Numerical Investigation of the Effect of Changing the Thickness of Airfoils used in Wind Turbines on the Lift to Drag Ratio

Omar A Abdulkareem, Ahmed F Khudheyer and Ali S Abbas

1University of Al-Nahrain, College of Engineering, Department of Mechanical Engineering, Baghdad, Iraq
E-mail: drahmed955@eng.nahrainuniv.edu.iq

Abstract. In this paper, the effect of changing the thickness of the airfoils on the lift and drag coefficients and lift to drag ratio for three types of airfoils is presented. The study was done by making a 2D simulation by using the commercial software ANSYS FLUENT 2019R1. The airfoils that were selected for the simulation were the NREL S830, SG6043, and SD7062. In order to study the effect of changing the thickness of airfoils on the aerodynamic characteristics, the thickness of NREL S830, SG6043, and SD7062 airfoils was increased and decreased by 20%. The method used to increase and decrease the thickness was done by modifying the coordinates of the upper and lower curves of the baseline airfoils in the y-direction. The simulation was done using the Spalart Allmaras turbulence model for low wind speed (5m/s) and a wide range of attack angles (from 0 to 15 degrees). The aerodynamic characteristics, which are the lift and drag coefficients, lift to drag ratio, and the optimum angle of attack, are presented in this study. It was concluded that reducing the thickness to 20% leads to a remarkable increase in the lift to drag ratio while increasing the thickness leads to a decrease in the lift to drag ratio. It was also concluded that reducing thickness will not always lead to maximizing the lift coefficient but will always reduce the drag coefficient. It was also found that for wind turbines operating in low wind speed regions, the SG6043 airfoil profile is the most suitable as it showed a good aerodynamic performance at low wind speed.

Keywords. Numerical, Airfoils, Wind turbines, Drag Ratio.

1. Introduction
Over the past few years, air pollution has increased rapidly due to non-renewable energy sources; this requires alternative resources that produce clean and efficient energy. The wind turbine is the most popular renewable power source, which converts the kinetic energy of wind into usable power. As a source of energy, wind energy is being widely favored as a substitutional to fossil fuels because it produces no greenhouse gas emissions, so it is clean and renewable. Approximately 3%-5% of the world’s energy is generated using wind turbines [1]. Basically, the wind turbine converts the wind’s kinetic energy into mechanical energy, which can be used to generate electricity. The main part of the wind turbine that captures the wind’s energy is the blade, and for this reason, it must have the best aerodynamic shape and material. The blade is designed with a combination of several airfoils with different thickness, chord length, and twist angle. The selection of the airfoil that forms the blade is an important task because it is...
responsible for determining the aerodynamic efficiency of the blade, thus to get the optimum blade shape that can extract as much energy as possible from the wind, the airfoil thickness, chord length, and twist angle must be optimized. Mohamed A. Sayed et al. [2] studied the aerodynamic performance of the S-series wind turbine blade profiles (from S809 to S835), which were developed by the National Renewable Energy Laboratory (NREL). A 2D simulation of the flow passing through the airfoils was done using the Computational Fluid Dynamics (CFD) method, depending upon the finite volume approach. The airfoils’ aerodynamic characteristics were extracted, which were the lift, drag coefficients, and lift to drag ratio, and the optimal angle of attack at different mean wind speeds. The SST turbulence model was used to detect the boundary layer accurately, and the governing equations were the Reynolds-Average-Navier-Stokes (RANS) equations. It was concluded that the CFD code could accurately predict the aerodynamic loads of the wind turbine blades. It was also concluded that the optimum profile selection was dominated by the angle of attack and was not affected by the wind speed. It was also noted that the most efficient blade profiles that were suitable for wind turbines operating at low and high wind speeds were the S825, S826, S830, and S831.

