A Computational Investigation on Aerodynamic Performance of the Cross-Section of a Flying Fish (Cypselurus Hiraii) in Comparison with Conventional Aerofoil Behaviour

Gayathri B1, Nasif Ali 2, Abhishek B V 3 and Vishnu Kumar G C4

1,2,3 Student, School of Aeronautical Sciences, HITS, Chennai -603103.
4 Assistant Professor, School of Aeronautical Sciences, HITS, Chennai -603103.
E-mail: gcvkumar@hindustanuniv.ac.in

Abstract. Bio-mimicry is one of the leading fields of research that has been around since the early ages of science. Aerospace industry, especially, takes most of its design inspiration from the aquatic life. In this paper a breed of flying fish scientifically known as Exocoetidae, Cypselurus Hiraii has been chosen to study. The aerodynamic characteristics of the fish were studied experimentally in a wind tunnel previously, but the plausibility of its design application in the aerospace industry hasn’t been explored yet. Therefore, in the present study the cross-section of the fish in the side view has been traced and its aerodynamic performance as a 2D infinite wing has been examined in ANSYS R15 (FLUENT). The performance of this fish inspired shape has been compared to the properties of conventional aerofoils in terms of coefficient of lift and coefficient of drag.

1. Introduction
The flying fish are aquatic animals that are equipped with certain morphological features that allow them to fly in air for an accountable period of time. Flying fish being the colloquial name, they scientifically termed as Exocoetidae.

There are all together 64 species of Exocoetidae. Flying fish can make strong, self propelled leaps out of the water where their long winglike fins allow gliding for large distances above the surface of the water, though they can not fly the same way a bird does. Flying fish flights are usually about 50 m, while updrafts at the leading edge of waves can be used to cover distances of up to 400 m. At speeds of more than 70 km/h, they can fly. The highest height is 6 m above the sea level. Flying fish sometimes land on the decks of smaller vessels inadvertently. The current study focuses of one breed of Exocoetidae known as Cypselurus Hiraii. The choice of this particular breed is due to the previously available studies and experiments. As in every flying fish this particular breed flies with the help of its pectoral, pelvic, dorsal and caudal fins. These fins expand and retract to form combinations called morphologies, each used in different flight conditions. The field of bio mimicry has come to play since the early evolutions of mankind. The word bio mimicry literally translates to ‘imitation of the living’.
It seeks to derive inspiration from nature's natural selection solutions and translate the concepts into human engineering. It is a technological-oriented technique focused on bringing the lessons of nature into motion.

The key idea is that nature has already solved many problems faced by society. Experienced engineers include animals, plants, and microorganisms. The primary belief of the bio mimicry method is that what did not work is now a relic after 3.8 billion years of research and growth, and what is around us is the secret to survival. According to Janine Benyus, a biologist and the co-founder of Bio mimicry Institute, bio mimicry sees nature as:

- A model. It studies nature’s models and imitates them or uses them as inspiration for designs or processes with the goal of solving human problems
- A measure. It uses ecological standards to judge the rightness of human innovations
- A mentor. It is a new way of observing, assessing and valuing nature

Aerospace industry has taken most of its inspiration from life and nature. Pioneers like Airbus and NASA have conducted detailed research on the topic and many of these aerial vehicles display the work successfully. From eagle winglets to sharkskin riblets on fuselages aerial and aquatic animals help in the implementation of the efficient flight design that we have today. The idea of the present work sprouts from an in depth study about the influence of aquatic life in aircraft design. Water and air both being fluids bare many similarities that allow efficient flow over similar shape with minute yet prominent changes. The area of research being bio mimicry of aquatic animals in aerospace, the aggregate of prior research is modest. However, the idea of the study is not entirely new, in the article ‘Fluid Mechanics of Fish Swimming’ many similarities between the fluid dynamics and propulsion of fish’s swimming and an aircraft’s flight have been stated. In a research paper titled ‘Aerodynamic Performance of Flying Fish in Gliding Flight’ detailed aerodynamic performance of Cypselurus Hiraii is given through a wind tunnel experiment of the stuffed biological fish body. This being an Engineering Fluid Dynamics (EFD) analysis, in ‘Numerical Predictions of
Aerodynamic Performance of Flying Fish in Gliding Flight’ is a succeeding paper elucidating the Computational Fluid Dynamics (CFD) analysis of the same Cypselurus Hiraii.

