Journal of Physics: Conference Series

PAPER • **OPEN ACCESS**

CFD Analysis of flow around fish

To cite this article: Swastika Palit *et al* 2019 *J. Phys.: Conf. Ser.* **1276** 012010

IOP Conf. Series: Journal of Physics: Conf. Series **1276** (2019) 012010 doi:10.1088/1742-6596/1276/1/012010

CFD Analysis of flow around fish

Swastika Palit[1], Shreya Sinha[2], Aylmer Britto R[2] and Arockia Selvakumar A[1]
[1]School of Mechanical and Building Sciences, VIT Chennai, Vandalur – Kelambakkam road Chennai – 600127, Tamil Nadu, India
[2]School of Electronics Engineering, VIT Chennai, Vandalur – Kelambakkam road Chennai – 600127, Tamil Nadu, India

Corresponding Author : arockia.selvakumar@vit.ac.in

Abstract. The study of fish like bodies in the liquid is a challenging research area in the fields of bio-locomotion and biomimetics. The power and maneuvering of swimming fish will help a lot in the field of underwater vehicles. Mainly the effect of tail oscillation and abdomen oscillation on fluid flow around such a body is highly unsteady, generating vortices and requiring detailed analysis of fluid-structure interactions. In the present context, a computational fluid dynamic (CFD) simulation of a tilapia fish has been developed to study about the fluid flows around this body when moving in water using the k-epsilon turbulence model. A parametric analysis of the variables that affect the flow surrounding the body is presented, along with flow visualizations. In the present study, a detailed unsteady and transient CFD analysis of the effect of tail and abdomen oscillation on the fluid flow around the fish has been studied. The simulation produces a motion of the tail and abdomen in the **(x,** y) plane, with both the tail and abdomen oscillating in the form of a sinusoidal wave. Detailed CFD analysis is done and results in terms of the aerodynamic force coefficients namely the lift and drag coefficients, pressure contour and velocity contour have been presented and discussed for the linear motion of fish along z-axis along with periodic oscillations of tail and abdomen in the (x,y) plane. It was observed that both the aerodynamic coefficients and forces came out to be very less which paves way for the motion of the fish with high efficiency. The analysis provided by the simulations has allowed the flow surrounding the tilapia fish undergoing periodic oscillations along with a linear motion to be studied in detail.

1. Introduction

Engineers are continuously finding faster, more efficient means of transportation especially in the area of underwater vehicles[1]. This has led a number of researchers to give attention to the power and maneuvering of swimming fish. Today underwater findings are mainly dominated by manned or unmanned submarine-type vehicles. The most successful in this respect is considered to be submarine but lacks flexibility, ease of maneuverability and energy efficiency which makes the robotic fish better adapted ecologically in comparison to submarines. Thus there is a requirement of research of propulsion

IOP Conf. Series: Journal of Physics: Conf. Series **1276** (2019) 012010 doi:10.1088/1742-6596/1276/1/012010

and maneuvering of different fishes. With a lot of development and optimization, the locomotion and propulsion of the modern fish are obtained. The propulsion method of fish is entirely different from the propulsion method used by different human developed underwater vehicles. Many new biological inspired designs are developed[2]. Recently a lot of work has been reported in the field of robotic fishes. Most of the works are reported in the field of mechanism and dynamics of robotic fish[3-4]. Chen[5] has worked on the application of Ionic Polymer-Metal Composite fins to propel the fish. Many studies are reported in the field of fluid-body interaction that leads to good speed and good efficiency[6-7]. Research works related to motion control algorithm has also been proposed[8].