Haci Sogukpinar and Ismail Bozkurt [3] worked on the wind turbine blade simulation for five wind speed cases to determine the optimum angle of attack to get the maximum lift to drag ratio. The airflow was numerically simulated around inclined NACA 632-215 airfoil using the SST turbulence model. The lift and drag coefficients, lift to drag ratio, pressure coefficient, and power coefficient were calculated and compared. The simulation was done at different velocities and also at various angles of attack. It was assumed that the flow was incompressible and turbulent over the entire airfoil. It was concluded that lift and drag coefficients were increased with the increase of wind speed, and the maximum lift to drag ratio was achieved at an angle of attack of around 4°, and the lift coefficient had a minus value at the angles of -4° to -7°. Also, it was concluded that the pressure difference between the upper and lower surface was increased with the increase of angle of attack. Jae-Ho Jeong and Soo-Hyun Kim [4] used the Genetic Algorithm with the CFD to optimize thick airfoils with a sharp trailing edge for wind turbines. The Akima curve-fitting method was used to generate the airfoil shape. The airfoils with 40% and 35% thickness values were selected as baseline airfoils. Optimization using GA was entirely dependent on the analysis results of the flow solver; thus, the experimental results of a thin airfoil (NACA4412) and a thick airfoil (FX77-W-500) were compared with the CFD analysis to validate the in-house solver. It was concluded that the drag coefficient was significantly dependent on the separation vortex nearby the trailing edge, and by comparing the baseline airfoils with the optimized airfoils with 40% and 35% thickness values, the optimized airfoils could increase the lift-to-drag ratio by 27.8 % and 42.8 %, respectively by suppressing the separation vortex around the trailing edge. It was also found that the enhanced airfoils could raise the blade torque coefficient by 5.7 % and 11.3 %, respectively, while the increase in the blade thrust coefficient was suppressed.

Albi et al. [5] studied the aerodynamic performance of a wind turbine blade airfoil using 2D modeling. This study's objective was to extract the aerodynamic performance of the airfoil, which was the lift to drag ratio at various angles of attack. The selected airfoil was NACA 4412, and the CFD analysis was done by using ANSYS FLUENT. The airfoil’s geometry was created using GAMBIT with a chord length equal to 0.1651m, and the analysis was done using the Spalart Allmaras turbulence model. It was concluded that the lift to drag ratio was increasing with the increase of the angle of attack in the range of 0 to 8 degrees. After an angle of attack of 8 degrees, the lift to drag ratio started to decrease. Also, the results showed a good agreement with the experimental results. Karim Oukassou et al. [6] made an aerodynamic analysis of two types of airfoils. The objective of this research was to extract the aerodynamics characteristics, which were the lift, drag coefficients, and lift to drag ratio of two different airfoils at different Reynolds numbers for a wide range of angles of attack. Another goal was to select the wind turbine type and the blade airfoil selection, and the chord length and twist angle distribution along the blade. Two airfoils were selected, which were NACA 0012 and NACA 2412, for the simulation. ANSYS FLUENT 16.2, 2014 was used for the simulation, and three turbulence models were selected for the analysis, which was the Spalart
Allmaras, $K$-ω-SST, and $K$-epsilon (RNG). The numerical results were compared with the experimental data available from the National Aeronautics Advisory Committee (NACA) wind tunnel, and the comparison showed a good agreement between the numerical results and the experimental data for a range of angles of attack between -5° to 5°. It was concluded that the airfoil type had a noticeable effect on the wind turbine efficiency, and at a tip speed ratio of 7; NACA 2412 airfoil had higher efficiency than NACA0012. It was also concluded that the maximum power output from the NACA2412 airfoil was more than the NACA0012 airfoil. Keriman oğuz and Nilay Sezer-Uzol [7] made an aerodynamic optimization of the wind turbine sectional airfoil and the design of new wind turbine blades. Two methods were used for the optimization process: the CLASS-shape Transformation (CST) method and the Parametric Section (PARASEC) method. The S809 airfoil was used as a baseline airfoil, and the thickness was increased and decreased by 20%. The BEM theory was used for the power calculation, and the results of the designed rotors were validated with the NREL Phase VI and NREL Phase III. The baseline airfoil was successfully optimized, and the optimized airfoils were used for designing new wind turbine blades. In this study, the effect of changing the airfoil thickness on the aerodynamic performance for the wind turbine blade profiles S830, SD7062, and SG6043 is studied numerically by using the finite volume method with commercial package ANSYS FLUENT 2019R1. A two-dimensional study of the selected airfoils is done using the CFD-RANS equations. The aerodynamic characteristics of the airfoils are extracted which are the lift and drag coefficients, lift to drag ratio, and the optimal angle of attack.

