World Academics Journal of Engineering Sciences
Vol.6, Issue.1, pp.01-06, October (2019)

Research Paper

E-ISSN: 2348-635X

Shape Amelioration of Hydrofoils to Maximize Lift and Minimize Drag

Sayan Saha[1*], Mainak Lodh[2], Sayani Sen[3]

[1,2,3]Dept. of Mechanical Engineering, Future Institute of Technology, Maulana Abul Kalam Azad University of Technology, Kolkata, India

[*]Corresponding Author: sayan.saha.fit.me15@teamfuture.in, Tel.: +91-9007166152

Available online at: www.isroset.org

Received: 20/Sept/2019, Accepted: 17/Oct/2019, Online: 31/Oct/2019

Abstract— This paper presents numerical study of water flow over 3D symmetrical and unsymmetrical hydrofoils. Steady flow of water around the hydrofoils is simulated using the k-ε transport equation-based model. The results focus on variation in lift and drag forces as an aftermath of shape variation, profile of the foil, angle of attack and velocity of water flow around it. The foils were tested at different angles of attack and different velocities of flow. Standard k-ε model without any modifications were used for simplicity. This was done to ascertain the most efficient foil shape which generates sufficient lift force while producing as little drag force as possible. After determining the most efficient shape, the power required to operate these foils on a boat was also calculated so as to get an idea of the reduction of required power.

Keywords—Hydrofoil, Lift force, Drag force, Numerical study, CFD.

I. INTRODUCTION

A hydrofoil is simply a wing or a vane placed underneath the hull of a boat, the primary function of which is to lift the hull of the boat out of the water. In doing so, the drag force created by the fluid friction between the hull of the boat and water is greatly diminished, enabling the boat to travel faster while using much less propulsion power. The primary objective of using a hydrofoil in a boat is to make it more efficient. While a hydrofoil reduces the drag created by the hull of the boat with water due to fluid friction, it cannot eliminate the drag entirely. The drag force of the hull is simply replaced by the drag force endured by the hydrofoil itself during its motion through the water [1]. The main purpose of this paper is to explore different shapes in order to obtain the most optimized shape so as to get a lift force while reducing drag as much as possible. We were able to create a shape capable of generating more than 10000 N of lift force while incurring only 314 N of drag force while moving through water at a speed of 10 m/s.

The article is divided into following sections. Section I contains the introduction of the article related to hydrofoils and how they work. Section II contains work previously done on hydrofoils. Section III contains the methodology used in the analysis. Section IV describes the CFD setup used in the analysis. Section V describes results and discussions. Section VI contains figures and tables. Section VII contains the recommendation regarding this subject for and Section VIII concludes research work.

II. RELATED WORK

Previously mostly numerical studies have been performed on this topic. Most of the studies usually focus on the fluid flow characteristics and how cavitation takes place on the foil [3].

III. METHODOLOGY

This study has been performed on 3D models of foils which were created using SOLIDWORKS. The 3D models were then analysed in ANSYS CFD in order to obtain the lift and drag forces acting on the foil due to flow of water around it. Steady state numerical analysis was conducted for each foil shape. Based on the lift and drag forces generated by the foil under varying conditions of angle of attack and flow velocity of water around the foil, the foil shape was changed accordingly and the same analysis was performed on the newer shape. The goal was to develop a shape capable of generating **more than 10000 N** of lift force while producing as little drag force as possible. Firstly, a NACA0012 foil was numerically analysed. After obtaining the lift and drag forces from that foil, its shape was altered. In this way 7 subsequent foil shapes were generated all of which were examined in exactly the same conditions. After obtaining the most efficient foil shape the power required to propel the foil through freshwater was calculated.

IV. CFD SETUP

The steady state flow of water around the foil was solved with the help of commercially available ANSYS CFD 16.0.

The inlet was specified with a velocity vector of water flow while the outlet was set up as a constant pressure boundary. The remaining sides of the enclosure were defined as a static no slip wall. Standard 2 equation k-ε transport model having the values of C1 and C2 epsilons as 1.44 and 1.92 respectively along with Prandlt number as 1 was used. Hybrid initialization was used to initialize the case and 100 iterations of the calculations were performed to obtain the necessary data. For the analysis, a 2D mesh with triangular mesh elements was selected the minimum edge length of which was set at 4.0555e-003 m.

