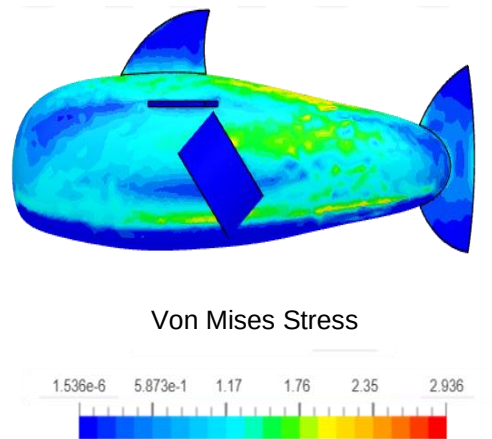


Figure 17. Von Mises Stress distribution

2.15.1 Total strain

Under 3 meter water depth figure 18 the total strain distribution of an underwater robot analyzed in SimScale. Total strain represents how much the material deforms under applied forces. The color scale at the bottom indicates the magnitude of strain in meters per meter (m/m). Where blue shades represent compression and red shades represent tension. In this result the rear part of the robot appears red and orange, showing the highest tensile strain of about 1.23×10^{-2} m/m, while the front and middle sections appear green and yellow, indicating moderate strain levels. The blue regions near the middle or lower part of the body show areas of compression or minimal deformation. Overall, this analysis suggests that the back portion of the robot experiences the greatest structural strain, likely due to pressure or fluid forces acting more intensely in that region.

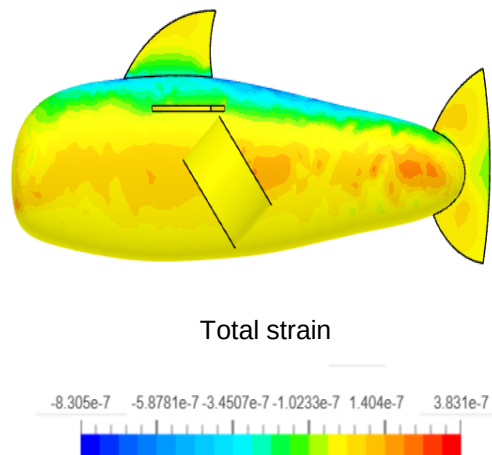


Figure 18. Total strain distribution of robot

2.16 Von Mises stress

The figure 19 shows FEA under 5 meters water depth and 50 kPa pressure. Our robot this simulation clearly show structural failure in the underwater robot's body due to high stress concentrations. The Von Mises stress touch approximately 77.6 MPa, which is significantly higher than the yield strength of material that we have used for our robot we used PETG materials the yield strength is around (around 40–50 MPa). This means the body has cross its elastic limit and the robot body will permanent deformation or cracking especially around the fin attachment areas and mounting sections where stress is very high. The high pressure load and real timing forces acting in these regions have created high stress so the resulting in a high risk of material failure. Hence our current design is not structurally safe for underwater the simulated conditions.

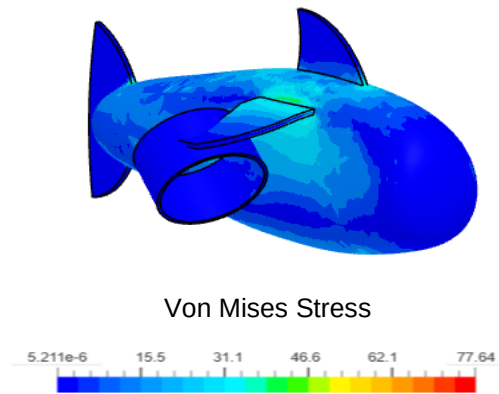


Figure 19. Von Mises stress front view

2.16.1 Strain distribution

The total strain distribution of the underwater robot figure 20 shows important deformation across the underwater robot body confirming that the material failure under high-pressure loading. The strain values around robot body from approximately 0.02 m/m, show critical stretching and strain, especially around the mid body and fin joints which is 6.6×10^{-4} to 1.2×10^{-2} . These regions are experiencing high stress transfer, leading to visible distortion and deformation in the structure. The high strain values show that the PETG plastic material cannot withstand the underwater pressure applied during the simulation, and causing body permanent deformation. This level of strain cross the safe elastic limit for typical PETG materials, showing that the design has failed under load. Structural reinforcement or using a stronger material.

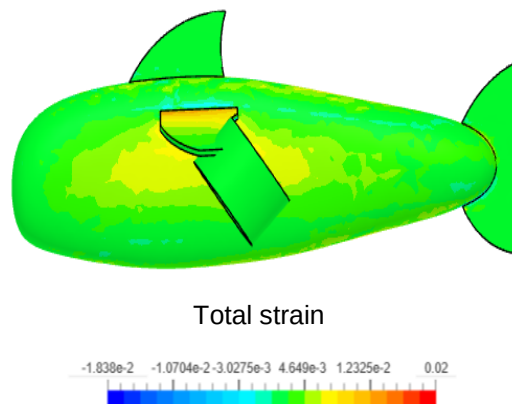


Figure 20. Strain distribution of robot

2.16.4 Structure deformed

Our simulation results below figure 21 shows the displacement for the underwater robot got from the static structural simulation in SimScale. The displacement values range approximately from -3.34×10^{-3} m to 1.79×10^{-3} m, show how much the structure deformed when applied to external forces 50 KPa. The color distribution from blue to red shows the magnitude of displacement where the blue region shows the maximum negative displacement and the red part shows the maximum positive displacement. It can be seen that the front and lower parts of the robot experienced high deformation, while the rear and upper sections remained comparatively stable. This uneven displacement pattern suggests structural instability under the given load, especially around the mid-body where the fin and thrust motor joints. As a result, the simulation result indicates that the structure failed to maintain uniform stiffness and stability along the X-direction.

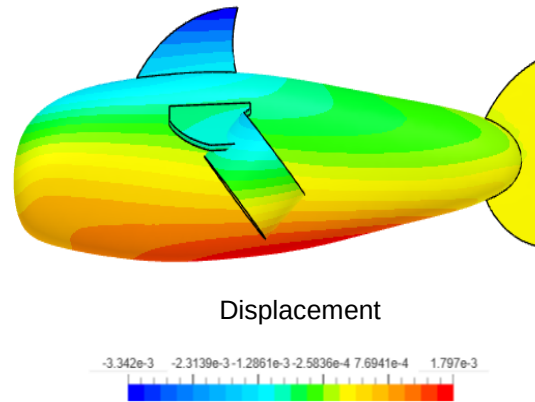


Figure 21. Structure deformed Displacement

2.17 Motion of Fish Tail

Our robotic fish tail swings side-to-side around its root, just like a real fish. The motion is governed by two main aspects:

- Angular displacement (θ): how far the tail swings from the centerline.
- Angular velocity ($d\theta/dt$): how fast the swing happens which determines the sideways transverse speed that pushes water and generates thrust.

In biological fish, the motion is smallest at the root and largest at the tip. We modeled the tail with a quadratic amplitude growth from root to tip, ensuring that most of the thrust comes from the faster-moving tip.

Key Assumptions

- Our Robot Tail length: $L = 0.20$ m (200 mm).
- Robot tail Maximum angular amplitude at the tip: 30° (0.5236 rad).
- Robot tail Frequency of oscillation: $f = 3$ Hz, corresponding to

$$\omega = 2\pi f = 18.85 \text{ rad/s} \quad (4)$$

2.17.1 Angular frequency

Let's consider

- $L = 0.20$ m or 200 mm
- $F = 3$ Hz
- $\theta_{tip} = 30^\circ$

$$\omega = 2\pi f \quad (5)$$

$$\omega = 2 \times \pi \times 3 = 18.84 \frac{rad}{s}$$

Amplitude distribution

$$\theta_{max}(x) = \theta_{tip}(x/L)^2 \quad (6)$$

We evaluate the points $x=0, x_{mid} = 0.10m, x_{tip} = 0.20m$

Root (x=0.00m)

$$\frac{x^2}{L^2} = \left(\frac{0.0}{0.20^2} \right) = 0$$

$$\theta_{max} = 0.523 \text{ rad} \times 0 = 0 = 0^\circ$$

Tip (x=0.20m)

$$\frac{x^2}{L^2} = \left(\frac{0.20^2}{0.20^2} \right) = 1$$

$$\theta_{max} = 0.523 \text{ rad} \times 1 = 0.523 \text{ rad} = 30^\circ$$

Mid (x=0.10m)

$$\frac{x^2}{L^2} = \left(\frac{0.10^2}{0.20^2} \right) = 0.25 \text{ rad} = 7.5^\circ$$

Angular velocity $\bar{\theta}(x,t)$

Formula

$$\bar{\theta}(x, t) = \omega \theta_{max}(x) \cos(\omega t) \quad (7)$$

Compute the peak angular velocity magnitude ($\cos \omega t = 1$)

$$\text{So} \quad \theta_{peak}(x) = \omega \theta_{max}(x) \quad (8)$$

Root

$$\theta_{peak}(0m) = 18.849 \times 0 = 0.0 \text{ rad/s}$$

Mid

$$(0.10m) = 18.849 \times 0.10 = 2.46 \frac{rad}{s}$$

Tip

$$\theta_{peak}(0.20m) = 18.849 \times 0.523 = 9.869 \frac{rad}{s}$$

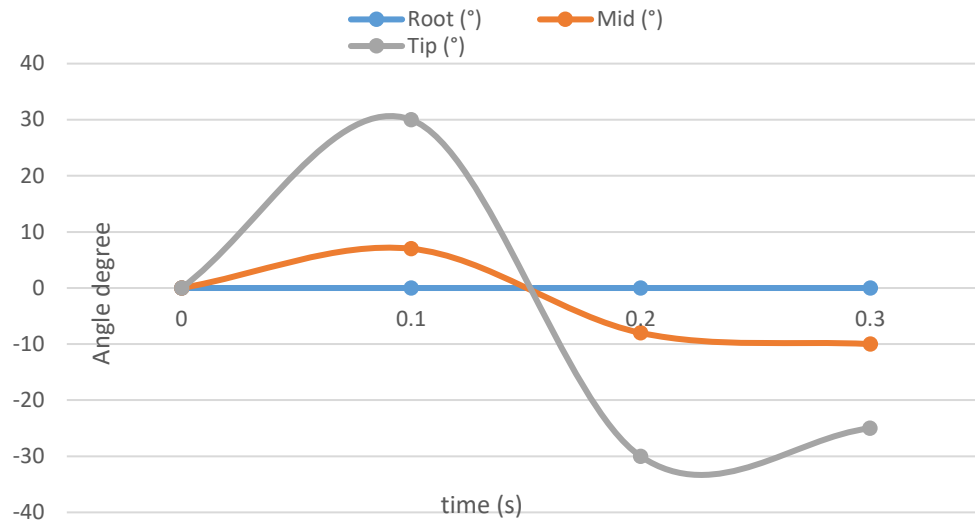


Figure 22. Motion of Fish Tail graph

2.17.2 Graph Explanation

The above graph figure 22 Angular Displacement θ vs Time graph, explains how much each section of the tail bends.