**Figure 4.** a) An aircraft flying with constant velocity. Thrust (T) from the engine balances the drag (D); lift (L) mainly from the wings balances the weight (W). b) The fish here the thrust is mainly from flapping tail. (FB)

**Figure 5.** Different types of fins of a fish

**Figure 6.** Cross-section of fish tail

This article gives a complete overview of fish swimming, a portion of this article is dedicated to considering the fish as a self-propelling body, here the author compares the fish to an aircraft, it being a self-propelling as well.

In general, a fish has six different types of fins, the coordinated movement of these different fins generate required lift and thrust to overcome drag and body weight. This is analogous to the coordinated movement of ailerons, elevators and rudder in an aircraft. Also the cross-section of every fin resembles an Aerofoil. This similarity is the foundation for the present study. The flying fish is an outstanding flying marine vertebrate that demonstrates effective air gliding capabilities. By a successive series of taxiing and takeoff, the flying fish glides over a total distance of as much as 400 m in 30 s, and its maximum flight speed is in the region of 10–20 m/s. Morphologically, pairs of fins are physically adapted to act as flight wings and some genera, such as Cypselurus, have a flattened bottom body to provide extra lift. While explanations have been proposed for the flight of flying fish as an escape from predators, the exact reason is still not clear. However, their behavioral adaptation for flight is rather distinctive. Some flying fish dip the lower lobe of the caudal fin and beat the tail, which is called taxiing, as they emerge from the sea or the gliding speed decreases at the end of one glide. They use this taxi to gain additional speed for successive gliding flights, assisted by the wings (stretched pectoral fins).

Wing-body configurations of the five model fish were selected as follows (1) both pectoral and pelvic fins spread (models L1-L3); (2) only pectoral fins spread with pelvic fins folded against the body (model M); (3) body only with both pectoral and pelvic fins folded against the body (model M); (3) body only with pectoral and pelvic fins folded (model N). In addition, to determine how it affects aerodynamic efficiency, we varied the lateral dihedral angle of the pectoral fins for models L1-L3. The flying fish's wings are made up of a soft membrane protected by fin rays. Their leading edge is rigid, but it is flexible on the trailing edge. On the lower wing surface, all the fin rays are positioned, so that the upper wing
surface is smooth and the lower surface is ribbed. By completely spreading out the flying fish fin, the wing shape of each model was decided.

Figure 7. Morphologies of Cypselurus Hiraii
The model was examined in a wind tunnel, a six axis force/torque sensor, was used for force measurements, centre of gravity and neutral point were depicted prior to the experiment. To introduce the ground effect on the flying fish, the base of the wind tunnel is filled with water to create an identical environment.

Figure 8. The wind tunnel experiment on Cypselurus Hiraii
Figure 9. (A) $C_L$ vs $\alpha$. (B) $C_D$ vs $\alpha$

Open circle, Model L1; open triangle, model L2; open square, model L3; closed diamond, model M; model N.

Figure 10. (a) model A, (b) model B and (c) model C morphometric parameters

For models L1–L3, M and N, the figure shows the variations of lift ($C_L$) and drag ($C_D$) coefficients with the angle of attack. The lift coefficients for models L1-L3 and M are greatest at 30-35°, as shown in Fig.5A. Since the largest lift force is needed to take off the flying fish, those findings are confirmed by the present result, assuming that the angle of take-off is close to the angle of the fish emerging from the sea. Interestingly, the lift coefficients of models L1-L3 and M do not drop sharply and display large peaks at high attack angles, even after stall, with little lift force loss. This observation coincides with the
characteristics of the thin-airfoil stall suggested by McCullough and Gault (McCullough and Gault, 1951) who listed the characteristics of the airfoil stall at low Reynolds number as the trailing-edge stall, leading-edge stall and thin-airfoil stall, respectively. The lift coefficient curve is relatively flat even after stall, with the thin airfoil stall occurring on a thin airfoil with a sharp leading edge.

Three separate flying fish models, the preliminary one of which, following a previous wind tunnel experiment, mimics Cypselurus Hiraii in pectoral fin morphology. By the computational fluid dynamics (CFD) method, their aerodynamic performances are numerically analyzed. At 35°, the maximum lift force coefficient of 1.03 is reached, and the maximum lift-to-drag ratio of 4.7 is achieved at 6°.