Navier-Strokes Equation is solved using Computational Fluid Dynamics(CFD). Many Researchers have found interest in the application of the CFD in different domains like biology, automobiles, buildings, sports etc. Some studies are reported in the field of application of CFD to analyze the hydrodynamics of 2D and 3D dimensional of swimming of Tadpole[9]. Other studies are reported to analyze the swimming in order to understand the propulsion and drag of underwater dolphin kicking using CFD [10]. There are mainly two types of the technique of propulsion. First is anguilliform motion and the other one Carangiform motion. In the anguilliform mode, wavelike motion takes place throughout the body from head to tail. Contrarily in the carangiform mode, the wavelike motion takes place in the rear half of the fish. Some of the studies are reported related to CFD analysis of biomimetic fish-like body but to the knowledge of authors, none of the studies are reported related to detailed CFD analysis of tilapia fish. In this context, a preliminary attempt is made in which detailed CFD analysis of flow around biomimetic fish like the body of tilapia was studied in water which follows a carangiform motion. The commercial package of Fluent 16.0 of CFD software was used.

2. Numerical Method

The velocity and pressure of a fluid flow are governed by the Navier-Stokes Equation. The k-epsilon (k-ε) turbulence model is the most generally perceived model used in computational liquid dynamics(CFD) for analyzing of mean stream attributes for turbulence stream conditions. It is a two-equation computational domain, that gives a portrayal of disturbance by means of two transport equations (PDEs). For the flow considered in this study, the conservation equations for mass and momentum are solved.

Mass Conservation Equation

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla . \rho \vec{v} = 0 \quad (1)$$

Momentum Conservation Equation

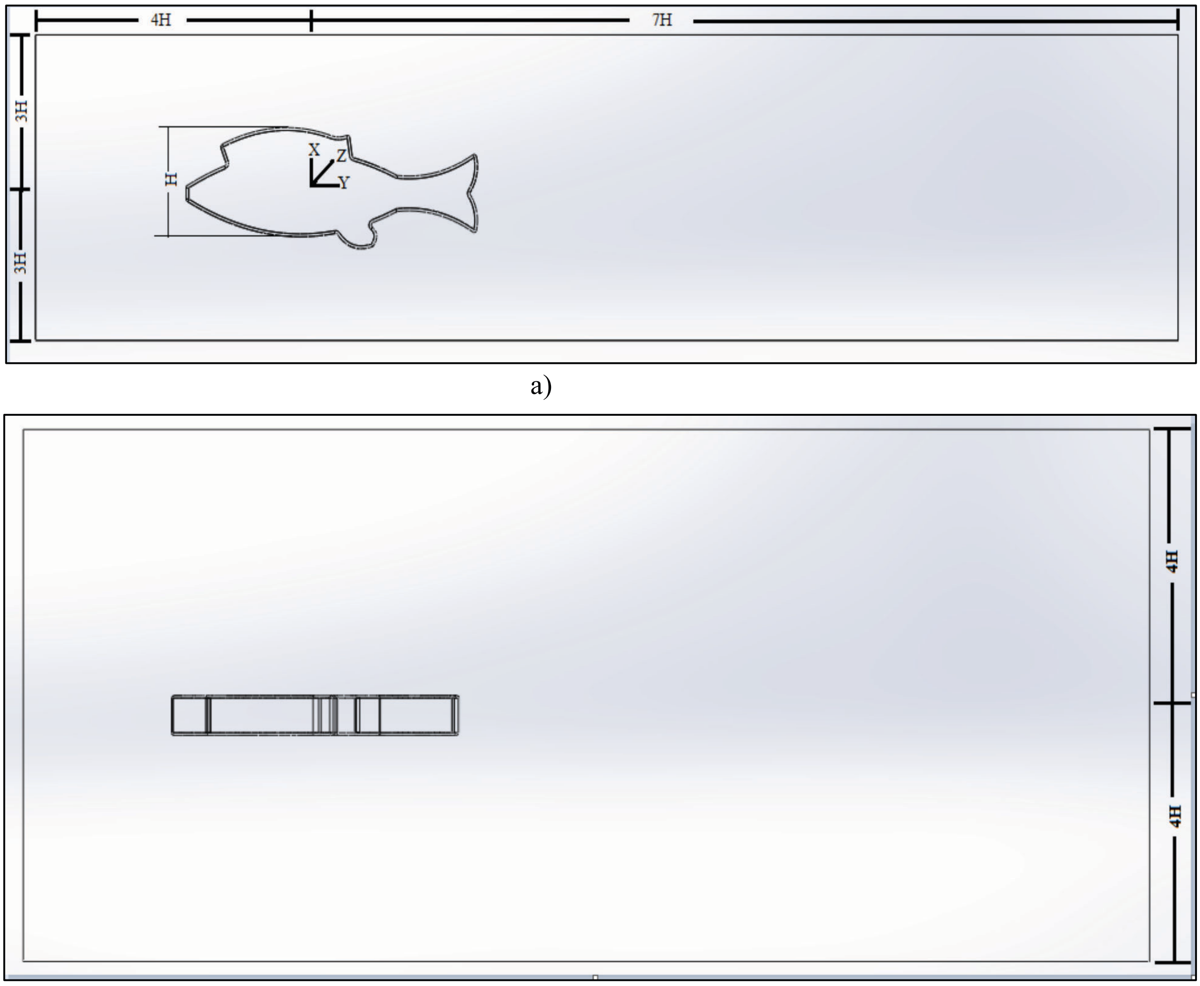
The conservation of momentum in an inertial (non-accelerating) reference frame is described by