- The root of robot tail ($x = 0.00$ m) remains at 0° , showing no angular displacement.
- The midpoint of robot tail ($x = 0.10$ m) oscillates between about $+7^\circ$ and -10° , showing moderate bending.
- The tip robot tail ($x = 0.20$ m) has the largest displacement, oscillating between about $+30^\circ$ and -30° .

This behavior shows that the bending increases from the root to the tip of our robot tail, forming a wave-like motion along the tail. The tip point end point contributes the most angular displacement, while the root remains nearly fixed, similar to how real fish tails bend during swimming.

2.18 Simulation under 2 m/s velocity

2.18.1 Pressure in Y direction

We performed simulation five times with fluid velocities ranging from 1 m/s to 5 m/s. The results in figure 23 shows below are based on the velocity of 2 m/s. The below results shown are based on the velocity of 2 m/s. At a fluid velocity of 2 m/s, the pressure variation around the underwater robot is more significant. The front side shows a strong pressure rise due to higher impact which is around 6.3 to 4.5 Kpa while the sides experience a sharper pressure drop as the fluid accelerates around the body. The contour lines indicate stronger gradients yellow and green 3.7 kPa to 4.0 kPa, meaning the flow is more energetic and creates higher drag forces compared to lower velocity. In figure 3.02 Velocity = 1 m/s) at 1 m/s velocity the pressure distribution was smoother and less intense. The nose still experiences the highest pressure, but the differences between both 1m/s and 2m/s velocity high and low pressure regions are smaller. The flow is also more stable with reduced turbulence near the tail, resulting in lower drag compared to the 2 m/s case.

2.18.2 Pressure X direction

The figure 23 (a) presents the pressure distribution around the underwater robot at a velocity of 2 m/s. The front nose experiences the highest pressure due to direct fluid impact, while the upper side shows lower pressure regions range 3.0 kPa and the lower side shows relatively higher pressure showing in scale 4.5 kPa. This demonstrates how the incoming flow creates a strong pressure gradient across the body.

Pressure Data (from range):

- Minimum Pressure = -3.326×10^{-2} kPa
- Maximum Pressure = 6.35 kPa

2.18.3 Pressure in Z direction

The figure 23 (c) shows the result of a robot fish CFD simulation performed in SimScale. The cutting plane displays the pressure distribution around the model. The pressure scale at the middle section indicates that the values range from approximately 2.3 to 5 kPa. This means that certain regions around the fish experience normal pressure while other areas. Such a distribution helps in analyzing the hydrodynamic behavior of the prototype, including drag forces and flow stability.

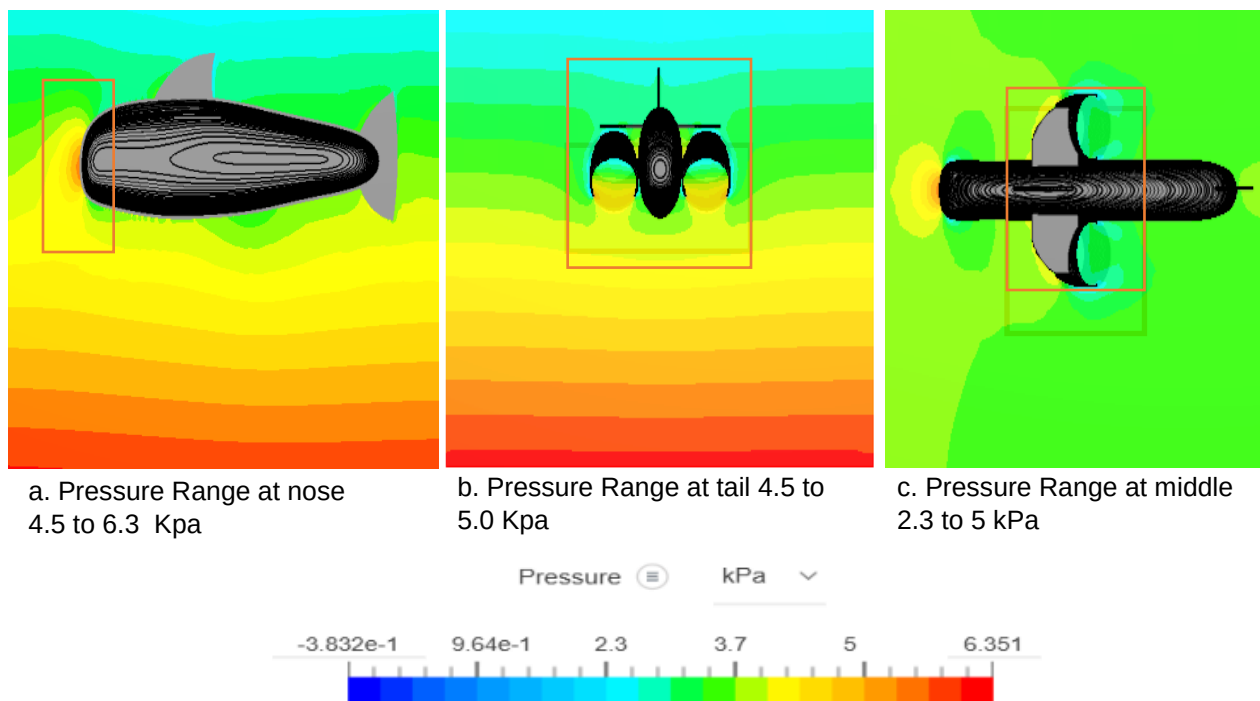


Figure 23. Pressure range distribution

2.18.4 Velocity X direction

The below simulation figure 24 (c) the contour plot shown the x-direction velocity distribution around our underwater robot in a CFD simulation. The color scale indicates the changes in fluid velocity, where blue color shown to negative values (flow moving in the opposite or leftward direction), and yellow to red shades show higher positive velocities range from 2.84×10^{-1} to 8.0×10^{-1} flow moving forward. The black central region shown the main body of our robot, The fluid flow pattern around the body shows clear velocity gradients, with the fluid velocity along the sides and scale range 2.8×10^{-1} . This indicates the presence of drag force and flow separation caused by the robot's geometry. The nearly color gradient along the outer boundary suggests a stable and consistent inlet flow. Overall, the slop demonstrates how the underwater robot disturbs the surrounding flow field, creating zones of high velocity around its sides.

2.18.5 Velocity Y direction

The figure 24 (a) simulation plot represents the Velocity m/s distribution around the underwater robot in a CFD analysis. The color scale at the below ranges from -1.936 m/s to +1.4 m/s shown the changes of the flow in the vertical (Y) direction.

- The dark blue part velocity -1.389 m/s represent the strong downward flow of water. The light blue to green regions velocity range -0.8 m/s to around 1.6 show normal downward or nearly neutral velocity.
- The yellow to red part of gradient velocity is +0.8 m/s to +1.3 m/s correspond to upward flow motion.

The gray shape in the center is the body of the underwater robot, where velocity equals 0 m/s due to the solid no-slip boundary condition. The contour lines near the body display the gradient of vertical velocity as the flow moves around it. The figure indicates that the robot's geometry induces both upward and downward velocity components in the surrounding flow field, with a maximum variation between -1.389 m/s and +1.4 m/s, representing the vertical fluid motion caused by its hydrodynamic interaction.

2.18.6 Velocity direction Z

The following simulation results figure 24 (b) shows the Velocity Z (m/s) contour of the underwater robot, shows the sideward flow in the CFD simulation. The color scale ranges from of velocity starts from -1.936 m/s (blue) to +1.559 m/s (red). The green area stable stable regions with nearly zero side velocity, meaning the flow remains mostly straight and uniform. The blue and red regions near the robot's sides show small lateral movements caused by the body shape and fin interaction with the fluid. The black contour lines highlight zones of slight sideward flow separation and vortex formation. The maximum fluid velocity is about +1.559 m/s, and the minimum is -1.936 m/s, indicating small side flow disturbances. The overall flow is symmetric and stable, showing that the underwater robot experiences very little lateral drift and maintains good directional stability in water...