2. Methodology
The process of the study is completely computational, hence materials and manufacturing was not involved. For the computational work, NX CAD was used for modelling and ANSYS FLUENT for analysis. 3D model of Cypselurus Hiraii was designed for validating the data acquired from the wind tunnel experiment in previous works (Park and Choi 2010 J. Exp. Biol. 213 3269–79). The 2D sheet model is designed for the study. The 2D side view cross-section if the fish, being the lift generating shape, is focused. The analysis involved the inspection of aerodynamic coefficients of the shape at four different Reynolds numbers and nine different angles of attack. The graphs of each aerodynamic coefficient are plotted against the angle of attack and the trends were observed.

NX CAD (version 10.0.0.24) is formerly known as “uni-graphics” which is an advanced high end CAD/CAM/CAE, which has been owned by the Siemens PLM software. The software is capable to design, to perform engineering analysis and to manufacture. CAD helps to develop innovative new products. With leading edge design tools and technologies, NX streamlines and accelerates the product development process. The modelling of the design includes the following steps:

- Rastor image and scaling
- Tracing of the image
- Final Model
- 2D Model

The top, side and front view images of the fish are taken from (Park and Choi 2010 J. Exp. Biol. 213 3269–79) given as model N. The images are then imported to NX CAD (version 10.0.24) by using the “Rastor image” option.

Menu=>Insert=>Datum point=>Rastor Image

As we enter the “Rastor image” option, a pop up bar becomes visible where there are following orientations:

![Rastor image bar](image)

**Figure 11. Rastor image bar**
The following are the parameter and axis’s used for the Raster image

Base point: “Bottom left” - It specifies the model base point.
Insertion point: “Inferred” - It specifies the insertion of the target body.
Reference direction: “Horizontal” - To design the model better.
Rotation: The rotation of the image is done horizontally to the respective planes
The images are imported with respect to the following planes:
Top view: YZ Plane
Side view: XY Plane
Front view: ZY Plane
Scaling of the images is done thereafter in order to find the dimensions.
Length of the model = 150mm
Width of the model = 28mm
Thickness to chord ratio = 14.6% (150mm x 28mm)

After importing the images and aligning the image onto respective planes, the images were traced by using “Spline” option.
Figure 14. Side View

For tracing the image, the respective view/plane is taken and by using “studio spline” option, the image of the fish is traced thoroughly by small splines and thereby the other views accordingly. All the individual sections are combined to become a complete profile.

To get a 3D model from these front, side and top profiles, “Studio Spline” was used. The “Studio Spline” tool dynamically creates and edits splines by dragging defining points or poles, and assigning slopes or curvature constraints at defining points.

Menu => Insert => Curve => Studio Spline

Before using Studio Spline, six datum planes were introduced, dividing the model to five sections, a 3D sheet for each sections was developed individually by the guide lines drawn on each datum plane and then combined to get the 3D model. Then “Edge blend” option was used to round the sharp edges of the model.

Figure 15. The final 3D model

Figure 16. 2D model

The cross section of the side view is taken to design the 2D model. The tracing of the 2D model is done in similar to the design of the 3D model. This is 2D sheet created for easier Boolean subtraction from the domain later on in ANSYS FLUENT.

3. Validation

The 3D model of Cypselurus Hiraii, whose design procedure is elucidated in the above chapter, is a solid model without sharp edges. It is 150mm in length. For the analysis the domain was 1500x600x600 mm cuboid. These measurements are chosen to keep the analysis, following the wind tunnel experiment (Park and Choi 2010 J.). Boolean subtraction was performed to remove the model from the domain, to observe the flow properties around the model. The mesh is unstructured with tetrahedral elements; the element size was chosen to be 2 mm surrounding the fish profile progressing to 5mm at the boundary.