V. RESULTS AND DISCUSSION

Firstly, a NACA0012 foil (foil 1) was tested in ANSYS CFD at 2 different speeds of water flow i.e. 5 m/s and 10 m/s and at three different angles of attacks 0°, 5°, and 10°. Each of the foils have a span of 1.5 m and a maximum chord length of 0.5 m. After testing NACA0012 its dimensions were altered so as to ameliorate the shape of the foil. The shape and dimensions of all the foils are shown in Figures. 1 to 8.

The comparison of foils under different conditions of flow velocity of water and angle of attack have been shown in Figures 9 to 14.

Table 1 contains the values of lift and drag forces obtained by analyzing the foils in different flow conditions.

Figure 15 is a graphical representation of the variation in required power, angle of attack of the foil and flow speed of water around the foil. Finally, we were able create a shape which produces **more than 10000 N** of lift force while incurring a drag force **as little as 314 N**.

Discussion

Once the NACA0012 foil [Fig.1] was tested in ANSYS CFD, the upper and lower camber radiuses of the foil were altered to 3 m and 9 m respectively in order to make the foil profile more aggressive to generate higher lift force. The resultant foil shape came in the form of foil 2[Fig.2]. Adding upper and lower cambers increased the generated lift force.
This shape was further modified to generate even higher lift by reducing the radius of upper and lower cambers to 1 m and 1.5 m respectively. The resultant shape obtained is foil 3[Fig.3].

While making the upper and lower cambers more aggressive increased the lift forces, the drag forces produced were also substantially higher. For this reason, the lower portion of the shape was altered so that it may be partially symmetrical to the upper camber in order to incur lower drag forces. The new shape is foil 4[Fig.4].

The shape of foil 4 reduced the drag forces generated but suffered an appreciable loss of the lift force. The lower symmetrical part of this shape was reduced to increase the lift. Furthermore, the circular front end was changed to an elliptical shape and the chord length was also reduced to 0.35 m. The upper and lower camber radius values were changed to 0.75 m and 2 m respectively This new shape is foil 5[Fig.5].

In order to further enhance the performance of the foil, the upper and lower camber radiuses of foil 5 were changed to 1 m and 2.5 m respectively so as to make the foil even thinner to reduce the drag force. This new shape is foil 6[Fig.6]. In an effort to further improve the design, the upper camber radius of foil 6 was increased to 1.75 m in order to make the foil even thinner for drag reduction purposes. Thinner foils generate lower the amount of drag force. This new foil shape is as foil 7[Fig.7].

Foil 7 was so thin that it was at the limit of practicality. Even though the drag force is reduced by making the foil thinner, the lift force gets substantially reduced as well. To solve this issue the upper camber radius of foil 7 was decreased to 1 m. The new foil shape is foil 8 [Fig.8].

In order to obtain the best possible shape of the hydrofoil, a compromise is necessary between aggressive profile of the foil and sleekness of the foil. For this reason, we have chosen foil 8 as the best possible foil shape in this study. Foil 8 has been considered as the best design. Foil 8 is the only foil design which produces more than 10000 N lift force despite the fact that it doesn't produce the least amount of drag force of all the foil designs analysed here.

VI. FIGURES AND TABLES

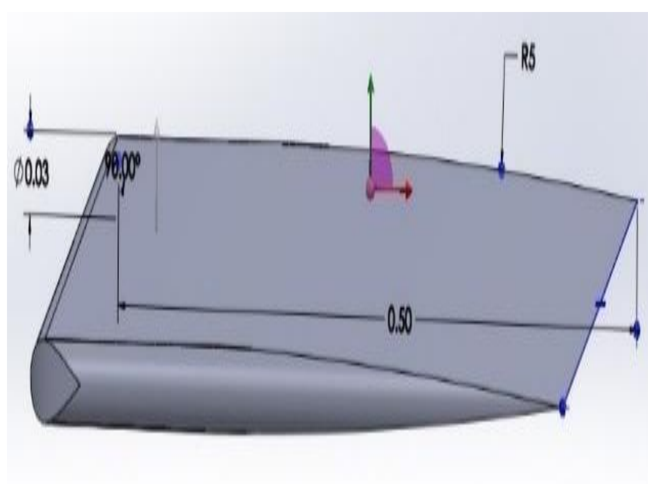

Figure 1 (Shape and dimensions of NACA0012 or foil 1)