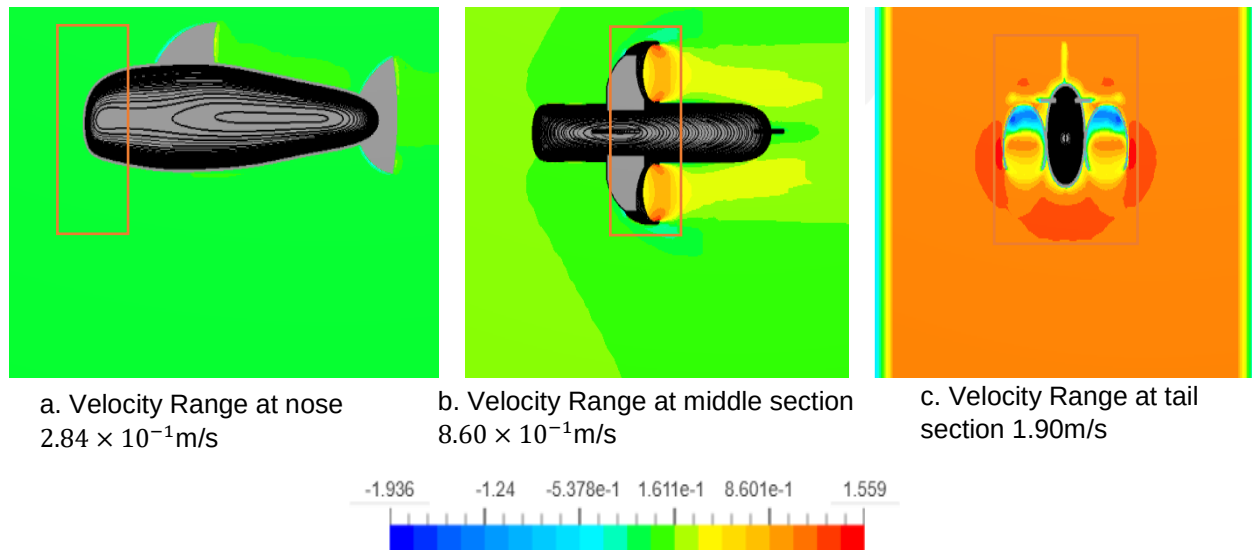


Figure 24. Velocity distribution around robot surface , a. Velocity Range at nose 2.84×10^{-1} m/s, b. Velocity Range at middle section 8.60×10^{-1} m/s, c. Velocity Range at tail section 1.90 m/s

Figure 25 the Velocity m/s distribution along a path passing through the underwater robot how velocity changes with distance along the path. CFD simulation. The contour plot highlights flow behavior while the graph below shows velocity changes with distance along the path. The velocity starts around 2 m/s at the inlet and gradually decreases to nearly 1 m/s near the robot's surface due to flow obstruction. After passing the body, the velocity slightly recovers as the flow reattaches downstream. Overall, the result shows a smooth velocity drop caused by drag and wake formation behind the underwater robot.

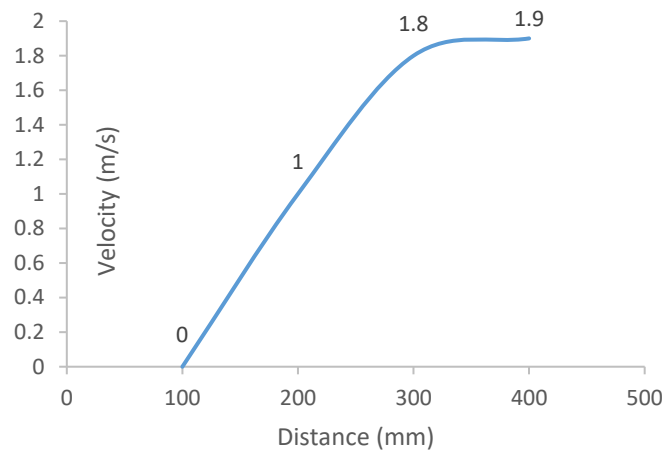


Figure 25. Velocity distribution around robot surface

2.18.7 Simulation Acceleration Analysis at the velocity of 2m/s

The velocity time figure 26 obtained from the CFD simulation shows the acceleration behavior of our underwater robot during its motion in the water. In this observation the total time interval was 1.5 seconds. The graph indicate that the robot starts from nearly zero velocity and slowly accelerates as the propulsion system generates thrust. According to the simulation observations the velocity increases continuously with time until it reaches a maximum value of velocity approximately 1.9 m/s. but the inlet flow condition of the simulation defined as 2 m/s the robot velocity reaches about 1.9 m/s due to the hydrodynamic resistant between the robot body and the the water. The slow increase in velocity show that the robot experiences an acceleration where thrust overcomes the hydrodynamic drag forces acting on the body of our robot. The slope of the velocity time curve represents the acceleration of the our underwater robot. As shown in the graph the velocity increases rapidly during the initial motion and then slowly balance near the maximum value this behavior show that the propulsion system provides enough thrust to accelerate the robot while the fluid resistance increases with velocity.

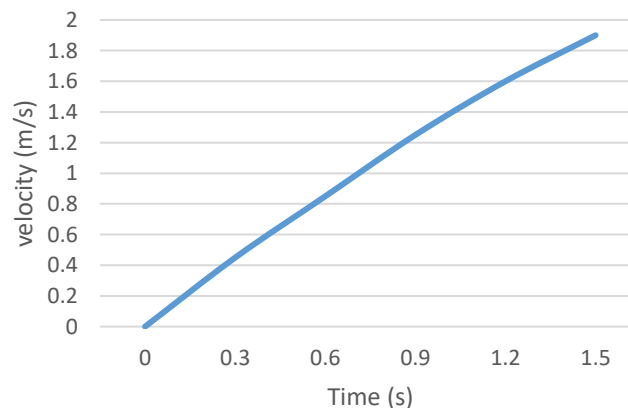


Figure 26.Simulation Acceleration Analysis at the velocity of 2m/s

The acceleration can be calculate using the basic relation:

$$a = \frac{\Delta V}{\Delta t} \quad (9)$$

$$a = \frac{1.9 - 0}{1.5}$$

$$a = 1.27 \text{ m/s}^2$$

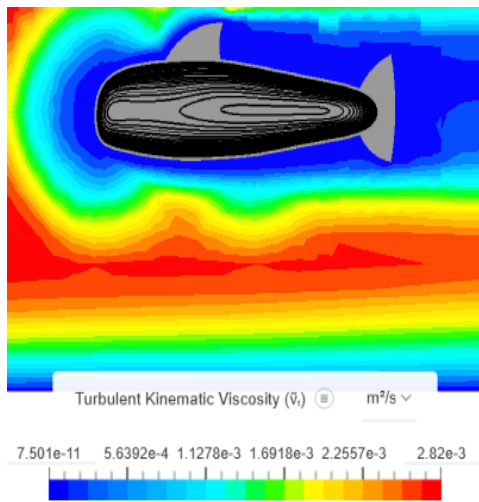
Therefore, the average acceleration of the underwater robot obtained from the simulation is approximately 1.27 m/s^2 .

2.18.7 Turbulent Kinematic Viscosity (V_t)

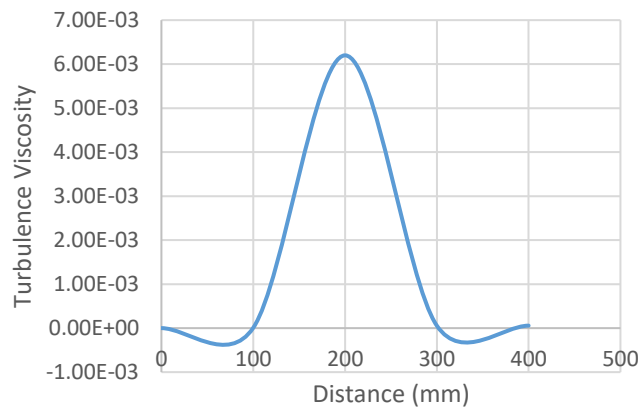
The following analysis figure 27 shows the Turbulent Kinematic Viscosity (V_t) of our robot in a CFD (Computational Fluid Dynamics) simulation at the velocity of 2 m/s. The color slope at the below represents the range of turbulent viscosity values, which from $7.50 \times 10^{-11} \text{ m}^2/\text{s}$ (blue) to $2.82 \times 10^{-3} \text{ m}^2/\text{s}$ (red).

- The red portion near the robot's nose and upper surfaces show areas of high turbulence generation, it is caused by strong interaction between the flow and the robot's body.
- The blue areas of colors back side of the robot show low turbulent viscosity, representing the wake region where the flow becomes more stable around our robot.

In summary, the result state that turbulence is strongest at the edges of the robot and decreases toward the backside which is typical for streamlined underwater bodies. This helps in understanding drag force and flow distribution behavior for current design and performance.



a. Turbulent Kinematic Viscosity 7.5×10^{-11} to $5.6 \times 10^{-4} \text{ m}^2/\text{s}$



b. Turbulent Kinematic Viscosity graph

Figure 27. Simulation Acceleration Analysis at the velocity of 2m/s, Turbulent Kinematic Viscosity 7.5×10^{-11} to $5.6 \times 10^{-4} \text{ m}^2/\text{s}$, **b.** Turbulent Kinematic Viscosity graph

Figure 27 (b) shows the variation of turbulent kinematic viscosity along a selected path in the flow field. The Y-axis represents the turbulent kinematic viscosity (v_t in m^2/s), and the X-axis shows the distance along the path (in meters). The line decreases gradually from about $0.003 \text{ m}^2/\text{s}$ to near zero, indicating that turbulence intensity reduces along the path likely moving away from the high-turbulence region near the robot's surface into a more stable flow region downstream.

2.19 Pressure Distribution under the velocity of 3 m/s

2.19.1 Pressure distributed X direction

CFD pressure, velocity, and turbulence analyzed at the velocity of 3 m/s. The figure 28 (a) represents the pressure distribution around the underwater robot during fluid flow analysis in SimScale. The pressure varies from -5.231 kPa to +8.584 kPa, where the front region of the robot experiences the highest pressure due to the direct impact which is around 5.8 kPa to 8.5 kPa of the approaching fluid as the water flows along the surface and moves toward the rear, the pressure gradually decreases reaching low-pressure zones at the tail pressure approach 3.0 to 3.5 kPa. These pressure regions as low as 5.2 kPa are formed due to flow separation and turbulence behind the robot. The difference between the high pressure at the front and the low pressure at the back contributes to the drag force, which resists the robot's motion. This pressure pattern indicates a balanced yet realistic hydrodynamic performance of the robot's design under simulated underwater flow conditions.