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla . (\rho \vec{v} \vec{v}) = -\nabla \mathrm{p} + \nabla . (\bar{\bar{\tau}}) + \rho \vec{g} \quad (2)$$

3. Computational Domain and Boundary Conditions

A computational domain was used for the study (Fig 1a and Fig 1b). There are one inlet and one outlet. The inlet velocity was taken as 0.1 m/s. 6H is the total length in the Y direction and for X and Z direction the total length is kept as 11H and 8H respectively. Here H is 0.015m. Except for the inlet and outlet, all the other walls were kept at a no-slip boundary condition.

IOP Conf. Series: Journal of Physics: Conf. Series **1276** (2019) 012010 doi:10.1088/1742-6596/1276/1/012010



b)

Figure 1.(a) Front view of the computational domain with fish; (b) Top view of the computational domain with fish

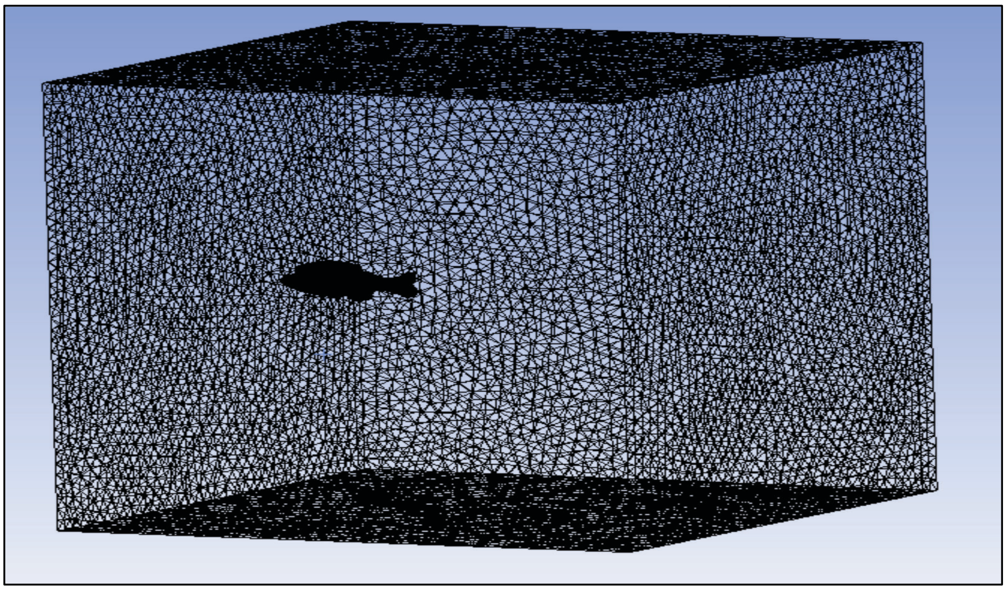
4. Dynamic Mesh and Mesh Structure

Meshing involves dividing the geometry into grids and small finite volumes. The sizes of these grids are such that the solutions to the Navier-Stokes equations can be reasonably approximated in the volume. Naturally, the finer the mesh size and smaller the volume of the grid, the more accurate will be the solution obtained. Meshing is shown in Figure 2a and Figure 2b. In the domain consist of the irregular positioning of the grid points in the flow field and also unstructured grid around the fish.