Irregular networking defines an unstructured grid. It's not possible to articulate it as a 2D or 3D series. Usually, these grids use 2D triangles and 3D tetrahedrals. The grids can be made to conform to any geometry desired. However, this generality comes with a price. The method of grid generation is not
entirely automatic and can require substantial user intervention to generate grids with appropriate local resolution levels while at the same time having a minimum of distortion of elements and tetrahedral elements will suit better for complex geometry. In setup, k-Omega model is selected as it is well suited for the surface analysis than any other model equations in ANSYS. This particular model is pre-defined to showcase and study the flow behavior over the surface of the section. The SST k-omega turbulence model is a two-equation eddy-viscosity model that is used for many aerodynamic applications. Combining the Wilcox k-omega and the k-epsilon models, it is a hybrid model. The blending feature, F1, in the free stream triggers the Wilcox model near the wall and the k-epsilon model. This ensures that in the flow sector, the right model is used. To simulate flow in the viscous sub-layer, the k-omega model is well adapted. The k-epsilon model is suitable in regions away from the wall for predicting flow behavior. After the model is selected and it is given with appropriate boundary conditions, Reference values. It is also asked to plot the graphs for continuity, lift and drag coefficients respectively. And along with these graphs, contours of velocity and pressure were obtained for the analysis performed. After defining the whole setup, it is processed to perform the analysis through iterative method to obtain the required values of lift, drag coefficients. The Reference values of this model include Operating Velocity, Gauge Pressure, Length of the section, Area, Depth, Density, Temperature, and Viscosity.

Table 1. Reference Values

| Parameter     | Value          |
|---------------|----------------|
| Area          | 0.15 $m^2$     |
| Density       | 1.225 $Kg/m^3$|
| Depth         | 1 m            |
| Length        | 0.15 m         |
| Gauge Pressure| 101325 Pa      |
| Temperature   | 288.16 K       |
| Velocity      | 12 m/s         |
| Viscosity     | $1.7894 e^{-05} Kg/m^s$ |

Velocity components are chosen as the input parameters. X, Y and Z components of the velocity are varied to give different angle of attacks. Velocity components corresponding to angle of attacks 0° to 45° with 5° degree intervals were given as inputs.

Aerodynamic parameters ($C_L$) and ($C_D$) are given as output parameters. This was to reduce the repetition of the setup process, for each input parameter the output is calculated at once.

The flow parameters and the turbulence model has been setup and the convergence has occurred. The output parameters for the corresponding inputs have been successfully calculated. After the result is obtained in the flow analysis at the operating velocity of 12m/s, this is to be compared and validated with the pre-existing results from the research data available from literature survey.
Figure 17. (A) $C_L$ vs $\alpha$. (B) $C_D$ vs $\alpha$

Open circle, Model L1; open triangle, model L2; open square, model L3; closed diamond, model M; model N. from (Park and Choi 2010 J)

The values of $C_L$ and $C_D$ at the angle of attacks 0, 10, 20, 30, 40 and 45 are deduced and graphs are plotted for lift and drag coefficients versus angle of attack comparing both the data. The viable observations made from this data deduced from the analysis are as follows:

Figure 18. $C_L$ versus $\alpha$
Figure 19. $C_D$ versus $\alpha$

Similar trend and behavior of the graphs is observed. The deviation of computational results from the experimental is inevitable, however similar stall characteristics are observed. Another reason for the deviation is due to the negligence of ground effect. In the wind tunnel experiment (Park and Choi 2010) a water base was introduced to study the ground effect. In the CFD analysis however, this parameter was neglected. Accounting the reasons for the deviation, the validation was considered successful.

The section is traced and image processed for the flow analysis over it in air medium at various velocities to decide the optimum velocity of the selected fish at 12m/s deduced from the literature survey. And this analysis is performed using ANSYS R15.0 software for all the values including coefficient of Lift, coefficient of Drag and coefficient of Moment. This also includes the respective graphs of above mentioned which are compared to the standard study graphs for the optimum and efficient result. This process of analysis includes following steps:

1. Importing & Geometry of the section
2. Meshing
3. Boundary Conditions of the section
4. Setup for Analysis
5. Solution & Result

3.1 Importing the geometry

Once the section is traced from the fish model, it’s imported into ANSYS using the import function and its geometry is to be defined for the software to progress further. As the section is imported into XY-plane, it is enclosed with a C-H type enclosure (Wind tunnel type) for a steady flow irrespective of the angle of attack and also for flow uniformity. After the section is enclosed, the shape of the section is deducted from the whole using the Boolean subtraction function. This is done for the better analysis of the section. Here, the geometry of the section is imported and other parameters are defined. Then enclosure defined for the inlet, outlet and the walls as well as section as the software input details.