2.19.2 Pressure distributed Y direction

The following CFD analysis figure 28 simulation in SimScale, show the pressure distribution around the underwater robot. The pressure field measured in kilopascals (kPa) its ranges varies from -5.231 kPa dark blue to 8.584 kPa red. The areas of lower pressure light blue on the sides of the thruster of robot which is 3.0 kPa signify the regions where static pressure is normal and fluid flow is faster. Conversely the areas of higher pressure yellow, are situated at the thruster of the robot the pressure range 4.5 kPa. The zones of flow that are relatively stable around the structure are represented by the green and yellow areas 3.0 kPa to 4.5 kPa which indicate medium pressure. In light of this, the pressure contour reveals the underwater robot's imposition on the surrounding fluid and provides valuable data on its hydrodynamic behavior, like buoyancy, drag, and stability.

2.19.3 Pressure distributed Z direction

The below underwater robot's pressure color plot got from a CFD simulation in SimScale figure 28 (a). The pressure analysis ranges from -5.231 kPa dark blue to 8.584 kPa red. The robot's front nose is where the water collides with the surface and creates a stagnation zone, thus, the high-pressure regions yellow to red are noticeably distributed in that area which is 5.0 kPa to 8.5 kPa. The middle of the robot is low-pressure regions green 3.1 kPa which means that there are flow separations and reduced static pressure regions due to the formation of the wake, according to the yellow area around it that indicates the free-stream water pressure, which is showing a uniform flow pattern far from the robot. In general, the pressure contour indicates the interaction of the underwater robot with surrounding fluid, thus, making it possible to identify drag zones, flow behavior, and pressure balance which are critical for optimizing hydrodynamic performance.

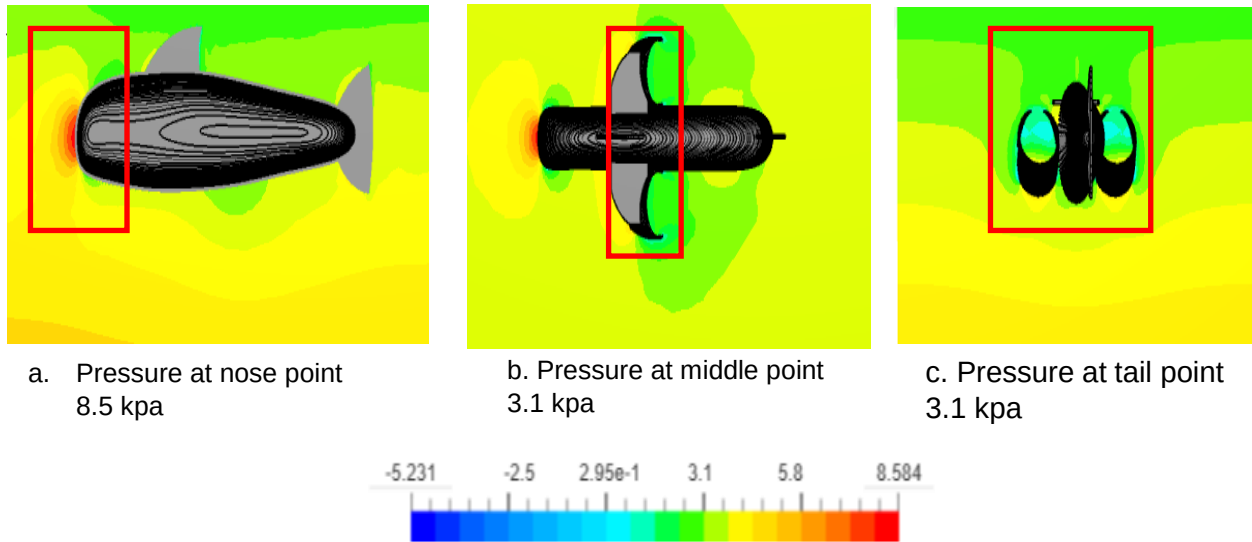


Figure 28. Pressure distribution ,(a) Pressure at nose point 8.5 kpa, (b)Pressure at middle point 3.1 kpa (c) Pressure at tail point 3.1 kpa

The figure 29 shows a pressure variation plot along a defined path across the underwater robot, obtained from a CFD simulation in SimScale. The selected field variable is Pressure (kPa), measured between two points $(-0.2, 0, -3e-3)$ and $(2.3e-1, 0, 1.2e-3)$. The pressure decreases gradually from approximately 6.8 kPa at the starting point to around 4.0 kPa at the end of the path, as shown in the plot-over-path graph below. this downward trend indicates a pressure drop along the flow direction, caused by the acceleration of water around the robot body and the corresponding reduction in static pressure. The analysis helps in understanding the pressure gradient and hydrodynamic flow behavior acting on the robot's surface, which is crucial for evaluating drag and stability performance.

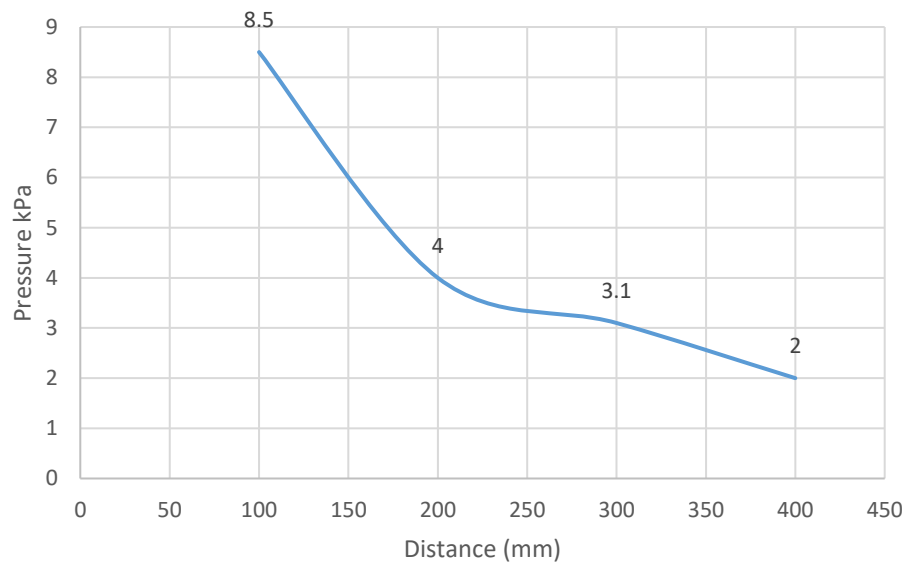


Figure 29. Pressure distribution

2.20 Velocity Distribution under the velocity of 3 m/s

The following figures present the side-based velocity distribution around the underwater robot using Velocity-Y, Velocity-Z, and Velocity contours. These views provide a detailed understanding of how water flow behaves at critical regions of the robot particularly at the nose, thruster section, and tail. Velocity values are expressed in meters per second (m/s).

2.20.1 Velocity analysis Y side

In the Velocity-Y view figure 30 (a), the flow around the nose of the robot appears mostly green, indicating relatively low velocity in the range of approximately -4.11×10^{-1} m/s to 4.28×10^{-1} m/s. This shows that the incoming water slows down significantly at the front due to direct interaction with the robot body. Moving toward the tail region, a light yellow wake can be observed, where velocity increases slightly to around 1.27×10^0 m/s. This increase indicates flow acceleration after passing the body, which is a typical wake behavior and suggests smooth downstream recovery of the flow.

2.20.2 Velocity analysis Z side

The Velocity-Z view figure 30 (b) clearly highlights the effect of the thrusters and mid-section geometry on the flow. Over by the thrusters, patches of yellow and orange stand out - these match speeds around 8.07×10^{-1} m/s up to 1.30×10^0 m/s. Because of the way the thruster housings sit and block airflow, turbulence builds, pushing velocities higher. Move to the nose tip, color shifts; now green dominates, near 2.43×10^{-1} m/s. That region resists sudden changes, hinting the water arrived undisturbed. Work your way back past the thrusters, speeds drop once more, shifting once more into green tones - this hints the flow regains calmness beyond the thruster zone.

2.20.3 Velocity analysis -X View

The Velocity-X view figure 30 (c) represents the primary flow direction and shows the strongest velocity variations. Blue-green stains mark where speed drops near the nose, ranging from minus six point six six billionth meters per second up to three point six zero billionth. That slowdown points to slowed airflow hitting head-on. On either side of the engines, color shifts toward orange and red - signs of faster movement. Speeds there jump into the tens of meters per second, between one point seven and two point seven billionth. That push suggests turbulence, sparked by how the thrusters angle and extend. Behind the robot, speed climbs fast - hitting about 3.74×10^0 m/s, which marks a quick surge in motion near the tail. That rise hints at how well momentum moves and how smoothly it leaves the area. Looking at velocities from the side, one thing stands out: the slowest movement happens right at the front, where traffic first meets the shape and begins slowing down. Close to the thrusters, changes in speed stand out the most because water moves and speeds up. Flow meets there in a way that mixes and pushes strongly. Acceleration plays its part here too. Moving further back, the last part shows higher speed behind. Then things level off - this hint suggests the wake has taken shape properly. Performance seems balanced because water moves as expected without sudden shifts.