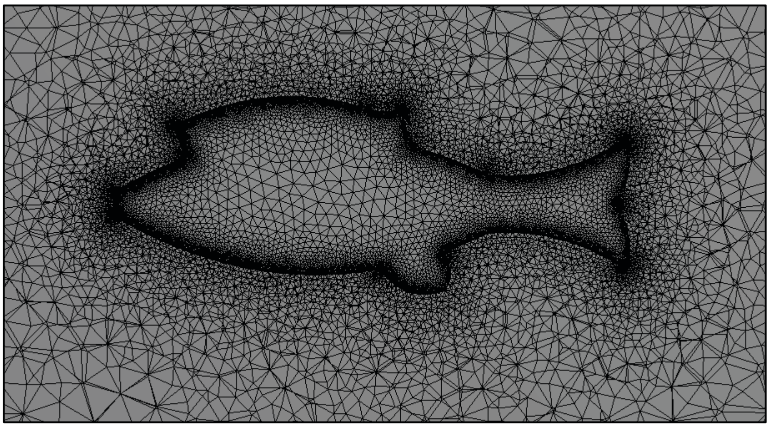
The dynamic meshing technique is used for simulation. It allows the displacement of domain boundaries in simulation, which form new mesh according to a new location. It is necessary to consider the grid velocity of boundaries in general conservation equations. Different dynamic mesh update methods are available in the literature. There are three fundamental methods: the smoothening method, layering method, and remeshing method. For current simulation, spring-based smoothening method and local remeshing method is used. In layering method layers are added or removed adjoining to the boundary in motion according to the layer's height adjoining to the surface in motion. In spring-based

IOP Conf. Series: Journal of Physics: Conf. Series **1276** (2019) 012010 doi:10.1088/1742-6596/1276/1/012010

smoothening method edges which are in between the nodes are considered as interconnecting springs network and local remeshing method is used to lessen the problem of convergence when the cells become extensively small and extensively large. In this simulation fine meshing is done and tetrahedral mesh is used around the geometry to avoid negative cell volume during the dynamic simulation.



a).



b).

Figure 2. (a) Mesh of Rectangular Computational Domain (b) Mesh near the fish body

4.1 User Defined Functions

The movement of the abdomen and tail in the current fish model was depicted using User Defined

IOP Conf. Series: Journal of Physics: Conf. Series **1276** (2019) 012010 doi:10.1088/1742-6596/1276/1/012010

Function (UDF). A UDF is basically a C program that can be dynamically loaded with ANSYS FLUENT to specify body motion. In this present context, UDF is applied using DEFINE macro that is DEFINE_CG_MOTION. The parameters used are linear velocity, angular velocity, current time and time step. Both linear and angular velocity are taken as 0.07 m/s

4.2 *Time – steps independence study*
For time independence study, 0.0001 s was chosen for the time step. Larger the time step, computational effort is lesser. The results from the simulation will be used to build a model for experimental studies.

5. Results and Discussions

5.1 Time-averaged flow characteristics
A three-dimensional transient (t=10 s) fluid flow analysis with fish velocity as 0.07 m/s has been carried out with the k-epsilon turbulence model. 10,000 time steps were used for averaging. Figure 3 depicts the 4 transverse planes which are parallel to the y-z plane (A, B, C, D) on which velocity profile is obtained.

