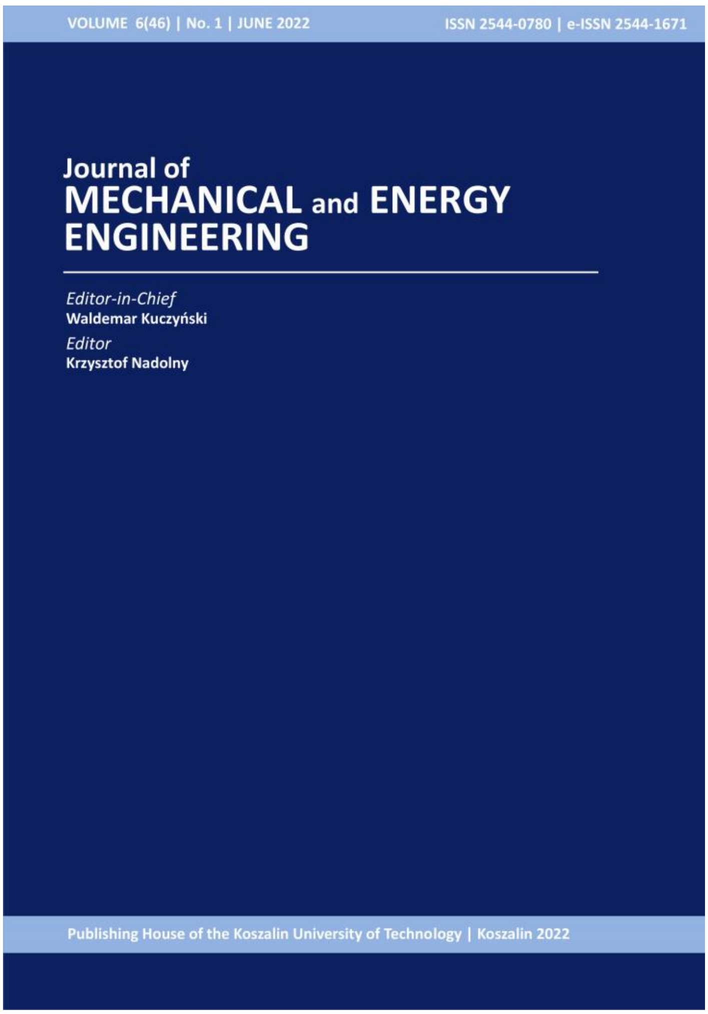
Study of flow pattern on a hydrofoil with structural and profile modification

Jithendra Sai Raja CHADA, B. S. V. Rama RAO

DOI: 10.30464/jmee.2022.6.1.27

Cite this article as:
Chada J. S. R.; Rao B. S. V. R. Study of flow pattern on a hydrofoil with structural and profile modification. Journal of Mechanical and Energy Engineering, Vol. 6(46), No. 1, 2022, pp. 27-32.



Journal of Mechanical and Energy Engineering

Website: jmee.tu.koszalin.pl

ISSN (**Print**): 2544-0780
ISSN (**Online**): 2544-1671
Volume: 6(46)
Number: 1
Year: 2022
Pages: 27-32

Article Info:
Received 21 February 2022
Accepted 7 March 2022

Journal of
MECHANICAL and ENERGY
ENGINEERING

ISSN: 2544-0780 | e-ISSN: 2544-1671
Vol. 6(46) | No. 1 | June 2022 | pp. 27-32
DOI: 10.30464/jmee.2021.6.1.27

STUDY OF FLOW PATTERN ON A HYDROFOIL WITH STRUCTURAL AND PROFILE MODIFICATION

Jithendra Sai Raja CHADA[1*], B. S. V. Rama RAO[2]

[1*] Junior Engineer, NS Engineering Company Pvt Ltd, Bachupally, Telangana, India,
email: csai0799@gmail.com
[2] HOD & Professor, Department of Mechanical Engineering, Pragati Engineering College, India

(Received 21 February 2022, Accepted 7 March 2022)

Abstract: Moving through water takes much more effort than walking through air, and this explains why ships travel much more slowly than automobiles and aircraft. Water is almost 1000 times denser than air, so most of the energy produced by a boat is taken up by dragging (water resistance). Hydrofoils travel much more quickly than ordinary boats, not by pushing through water but by raising the hull (the main body) of the boat upward so it can glide above the waves. Hydrofoil is one of the typical factors that affect the vortex structure and flow characteristics of hydraulic machinery. In order to enhance the utilisation efficiency of hydraulic machinery in marine energy, parallel grooves are proposed and applied to the hydrofoil. Following that, a numerical analysis is performed using the SST k- turbulence model, and the effects of the hydrofoil profile, the angle of attack and the flow are investigated. The profiles of NACA 0066, NACA 8412, NACA M2 and RAE 104 are considered for the study. The performance is analysed based on the lift to drag ratio. The best model from this is given with surface modification and the flow study is carried out at different angles of attack. The modified profile of NACA 8412 with parallel groves has shown the highest lift to drag ratio at a 12 degree angle of attack.