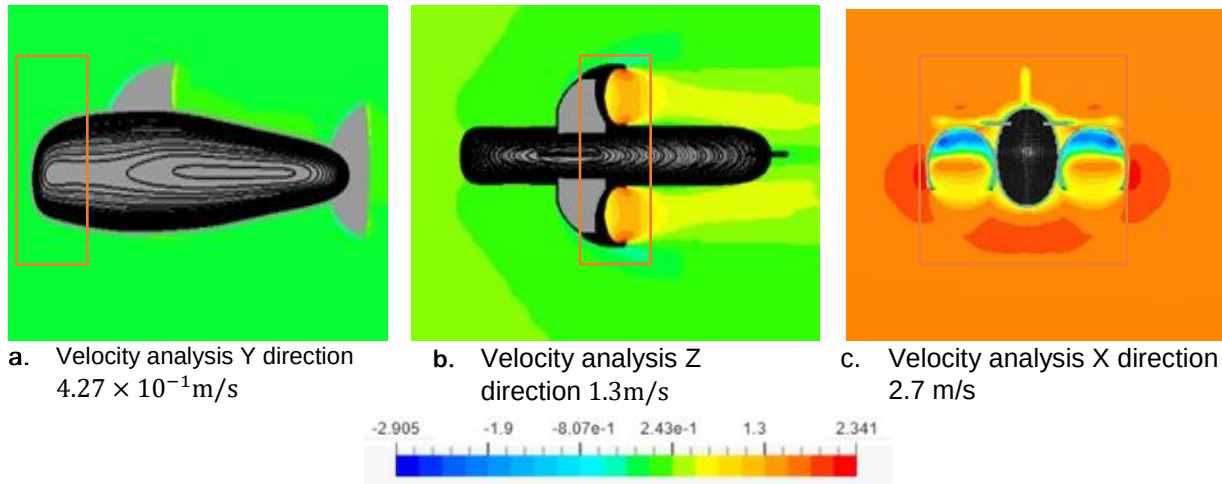


Figure 30. CFD velocity Analysis around robot surface

2.20.4 Graph Explanation

The following figure 31 slope shows the change of Velocity X (m/s) during a specified path in our underwater robot flow area. The fluid flow increases slowly from scale 2.14 m/s to close to 2.32 m/s over a distance of about 0.40 meters, where the total flow path lasts about 0.40 meters. Such a minor increase in velocity indicates that the fluid moved through the way extremely slowly, very gradually and very smoothly; hence it can be concluded that the flow in that sector was steady and laminar. The results indicate that the flow characteristics are so close to the surface of the robot that they are uniform and static, as no sudden changes or vortices were detected during the process of the robot moving to-and-fro along the designated path.

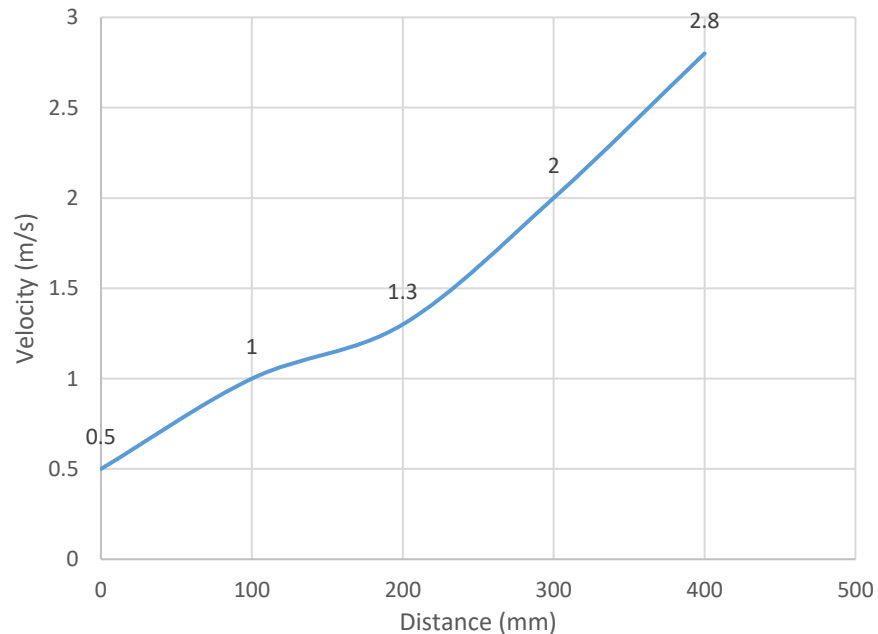


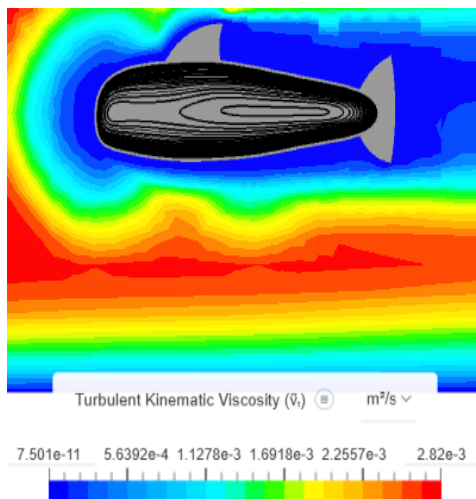
Figure 31. Velocity analysis graph

2.21 Turbulence Distribution under the velocity of 3 m/s

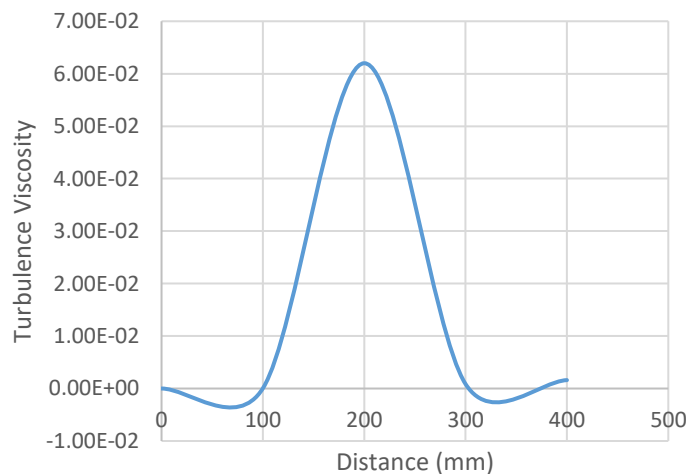
2.21.1 Turbulence analysis Y view

The figure 32 (a) the turbulent kinematic viscosity contour reveals how fluid moves near the robot's nose underwater. This pattern covers movement shaped by shape and direction of force. Values stretch from nearly zero at $3.09 \times 10^{-10} \text{ m}^2/\text{s}$ (dark blue) up to $4.23 \times 10^{-3} \text{ m}^2/\text{s}$ (red). Along the curved tip where water hits directly, values stay between $1.69 \times 10^{-3} \text{ m}^2/\text{s}$ and $3.33 \times 10^{-3} \text{ m}^2/\text{s}$. That narrow band shows strong mixing caused by paused flow - the kind that happens when currents stop abruptly on impact. Flow near the front experiences high turbulence, which grows less along the surface.

Figure 30 (a) the turbulent flow distribution around the thruster region. The overall From place to place, the whole mix of movement and mixing changes quite a bit. It moves slowly across the system, sitting between $3.09 \times 10^{-10} \text{ m}^2/\text{s}$ and $4.23 \times 10^{-3} \text{ m}^2/\text{s}$. Over the thruster parts and close by, things get more active. There, values climb up, mostly between $2.54 \times 10^{-3} \text{ m}^2/\text{s}$ and $4.23 \times 10^{-3} \text{ m}^2/\text{s}$. What drives this? It's how the flow twists and collides when spinning, splitting, and colliding in trails behind the thruster each twist draining more motion away. the contour presents the turbulent kinematic viscosity near the tail of the robot. From lowest to highest, the range of turbulent kinematic viscosity covers $3.09 \times 10^{-10} \text{ m}^2/\text{s}$ up to $4.23 \times 10^{-3} \text{ m}^2/\text{s}$. Over on the tail side, that same measure stays relatively low, situated between $8.46 \times 10^{-4} \text{ m}^2/\text{s}$ and $2.54 \times 10^{-3} \text{ m}^2/\text{s}$. Behind the robot, swirling currents play a key role - wake patterns and re-circulating flows take shape here. Because of these shifts, keeping steady direction and balance becomes harder under certain conditions.



a. Turbulence analysis simulation view



a. Turbulence analysis graphi

Figure 32. Turbulence Distribution under the velocity of 3 m/s

2.21.4 Acceleration analysis under 3 m/s of robot velocity

Figure 33 Under the motion of 3 m/s the underwater robot the velocity changes slowly as the robot moves forward in the water during the 1 second interval the velocity increases from 0 m/s to approximately 2.8 m/s due to the thrust generated by the propulsion system. This gradual increase in velocity indicates that the robot experiences continuous acceleration while moving through the fluid environment. The change in velocity at different time intervals and the corresponding acceleration values are shown in the table below.

Acceleration is calculated using:

$$a = \frac{\Delta V}{t} \quad (10)$$

Since the total time is 1 seconds and the velocity values are divided into 6 intervals, the time step between each point is approximately 0.5 s.

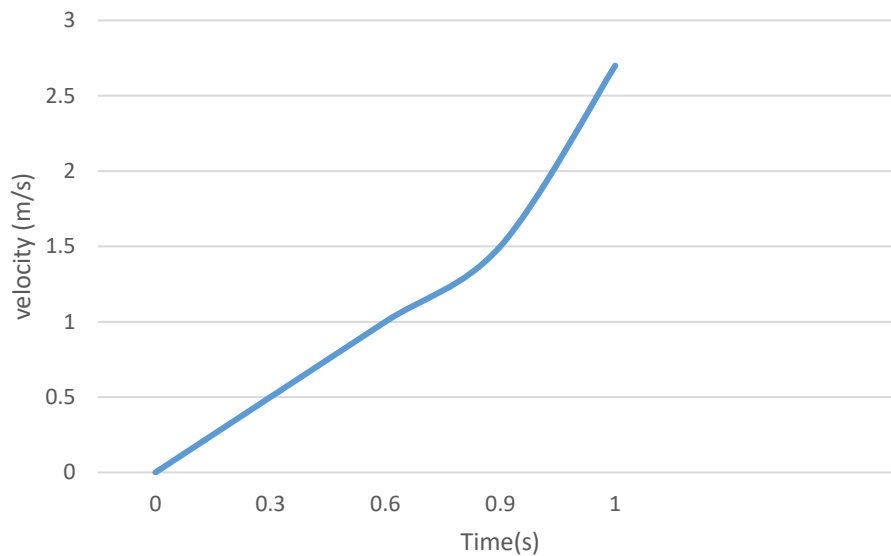


Figure 33. Acceleration analysis under 3 m/s of robot velocity

2.22 Pressure Distribution under the velocity of 4 m/s

2.22.1 Pressure analysis Y direction

CFD pressure, velocity, and turbulence analyzed at the velocity 4 m/s In the figure 34 a simulation shows a clear and realistic representation of the pressure distribution around the underwater robot. The pressure ranges from -21.23 kPa to 17.53 kPa, with the highest pressure concentrated at the front nose and the lowest pressure forming at the rear wake region range from 17.53 kPa to 2.0 kPa. This pattern is expected because the front of the robot directly faces the incoming fluid flow, creating stagnation points where water pressure builds up. The high pressure area at the front nose of robot 17.53 kPa can significantly affect the robot's hydrodynamic performance. But some pressure is necessary for stability and forward motion of robot high pressure increases form drag which require more propulsion power and reduces energy efficiency. Additionally if not properly balanced by the shape and buoyancy distribution, high pressure can cause structural stress on the robot's front body. Overall, this result is realistic and valuable it highlights how the robot's streamlined design successfully channels the pressure smoothly along its body, minimizing turbulent separation and drag at the rear.

