Jurnal Kejuruteraan SI 4(1) 2021: 71-75
https://doi.org/10.17576/jkukm-2021-si4(1)-09

Computational Fluid Dynamics Analysis of Seven Selected Hydrofoils for Different Angle of Attack

(Analisa Pengiraan Dinamik Bendalir Bagi Tujuh Hydrofoil Terpilih untuk Sudut Serang Berbeza)

Puteri Nurfarah Adawiyah Taslin[a], Aliashim Albani[a*], Mohd Zamri Ibrahim[a], Mohd Azlan Musa[a], Zulkifli Mohd Yusop[a] & Mohd Afifi Jusoh[a]

[a]*Renewable Energy & Power Research Interest Group (REPRIG), Eastern Corridor Renewable Energy Special Interest, Faculty of Ocean Engineering Technology and Informatics, University Malaysia Terengganu, Kuala Nerus, Terengganu, Malaysia*

**Corresponding author: a.albani@umt.edu.my*

Received 28 March 2021, Received in revised form 24 April 2021
Accepted 30 August 2021, Available online 30 September 2021

ABSTRACT

This paper aims to conduct a Computational fluid dynamics (CFD) analysis of seven selected hydrofoils at different angles of attack (AOA) for application in ocean current energy converter blades. The hydrofoil models are NACA0012, NACA0015, NACA2412, NACA2414, NACA2415, NACA 4412, and E387. The geometry was managed using the Design Modeler tool. The fluid flow simulation was carried out using ANSYS Fluent. The result showed the performance of E387 is better than other hydrofoils as it gives better lift force with the least drag force, resulting in better hydrodynamics of turbine blade. Overall, the value of lift and drag coefficients at 10^o was more consistent than 0^o and 20^o AOA.

Keywords: CFD; hydrofoil; angle of attack; fluid dynamics; lift and drag coefficient

ABSTRAK

Kertas kerja ini bertujuan untuk menjalankan analisa pengiraan dinamik bendalir (CFD) terhadap tujuh hydrofoil terpilih pada sudut serang (AOA) yang berbeza untuk pengunaan bilah dalam penukaran tenaga arus lautan. Model hydrofoil yang dikaji adalah NACA0012, NACA0015, NACA2412, NACA2414, NACA2415, NACA 4412, dan E387. Geometri disediakan menggunakan perisian Ansys Design Modeler. Simulasi aliran bendalir dijalankan menggunakan perisian ANSYS Fluent. Dapatan kajian menunjukkan prestasi E387 adalah lebih baik daripada hydrofoil lain kerana ia memberikan daya tujah yang baik dengan daya seretan paling sedikit, menyebabkan hidrodinamik yang baik pada bilah turbin. Secara keseluruhan, pekali tujah dan seretan pada 10^o lebih stabil daripada sudut serang 0^o dan sudut serang 20^o.

Kata kunci: Hydrofoil; sudut serang; dinamik bendalir; pekali tujah dan seretan

INTRODUCTION

An hydrofoil is a curved surface pattern that provides the most significant lift and drag forces required by the propeller blades, airplane wings, renewable energy converter blade, etc. The hydrofoil is also very helpful since it suspends the whole heavy aircraft weight in the air. When moved at the necessary velocity, the hydrofoil creates a beneficial aerodynamic force (lift and drag) due to the relative movement between the pressure difference (perpendicular). The lift force acting on the hydrofoil and the drag force acting opposite the motion direction is due to the shear force (parallel), resulting in more energy consumption to overcome the drag force (Yunus, 2014). The basic principle used is Bernoulli's principle because the flow of air over the hydrofoil creates a low- pressure zone on the upper surface and a high-pressure zone on the lower surface, so due to the pressure difference, the generated force is known as lift force (Jain et al., 2016).