Keywords: hydrofoil, groves, angle of attack, CFD, fine mesh, validation

1. INTRODUCTION

Moving through water takes much more effort than walking through air, and this explains why ships travel much more slowly than automobiles and aircraft. Water is almost 1000 times denser than air, so most of the energy produced by a boat is taken up by dragging (water resistance). Hydrofoils travel much more quickly than ordinary boats, not by pushing through water but by raising the hull (the main body) of the boat upward so it can glide above the waves. Hydrofoils are among the fastest boats on the water, with top speeds of around 100–110 km/h (60–70 mph). Hydrofoils have been widely used as high-speed ferries and fast military patrol boats. The use of hydrofoils in fast boat applications is very useful because the travel time could be shortened. The patrol boat, the fishing vessel and the passenger boat are some of the ship types that could apply the hydrofoil as a device to support the speed increase of a ship. A hydrofoil ship uses foil that is laid beneath the surface of the water. This idea was adopted from the aerofoil used by aeroplanes. There is a similarity between hydrofoil and aerofoil working principles that is to increase the lift force. Using a hydrofoil, the displacement of a ship would decrease because the lift force supports the ship upraised. Added resistance is one of the problems in hydrofoil application. The ship would experience greater resistance compared to without the hydrofoil at a low speed. The added resistance will be reduced because the lift force will increase the displacement of the ship.

2. LITERATURE

Yue Chen *et al.* have studied the flow over a NACA 0009 hydrofoil by placing parallel groves at different positions. The hydrofoil made of 1023 carbon steel gave greater stability of flow compared to the L3T1 model [1]. Jithendra Sai Raja Chada *et al.* have studied the flow over NACA 6412 with profile modifications. The different conjugations considered are circular dimples, circular dimples above and below the surface, triangular dimples, and triangular dimples above and below the surface. Circular dimples above and below the surface have shown better performance [2]. Akhil Yuvaraj Manda *et al.* have studied the flow over NACA

6412, NACA 7412, and NACA 8412. The pressure and velocity characteristics were studied and it was concluded that NACA 8412 has greater stability of flow compared to the other two ones [3]. Jithendra Sai Raja Chada *et al.* have studied the effects of the position of the surface modification and the extension of the surface modification. The study concluded that the modifications at the rear end with 30% of extension have superior aerodynamic characteristics [4].

Hatem Kanfoudi *et al.*, have studied the cavitation characteristics and concluded that the cavitation depends on the turbulence of the flow [5]. Greeshma P Rao *et al.* have studied the cavitation effect on aerofoils with respect to lift and drag. The study stated that the cavitation effect increases with a decrease in lift and an increase in drag [6]. L G Sun *et al.* have investigated the flow over the NACA 66 hydrofoil by varying the mesh size. The study concluded that medium or fine mesh is preferred as the error is very low compared to the experimental results [7].

Jiafeng Hao *et al.* have conducted an experimental study on the effect of surface materials on the flow characteristics. Among stainless steel and aluminium, stainless steel has shown better characteristics [8]. Y. Lemini Malathi *et al.* have studied the flow parameters of the NACA 63-015 hydrofoil by using stainless steel, aluminium and brass. The stresses and deformations are smaller when structural steel is used. Yet, the hydrofoil weight increases when structural steel is used, so when aluminium is used, weight decreases and stresses are within range. So aluminium is good for hydrofoils [9].

3. DETAILED PROCEDURE

A GOE 527 hydrofoil [10] is considered for the validation of software. The validated software is applied to study the flow over NACA 66, NACA 8412, NACA M2 and RAE 104, which are considered from references. The best model is applied with surface modifications and the angle of attack is varied. A standard tank is considered to study the flow. The length of the tank is accepted as 750mm, the width of the tank is 150mm, and the height of the tank is 150mm. All the aerofoils are placed at a distance of 200 mm from the inlet to the leading edge. The chord and span of the aerofoil are accepted as 100 mm and 146 mm, respectively.