As per the given input, the flow will be directed on to the section with respective inlet and outlet for the velocity input given throughout the enclosure. This allows performing the flow analysis at required velocities under given pressure and other parametric conditions. Further, this is processed for meshing.
The Named Selections to the domain boundary, the curved profile corresponding to inlet, the opposite edge as the outlet, rest of the edges were named walls including the fish profile. The gauge pressure is at 101325 Pa.

3.2 Meshing

Meshing involves developing of grid with shape in any pre-defined pattern like triangle, hexagon, rectangle etc., for the flow to pass through. This is done so as to particulate the flow on to and around the section using Nodes and elements. In this Meshing, the enclosure is defined with following meshing parameters:

Type of Grid: “Unstructured”- This is preferred because of the uneven surface of the section and hence the structured grid cannot be aligned properly for uneven surfaces, this is selected. Element pattern: “Tetrahedrons”- All triangular method is used as it involves a simple, accurate process and progress for this flow analysis.

After the grid is developed, using the sizing function, it is prepared and defined for Body sizing (sizing of the enclosure elements involved) and the edge sizing (sizing of the section elements involved). In this meshing, certain range of inflation of the grid- 10 layers is also given to ensure the flow
uniformity around the section using pre-defined inflation function. And this meshed enclosure will be further processed to setup to include the boundary conditions.

Boundary conditions of this model include Operating Velocity, Gauge Pressure and Length of the section, Area, Depth, Density, Temperature, and Viscosity.

![Figure 22. Mesh Domain](image)

Table 2. Reference Values

| Parameter     | Value          |
|---------------|----------------|
| Area          | $0.15 \text{ m}^2$ |
| Density       | $1.225 \text{ Kg/m}^3$ |
| Depth         | $1 \text{ m}$ |
| Length        | $0.15 \text{ m}$ |
| Gauge Pressure| $101325 \text{ Pa}$ |
| Temperature   | $288.16 \text{ K}$ |
| Velocity      | $12 \text{ m/s}$ |
| Viscosity     | $1.7894 e^{-05} \text{ Kg/m*}\text{s}$ |

Velocity components are chosen as the input parameters. X and Y components of the velocity are varied to give different angle of attacks. The model was analysed for four velocities $12 \text{ m/s}$, $30 \text{ m/s}$, $60 \text{ m/s}$ and $150 \text{ m/s}$. Once the iterative method progress, the behaviour of the graphs can be observed with the number of iterations being done. And this iteration number is dependent on the element sizing (both body & edge). As the number of iterations increases, the model equation will be calculated those many times until the continuity graph is seen to be converged.

The graph is seen to be converged, then it can be seen as parametric table with the values of lift and drag coefficients also with the operating velocity. The parametric table is used to validate the values of lift and drag coefficients with varying angles of attack either positive or negative. This is done by altering the flow direction with respect to the angle of attack chosen by resolving the operating velocity based on the angle into X and Y components namely on x-axis and y-axis instead of changing the orientation of the section. Further stage of this flow analysis includes the validation w.r.t the standard results from the literature survey performed. And then, this whole analysis is subjected for grid independency test for the standard, stable and accurate result which is considered final. Once the analysis is done for a
particular velocity, body sizing of the elements, and edge sizing of the elements, it is liable for grid independent testing which is done in this flow analysis. In this study, at an operating velocity of 12m/s, this test has taken place. Throughout the test, the velocity maintained is 12m/s. The angle of attack of the flow is at 0° on to the whole enclosure. The element size in body sizing varied from 2 mm to 0.5 mm and is observed unchanged at 0.5 mm. The element size in edge sizing has also varied by maintaining it at 1 mm but altering the number of divisions each element is sub divided into. These element sub divisions have varied from 1000 to 700 divisions and got stabilized at 700 divisions and the Number of Nodes varied from 1,32,000 (approx.) to 4, 96,000(approx.)