The lift coefficients and drag coefficient on various hydrofoil sections were studied in (Chumbre, et al., 2015). They concluded that the higher the AOA resulting in, the higher lift power, but simultaneously higher the drag force. The practical and CFD analysis performed by (Chandrakant et al., 2012) stated that the lift coefficient increased by raising the lift angle at the Reynolds number's low value. The lift coefficient decreases after a 12^{o} AOA and a small increment of the drag coefficient. The CFD analysis on one particular hydrofoil, NACA0012, with different AOA, showed the lift coefficient increased rapidly (Dash, 2016). The drag coefficient also increased but not as rapidly as the coefficient of lift. The coefficients increased up until 10^{o} of angle before starting to decrease (Dash, 2016). A study on NACA0012 and NACA2412 using three different methods, Spallart-Allamaras, k-omega, and k-epsilon, was conducted by (Oukassou et al., 2019), where the result revealed that NACA2412 had a higher overall power output than the NACA0012. NACA2412 was created as a powerful turbine blade and is more efficient than NACA0012 (Oukassou et al., 2019).

The aerodynamic efficiency of NACA0012, NACA0015, NACA0018, and NACA0021 hydrofoils used in vertical axis wind turbines were studied by (Sauvageat & Rolin, 2016). The aerodynamic performance of the hydrofoil was analyzed by (Liang & Li, 2018) by combining the XFOIL program and Viterna–Corrigan poststall model. The hydrofoil's performance was validated with computational fluid dynamic simulations, where the results showed that, compared to an unoptimized NACA0015 hydrofoil, the optimized hydrofoil's lift to drag ratio was improved over a wide range of attack angles, and the stall performance was gentler (Liang & Li, 2018).

NACA2412 was found as the best hydrofoil after the different AOA affected the lift, and the drag coefficient was studied by (Kulshreshtha et al., 2019). The increment of the AOA, resulting in the decrement of the ratio of the coefficient of lift to the coefficient of drag (Kulshreshtha et al., 2019). NACA2415 and NACA4410 were examined in (Rajakumar & Ravindran, 2012), using the k-omega SST method, which was carried out using various AOA in the range from 1o to 20o. The E387 was studied by (Emmerson & Verstraete, 2020) on the selected AOA invariant of -32^{o} to 33^{o} using Reynolds number of 300 000. The drag coefficient slowly increased by 0.002 from 5.3^{o} of AOA, and towards zero lift coefficient with the pressure measurement accurately predicts the moment coefficient, although around stall is underpredicted by 0.02. (Emmerson & Verstraete, 2020).

METHODOLOGY

COMPUTATIONAL FLUID DYNAMIC ANALYSIS

In this study, ANSYS Fluid Flow (Fluent) is used as a solver domain. The coordinates required to generate the hydrofoil are imported from the National Advisory Committee for Aeronautics (NACA) website. The models are developed using Design Modeler, meshing, and Spallart-Allmaras turbulence method simulated in Fluent for data collection. The most fundamental aspect of every CFD issue is the details of the boundary conditions thereof. Boundary conditions for the problem and initial inputs are taken to perform the simulation, and this is presented in Table 1.

TABLE 1. Input boundary conditions

No	Input	Value
1	Velocity of flow	1.0 m/s
2	Operating pressure	101325 Pa
3	Model	Spallart-Allmaras
4	Density of fluid	1023.6 Kg.m^3
5	Kinematic viscosity	1.07x10^{-5} Kg/m. s
6	Fluid	Seawater as Ideal

MESHING DETAILS

Meshing is an essential part of the simulation method in which complex geometries are divided into simple components that can be used as separate local approximations of the larger domain. The mesh affects the simulation's precision, convergence, and speed. Besides that, since meshing usually takes a large portion of time to achieve simulation outcomes, the quicker and more precise the solution, the better and more automated the meshing tools (Thompson & Thompson, 2017).

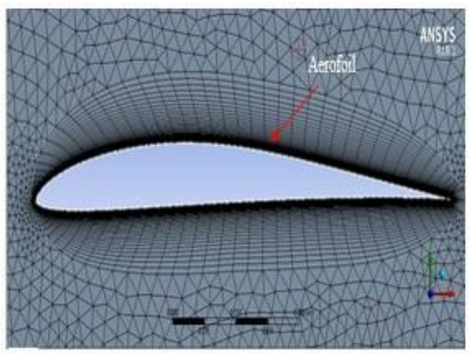

FIGURE 1. Meshing structure using ANSYS on selected hydrofoil

The meshing is developed to be fine as it was closer to the hydrofoil and coarser more remote far from the hydrofoil. The meshing type is the triangle method followed by sizing down the hydrofoils area and boundary area. The inflation method is also applied to the face for both hydrofoils and boundary.

