Lift-drag coefficient and form factor analyses of hydrofoil due to the shape and angle of attack

Article *in* International Journal of Applied Engineering Research · January 2017

2 authors:

Teguh Putranto
Institut Teknologi Sepuluh Nopember

Aries Sulisetyono
Institut Teknologi Sepuluh Nopember

International Journal of Applied Engineering Research ISSN 0973-4562 Volume 12, Number 21 (2017) pp. 11152-11156

Lift-Drag Coefficient and Form Factor Analyses of Hydrofoil due to The Shape and Angle of Attack

Teguh Putranto[1] and Aries Sulisetyono[2]

[1,2]Department of Naval Architecture, Sepuluh Nopember Institute of Technology, Surabaya, Indonesia.

[1]Orcid: 0000-0003-2526-1963, [2]Orcid: 0000-0003-0798-130X

Abstract

The objective of this research is to obtain the NACA foil type which is appropriate to be applied in a ship using hydrofoil. There are three parameters that are used as criterion to determine the best foil. Lift-drag coefficient effect on the lift ability and the resistance of foil. Form factor means that the block coefficient of foil which is as the part of drag coefficient. This research is carried out using numerical method. There are three foils based on the NACA series comprise NACA 2212, NACA 2309, and NACA 4712. Meanwhile, the angle of attack is varied in 0, 5, 10, 15, 20, and 25 degrees. For the form factor analysis, the fluid flow is simulated in low speed. For the lift-drag coefficient, the fluid is run in high speed. The characteristics of optimum foil cover: 1) at the specified speed, the foil have the maximum lift force and is able to withstand the drag; 2) the foil have minimum form factor; and 3) the stall phenomenon does not occur in the selected NACA foil. From these characteristics and analyses of the foil, the NACA 4712 with the angle of attack 15 degrees is the best foil to be applied in hydrofoil.

Keywords: lift-drag coefficient, NACA foil, fluid flow, angle of attack, stall

INTRODUCTION

The use of hydrofoil in fast boat application is very useful because the traveling time could be shortened. Patrol boat, fishing vessel, and passenger boat are some of ship types that could apply the hydrofoil as the device to support the speed increase of a ship (Wu, 2017). Figure 1 shows a hydrofoil concept.

Hydrofoil is a ship using foil which is laid beneath the surface of the water. This idea is adopted from an aerofoil used by aeroplanes. There is a similarity of hydrofoil and aerofoil working principle that is to increase lift force (Ramesha, 2011). Using hydrofoil, the displacement of ship would decrease because lift force supports the ship upraised.

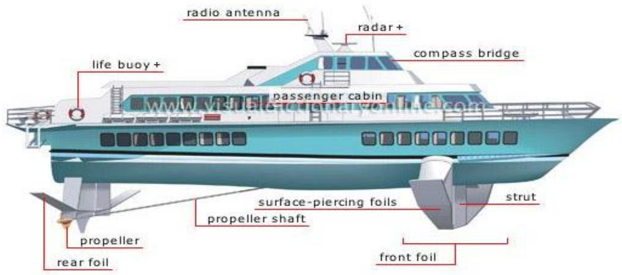


Figure 1: A Hydrofoil Concept

Added resistance is one of the problem in hydrofoil application. The ship would experience greater resistance compared to without hydrofoil one in the low speed. The added resistance will reduce because the lift force will raise the displacement of the ship (Putranto, 2015).

LITERATURE REVIEW

Computational Fluid Dynamics (CFD) is a method usually used to solve the fluid flow problem (Lakshman et. al., 2014). The fluid flow analysis could be applied to determine the foil shape producing the optimum lift and drag force. In the case of wind turbine development, the selection of foil could be able to improve the energy efficiency from the various of MARIN designed turbines (Make, 2015).

There are several steps to vary the foil shape that is by changing the chord and chamber. The first step is by changing the chamber width only. The proper selection of chamber will give a contribution for the enhancement of lift force but the drag force will increase as the chamber widened (Hoke, 2015). The effect of fluid structure interaction is analyzed in order to obtain the influence of the lift and drag force changes. The second step is by changing the chord length and chamber width together. Lift and drag coefficient are obtained based on the variation of both components (Shinde et. al., 2014). On the other hand, the angle of attack is also varied to produce the same magnitude of lift force although the length and width of the foil are different. In addition, lift and drag force are considered, there is one more phenomenon that should be avoided is the stall. This phenomenon causes that the stability of foil is unstable because the ratio between drag and lift coefficient is the most of its value (Putranto, 2017).