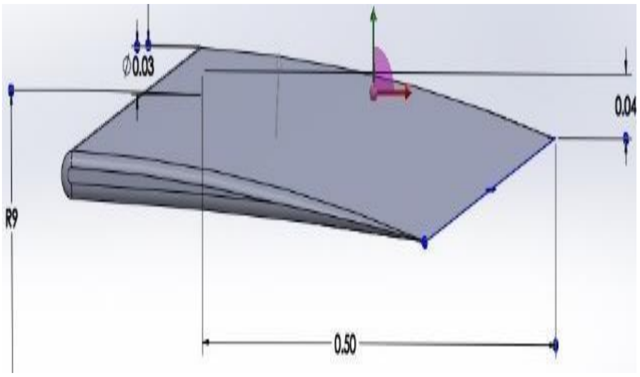

Figure 2 (Shape and dimensions of foil 2)

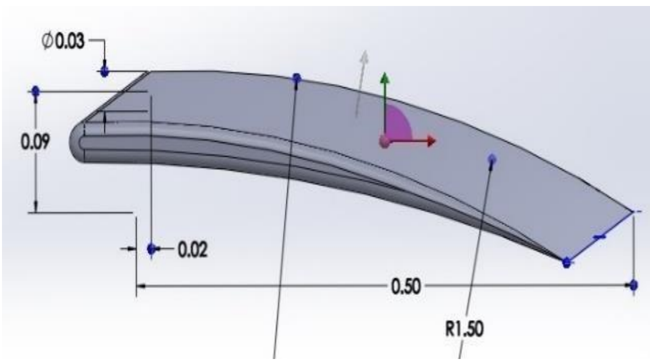

Figure 3 (Shape and dimensions of foil 3)

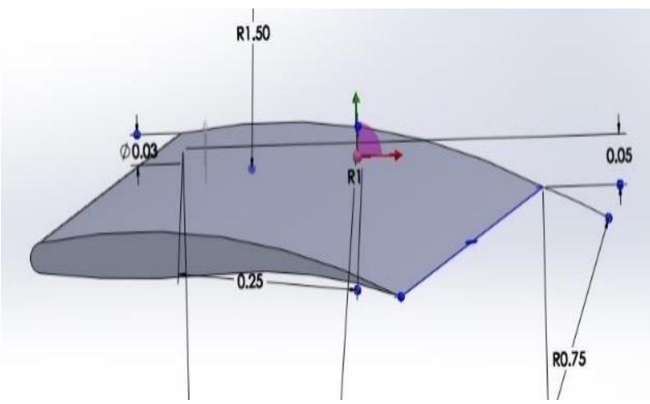

Figure 4 (Shape and dimensions of foil 4)

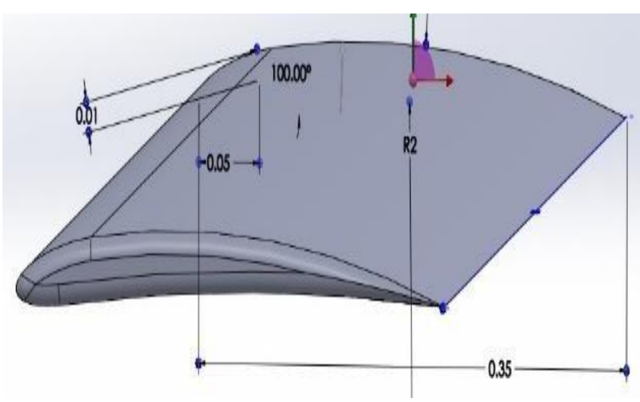

Figure 5 (Shape and dimensions of foil 5)