TABLE 2. Meshing details

Hydrofoils	Number of nodes	Number of elements
NACA0012	170689	86997
NACA0015	287700	26604
NACA2412	172362	88500
NACA2414	155302	35750
NACA2415	150982	77183
NACA4412	170689	86997
E387	132809	70056

RESULT AND DISCUSSION

The results were obtained using k-omega SST method, input values such as density and kinematic viscosity of seawater, and seawater as our ideal fluid. The simulation was conducted in ANSYS Fluid Flow (Fluent). After investigating various hydrofoils and design modifications, the comparison of lift and drag coefficients obtained from three selected AOA varies from 0o to 20o are shown in Table 3, 4, and 5.

TABLE 3. 0° of Angle of Attack

Hydrofoils	C_l	C_d
NACA0012	0.0038	0.0126
NACA0015	0.0106	0.0159
NACA2412	0.2381	0.0128
NACA2414	0.1908	0.0157
NACA2415	0.4132	0.0210
NACA4412	0.0038	0.0126
E387	0.3976	0.0138

TABLE 4. 10° of Angle of Attack

Hydrofoils	C_l	C_d
NACA0012	0.9599	0.0472
NACA0015	1.1749	0.0446
NACA2412	1.3430	0.0343
NACA2414	1.3157	0.0475
NACA2415	1.7660	0.1381
NACA4412	0.9599	0.0472
E387	1.6346	0.0702

TABLE 5. 20° of Angle of Attack

Hydrofoils	C_l	C_d
NACA0012	1.2618	0.4429
NACA0015	1.3733	0.2345
NACA2412	1.1221	0.4153
NACA2414	1.1669	0.4866
NACA2415	3.4794	0.4993
NACA4412	1.2618	0.4429
E387	1.8474	0.6502

Figure 2 to Figure 4 present the lift and drag coefficient value for different hydrofoils at different AOA. The blue line is representing the lift coefficient, and the orange colour line is representing the drag coefficient.

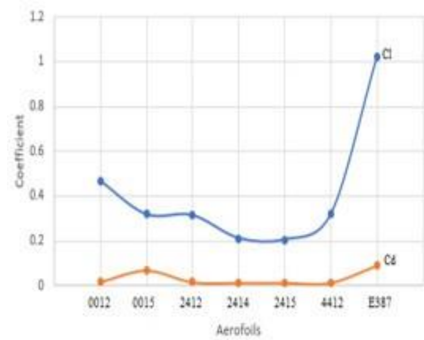

FIGURE 2. Hydrofoils at 0° of AOA vs coefficient

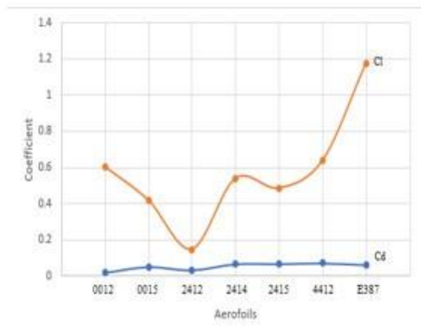

FIGURE 3. Hydrofoils at 10° of AOA vs coefficient

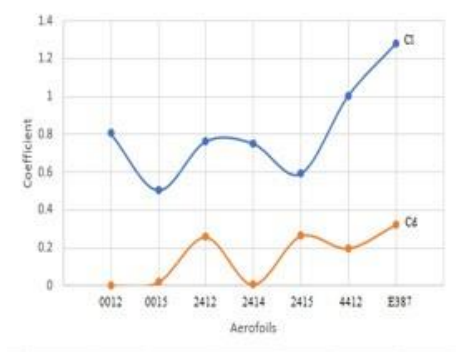

FIGURE 4. Hydrofoils at 20° of AOA vs coefficient

The coefficient of lift is increased until it reaches the stall angle. The angle at which the coefficient of lift attends maximum value is known as the stall angle. Also, the coefficient of drag for seven types of hydrofoils at three different AOA is not much varied, but after a certain degree of AOA, the value was continuously decreased.