Angle of attack is the angle between a reference line on a body (often the chord line of an airfoil) and the vector representing the relative motion between the body and the fluid through which it is moving. The critical angle of attack is the angle of attack which produces maximum lift coefficient. This is also called the "stall angle of attack". Below the critical angle of attack, as the angle of attack increases, the coefficient of lift (Cl) increases. Conversely, above the critical angle of attack, as angle of attack increases, the air begins to flow less smoothly over the upper surface of the airfoil and begins to separate from the upper surface.

RESEARCH METHOD

LIFT AND DRAG FORCE BY USING CFD

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows.

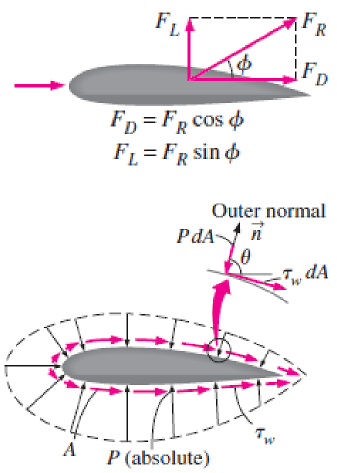


Figure 2: Pressure and Viscous Force Acting on a Two-Dimensional Body and the Resultant Lift and Drag Force

The Navier-Stokes equations are the basic principal of CFD problem to solve the fluid flow analysis. These equations can be simplified using approximation considering the creeping flow, inviscid region flow, irrotational flow, and boundary layer. Finite volume is the method based on the Eulerian approach that is useful to accomplish these equations. Figure 2 shows the illustration of force acting around the foil.

Equation 1 and 2 are formula to calculate the lift (F_L) and drag (F_D) force as function of the normal and shear stress. Shear stress is the component of stress coplanar with a material cross section. Shear stress arises from the force vector component parallel to the cross section. Normal stress, on the other hand, arises from the force vector component perpendicular to the material cross section on which it acts.

$$F_D = \int_A dF_D = \int_A \left(-p\cos\theta + \tau_w \sin\theta\right) dA \qquad (1)$$

$$F_L = \int_A dF_L = \int_A \left(p\cos\theta + \tau_w \sin\theta\right) dA \qquad (2)$$

where:

p = normal stress (N/m^2)

τ_w = shear stress (N/m^2)

In a continuum fluids, the Froude Number (Fr) is a dimensionless number obtained from the ratio of the flow inertia to the external field flow. This one can be used to determine the classification of ship speed. Equation 3 shows the Froude Number formula.

$$F_r = \frac{u}{\sqrt{gL}} \qquad (3)$$

where:

u = ship speed (m/s)

g = gravity (m/s^2)

L = length of waterline of ship (m)

FORM FACTOR

Using CFD, the viscous force of foil can be known which is usually called as drag force. The simulation has to be applied in one fluid so that the viscous resistance is only occur in this case. After the viscous force is obtained, the viscous coefficient can be calculated using Equation 4 as follows.

$$C_V = \frac{R_V}{\frac{1}{2}\rho S V^2} \qquad (4)$$

Friction coefficient affected by the normal force acting the model. International Towing Tank Conference (ITTC) give a formula to calculate the friction coefficient that can be shown in Equation 5.

$$C_F = \frac{0.075}{\left(\log R_e - 2\right)^2} \qquad (5)$$

Because friction coefficient is only caused by the normal pressure, viscous coefficient can be calculated by adding the form factor at the friction coefficient calculation. The relation between viscous and friction coefficient can be shown in Equation 6 which the form factor is a part from the two coefficients.

$$C_V = (1+k)C_F \qquad (6)$$

where:

C_V = viscous coefficient

C_F = friction coefficient

$1+k$ = form factor

In the other hand, the form factor is given as a function of the thickness (t) per chord (c) ratio of the section from Hoerner which is shown in Equation 1. This equation is only used to the angle of attack 0 degree. For any degree, this formula cannot be used because the variable of degree is not given. The value of form factor from Equation 1 will be compared with the CFD.

$$(1+k) = \left(1 + 2\left(\frac{t}{c}\right) + 60\left(\frac{t}{c}\right)^4\right) \qquad (1)$$

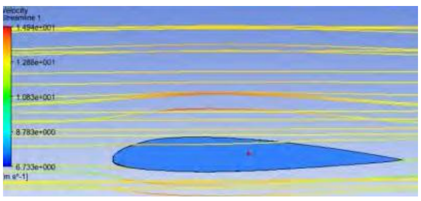
For 5, 10, 15, 20, and 25 degrees, the form factor can be only obtained from the viscous per friction coefficient ratio.

RESULT AND ANALYSIS

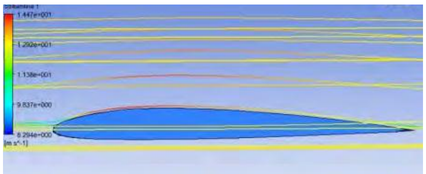
In order to obtain the optimum NACA foil, this research divides the NACA foil into the several of its shape. There are three kinds of NACA foil which is described as can be shown in Table I:

Table I: NACA Foil Specifications

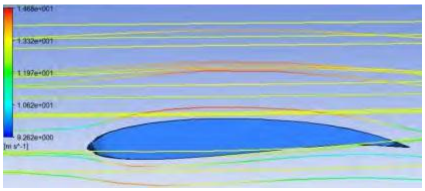
No.	Specifications	NACA 2212	NACA 2309	NACA 4712
1.	Span	3.02 m	3.02 m	3.02 m
2.	Chord Length	2.85 m	2.85 m	2.85 m
3.	Area	20.1 m^2	19.7 m^2	21.1 m^2



(a)

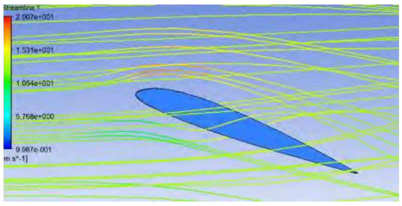


(b)

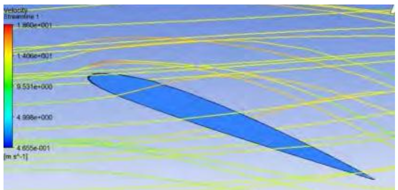


(c)

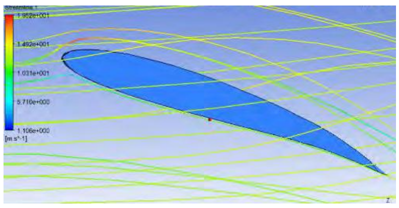
Figure 3. Type of Foil a) NACA 2212, (b) NACA 2309, and (c) NACA 4712 at the Angle of Attack 0 degree



(a)



(b)



(c)

Figure 4. Type of Foil a) NACA 2212, (b) NACA 2309, and (c) NACA 4712 at the Angle of Attack 20 degrees

The angle of attack of foils are consisted of 0, 5, 10, 15, 20, and 25 degrees. The speed of fluid flow is varied in 0.25, 0.5, 0.75, and 1.00 m/s for low speed and 2.5, 5.0 7.5, 10.0, and 12.5 m/s for high speed.