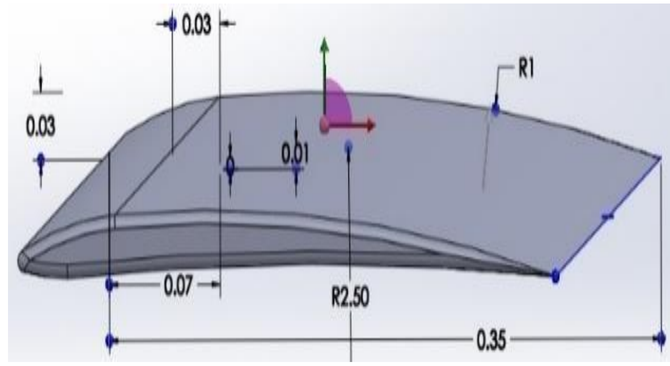

Figure 6 (Shape and dimensions of foil 6)

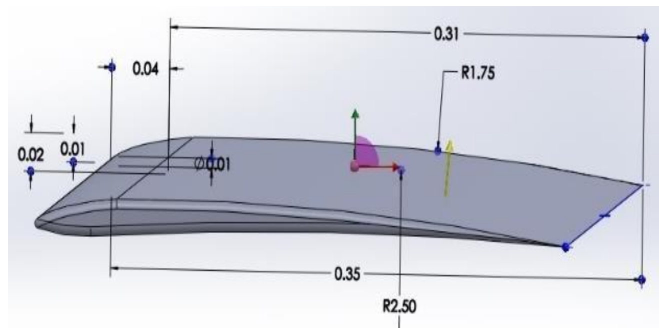

Figure 7 (Shape and dimensions of foil 7)

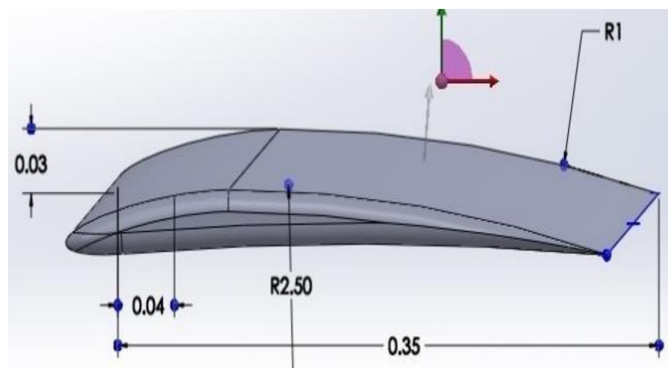

Figure 8 (Shape and dimensions of foil 8)

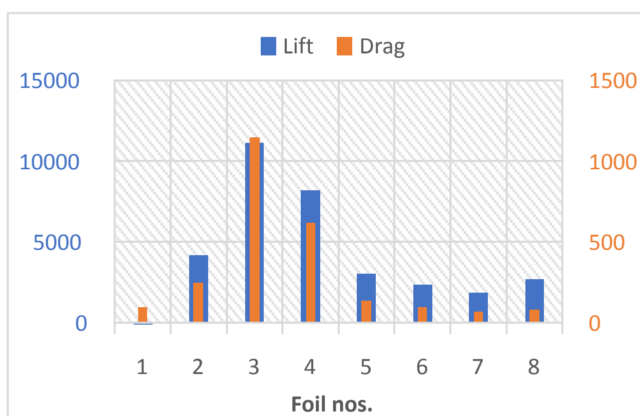

Figure 9 (Graphical comparison of lift and drag forces of all the foils at 0° angle of attack and 5m/s)

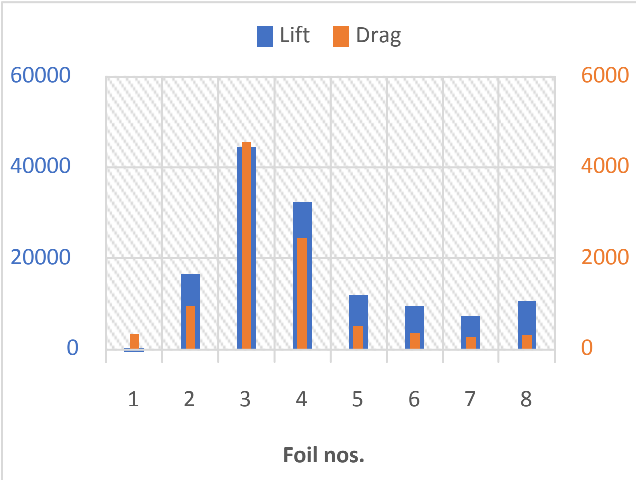

Figure 10 (Graphical comparison of lift and drag forces of all the foils at 0° angle of attack and 10 m/s)