Figures 5 and 6 presented the contours gained from selected hydrofoils, specifically E387, and the velocity magnitude and pressure magnitude calculated using ANSYS fluid flow (Fluent) from the AOA value of 10^{o}. The velocity magnitude and pressure magnitude of NACA0012 and NACA2412 are not uniform and did not spread evenly. The other five hydrofoils velocity and pressure magnitude are distributed in uniform and is accurate. The velocity and pressure magnitude of NACA4412 and E387 is the most uniform.

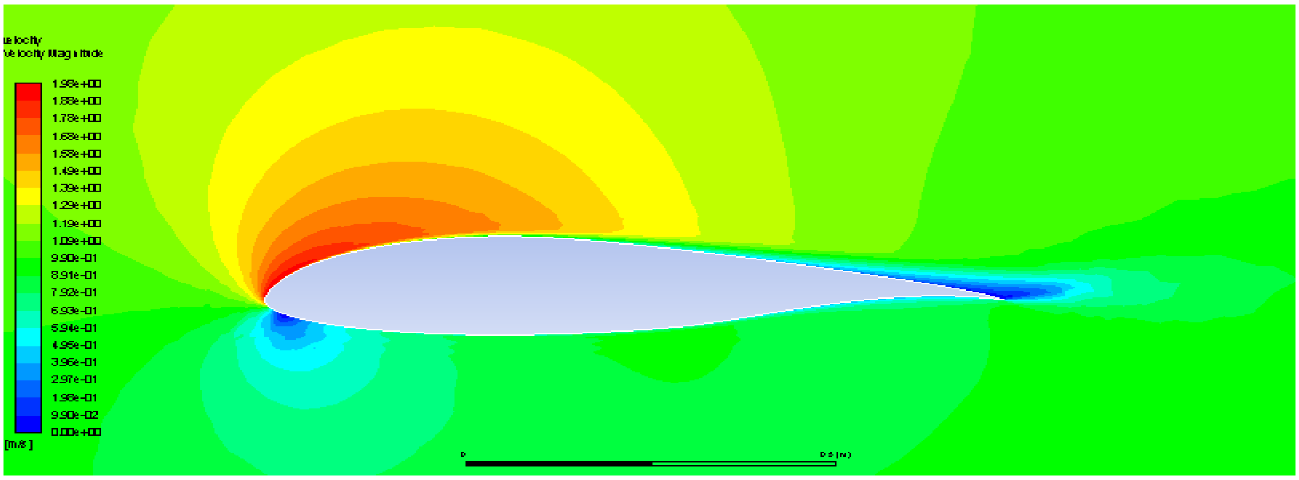

FIGURE 5. Velocity magnitude contour of E387

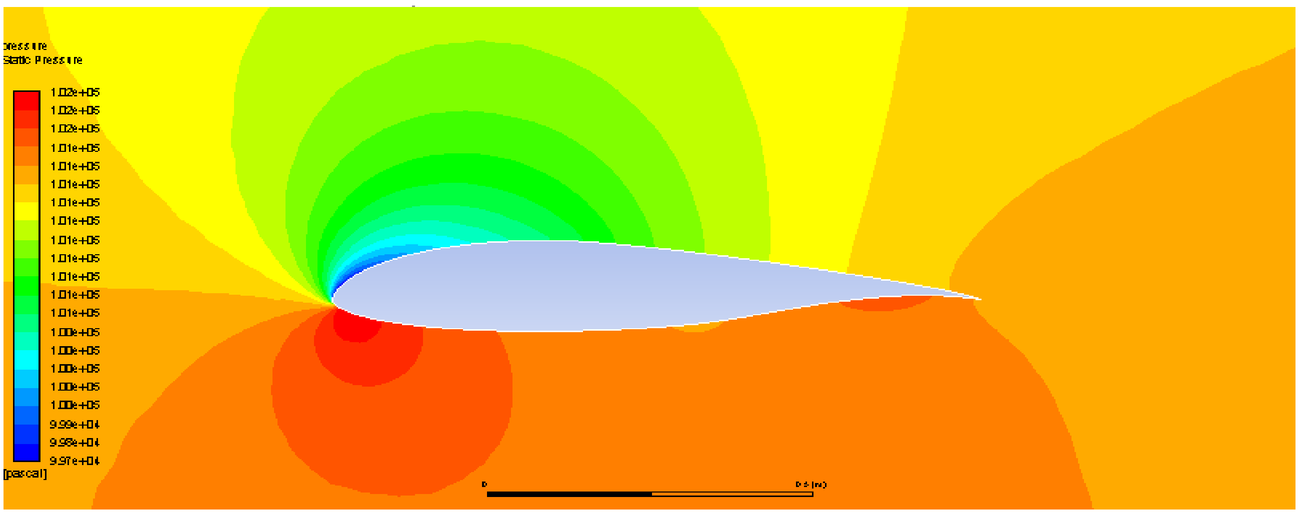

FIGURE 6. Pressure magnitude contour of E387

CONCLUSION

This paper report on lift and drag coefficient, which showed that NACA0015 and NACA2412 give the minimum drag force among all studied hydrofoils, but the lift force that it generates is lower. NACA0012 has a higher lift force but creates a lower drag force. NACA2414 and NACA2415 give a median rate of lift force but produced slightly higher drag force; hence it is considered that both hydrofoils are moderated in either performance. As for NACA4412, this type of hydrofoils gives a higher lift force and a higher drag force compared to the other six hydrofoils. We also concluded that E387 gives an intermediate value of lift force, while drag force produced by this type of hydrofoil is also relatively low, and in fact, at a certain angle, the drag force is continuously increased. Overall, the lift and drag coefficient at 10º are more consistent compare to 0º and 20º AOA. Values produced by 10º of AOA are quite similar to experimental data from previous research with accuracy in the range from 1-10%, probably caused by seawater usage as ideal fluid instead of air, which is commonly used.

ACKNOWLEDGEMENT

This study was funded by the Malaysia Ministry of Higher Education (MOHE) through the Fundamental Research Grant Scheme (FRGS), grant number (VOT. 59602), FRGS/1/2019/TK07/UMT/02/1.