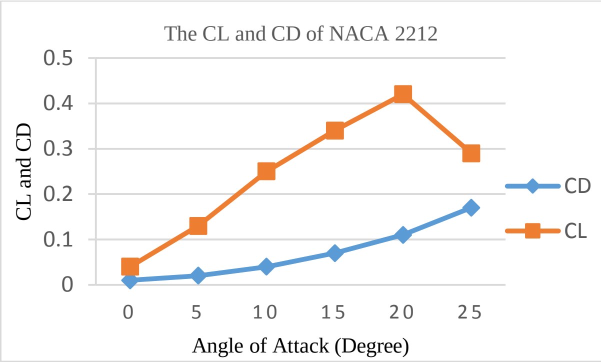


Figure 5. Lift and Drag Coefficient for NACA 2212 in Various of Angle of Attack

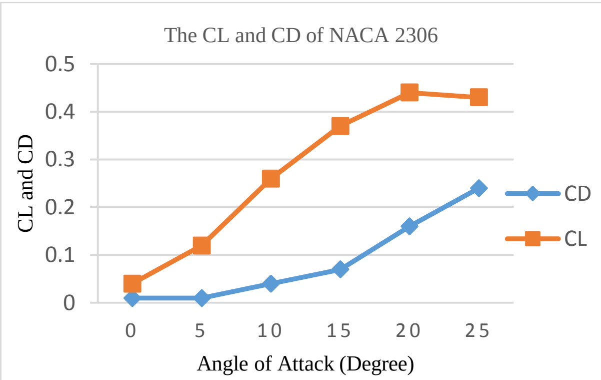


Figure 6. Lift and Drag Coefficient for NACA 2309 in Various of Angle of Attack

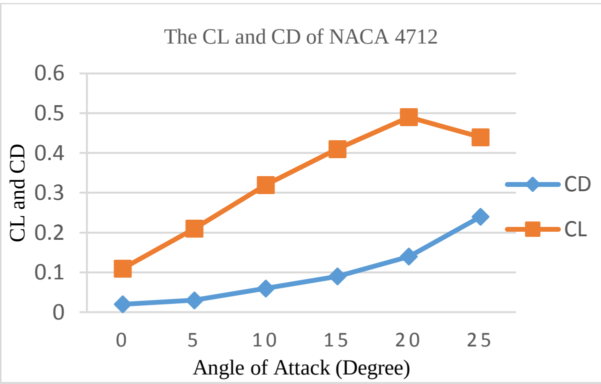


Figure 7. Lift and Drag Coefficient for NACA 4712 in Various of Angle of Attack

Figure 3 and 4 are the plot of streamline meaning that the velocity fluid around the foil. There are streamlines that intersect with foil because the flow does not pass through the flow. It means that the fluid flowing outside the foil. Because of the speed difference on each side, the pressure arises and causes the lift and drag.

The potential velocity is greater at the angle of 0 degree than the angle of 20 degrees. Conversely, the stream function is greater at the angle of 20 degrees than the angle of 0 degree.

The lift force of NACA 4712 is greatest than the other one. Although the drag force is also great, the ship speed does still reach in the service speed. Adding the drag force of foil, the displacement of ship reduces because lift force will raise the ship hull.

In order to obtain the effective of chord length of NACA 4712, there are 3 (three) various of the chord length consisted of 2.5, 3.0, and 3.5 meters. Figure 8 shows the graph of CL and CD of NACA 4712 in various of chord length.

The fluid flow is simulated in the low speed in order to obtain the viscous resistance which is without wave resistance. From CFD, the magnitude of drag force is viscous resistance. Then, the form factor can be calculated by dividing between viscous coefficient to friction coefficient.

By using formula from the ITTC, friction coefficient can be obtained as can be shown in Table II which the flow speed is 1 m/s.

Table II: The C_F of NACA 4712

No.	Chord Length	Re ($x10^5$)	Fr	C_F
1.	2.85 m	27.94	0.187	0.0168
2.	3.00 m	29.41	0.182	0.0167
3.	3.50 m	34.31	0.169	0.0165
4.	2.50 m	24.50	0.200	0.0170

Table II shows that the maximum Reynolds Number (Re) is still in laminar flow region. The laminar flow is actually determined by the length of model and speed if the viscosity of fluid is assumed in the same condition. The length of the model is getting longer, the flow region will be in turbulent condition. The speed of the model is getting faster, the flow region will also be in turbulent condition. For the low speed case, the foil has to be simulated in the low Froude Number (Fe). From Table II the maximum Fe is still in the criterion of low speed 1 m/s.