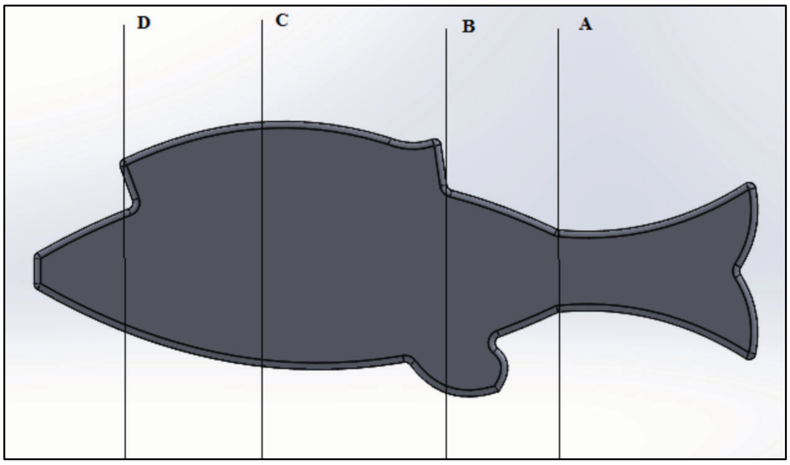


Figure 3. Four different planes parallel to x-y plane

Figure (4a, 4b, 4c, 4d) shows the velocity vectors around the fish at 4 different planes parallel to y-z plane Figure 3. It was observed that velocity vectors at four different planes were very low. It varied from 8.423e-003 ms^{-1} to 1.685e-002 ms^{-1}.The range of velocity vectors for all four planes was observed to be similar. There is a mixture of transverse and longitudinal characteristics due to the periodic oscillation of tail and abdomen. Figure 5 shows three-dimensional view of the flow around the fish at four different planes.

