International Journal of Engineering & Applied Sciences (IJEAS)
Vol.9, Issue 3 (2017) 75-86
http://dx.doi.org/10.24107/ijeas.332075
Int J Eng Appl Sci 9(3) (2017) 75-86

Numerical Investigation of Multi Airfoil Effect on Performance Increase of Wind Turbine

Haci Sogukpinar

Department of Electric and Energy, Vocational School, University of Adiyaman, Adiyaman 02040, Turkey
*E-mail address: hsogukpinar@adiyaman.edu.tr

ORCID numbers of authors:
0000-0002-9467-2005

Received date: August 2017
Accepted date: September 2017

Abstract

In this study, numerical calculations are conducted by using SST turbulence model to investigate effect of multi airfoil on aerodynamic efficiency of MW scale wind turbine blade. For the numerical calculation S type airfoils developed by NREL are used. Initially, numerical calculations are performed for S825 airfoil, and obtained results are compared with experimental data to validate the simulation accuracy of this modeling. The comparisons show good agreement for the numerical approach with experiment in the lift coefficients at the angle of attack from -2 to 3 degree, which is the normal operation angle of wind turbine. For the root part of the wing S826, for the body S825, which is slightly thinner and for the tip section S814 airfoil are selected and designed in 2D and 3D shape. Lift coefficients, lift to drag ratios and pressure coefficient along the surface for S 814, S 825 and S 826 airfoil are calculated, and compared.

Keywords: S825, S814, S826, airfoil, lift, drag, numerical calculation, SST

1. Introduction

The primary component of a wind turbine is the rotor which transforms the kinetic energy of air into mechanical energy. The capability of rotor to convert a maximum ratio of wind energy to mechanical is obviously depends on aerodynamic properties of blades which determine overall efficiency of wind turbine. Efficiency of the blades is prime importance for the overall economics of the system. Therefore the blade is the one of the key components of a wind turbine which compromises ideal aerodynamic shape. With the increasing power harnessed by wind turbine, the size and weight of the blades also increase [1]. The majority of the airfoils in use on horizontal-axis wind turbines were initially developed for aircraft but, with the development of the wind turbine industry, more efficient airfoil especially only for wind turbine has been designed for last 30 years [2]. Airfoil design and wind tunnel test is complex and requires significant expertise because it has a very time consuming process [3]. But today, with the help of high speed and powerful computer, numerical calculation can be done close to the experimental results. Therefore many airfoil design code has been develop and are being used for performance calculation of airfoil. Therefore numerical calculation is very helpful to estimate aerodynamic properties of the airfoil by using turbulence model like k- ε, k-w, Spalart–Allmaras and SST. NACA 0012 airfoil has been subjected to many researches due to its aerodynamic properties and has been subjected to sensitive experimental and theoretical studies. Numerical simulation of 4-digit inclined NACA 00xx symmetric airfoils to find optimum angle of attack for airplane wing was performed and obtained data compared with experiment data to validate the simulation accuracy of the

Computational Fluid Dynamics (CFD) approach [4].Correlation between numerical calculation and experimental study is very good. Numerical calculation of airfoil NACA 63_2-215 was performed to determine optimum angle of attack for horizontal axis wind turbine by using SST turbulence model and lift, drag coefficient, lift to drag ratio and pressure coefficient around the airfoil were calculated and compared with different velocity [5]. Numerical Analysis of NACA64-418 Airfoil with Blunt Trailing Edge was conducted and obtained results were compared with experimental data validate simulation accuracy of CFD then other airfoil were investigated. The transport equations for the transition of SST model is based on the Wilcox k-ω model, is good to predict the transition point [6]. The numerical simulation of horizontal axis wind turbines airfoil S809 with untwisted blade was performed with k- ε model and compared with experimental data to determine the optimal angle of attack that produces the highest power output [7]. The performance of NREL S series airfoils with different wind speeds was investigated with SST turbulence model and the optimum blade profile for each wind speed is determined based on the maximum lift to drag ratio and results showed that the CFD code can accurately predict the wind-turbine blades aerodynamic properties [8]. Steady-state, two-dimensional CFD calculations were performed for the S809 laminar-flow and calculations show that the k- ε model, is not appropriate at angles of attack with flow separation [9]. A mathematical model for airfoil design based on the blade element momentum theory for S809 airfoil was implemented and compared with experimental data to evaluate turbine performance with a wide range of wind velocities [10]. Aerodynamic performances of S-series wind turbine airfoil of S 825 was numerically investigated by using SST turbulence model to get maximum aerodynamic efficiency for a wind turbine and the comparison shows good agreement for the numerical approaches [11].Therefore SST turbulence model is reliable to investigate aerodynamic properties of other airfoils in this study.