Figure 23. Convergence curve

| S. No | Nodes  | Elements | \( C_L \) at \( \alpha = 0 \) | \( C_D \) at \( \alpha = 0 \) | \( C_M \) at \( \alpha = 0 \) |
|-------|--------|----------|------------------------------|----------------|----------------|
| 1     | 132674 | 165888   | -0.03927                     | 0.03587        | -0.01935        |
| 2     | 151347 | 187163   | -0.08224                     | 0.02652        | -0.03271        |
| 3     | 170925 | 209643   | -0.08230                     | 0.02784        | -0.03275        |
| 4     | 212756 | 292813   | -0.079972                    | 0.027184       | -0.03154        |
| 5     | 314311 | 495266   | -0.07933                     | 0.02747        | -0.02985        |

The highlighted row in the table3 shows the values where the grid independency test is achieved. And as we can see the values of coefficient of lift, coefficient of drag for the highlighted portion and the values of the consecutive portion are almost the same. Henceforth, the grid independency test is achieved for this design at the considered operating velocity with all the boundary conditions. The precision for the test is \( 10^{-2} \).

The analysis was run at different values of velocities, 12m/s such as 30m/s, 60m/s, 150m/s and respective values of coefficient of lift and coefficient of drag are calculated. Thus using these values obtained at four different Reynolds number is plotted against angle of attack for observation.
4. Results and conclusion

4.1. $C_L$ Behaviour (Characteristic Curve)

At all the Reynolds number calculated, the stall angle is observed at 25°. The coefficient of lift here is seen increasing proportionally with the angle of attack. This trend is fairly close to the behavior of an aerofoil. Since the analysis is done using K-omega turbulence model, the viscosity plays a crucial role in the determination of $C_L$. That is also the reason why $C_L$ depends on Reynolds Number. The contours of velocity and pressure are at 12m/s, 30m/s, 60m/s and 150m/s. The most efficient performance is seen at the Reynolds number $1.54 \times 10^6$.

4.2. $C_D$ Behaviour

The contours of velocity and pressure is same as the above graph. $C_D$ reaches a local maximum at 25° which is the stall angle. The $C_D$ increases with $C_L$ proportionally; however, it is seen with greater
increasing rate at high angles of attack due to the boundary layer separation due to the difference in upstream and downstream pressures.

5. Conclusion

The characteristic of the side view cross-section of Cypselurus Hiraii is fairly similar to the characteristics of an aerofoil. The area of application of this shape and the modifications required for practical use would be a progressive study that we would like to pursue in the future. The sideward cross section traced image of Cypselurus Hiraii is traced and image processed in NX CAD, the computational flow analysis on the model was done using ANSYS R15.0 (Fluent) software. This analysis is performed mainly to observe the behaviour of lift and drag coefficients with respect to various angles of attack on the design considered in this study. This is a 2D flow analysis which aims to imply that this cross sectional design of the fish considered can be an aerofoil. But, it is an uneven surface and the results are to be taken with at most care.

The fish section is of length 0.15m which operates at the above mentioned operating velocity, and a gauge pressure of 101325 Pa. And the meshing of this part includes an unstructured mesh because of its uneven surface nature, node pattern as triangle as it is the simplest and most convenient for an unstructured grid and as well for the section. The sizing of nodes, elements are altered to get a result with perfect grid independency. And this flow analysis has happened with K-Omega equation for efficient results, suitable nature as this is a surface analysis. From this, the analysis has performed for 12m/s, 30m/s, 60m/s, 150m/s for angles of attacks 0° to 45° with 5° intervals.

The contours of velocity and pressure at 12m/s and 150m/s are taken. Specifically, 150m/s is considered because it is the optimum & desirable velocity for this fish section. It is to be noted that 150 m/s is the average speed of commercial aircrafts. Through the graphs plotted from this analysis of C_L and C_D versus angles of attack, which are compared to standard graphs of aerodynamic coefficients of an aerofoil, the performance trend is similar. With these values of lift and drag coefficients obtained and the stall angle (25°) is observed. The results show the behaviour of the model to be similar to that of an aerofoil, supporting the ideology of the study.

6. References

[1] Jaywant H Arakeri, 2009. Fluid mechanics of fish swimming, Resonance 14(1): 32-36
[2] Park Hyungmin, Haecheon Choi. 2010. Aerodynamic characteristics of flying fish in gliding flight, Seoul. Journal of Experimental Biology 213; 3269-3279
[3] Jian Deng, Lingxin Zhang, Zhiyou Liu and Xuerui Mao. 2019. Numerical prediction of aerodynamic performance for a flying fish during gliding flight, Hangzhou. Bio inspiration & Biomimetics 14(4).