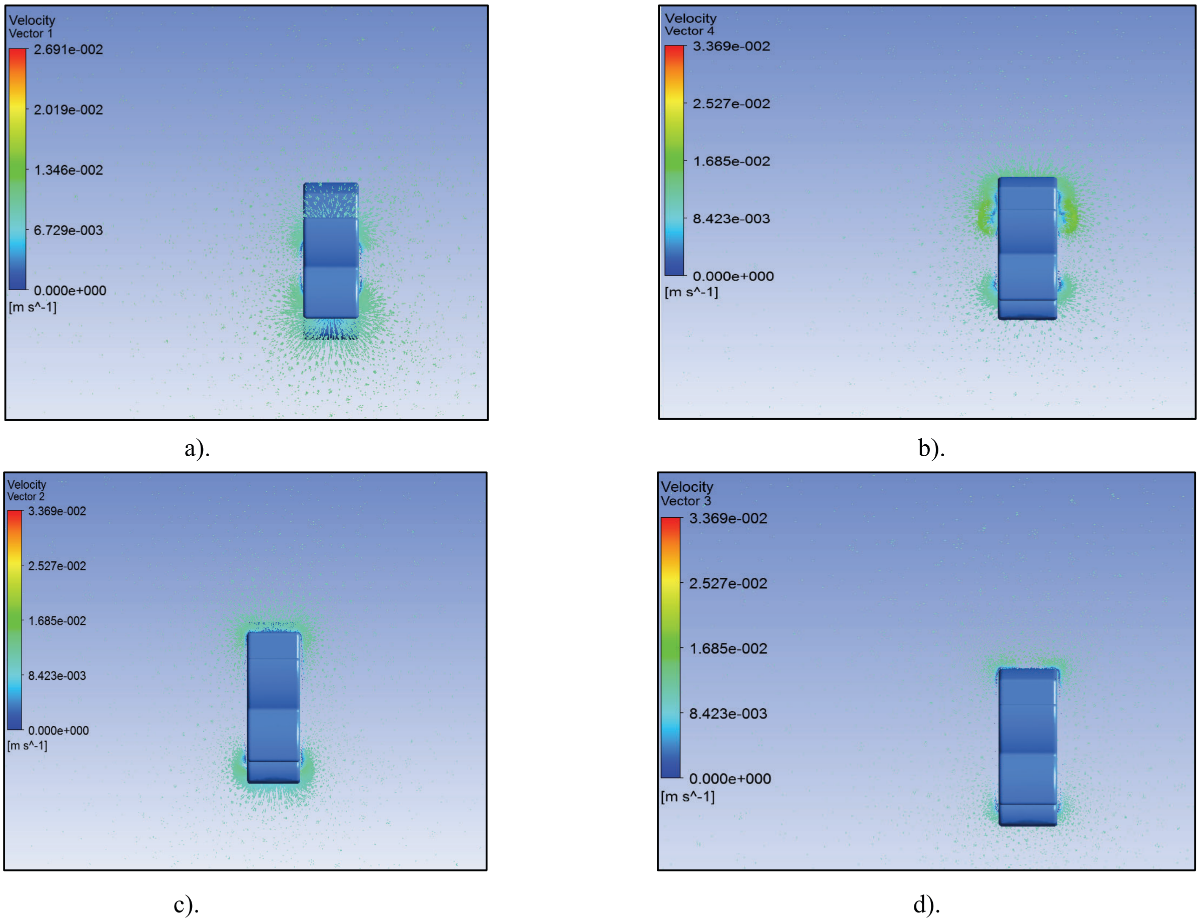


Figure 4. Velocity Vector at (a) Plane A (b) Plane B (c) Plane C (d) Plane D

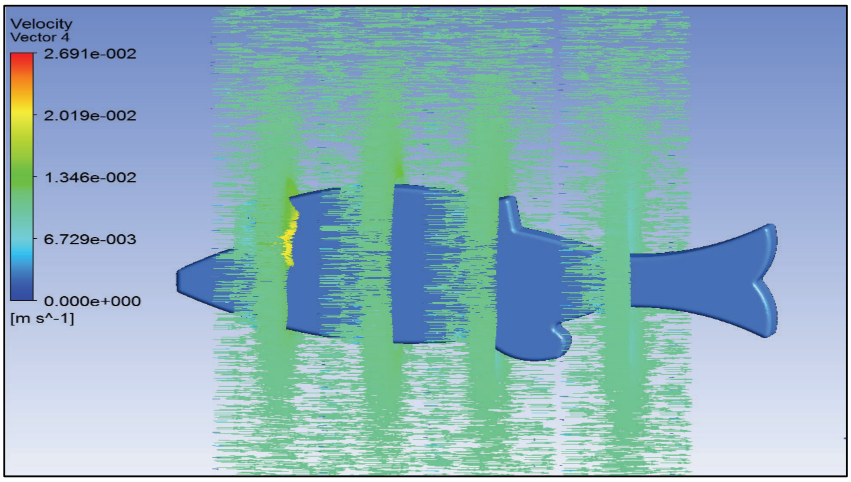
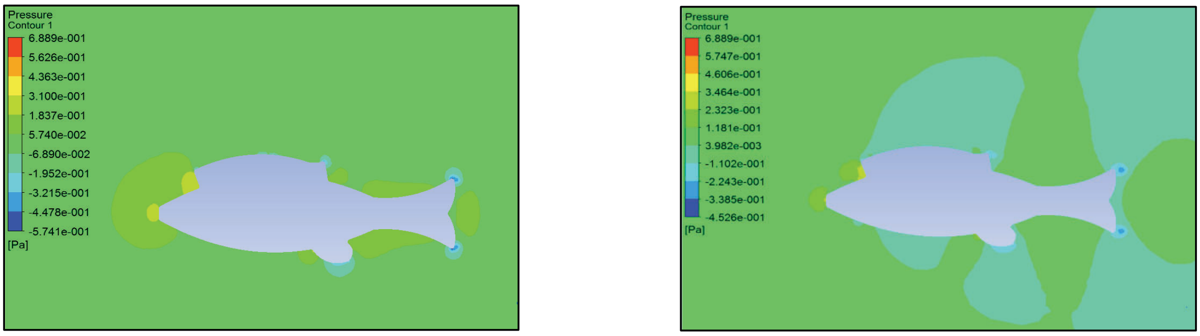