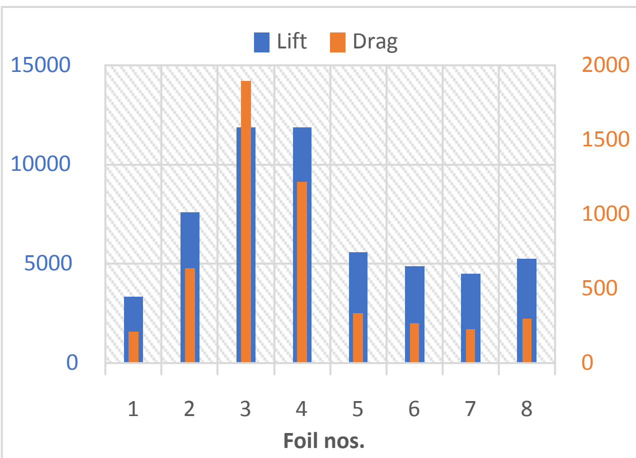

Figure 11 (Graphical comparison of lift and drag forces of all the foils at 5° angle of attack and 5 m/s)

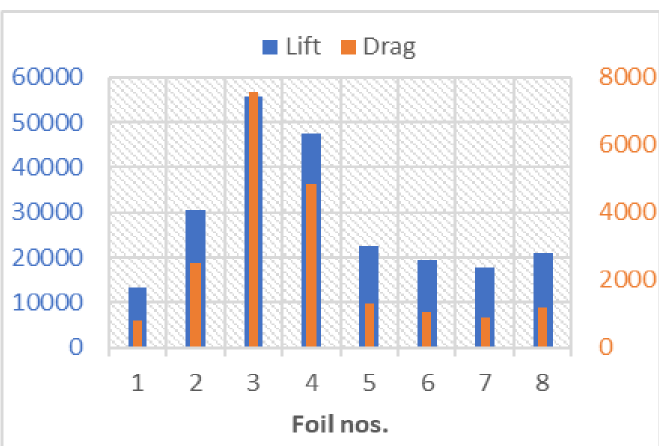

Figure 12 (Graphical comparison of lift and drag forces of all the foils at 5° angle of attack and 10 m/s)

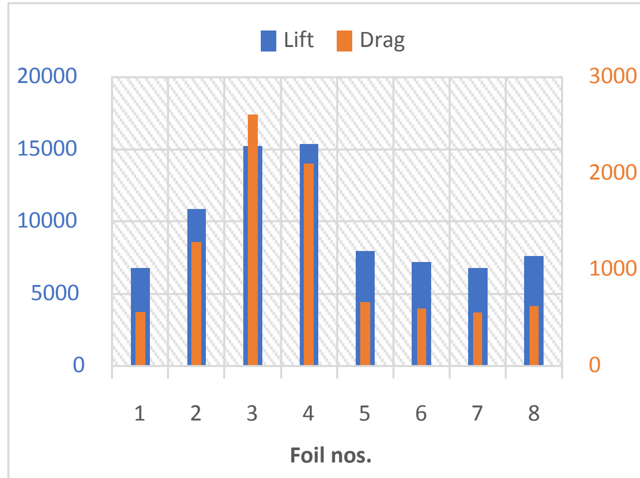

Figure 13 (Graphical comparison of lift and drag forces of all the foils at 10° angle of attack and 5 m/s)