2. Airfoil selection
The first step in designing a wind turbine blade is selecting the airfoil type responsible for determining the rotor's aerodynamic efficiency. Modern wind turbines use different airfoils along the blade that are a thick airfoil in the root, and thin airfoil at the tip, while in small wind turbines, the same airfoil type is used along the blade. For the present study, three airfoils are selected to model the wind turbine blades, which are the NREL S830, SG6043, and SD7062. The NREL S830 is selected because it is one of the most efficient NREL profiles, and it is suitable for wind turbines operating at low and high wind speeds [2]. The SG6043 airfoil is selected because of its high lift to drag ratio, and it is designed especially for small scale wind turbines by Giguere and Selig [8]. The SD7062 airfoil is selected because it has some structural advantages and performs well at low wind speeds [9]. In order to study the effect of changing the thickness of airfoils on the aerodynamic characteristics, the thickness of each baseline airfoil is increased and decreased by 20%. The selected airfoils are shown in Figure 1.
3. Numerical analysis

In this study, the aerodynamic performance of three types of airfoils is obtained by using the Spalart-Allmaras turbulence model for a wide range of angles of attack by using the commercial software ANSYS FLUENT 2019R1. The coordinates of the airfoils are downloaded from the (Airfoiltools) database [10], and the geometries of the airfoils are created in ANSYS Design Moduler with a chord length equal to 0.09m and a span of 0.25m. The C-type fluid domain which surrounds the airfoil is created in ANSYS DESIGN Modeler, and it consists of a semicircle with a radius of 200C (where C is the chord length) and a rectangular domain with a size of 400Cx300C. The boundary conditions and the dimensions are shown in Figure 2; the blue arrows represent the inlet boundary condition while the red arrows represent the outlet boundary condition.
Figure 2. C-type fluid domain.

After creating the fluid domain, the airfoil is subtracted from its domain by using the Boolean function, and the fluid domain is sliced into four sections to allow for structured mesh around the airfoil. One of the requirements to achieve accurate simulation results is a fine mesh. The C-type grid is an effective meshing method for airfoil geometry. A fine mesh is accomplished by defining edge sizing and bias factor on the edges of the airfoil and the boundaries of the domain. As seen in Figures 3 and 4, high-resolution structured grids are applied around the airfoil.

Figure 3. Hexahedral mesh around the airfoil.  Figure 4. Fluid domain mesh.

The first cell thickness is kept so that the y+ value is in order of 1, which is the required value for the Spalart Allmaras turbulence model. For each model, mesh independence is done for four different cell sizes (40000, 160000, 360000, 640000, and 1000000) at the angle of attack of zero, and the judging parameters are the lift and drag coefficients. The average skewness for all the models is ranging between 0.014161 and 0.019033. For all the models, 640000 cells and 641600 nodes are used as the lift and drag coefficients are no longer changing after 640000 cells. A sample of mesh independence for the SD7062 (20% thinner) airfoil is shown in Table 1.
Table 1. Mesh independence of SD7062 (20% thinner) airfoil.