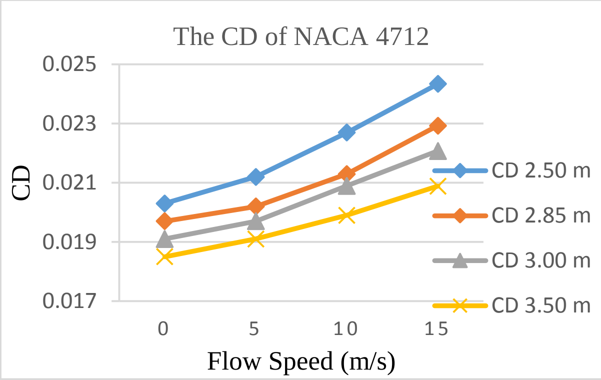


Figure 8. Lift and Drag Coefficient for NACA 4712 in the Various of Chord Length

The drag coefficient, which can be called as viscous coefficient, is used to calculate the magnitude of form factor. The calculation of form factor is easily obtained from the dividing between viscous coefficient to friction coefficient. Table III shows the form factor of foil NACA 4712 obtained from CFD and empirical formula, Hoerner.

Table III: The Form Factor of NACA 4712

No.	Chord Length	Form Factor		Error
		CFD	Hoerner	
1.	2.50 m	1.432	1.496	4.27%
2.	2.85 m	1.365	1.407	3.03%
3.	3.00 m	1.322	1.379	4.17%
4.	3.50 m	1.266	1.310	3.41%

CONCLUSION

From the result and analysis, the conclusion of the research is as follows:

1. There is a maximum point of C_L to the various of angle of attack which is a stall phenomenon occurred in the angle of 20 degrees in each of NACA foil type.
2. NACA 4712 has the greatest value of C_L which the magnitude is 0.49 meaning that the force is 49% from the multiple between the velocity pressure and the surface applied the force.
3. The form factor of NACA 4712 is smaller when the chord length is longer. It occurs because the chamber of the chord length varied is same in 0.5 m. The foil shape does close to the slender form which the drag force decreases.

ACKNOWLEDGEMENT

This research was supported by Institute for Research and Community Services (LPPM) Sepuluh Nopember Institute of Technology (ITS), Indonesia through Pemula Grant in 2017. We thank our colleagues from Department of Naval Architecture ITS who provided insight and expertise that greatly assisted and advised the research.

REFERENCES

[1] Hoke, C. M., Young, J., Lai, J. C. S., 2015, Effects of time-varying camber deformation on flapping foil propulsion and power extraction, Journal of Fluids and Structures, Volume 56, July 2015, Pages 152 – 176.

[2] Hoerner, S.F., 1965, Fluid-dynamics drag, Published by the author, New York

[3] Lakshman, A., and Sivakumar, R., 2014, CFD Analysis of Double Ramp in Hypersonic Flows, International Journal of Applied Engineering Research, Volume 9, Number 26, Pages 8883 – 8886.

[4] Make, M., and Vaz, G., 2015, Analyzing scaling effects on offshore wind turbines using CFD, Renewable Energy, Volume 83, November 2015, Pages 1326 – 1340.

[5] Putranto, T., Suastika, K., and Gunanta, J., 2017, Intact Stability Analysis of Crew Boat with Variation of Deadrise Angle, IPTEK Journal of Proceedings Series, Volume 2, Pages 124 – 127.

[6] Putranto, T., and Sulisetyono, A., 2015, *Analisa Numerik Gerakan dan Kekuatan Kapal Akibat Beban Slamming Pada Kapal Perang Tipe Corvette*, Jurnal KAPAL Volume 12, Number 3, Pages 158 – 164.

[7] Ramesha, D. K., Kumarswamy, N.S., Kishore, H.N., Vidya S., and Premkumara, G., 2011, Numerical Simulation of Cavitating Flow over Oscillating Hydrofoils, International Journal of Applied Engineering Research, Volume 6, Number 10, Pages 1273 – 1283.

[8] Shinde, S. B., and Sivakumar, R., 2014, CFD Analysis of Fluid Flow Around An Elliptic Cylinder, International Journal of Applied Engineering Research, Volume 9, Number 26, Pages 8887 – 8890.

[9] Wu Q., Wang, Y., Wang, G., 2017, Experimental Investigation of Cavitating Flow-Induced Vibration of Hydrofoils, Ocean Engineering, Volume 144, Pages 50 – 60.