Figure 5. 3-D view of flow around the fish

IOP Conf. Series: Journal of Physics: Conf. Series **1276** (2019) 012010 doi:10.1088/1742-6596/1276/1/012010

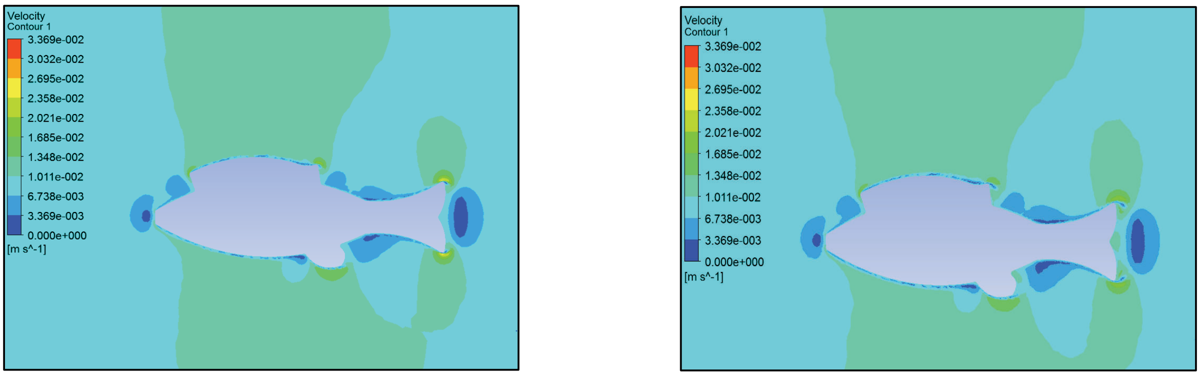
Figure 6a and 6b depicts the pressure contours around the fish at two different time that is at t = 5 sec and t = 10 sec respectively in x – y plane. It was observed at t = 5 sec the magnitude of pressure is high near the mouth of the fish. The magnitude of pressure is high near the tail and rear side of the body due to turbulence and strong vortex effect. The low-pressure magnitude was observed near the corners and fins of the fish. With the increase in time, the magnitude of pressure becomes less around the fish whereas high-pressure magnitude is still there near the mouth of the fish and on the rear side of the fish. Lowest pressure magnitude was found near the corners of the tail. But behind the tail, the magnitude of pressure was high due to the predefined reasons of turbulence and strong vortex.

Figure 7a and 7b depicts the velocity contours around fish at t = 5 sec and t = 10 sec respectively in x – y plane. It was observed that almost similar type of velocity magnitude was there at both the time instants. Much difference was not observed. The high magnitude of velocity was observed at corners of the fins and tails due to the sudden change in shape. It was observed that the magnitude of velocity is very low near the mouth, near the tail and on the rear side of the body. Around the fins also the low magnitude of velocity was observed.



a). b).

Figure 6 . Pressure Contours at (a) t = 5 sec (b) t = 10 sec



a). b).

Figure 7 . Velocity Contours at (a) t = 5 sec (b) t = 10 sec

5.2 Aerodynamic Force Coefficients

The force that acts in the opposite direction of the fish's motion is the drag. It is unfavorable to the fish's performance. C_d is called the coefficient of drag. Force acting perpendicular to fish is lift. Similar to C_d, C_l is called the Coefficient of Lift. The lesser the C_d and Drag more efficient is the motion of the fish. The C_d and C_l for this CFD Analysis came out to be 0.0140 and 0.000309 respectively. The drag force and lift force came out to be 0.00105 N and 0.000309 N respectively. It was observed that both the aerodynamic coefficients and forces came out to be very less which paves way for the motion of the fish with high efficiency.