2.22.2 Pressure analysis X direction

The below simulation figure figure 34 (b) shows the pressure distribution around the underwater robot during the interaction with the fluid. The pressure shows a variation of -21.23 kPa and 17.53 kPa, which covers all the low and high pressure regions along the entire body of the robot. The robot's front and side parts are marked as high-pressure zones in yellow to red shades 1.0 kPa to 9.88 kPa due to the direct impact of the water flow on the surface. This is where the fluid is decelerated and compressed hence, a stagnation point is formed and higher pressure is produced. The high-pressure effect on the robot middle section which reaches 9.8 kPa it leads to a rise in hydrodynamic drag, thus the robot will require more power for maintaining the forward motion. If the pressure is not properly controlled, then over time, continuous exposure to high-pressure zones may lead to the structural stress of the body or joints.

2.22.3 Pressure analysis Z direction

This SimScale pressure contour in figure 34 (c) shows the underwater robot experiencing pressure values ranging from -21.23 kPa to 17.53 kPa. The front nose of robot region experiences the highest pressure its around 17.5 kPa due to direct impact with the incoming fluid while the back wake region shows low pressure zones it as low as 3.0 kPa caused by flow separation. The high-pressure force at the front slightly increases drag force, which can reduce propulsion efficiency and cause minor structural loading over time. The overall flow pattern remains stable, indicating that the robot's streamlined design effectively reduces turbulence. However, optimizing the front curvature or fin geometry could help minimize high-pressure buildup and further enhance hydrodynamic performance.

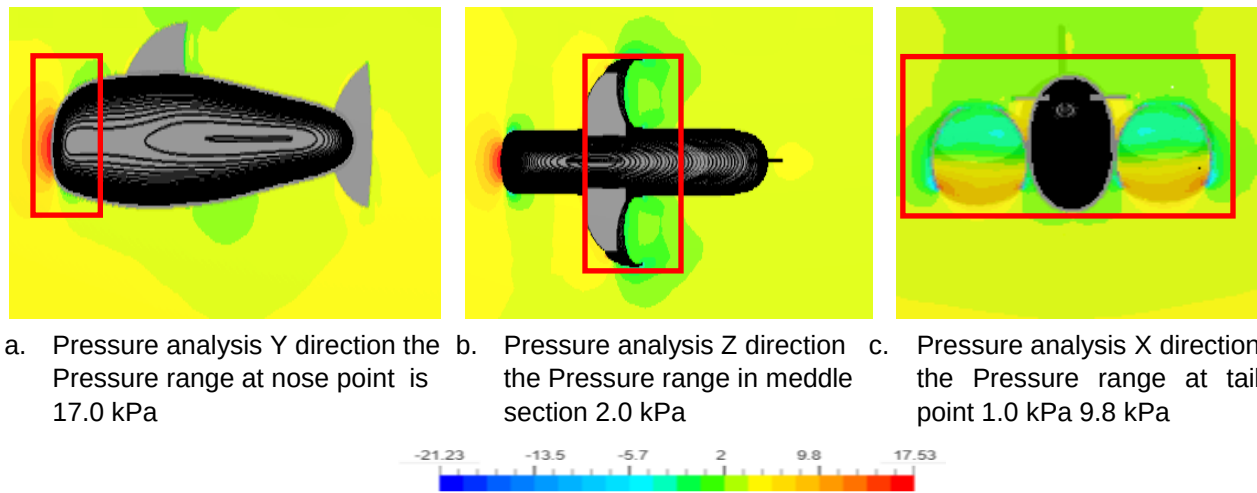


Figure 34. Pressure distribution around robot surface

2.23 Velocity Distribution under the velocity of 4 m/s

2.23.1 Velocity contour (Y-direction)

This following simulation figure 35 (a) represents the velocity contour Y direction around the underwater robot obtained from a CFD simulation in SimScale at velocity of 4 m/s. The velocity distribution ranges between -3.536 m/s and +3.532 m/s indicating the variations in flow movement due to the robot's hydrodynamic interaction with the surrounding fluid. From the analysis, the front region of the robot shows low velocity zones 7.05×10^{-1} m/s caused by flow stagnation as the fluid directly impacts the robot's nose. The sides and rear regions display higher velocity gradients which is around 1 to 1.5 m/s particularly near the fins where flow separation and acceleration occur.

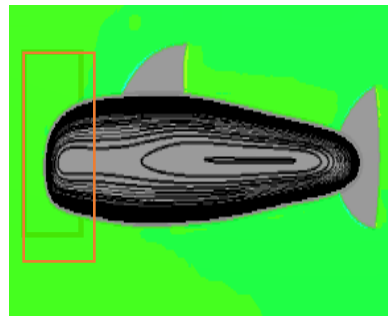
2.23.2 Velocity contour X-direction

The below results figure 35 (b) depicts the underwater robot's velocity X direction contour that has been analyzed using Computational Fluid Dynamics (CFD) in SimScale. The range of velocity distribution is between -3.536 m/s and +3.532 m/s, which directs our attention to the water that flows around the robot from a X view. The results depict a little non symmetrical velocity pattern on the robot's left and right sides on the left thruster the maximum velocity is 1.0 to 2.1 m/s and left thruster its around 1.0 m/s which attests to a hydrodynamic design that is may be it cause little unbalanced. Areas in blue and red that are close to the fins signify opposite directions of flow, which means that there are small vortices or lateral flow deviations caused by fin movement. The central body region is mostly colored green which corresponds to a low lateral velocity and implies a stable and streamlined flow with very little side forces.

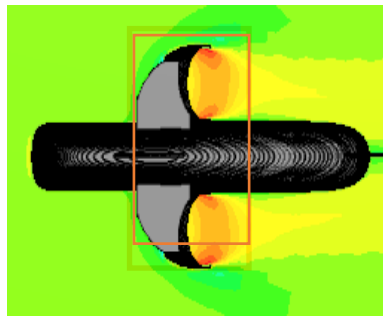
2.23.3 Velocity contour (Z-direction)

The below figure 35 (c) represents the velocity contour Z-direction around the underwater robot simulated in SimScale using Computational Fluid Dynamics (CFD). The velocity range varies from -5.355 m/s to +3.765 m/s, visualized through a color gradient from blue (low or reverse flow) to red high forward velocity. The contour shows how the water accelerates and decelerates as it flows past the robot. The green and yellow regions around the body indicate areas of uniform flow, while the red zones at the rear suggest increased velocity caused by flow convergence and jet-like acceleration. The blue areas near the sides and front indicate flow separation or low-velocity zones due to pressure buildup at the leading edges.

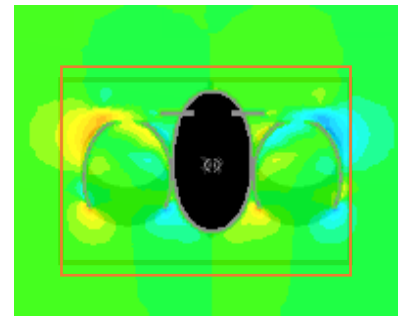
From a scientific perspective, this pattern demonstrates the pressure-velocity relationship defined by Bernoulli's principle. As velocity increases behind the robot, static pressure decreases, leading to a low-pressure wake zone. Conversely, high-pressure areas form at the front where velocity drops. This high-pressure gradient shows that the robot body faces strong hydrodynamic resistance at the nose but maintains smooth flow along its length, proving its shape is aerodynamically and hydro dynamically optimized for underwater operation.



a. Velocity analysis y direction
the velocity range at noes
 7.09×10^{-1}



b. Velocity analysis range at
middle point 1.5 to 3.0 m/s



c. Velocity analysis X direction
the velocity range at tail point
 -7.09×10^{-1} to 2.1 m/s



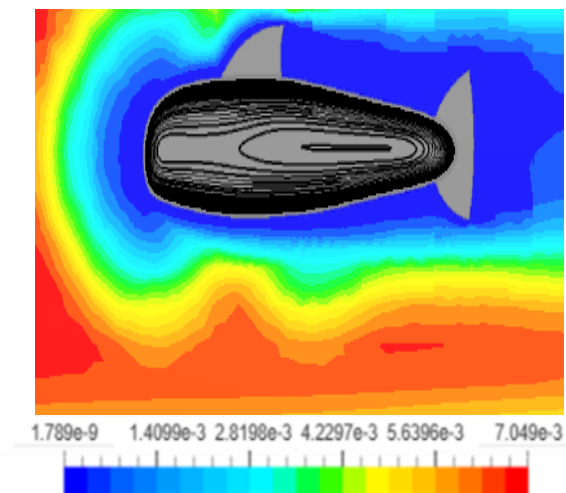
Figure 35. Velocity distribution around robot surface