The inlet condition velocity is taken as 10 m/s in the entire study. Al 7075 material is applied to the aerofoils under study. The simulation is carried out in Solidworks Flow simulation software by creating a tank of 750mm×150mm×150mm.

4. STRATEGIES AND CONDITIONS

Geometries are applied with such principles as the continuity principle, the Bernoulli's principle and such equations as the energy equation and the kappa – epsilon equation. The calculation and evaluation are termed as follows:

1. Continuity Equation

The constitution of the fluid entering the intended profile must be same as the constitution of the fluid leaving the profile:

$$m_1 = m_2, \tag{1}$$

$$\frac{dm1}{dt} = \frac{dm2}{dt}, \tag{2}$$

$$\rho_1 A_1 U_1 = \rho_2 A_2 U_2, \tag{3}$$

$$A_1 V_1 = A_2 V_2. \tag{4}$$

2. Momentum Equation

The rate at which the momentum of a fluid particle changes must be equal to the forces acting along the flow stream

$$F = mass \times acceleration. \tag{5}$$

Now, let us consider a functional sample from the depicted fluid flow.

So, $\underline{dA}$ - cross sectional area of the functional fluid sample considered, dL - length of the functional fluid element, dW - weight of the functional fluid element, u - velocity of the functional fluid element, p - pressure of the functional fluid element.

Let us assume that the fluid is steady, non-viscous and incompressible so that the frictional losses are zero and the density of the fluid is constant.

The different forces acting on the fluid are as follows:

- pressure force acting in the direction of the flow (PdA),
- pressure force acting in the opposite direction of the flow [$(P+dP)dA$],
- gravity force acting in the opposite direction of the force ($dWsin\ \theta$).

Therefore,

$$Total\ force = gravity\ force + pressure\ force. \tag{6}$$

The pressure force is considered in the direction of the flow:

$$F_p = P\ dA - (P + dP)dA\ . \tag{7}$$

The gravity force considered in the direction of the flow:

$$\begin{aligned} F_g &= -dw\ sin\theta \\ &[W = mg = \rho\ dA\ dL\ g] \\ &= -\rho g\ dA\ dL\ sin\theta\ [sin\theta = \tfrac{dZ}{dL}] \\ &= -\rho g\ dA\ dZ. \end{aligned} \tag{8}$$

The net force is considered in the direction of the flow.

$$F = m\,a\,[m = \rho\,dA\,dL] \quad (9)$$

$$= \rho\,dA\,dL\,a. \quad (10)$$

We have:

$$\frac{dP}{\rho} + u\,dU + dZ\,g = 0, \quad (11)$$

(Euler's equation of motion).

On integrating the Euler's equation, we obtain the Bernoulli's equation

$$\int \frac{dP}{\rho} + \int U\,dU + \int dZ\,g = constant, \quad (12)$$

$$\frac{P}{\rho} + \frac{U_2}{2} + Zg = constant, \quad (13)$$

$$\frac{\Delta P}{\rho} + \frac{\Delta U_2}{2} + \Delta Z\,g = 0, \quad (14)$$

(Bernoulli's equation).

5. RESULTS AND DISCUSSION

5.1. Validation results

A GOE 527 hydrofoil [9] is considered for the validation of software. The same boundary conditions are considered to observe the error between the software and experimental data. The results are as shown in Table 1. Figure 1 shows CFD vs. experimental results validation graph.

5.2. Flow over different hydrofoils

The flow analysis is conducted by applying boundary conditions by inserting the hydrofoil in a predefined dimension of the tank. The inlet velocity is accepted as 10 m/s. The flow parameters like max velocity, pressure variation, lift and drag parameters are obtained from CFD simulation as listed in Table 2.

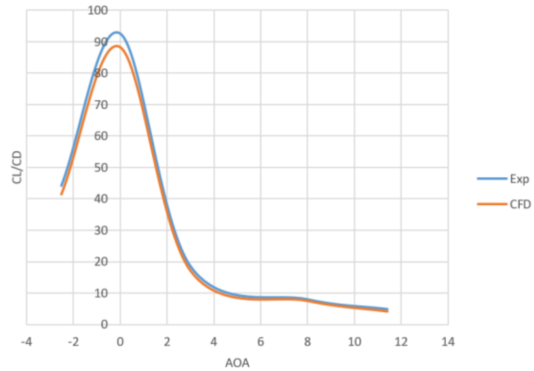


Fig. 1. CFD vs. Experimental results validation Graph

Tab. 1. Validation results

Angle of attack	-2.5	0	3	8	11.4
Cl/Cd exp	44.1	92.6	18.5	7.9	4.8
Cl/Cd CFD	41.42	88.285	17.13	7.487	4.136
% error	6.08	4.65	7.405	5.23	13.83

The maximum percentage error is observed as 13.83%. The average error between the CFD results and experimental results is observed as 7.439%.