| No. of elements | $C_L$    | $C_D$    | Percentage % difference in $C_L$ | Percentage % difference in $C_D$ |
|-----------------|----------|----------|----------------------------------|----------------------------------|
| 40000           | 0.2851085| 0.02931032| 0                                | 0                                |
| 160000          | 0.28807283| 0.026781495| 2.625023679                     | 8.723404821                     |
| 360000          | 0.29076455| 0.025738712| 0.925738712                     | 2.574460462                     |
| 640000          | 0.29149269| 0.025844916| 0.249797002                     | 0.947032993                     |
| 1000000         | 0.29180258| 0.025710661| 0.106198513                     | 0.51943867                      |

3.1. Governing equations

In this study, the flow is assumed to be steady, two-dimensional, and incompressible. The simulation is done for a Re number of 25148.094, and the governing equations are the continuity equation and the Navier-Stokes equations. They can be written as follows:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$  (1)

Equation (1) represents the continuity equation.

$$\rho \left( u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = -\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)$$  (2)

Equation (2) represents the X-direction Navier-Stokes equation.

$$\rho \left( u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) = -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)$$  (3)

Equation (3) represents the Y-direction Navier-Stokes equation.

Since the flow is assumed to be incompressible, the air density and kinematic viscosity are constant and taken as 1.225 kg/m$^3$ and 1.7894e-05 Kg/(m.s), respectively. The equations that are used to calculate the lift and drag coefficients are [11]:

$$C_L = \frac{L}{\frac{1}{2} \rho U^2 C_l}$$  (4)

Equation (4) represents the lift coefficient.

$$C_D = \frac{D}{\frac{1}{2} \rho U^2 C_l}$$  (5)

Equation (5) represents the drag coefficient.

3.2. Turbulence model

The turbulence model chosen for the simulation is the Spalart-Allmaras turbulence model, specifically designed for aerospace applications containing wall-bounded flows. It showed promising results for boundary layers exposed to an adverse pressure gradient. It is a comparatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. In its primary form, it is constructively a low-Reynolds number model, demanding the viscous-affected region of the boundary layer to be appropriately resolved [6]. The transport equation is given as [12]:

$$\frac{d\theta}{dt} = c_b \left( 1 - f_{t2} \right) \bar{v}\bar{s} + \frac{1}{\sigma} \left[ \nabla \cdot \left( (v + \bar{v}) \nabla \bar{v} \right) + c_{b2} \left( \nabla \bar{v} \right)^2 \right] - \left( c_{w1} f_w - c_{b1} \frac{c_{b1}}{k^2} f_{t2} \right) \left( \frac{\theta}{\alpha} \right)^2 + f_{t1} \Delta U^2$$  (6)

3.3. Boundary conditions

The boundary condition applied at the airfoil surface is set as a wall with a no-slip boundary condition. Inlet and outlet turbulence are assumed with a turbulent viscosity ratio of 10 and an inlet velocity.
magnitude of 5m/s. At the outlet, the boundary condition is set to pressure-outlet with zero gauge pressure. All the boundary conditions can be seen in Figure 2.

3.4. Methods
The pressure-based coupled algorithm is used to solve the incompressible RANS (Reynolds-Averaged Navier-Stokes) equations in which the momentum and pressure-based continuity equations are solved in a nearly coupled manner. The pressure-based coupled algorithm remarkably enhances the convergence rate compared to the pressure-based segregated algorithm, in which the momentum and pressure-based continuity equations are solved separately [13]. Second-order pressure is used for the discretization, and the momentum and the modified turbulent viscosity are set to second-order upwind.

3.5. Convergence criteria
One of the mainly used criteria for judging the CFD solution convergence is the residual. In this study, the residual values of four variables (i.e., continuity, x velocity, y velocity, and nut) are monitored during the calculation process. These residual values, which consist of some numerical errors, are first calculated. Then, these errors decrease more and more as the system proceeds from iteration to another. After that, the residual values are divided by the most considerable residual value obtained from the first iteration to provide a normalized value [14]. The residual values are set to $10^{-6}$, and the number of iterations is set to 500 iterations. A sample for convergence history for SD7062 (20% thinner) airfoil at an angle of attack equal to zero is shown in Figure 5, which illustrates that all the residual values are reached to $10^{-6}$ and from Figure 6 and Figure 7, it can be seen that the lift and drag coefficients are also converged.