2.24 Turbulence Distribution under the velocity of 4 m/s

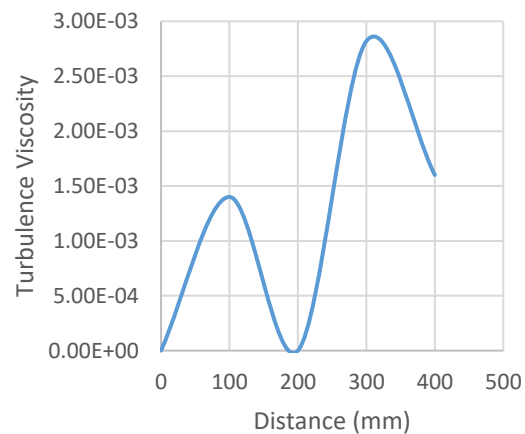
2.24.1 Turbulent kinematic viscosity

The below visual figure 36 represent depicts the phases of turbulent kinematic viscosity around the submersible robot as it was simulated in SimScale. The color scale from 1.789×10^{-9} to $7.049 \times 10^{-3} \text{ m}^2/\text{s}$ shows the changes of turbulence intensity throughout the flow field. The red and orange regions represent areas of high turbulence intensity, whereas the blue regions symbolize slower and more laminar flow behavior. Around the robot surface, particularly along the streamlined body, the turbulent kinematic viscosity is observed mainly in the lower range of approximately 1.40×10^{-9} to $2.82 \times 10^{-3} \text{ m}^2/\text{s}$, which is indicated by blue to light-green contours. This shows that the flow remains relatively smooth and stable around most of the robot's body, reducing excessive turbulence near the surface.

The high-turbulence areas are mainly found at the robot's nose and in the wake region, where the water flow undergoes strong changes in velocity and pressure. The turbulent layers are formed due to boundary layer separation and the release of vorticity behind the body, which is characteristic of streamlined underwater vehicles. The front part of the robot exhibits comparatively lower turbulence due to smooth fluid attachment, while the rear section forms stronger eddies as a result of flow detachment and mixing. From a scientific perspective, this behavior is a positive indicator for the robot's hydrodynamic design, as reduced turbulence at the front leads to lower drag and improved flow stability. Although the wake region shows increased turbulent viscosity common in bluff body flows the overall trend suggests that the robot experiences moderate hydrodynamic resistance with efficient energy transfer and sufficiently stable flow characteristics for underwater navigation.



a. Turbulent kinematic viscosity range 1.78×10^{-9}



b. Turbulent kinematic viscosity graphic view

Figure 36. Turbulent kinematic viscosity range 1.78×10^{-9}

2.25 Pressure Distribution under the velocity of 5 m/s

2.25.1 Pressure Analysis Y direction

CFD pressure, velocity, and turbulence analyzed at velocity of 5 m/s. This below figure 37 (a) shows how pressure is distributed around the underwater robot, scale from -32.17 kPa to 23.63 kPa. The red and yellow areas show high pressure zone at the front robot nose which is 23.63 kPa where water hits the surface head on, causing high pressure. The blue area at the back shows a low-pressure 1.31 kPa wake zone due to the flow separating and vortices forming. The pressure smoothly changes from high to low along the body 23.63 kPa to 1.31 kPa, suggesting the robot is well-streamlined. This reduces drag and improves how it moves underwater. The pressure change around the robot shows that the flow reattaches in a stable manner, cutting energy losses and keeping the robot stable as it moves forward. The pressure pattern shows the design handles pressure well, giving the robot better propulsion and less resistance when moving underwater.

2.25.2 Pressure Analysis Z direction

The below visual figure 37 (b) represents a CFD Computational Fluid Dynamics pressure contour of our underwater robot moving through water. The color scale at the bottom ranging from -32.17 kPa blue to +23.63 kPa red shows the pressure distribution around the body. The front of the robot experiences high-pressure red 23.63 kPa region due to water resistance, while the middle section shows a low-pressure zone blue region 1.0 kPa to 1.31 kPa indicating flow separation and drag formation. The green and yellow areas around the fins display moderate pressure 1.31 kPa to 10.5 kPa variations, suggesting that these surfaces play a role in stabilizing the flow. Overall, the pressure pattern demonstrates typical hydrodynamic behavior, where high pressure at the nose and low pressure at the tail contribute to thrust resistance and affect the vehicle's overall stability and efficiency.

2.25.3 Pressure Analysis X direction

This below gradient figure 37 (c) shows a front view of an underwater robot's pressure levels 23.63 kPa, examined with CFD simulation in SimScale. The below Pressure ranges from -32.17 kPa blue to +23.63 kPa red. Areas show high pressure red and orange are on each side of the robot's front and tail meaning these front experience high pressure 23.63 kPa direct water resistance when moving forward. The robot's middle has moderate pressure green and yellow 1.31 kPa to 12.5 kPa which suggests smoother water flow around the hull. Low-pressure areas blue around the edges suggest faster water flow and possibly vortices. In general, the pressure shows the robot's design splits water flow evenly which helps with balance, lowers drag, and keeps it steady underwater Movement.

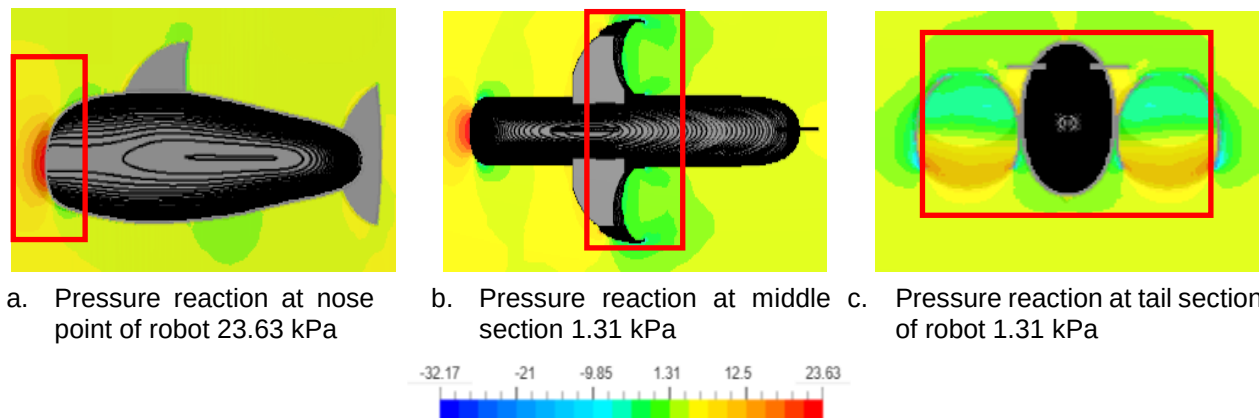


Figure 37. Pressure distribution around robot surface

2.26 Velocity Distribution under the velocity of 5 m/s

2.26.1 Velocity Analysis Y Direction

The below is an analysis figure 38 (a) of the water speed surrounding an underwater robot based on a SimScale simulation. The graphic shows different colors to represent water velocity in the Y direction. In the nose of robot region, blue colored zones indicate a velocity of approximately -4.244 m/s, showing reverse or downward flow caused by stagnation pressure at the leading edge of the robot. As the flow moves slightly away from the nose surface, the color transitions to cyan and light green color representing velocities between -2.5 m/s to -0.85 m/s, indicating gradual flow recovery. The green region around -0 to +0.85 m/s near the nose sides shows smooth and attached flow, shows that the robot's front geometry effectively distribute the water without strong separation. The absence of intense red regions at the nose indicates controlled acceleration which helps decrease pressure drag and ensures stable forward entry into the water.

2.26.2 Velocity Analysis Z Direction

The below analysis figure 38 (b) shows the velocity changes around the mid-section of the underwater robot in the Z direction. The velocity change in scale ranges from -4.244 m/s blue to +4.239 m/s red. Near the fins and mid-body surfaces, yellow to light red regions are visible represent to velocity values of approximately +2.5 m/s to +4.239 m/s showing accelerated lateral flow due to flow changes around the robot surfaces. Meanwhile, cyan and blue regions -2.5 m/s to -4.244 m/s show just downstream of the fins representing localized low velocity and recirculation zones caused by flow separation. The surrounding green region of underwater robot 0 m/s indicates stable far field flow with minimal disturbance. The symmetric distribution of these velocity contours on both sides confirms balanced lateral forces and effective hydrodynamic control at the mid-body region.

2.26.3 Velocity Analysis X Direction

The below simulation figure 38 (c) shows the underwater robot moving forward and shows the X-direction velocity distribution near the tail rear region. The velocity scale ranges in the below image from -2.737 m/s blue to +7.613 m/s red. Around the tail and immediate downstream region orange to red zones indicate high-speed flow ranging from +4.0 m/s to +5.0 m/s caused by flow acceleration and jet-like behavior as water exits the rear of the robot. Adjacent to the fins and wake core, blue and cyan regions -2.737 m/s to -0.667 m/s indicate low-velocity zones where water loses momentum after passing the robot, contributing to wake formation and drag. The presence of similar velocity patterns on both sides of the tail confirms symmetric wake development, indicating that the robot maintains directional stability and efficient thrust alignment during forward motion.