5.3 Validation

The current analysis simulated using FLUENT 16.0 was validated by comparison with studies reported by Rajeev et al [11], in that they have done the design and fabrication of a biomimetic fish. The C_d and C_l obtained by our simulation were similar to the aerodynamic coefficient obtained by their simulation. It was also compared with the work reported by Adkins [12], in which CFD analysis was done for fish Tuna which performs carangiform motion which is similar to the motion of tilapia. The numerical results obtained from tail and abdomen periodic oscillation were compatible with works reported in their studies and previous works in the field.

6. CONCLUSION

The CFD analysis was carried out to study the hydrodynamic performance of the biomimetic fish like the body of tilapia in water, which follows carangiform motion. Analysis of flow around the fish moving with a speed of 0.07 m/s with periodic simulations of tail and abdomen was done. This work currently doesn't consist of complicated mechanisms like tail flexing and retraction of the fin which are performed by real fish. This analysis consisted of a fish which was moving in the x-y plane with a periodic oscillation of tail and abdomen along the z-axis. The fine-mesh method is used with 0.9 million cells. To provide optimized time-step value, time-step independence study was carried out.

- The optimized time-step value provides less computational efforts. The largest possible time-step which successfully carried out simulation for the selected mesh is 0.0001 s.
- Behind the tail, the magnitude of pressure was found high due to the predefined reasons of turbulence and strong vortex.
- It was observed that the magnitude of velocity is very low near the mouth, near the tail and on the rear side of the body. Around the fins also the low magnitude of velocity was observed.
- The Cd and drag force when the fish was traveling with velocity 0.07 m/s came out to be 0.0140 and 0.000309 N respectively which paves way for the motion of the fish with high efficiency.

7. REFERENCES

[1] Techet A H and Triantafyllou 1999 M S Boundary layer relaminarization in swimming fish *Proceedings of the Ninth International Offshore and Polar Engineering Conference*, Brest, France

[2] L J Rosenberger 2001 Pectoral Fin Locomotion in Batoid Fishes: Undulation Versus Oscillation *The Journal of Experimental Biology* **204** 379–394

[3] N Kato, M Furushima 1996 Pectoral Fin Model for Maneuver of Underwater Vehicles. In *Proceedings of the Symposium on Autonomous Underwater Vehicle Technology*. Monterrey, CA 49–56

[4] K. H. Low, A. Willy 2005 Development and Initial Investigation of NTU Robotic Fish with Modular Flexible Fins. In *Proceedings of the IEEE International Conference on Mechatronics & Automation (ICMA2005)*, Niagara Falls, Canada 958–963

[5] Chen Z, Shatara S and Tan X 2010 Modeling of biomimetic robotic fish propelled by an ionic polymermetal composite caudal fin *Mechatronics IEEE/ASME Transactions* **15(3)** 448-459
[6] Kelly S and Murray R 2000 Modeling efficient pisciform swimming for control *International Journal of Robust and Nonlinear Control* **10(4)** 217-241
[7] Hong C and Chang-an Z 2005 Modeling the dynamics of biomimetic underwater robot fish *Robotics and Biomimetics (ROBIO) IEEE International Conference* on 478-483
[8] Shang L, Wang S, Tan M and Dong, X 2009 Motion control for an underwater robotic fish with two undulating long-fins *Decision and Control* 6478- 6483
[9] H. Liu and K. Kawachi 1997 The Three-dimensional Hydrodynamics of Tadpole Locomotion *The Journal of Experimental Biology* **200** 2807–2819
[10] Lyttle AD Blanksby B **A** Elliot B C Lloyd D *G* 2000 Net forces during tethered simulation of underwater streamlined gliding and kicking techniques of the freestyle turn *Journal of Sports Science,* **18** 801-807
[11] Sidharth R, Sandeep K S and Sheeja J 2016 Design and Fabrication of a Bio-Inspired Fish-Shaped Autonomous Underwater Vehicle
[12] D Adkins and Y Y Yan 2006 CFD Simulation of Fish-like Body Moving in Viscous Liquid *Journal of Bionic Engineering* **3** 147 - 153