![Figure 5. Residual convergence.](image)

![Figure 6. Lift coefficient convergence.](image)
4. Results and discussion

4.1. Code validation
This work was validated with reference [2]. For validation, the selected airfoil profile is S830, and the flow velocity is 5m/s, and the same boundary conditions are used as the reference, and the simulation was done at angles of attack ranging from 0 to 4 degrees. Figure 8 shows the validation results in which it shows a good agreement with reference [2] with a percentage difference ranging from 0.229% to 3.846%.

4.2. CFD Simulation results
In this study, the aerodynamic characteristics, which are the lift and drag coefficients and lift to drag ratio of S830, SG6043, and SD7062 airfoils, are extracted by using (ANSYS FLUENT 19.0 R1). The simulation is done for a wind speed of (5m/s) and a wide range of attack angles (from 0 to 15 degrees). Figure 9 shows the \( \frac{C_L}{C_D} \) vs. the angle of attack for the S830 airfoils. The results show that the thinnest airfoil has the highest \( \frac{C_L}{C_D} \). As the thickness increases, the \( \frac{C_L}{C_D} \) decreases sharply and vice versa. From the figure, it is observed that the baseline and the 20% thinner airfoils have the highest \( \frac{C_L}{C_D} \) ratio at the same angle of attack (which is 6 degrees), while the 20% thicker airfoil has the highest \( \frac{C_L}{C_D} \) at an angle...
of attack of 7 degrees. Figure 10 shows the lift and drag coefficients vs. the angles of attack for the S830 airfoils; it can be seen that the lift and drag coefficients are increasing with the increase of angles of attack. As the thickness of the airfoil decreases, the lift coefficient shows good behavior as it has the highest values among the baseline and the thicker airfoils.

This shows that the maximum thickness has a significant effect on the lift and drag coefficient values. The maximum $C_L/C_D$ and the optimal angle of attack are shown in Table 2.

| Airfoil type        | Maximum $C_L/C_D$ | The optimal angle of attack (degree) |
|---------------------|-------------------|-------------------------------------|
| S830 (20% thinner)  | 20.451            | 6                                   |
| S830 baseline       | 13.969            | 6                                   |
| S830 (20% thicker)  | 7.077             | 7                                   |

Figure 11 shows the velocity streamlines at the optimum angle of attack for the S830 airfoils. As expected for all models, the velocity is high at the airfoil's upper surface and low at the lower surface. For the thinnest airfoil, it can be seen that the velocity on the upper surface is much higher than the other airfoils; this means that more pressure is formed on the lower surface, which generates more lift. Also, the separation is very close to the trailing edge with little vortices forming at it, which will generate less drag force than the other airfoils. For the baseline airfoil, it can be seen that the vortices are more significant than the thinner airfoil, and the separation begins at approximately 60% of the chord length. The thicker airfoil has the lowest lift to drag ratio with a percentage difference from the baseline airfoil of 49.338%. This huge difference is because of the earlier separation, which begins at the maximum thickness, and the generation of the vortices accompanied by the formation of the laminar separation bubble, which can be noticed at the airfoil's lower surface in Figure 11c. Due to the intense pressure gradient, the laminar boundary layer is separated from the surface, and the shear layer is shortly reattached downstream. This separation and reattachment form a region between them called the separation bubble [15].
Figure 11. Velocity streamlines. (a) S830 20% thinner, (b) S830 baseline and (c) S830 20% thicker.