DECLARATION OF COMPETING INTEREST

None

REFERENCES

Chandrakant, S., Mane, P. & Gawali, B. S. 2012. Experimental and Cfd analysis 1(3).

Dash, A. 2016. CFD Analysis of wind turbine airfoil at various angles of attack. *IOSR Journal of Mechanical and Civil Engineering* 13(4): 18–24.

Emmerson, B.A. & Verstraete, D. 2020. A post stall experimental study of an Eppler 387 airfoil at a Reynolds number of 300,000 2020-2215.

Jain, R., Jain, S. & Bajpai, L. 2016. Investigation on 3-D wing of commercial aeroplane with hydrofoil NACA 2415 using CFD Fluent. *International Research Journal of Engenieering and Technology* (IRJET) 4(6): 2321–0613.

Kulshreshtha, A., Gupta, S. K., & Singhal, P. (2019). FEM/CFD analysis of wings at different angle of attack. *Materials Today: Proceedings*, 349–353.

Liang, C. & Li, H. 2018. Hydrofoil optimization for improving the power performance of a vertical axis wind turbine using multiple streamtube model and genetic algorithm. *Royal Society Open Science* 5:180540.

Oukassou, K., El Mouhsine, S., El Hajjaji, A. & Kharbouch, B. 2019. Comparison of the power, lift and drag coefficients of wind turbine blade from aerodynamics characteristics of NACA0012 and NACA2412. *Procedia Manufacturing* 32: 983–990.

Rajakumar, S. & Ravindran, D. 2012. Iterative approach for optimising coefficient of power, coefficient of lift and drag of wind turbine rotor. *Renewable Energy, RE* 38: 83–93.

Sauvageat, E. & Rolin, V. 2016. Prediction and comparison of low-Reynolds airfoil performance 1–97. https://infoscience.epfl.ch/record/225475

Thompson, M.K. & Thompson, J.M. 2017. Introduction to ANSYS and finite element modeling. *ANSYS Mechanical APDL for Finite Element Analysis* 1–9.

Vinayak Chumbre, T. Rushikesh, Sagar Umatar, S.M.K. 2015. CFD analysis of airfoil sections.05(07). https://ijmter.com/published_special_issues/07-02-2015/cfd-analysis-of-airfoil-naca0012.pdf

Yunus, A.C., Cimbala, J.M. 2014. Yunus A. *cengel.pdf.* New York: Tata Mc Graw Hill Publication.