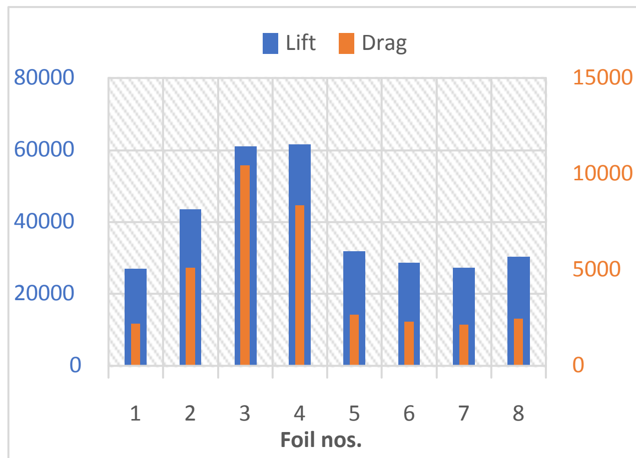

Figure 14 (Graphical comparison of lift and drag forces of all the foils at 10° angle of attack and 10 m/s)

Table 1 (Data containing lift and drag forces and power requirement as obtained from analysis of two foil 8 hydrofoils in ANSYS CFD)

Angle of attack (α)	Speeds						Monitors
	5 m/s	6 m/s	7 m/s	8 m/s	9 m/s	10 m/s	
0°	5325.94	7670.008	10440.52	13637.92	17261.864	21331.64	Lift (N)
	158.8	228.58	310.08	404.028	510.34	629.08	Drag (N)
	1.064	1.838	2.908	4.332	6.156	8.432	Power (hp)
2.5°	7918.256	11405.72	15527.58	20284.294	25676.238	31703.36	Lift (N)
	369	526.44	711.16	923.116	1162.16	1428.274	Drag (N)
	2.472	4.234	6.672	9.898	14.02	19.144	Power (hp)
5°	10486.66	15105.71	20565.84	26866.392	34008.28	41990.714	Lift (N)
	594.44	850.54	1151.8	1498.04	1889.3	2325.46	Drag (N)
	3.984	6.8408	10.808	16.064	22.792	31.172	Power (hp)
7.5°	12953.46	18661.18	25408.59	33194.904	42020.3	51886.56	Lift (N)
	894.736	1282.16	1738.34	2263.306	2856.92	3503.02	Drag (N)
	5.996	10.312	16.31	24.27	34.466	46.956	Power (hp)
10°	15173.92	21858.25	29766.87	38896.89	49243.66	60808.6	Lift (N)
	1254.74	1798.91	2440.46	3178.96	4014.114	4945.76	Drag (N)
	8.408	14.468	22.898	34.09	48.426	66.28	Power (hp)

An optimal combination of speed, required power and angle of attack is required, such that a boat with hydrofoils attached to its hull can take off with the lowest combination of speed and power. For stability purposes and ease of analysis, it is assumed that **two foil 8 hydrofoils** are attached to the boat hull. The take-off speeds are calculated by means of interpolation from the data obtained from ANSYS CFD. The variation of speed power and angle of attack is presented in the following graph i.e. Figure 15.

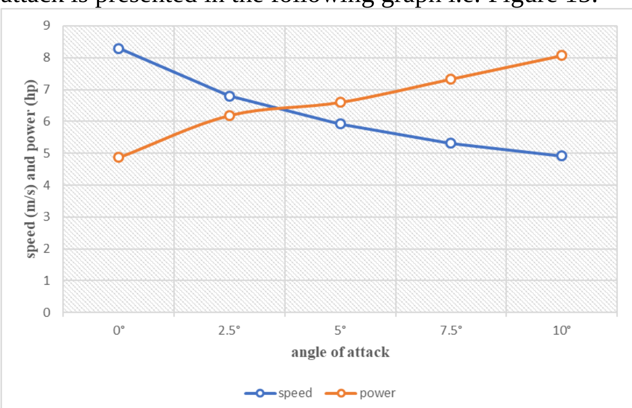

Figure 15 (Variation of speed and power required with angle of attack of the foils)

VII. RECOMMENDATIONS

Hydrofoils are the key to efficient transportation via water borne vehicles. Unlike conventional boats hydrofoils produce much less drag and hence are perfect for future boat designs. Once the optimal design is obtained, hydrofoils can be used in all sorts of boats, either to propel them with great speed or use less power to reduce environmental pollution. Numerical studies are instrumental in understanding the flow characteristics of water around the foil. Transient flow study of water flow around the foil might be instrumental for designing future more complex and more efficient foils.