Figure 12 shows the lift to drag ratio vs. angle of attack for the SD7062 airfoils. The graph shows that the SD7062 (20% thinner) airfoil has the highest lift to drag ratio. The $C_L/C_D$ for the 20% thinner airfoil shows an increment of (13.151%) compared to the baseline airfoil, while for the 20% thicker airfoil, the $C_L/C_D$ is decreased by (14.820%) when compared to the baseline airfoil. This indicates that the thickness has a significant influence on the lift to drag ratio, and decreasing the thickness of the same airfoil thickness gives a remarkable increase in the $C_L/C_D$. Figure 13 shows the lift and drag coefficients vs. angle of attack. As the attack angle increases, the lift coefficient also increases for all the airfoils until it starts to decrease, which means that the airfoil begins to stall, and the drag coefficient values become greater.

Figure 12. Lift to drag ratio vs. angles of attack.  
Figure 13. Lift and drag coefficients vs. angles of attack.
Table 3 shows the maximum $C_L/C_D$ and the optimal angle of attack for each airfoil.

**Table 3.** Maximum $C_L/C_D$ and optimal angle of attack for the SD7062 airfoils.

| Airfoil type          | Maximum $C_L/C_D$ | The optimal angle of attack (degree) |
|-----------------------|-------------------|--------------------------------------|
| SD7062 (20% thinner)  | 25.071            | 7                                    |
| SD7062 baseline       | 21.774            | 6                                    |
| SD7062 (20% thicker)  | 18.547            | 5                                    |

Figure 14 shows the velocity streamlines for the SD7062 airfoils. For the thinner airfoil, it is evident that the flow stalls near the trailing edge at an angle of attack of $7^\circ$, and the stall is not accompanied by any vortices, while for the baseline and the 20% thicker airfoils, the stall is covering a more significant area of the upper surface of the airfoil and happens at angles of attack of $6^\circ$ and $5^\circ$ respectively.

Figure 15 shows the lift to drag ratio vs. angle of attack for the SG6043 airfoils. As with the previous airfoils, the lift to drag ratio for the thinnest airfoil is higher than the thicker airfoils, which also can be seen in Table 4. The $C_L/C_D$ of the thinner airfoil (20% thinner) is increased by (6.969%) as compared to the baseline airfoil, while for the thicker airfoil (20% thicker), the $C_L/C_D$ is decreased by (12.900%) as compared to the baseline airfoil. However, when looking at Figure 16, it is observed that the $C_L$ values of the thinnest airfoil (20% thinner) are less than the $C_L$ values of the thinner airfoil (20% thicker) and the baseline airfoil, while the drag coefficient of the thinner airfoil is the smallest among them. This means that reducing the thickness will not always maximize the lift force, but as seen for all the models reducing thickness will always decrease the drag coefficient.
Figure 15. Lift to drag ratio vs. angles of attack.

Figure 16. Lift and drag coefficients vs. angles of attack.

Table 4. Maximum $C_L/C_D$ and optimal angle of attack for the SG6043 airfoils.

| Airfoil type                  | Maximum $C_L/C_D$ | Optimal angle of attack (degree) |
|-------------------------------|-------------------|----------------------------------|
| SG6043 (20% thinner)          | 28.455            | 6                                |
| SG6043 baseline               | 26.472            | 6                                |
| SG6043 (20% thicker)          | 23.057            | 5                                |

The CL/CD for the thinner airfoil is higher than the thicker airfoils because airflow separation happens at the trailing edge with no vortices, which can be noticed in Figure 17a leads to less drag force as compared to the other airfoils. For the baseline and thicker airfoils, which can be seen in Figures 17 b and c, the stall covers a larger area of the upper surface of the airfoils with the formation of vortices, which increases the drag force and thus reduces the $C_L/C_D$.