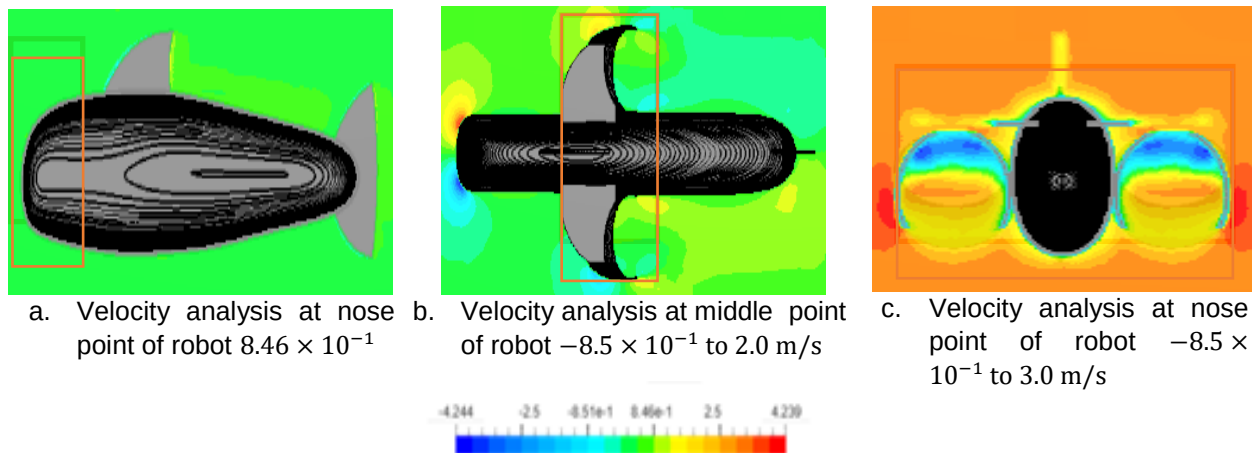


Figure 38. Velocity Analysis around robot surface

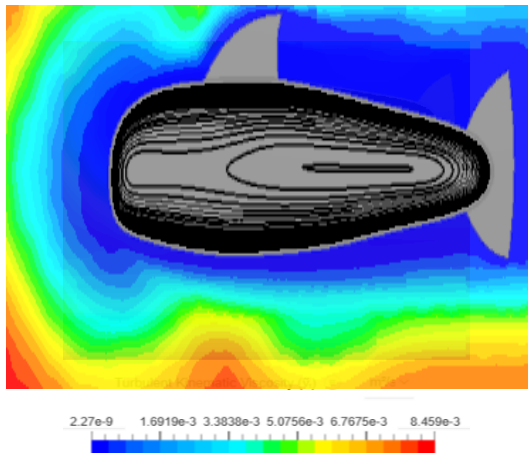
2.27 Turbulence Distribution under the velocity of 5 m/s

2.27.1 Turbulent kinematic viscosity

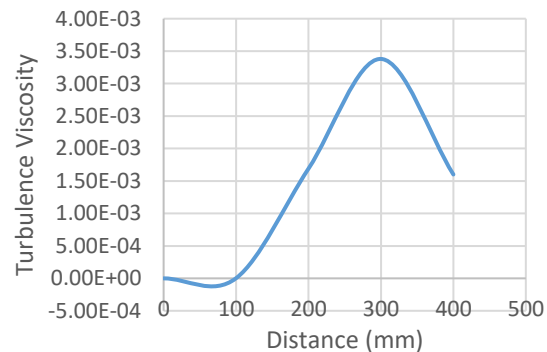
The below simulation figure 39 show that the turbulent kinematic viscosity distribution around the underwater robot obtained from the SimScale. The color contour represents changes in turbulent viscosity, indicating the intensity of turbulence in the surrounding water flow. The total turbulent kinematic viscosity range shown in the scale extends from approximately $2.27 \times 10^{-9} \text{ m}^2/\text{s}$ dark blue to $8.46 \times 10^{-3} \text{ m}^2/\text{s}$ red. The red and orange regions upstream and along the outer flow show high turbulence intensity, caused by strong velocity gradients and flow interaction with the robot geometry.

Around the nose and near-body region near to tail, a light blue to cyan color band is clearly visible, which represents normal turbulence levels. This light blue region corresponds to turbulent kinematic viscosity values in the approximate range of $1.69 \times 10^{-3} \text{ m}^2/\text{s}$ to $3.38 \times 10^{-3} \text{ m}^2/\text{s}$. These turbulence levels occur due to robot surface development and flow acceleration as water passes around the curved nose of the robot. The slow change from light blue to green 3.38×10^{-3} to 5.07×10^{-3} indicates controlled turbulence growth, showing that the robot's streamlined shape helps limit normal turbulent mixing while maintaining stable flow attachment.

Downstream of the robot, the color changes again toward blue shades 1.69×10^{-3} indicating a decrease in turbulent viscosity as the flow stabilizes in the wake region. The symmetric distribution of turbulent viscosity around the robot show the balanced hydrodynamic behavior and controlled turbulence generation, which is beneficial for reducing drag and improving overall underwater motion efficiency.



a. Turbulent kinematic viscosity around robot



b. graph of Turbulent kinematic viscosity around robot

Figure 39. Turbulent kinematic viscosity 1.6×10^{-3} , **(a)** Turbulent kinematic viscosity around robot **(b).** graph of Turbulent kinematic viscosity around robot

2.28 Electronic Components

2.28.1 Power Distribution Board (PDB)

For our underwater robot, we used a custom Power Distribution Board (PDB) rated at 40A. The primary function of this board is to efficiently distribute power from the lithium battery to all connected components. The PDB supplies equal and stable power to all Electronic Speed Controllers (ESCs) ensuring consistent performance of the system. It supports the integration of a battery up to 40A capacity making it suitable for high-load underwater operations. In addition to power distribution the board also helps with management. It helps send control signals, between the transmitter and receiver these signals then go to the ESCs for motor control. This coordinated power and signal distribution helps control all four motors. The underwater robot can then operate smoothly. Navigate efficiently beneath the water surface.

a. Power Distribution Board (PDB)



Figure 40. Power Distribution Board (PDB)

b. ESC's 30 A



Figure 41. ESC's 30 A

2.28.2 The LM2596S step down module

The LM2596S is a high efficiency DC-DC buck converter module that is used to reduce the higher input voltages to a constant and adjustable lower output voltage. It has an input range of 3V to 40 V and an output range of 1.5 V to 35 V, so it can be used in a vast range of electronic applications. The regulator IC board, LM2596S is used in the module to provide the efficient conversion of the power with minimal heat loss. The multi-turn potentiometer is blue to give the fine voltage adjustment. This module may be used to supply power to microcontrollers like the ESP32, Arduino and other sensitive electronics by safely decreasing high battery voltages by say, reducing 11.1V LiPo battery to 5V to ensure safe operation. It is a stable, efficient, and compact-sized embedded system with a solid foundation in embedded systems, robotics, DIY electronics, and battery operated applications.

a. LM2596S step down module

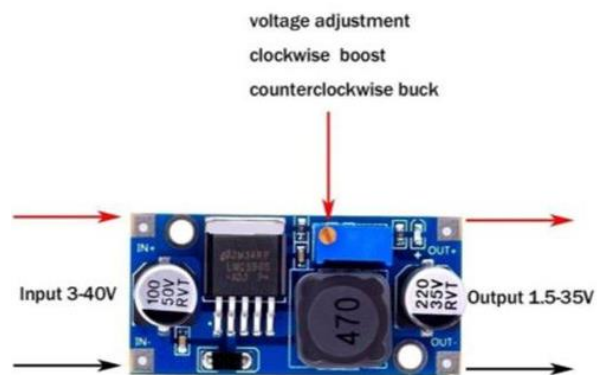


Figure 42. LM2596S step down module

2.28.3 Radio Master Transmitter

Figure 43 is one of the radio masters transmitters we utilized as the primary remote control system on our underwater robot. Control signals are transmitted wirelessly to the receiver and then to the ESP32 by the transmitter and we can then manually run all the four motors. All the motors allow the robot to move vertically in water as two of them are applied in vertical motors. The third motor is used to move the robot forward and backward giving it a horizontal movement. The tail mechanism is controlled by the fourth motor to give the motion of a realistic fish to make the fish maneuverable. All of these operations are directly operated using this transmitter through which we can have good and accurate control on the motion and actions of our underwater robot.

a. Radiomaster transmitter



Figure 43. Radiomaster transmitter system

2.28.4 Thruster for underwater robot

In the case of our underwater robot, we adopted the use of brushless ROV thruster motors, which are the same as those depicted in the figure 44. They are underwater motors that are not only specially designed but also constructed with stainless-steel bearings and a robust propeller housing made of PEGT in order to avoid corrosion. All four motors were in operation: two motors were fixed on the vertical thrust to enable the robot to dive and rise in the water, one motor was applied to give the forward movement and the remaining motor was attached to the tail mechanism to mimic a real swimming motion. Though the product specifications were 12 volts and 480 KV, in our project we used 12 volts and operated the motors at 1000 KV giving enough speed and torque to operate under water. These brushless thrusters provided potent and efficient propulsion, and, therefore, they were suitable in the maneuverability of our ROV and the performance in general.

a. Thruster motor

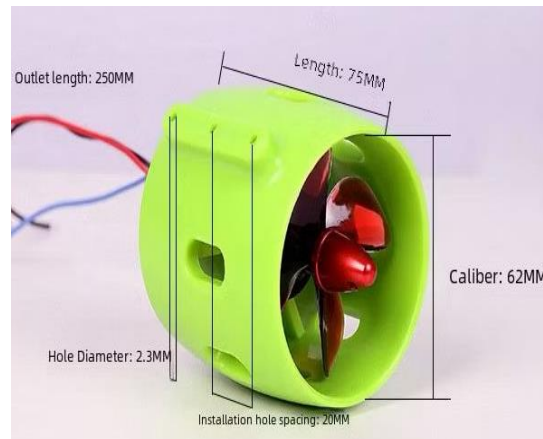


Figure 44. Thruster and its dimensions

2.28.5 Power Source for Underwater Robot

To give power to our underwater robot we used a 3S (Lithium Polymer) LiPo battery figure 45 of capacity 4000 mAh that supplied a constant power of 11.1 volts. This kind of battery is popular in the RC models due to its ability to provide a lot of current thus it is applicable in driving several motors at a time. The battery can provide sufficient power to the thruster motors, tail motor, and all the electronic components attached to the PDB with a rate of 50C. The 4000 mAh capacity also ensures a longer operating time and the robot can be used in underwater operations and last long enough. All in all, this LiPo battery was a good and effective source of power to the whole system.

a. Lipo 3s battery



Figure 45. LiPo 3s battery (Lithium Polymer)