Tab. 2. CFD simulation results of different aerofoils

	Max Velocity, m/s	Max pressure, Pa	Min pressure, Pa	*Cl*	*Cd*	*Cl/Cd*
NACA 66	11.37	155759.414	83863.792	0.004	0.011	0.403
NACA 8412	12.368	154885.06	39953.478	0.112	0.026	4.251
NACA M2	10.854	130938.249	77698.148	0.001	0.002	0.053
RAE 104	10.848	132952.724	78008.371	0.0006	0.0041	0.138

Based on the results, it is observed that NACA 8412 possesses superior characteristics among the hydrofoil profiles under consideration.

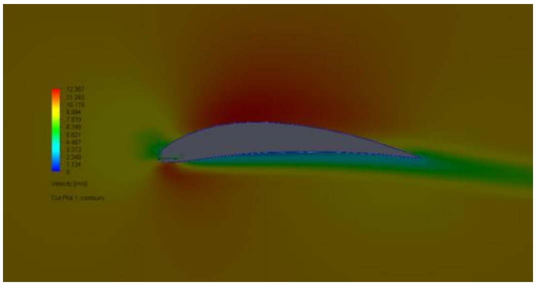


Fig. 2. Velocity distribution of NACA 8412

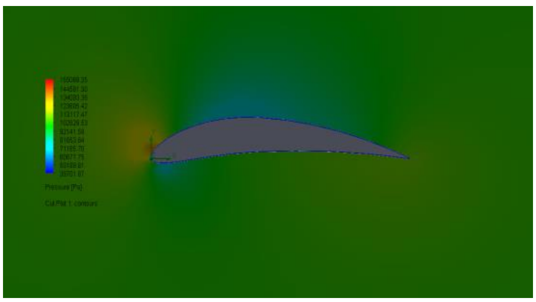


Fig. 3. Pressure distribution of NACA 8412

Now, surface modifications are applied to the hydrofoil. The parallel groves and the circular dimples above and below the surface are made on the rear end of the hydrofoil NACA 8412. It is observed that the lift and drag characteristics are enhanced compared to NACA 8412. Table 3 shows parameters for CDF simulation of NACA 8412 and Mod-8412 profiles.

5.3. Flow variation with the angle of attack

Under the same conditions, the angle of attack of the hydrofoil is varied at an interval of 5 degrees from 0 to 15. The velocity, pressure and lift and drag characteristics are compared with respect to the angle of attack. The results are tabulated in Table 4.

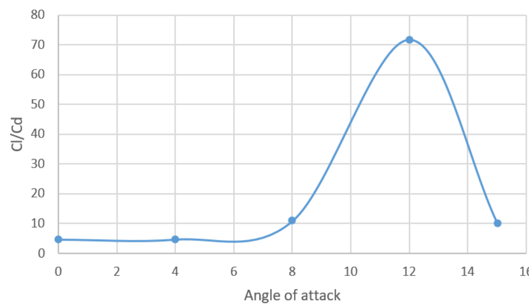


Fig. 4. *Cl/Cd* vs angle of attack graph

Tab. 3. CFD simulation of NACA 8412 and Mod-8412 profiles

	Max Velocity, m/s	Max pressure, Pa	Min pressure, Pa	*Cl*	*Cd*	*Cl/Cd*
NACA 8412	12.368	154885.06	39953.478	0.112	0.026	4.251
Mod-8412	12.208	155628.756	38108.319	0.114	0.025	4.55

Tab. 4. Results of flow variation with the angle of attack

AOA	Max Velocity, m/s	Max pressure, Pa	Min pressure, Pa	*Cl*	*Cd*	*Cl/Cd*
0	12.208	155628.756	38108.319	0.114	0.025	4.55
4	12.141	231461.632	54005.900	0.134	0.029	4.567
8	12.057	131801.769	61545.710	0.185	0.017	10.802
12	12.625	148133.996	46221.885	0.296	0.004	71.708
15	11.356	229293.661	45748.29	0.243	0.023	10.153