Figure 17. Velocity streamlines (a) SG6043 20% thinner (b) SG6043 baseline and (c) SG6043 20% thicker.
5. Conclusion
In this study, the aerodynamic performance of nine wind turbine blade profiles was extracted which are the lift and drag coefficients, lift to drag ratio, and optimal angle of attack by using the commercial software ANSYS FLUENT. It is concluded that the airfoil thickness has a strong influence on the lift to drag ratio, and for the thick airfoils, a 20% decrease in the airfoil thickness significantly improves the lift to drag ratio. For all the airfoils, the percentage increase in the lift to drag ratio that was achieved from decreasing the airfoil thickness is ranging between 6.969% to 31.695%, while the percentage decrease of the lift to drag ratio that was achieved from increasing the airfoil thickness is ranging between 12.900% to 49.338%. It is also concluded that increasing the airfoil thickness leads to an increase in the drag coefficient and leads to an earlier separation of the flow on the airfoil's upper surface. Moreover, it is concluded that for wind turbines operating in low wind speed regions, the SG6043 airfoil profile is the most suitable as it showed a good aerodynamic performance at low wind speed.

6. References
[1] Kapdi R Dahiya R and Naranje V 2016 Analysis and Optimization of Horizontal Axis Wind Turbine Blade Profile (IJERT) (Dubai, U.A.E)) vol 05 P 1
[2] Sayed M A and Kandil H A 2012 Aerodynamic Analysis of Different Wind-turbine-blade Profiles Using Finite-volume Method (Energy Convers. Manag, (Cairo, Egypt)) vol 64
[3] Sogukpinar H and Bozkurt I 2015 Calculation of Optimum Angle of Attack to Determine Maximum Lift to Drag Ratio of NACA 632-215 Airfoil (Journal of Multidisciplinary Engineering Science and Technology) vol 2
[4] Jeong Jand and S Kim 2018 Optimization of Thick Wind Turbine Airfoils Using a Genetic Algorithm (Journal of Mechanical Science and Technology, (Yuseoung-gu, Daejeon, Korea)) vol 32
[5] Albi, Anand M D and Herbert G M 2018 Aerodynamic Analysis on Wind Turbine Aerofoil (Int. J. Eng. Technol. (Kumaracoil, Tamil Nadu, India)) vol 7
[6] K. Oukassou, S El Mouhsine, A El Hajjaji and B Kharbouch 2019 Comparison of the Power, lift and Drag Coefficients of Wind Turbine Blade from Aerodynamics Characteristics of Naca0012 and Naca2412 (Procedia Manuf. (Morocco)) vol 32
[7] Oğuz K and Uzol N S 2019 Aerodynamic Optimization of Horizontal Axis Wind Turbine Rotor by Using BEM, CST Method and Genetic Algorithm (10th Ankara International Aerospace Conference (Ankara, Turkey))
[8] Pourrajabian A Ebrahimi R and Mirzaei M 2014 Applying Micro Scales of Horizontal Axis Wind Turbines for Operation in Low Wind Speed Regions (Energy conversion and management (Tehran, Iran)) vol 87 p 2
[9] Sessarego M and Wood D 2015 *Multi-dimensional Optimization of Small Wind Turbine Blades* (Renewables: Wind, Water, and Solar (Calgary, AB, Canada)) p 1

[10] http://www.airfoiltools.com

[11] J F Manwell and J G McGowan *Wind Energy Explained: Theory, Design and Application* (Washington, USA) p 103

[12] Spalart P R and Allmaras S R 1992 *A One Equation Turbulence Model for Aerodynamic Flows* (AIAA 30th Aerospace Sciences Meeting and Exhibit (USA)) p 20

[13] Wang L, Quant R and Kolios A 2016 *Fluid Structure Interaction Modelling of Horizontal-axis Wind Turbine Blades Based on CFD and FEA* (Journal of Wind Engineering and Industrial Aerodynamics) vol 158 p 11

[14] Abbas A S, Khudheyer A F and Hussain J M 2018 *Investigation the Effect of Flap Design Optimization on Wake Propagation Behind Wings of Transport Plan* (ARPN Journal of Engineering and Applied Sciences, (Baghdad, Iraq)) vol 13 p 4

[15] Dongli M, Yanping Z, Yuhang Q and Guanxiong L 2015 *Effects of Relative Thickness on Aerodynamic Characteristics of Airfoil at a Low Reynolds Number* (Chin J Aeronaut) vol 28 p 2