VIII. CONCLUSIONS

1. Unsymmetrical foils generate higher lift and drag forces than a symmetrical foil.
2. Making the foil profile more aggressive yields higher lift forces but suffers from high drag forces as well.
3. With hydrofoils a boat uses only around 5% to 25% of power it originally required to travel through water.
4. Foil 8 is capable of generating more than 10000 N of lift force when subjected to a fluid flow of 10 m/s speed.

5. Further optimization is possible and is essential to the success of hydrofoils in the commercial industry.

ACKNOWLEDGMENT

We would like to thank all the faculty members of the department of Mechanical Engineering of Future Institute of Technology for supporting this project. We express our sincerest gratitude to Prof. Bivas Mandal for guiding us with resources and ideas.

REFERENCES

[1] Antoine Ducoin[1] Biao Huang[2] and Yin Lu Young[1], "Numerical Modelling of unsteady cavitating flows around a stationary hydrofoil", International Journal of Rotating Machinery, Volume **2012**, Article ID **215678**, pp-17. doi:10.1155/2012/215678

[2] Omer Kemal Kinaci, "A numerical parametric study on hydrofoil interaction in tandem", International Journal of Naval Architecture and Ocean Engineering, Volume **7,** Issue **1**, pp 25-40, January 2015.

[3] Carlo Negrato[1], Thomas Lloyd[1], Tom Van Terwisga[2], Guilherme Vaz[2], Rickard Bensow[*], "Numerical study of cavitation on a NACA0015 Hydrofoil: Solution Verification", VII International Conference on Computational Methods in Marine Engineering, France, MARINE 2017.

[4] J. A. Szantyr, R. Biernacki, P. Flaszyński, P. Dymarski, M. Kraskowski, M.Sc."An experimental and numerical study of the vortices generated by hydrofoils", POLISH MARITIME RESEARCH 3(61) Vol **16**; pp. **11-17**, **2009** doi:10.2478/v10012-008-0027-3.

[5] Sarraf, Christophe and Djeridi, Henda and Prothin, Sebastien and Billard, Jean-Yves Thickness effect of Naca foils on hydrodynamic global parameters, boundary layer states and stall establishment. Journal of Fluids and Structures, Vol. **26** pp. **559-578**, **2010,** ISSN 0889-9746.

[6] Teguh Putranto[1] and Aries Sulisetyono[2], "Lift-Drag Coefficient and Form Factor Analyses of Hydrofoil due to The Shape and Angle of Attack", International Journal of Applied Engineering Research Vol. **12**, pp. **11152-11156**, Number 21 **(2017)** ISSN 0973-4562

[7] Andrun, Martina & Šarić, Božidar & Basic, Josip & Blagojević, Branko, "CFD Analysis of Surface-Piercing Hydrofoil Ventilation Inception", Article Id-308948321, XXII Theory and Practice of Shipbuilding, At Trogir, Croatia 2016.

[8] Font, D., Tresanchez, M., Siegenthaler, C., Teixido, T., Palleja, T., Pradalier, Cédric, Palacin, J., "Experimental determination of the hydrofoil's angle of attack in the case of a turtle-like Autonomous Underwater Vehicle", IEEE Spain oceans, pp 1 – 5, 2011.

[9] Qin Wu, Biao Huang, Guoyu Wang, "Numerical simulation of transient flows around a 3D pitching hydrofoil", Advances in Mechanical Engineering, Vol. **7,** issue 2, November 2014, doi: 10.1155/2014/808034.

[10] Reichel, Maciej & BEDNAREK, A,"The experimental studies on hydrofoil resistance" Archives of Civil and Mechanical Engineering. Vol **7** pp 167–175. doi:10.1016/S1644-9665(12)60024-7, 2007.

AUTHORS PROFILE

All the authors pursued B.Tech in Mechanical Engineering from Future institute of Technology in Kolkata West Bengal. They were batchmates and completed their engineering in 2019.