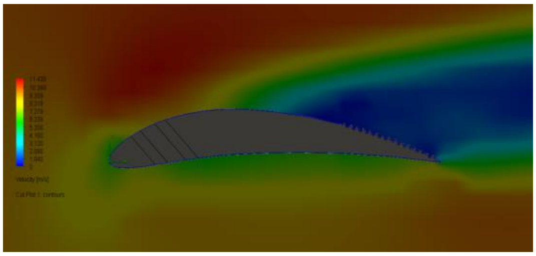


Fig. 5. Velocity distribution on Mod-NACA 8412 at 12°

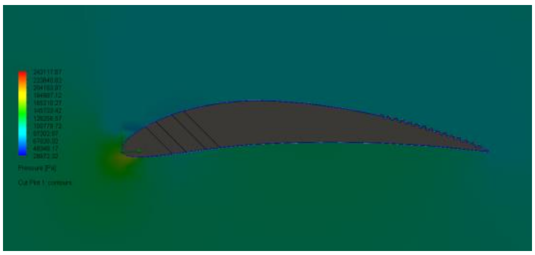


Fig. 6. Pressure distribution on Mod-NACA 8412 at 12°

6. CONCLUSIONS

The work presented targets the evaluation of the flow behaviour over the surface of a hydrofoil with dimples spread at the rear end of the geometry and parallel groves. The intention is to generate a hydrofoil geometry with optimum values in terms of all the possible conditions. The hydrofoil geometries are analysed for velocity lineation, pressure lineation, coefficient of lift, and coefficient of drag. The modified profile of NACA 8412 with parallel groves has shown enhanced performance at the 120 angle of attack. The cl/cd ratio is noted as 71.70882044. This appreciable increase in terms of all the applicable conditions makes it the optimum design of the hydrofoil geometry.

References

1. Chen, Y.; Zhang, W.; Fang, D.; Sun, M.; Liu, J.; Song, D.; Zhang, X. Vortex Suppression and Flow Pattern Analysis of a Hydrofoil with Parallel Grooves. Processes 2021, 9, 816.
2. Chada, J. S. R., Satyanarayana, K. V. V., Kumar, G. V., Shaheen, M., and Bhaskar, A. P. (2020). Flow Variation on NACA 6412 Aerofoil with Surface Modifications. i-manager's Journal on Mechanical Engineering, 10(3), 31-37.
3. Manda Akhil Yuvaraj, Chada Jithendra Sai Raja, Surapaneni Sambhu Prasad, Geeri Satish Flow behaviour on aerofoils using CFD. Journal of Mechanical Engineering, Automation and Control Systems, Vol. 1, Issue 1, 2020, p. 26-36.
4. S. Geeri, S. P. Surapaneni, J. S. R. Chada, and A. Y. Manda, "Studies Based on CFD Behaviour of Aerofoil and Regression Analysis", Engineering Science & Technology, vol. 3, no. 1, pp. 55-68, Oct. 2021.
5. Hatem KANFOUDI, Ridha ZGOLLI, Simulation of unsteady cavitating flow aound an hydrofoil, The fifth International Renewable Energy Congress IREC 2014.
6. Greeshma P Rao , Likith K, Mohammed Naveed Akram,Adarsh Hiriyannaiah, Numerical Analysis of Cavitating Flow over A2d Symmetrical Hydrofoil, International Journal Of Computational Engineering Research (ijceronline.com) Vol. 2 Issue. 5.
7. L G Sun et al, Numerical investigation into cavitating flow around a NACA66 hydrofoil with DCM models,IOP Conf. Ser.: Earth Environ. Sci.2019.
8. Hao, J., Zhang, M. & Huang, X. Experimental Study on Influences of Surface Materials on Cavitation Flow Around Hydrofoils. Chin. J. Mech. Eng. 32, 45 (2019).
9. Y.LEMINI MALATHI , B.GEETA CHANDRA SEKHAR, Design and Optimization of Hydrofoil using CFD and Structural Analysis, International Journal of Scientific Engineering and Technology Research Volume.05, IssueNo.44.
10. Byrne G, Persoons T, Kingston W. Experimental validation of lift and drag forces on an asymmetrical hydrofoil for seafloor anchoring applications. Journal of Ocean and Climate. January 2019.

Biographical notes

Biagraphical notes were not provided.