In this paper a sample blade is designed by using three different airfoils (S814, S825, S826) then aerodynamic properties of each airfoil are calculated by using SST turbulence model. Such types of airfoils were proposed for megawatt-scale wind turbines by NREL. The lift and drag coefficients of airfoil S825 are calculated with the SST turbulence model, and obtained results are compared with experimental data to validate the simulation accuracy of the Computational Fluid Dynamics (CFD) approach then S814 and S826 are calculated and compared with different velocity. According to the calculations, to achieve the highest torque for this blade design S814 should place at the root, S825 is at the primary part and S826 should place at the tip of the blades.

2. Computational Approach

For this calculation flow is assumed that fluid is incompressible and two-dimensional (2-D) Navier–Stokes equations written as:

$$\rho\frac{\partial u}{\partial t} + \rho(u.\nabla)u = \nabla.[-pI + \mu(\nabla u + (\nabla u)^T)] + F \quad (1)$$

$$\rho\nabla.U = 0 \quad (2)$$

Any solid objects with any shape, if it was subjected to fluid stream, object experience a force from the flow. The sources of this force are from viscous and pressure effects on the surface of the object. Total force on the surface of airfoil is written as:

$$F = \int p dA + \int \tau_w dA \quad (3)$$

This force can be divided in two parts as lift and drag force. If we take airflow along the x direction, drag force is in the same direction with airflow but lift is in the y direction. Lift and drag force can be written as:

$$L = -\int p sin\theta dA + \int \tau_w cos\theta dA \quad (4)$$
$$D = \int (pdA) cos\theta + \int (\tau_w dA) sin\theta \quad (5)$$

Lift and drag coefficients are dimensionless quantities and used to measure the aerodynamic properties of an object which vary with the angle and the shape of the airfoil. Lift, drag and pressure coefficients defined as:

$$C_L = \frac{L}{1/_2 \rho U_\infty{}^2 A} \quad (6)$$
$$C_D = \frac{D}{1/_2 \rho U_\infty{}^2 A} \quad (7)$$
$$C_P = \frac{p - p_\infty}{1/_2 \rho U_\infty{}^2} \quad (8)$$

Where, $1/_2 \rho U_\infty{}^2$ is dynamic pressure. Generally drag and lift coefficients of an object was only measured with wind tunnel tests. But with development of efficient and cost effective CFD software and rapid decrease in the cost of computations, CFD is replacing the wind tunnel tests due to the rapid increase in the cost of experimentation.

3. Method of modeling

The wind turbine blade designed and modeled in this study are composed of three different airfoils and are shown in Fig. 1. One of the airfoil is intentionally chosen because which has experimental data to compare the accuracy of the simulation technique. For the numerical analysis, commercial CFD program COMSOL is used for the shape modeling, grid generation and aerodynamic analysis. The flow field around S814, S825 and S826 is assumed as incompressible viscous flow. To eliminate the effect of the domain size on the results, computational domain is extended 300x200 times the chord length of the airfoil.

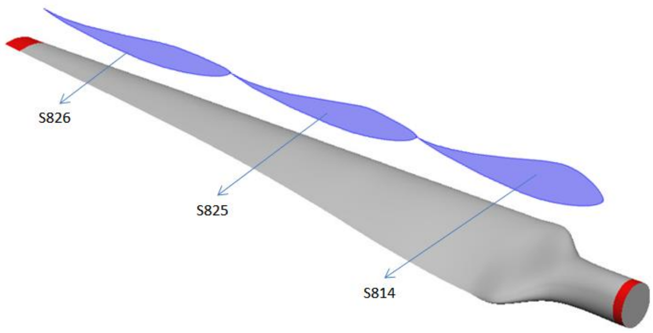


Fig. 1. Aerodynamic shape of a wind turbine blade

As the boundary conditions, velocity inlet and open boundary are selected. The inlet port is set as a velocity inlet and determined by Reynolds number, the output port is set open boundary with zero pressure. The computational domain is consists of a semicircle of the diameter of 200c and a rectangular domain size of 200x200c, the airfoil is locates at the semicircular center and flow domain and boundary conditions is shown in Fig. 2. No slip boundary conditions are applied on the airfoil surface. The pressure is assumed to be 1 atm and temperature is 20°C. SST turbulence model is adopted and turbulent intensity, 0.005, turbulence length scale, 1 m and reference velocity scale length is set to 1 m/s.

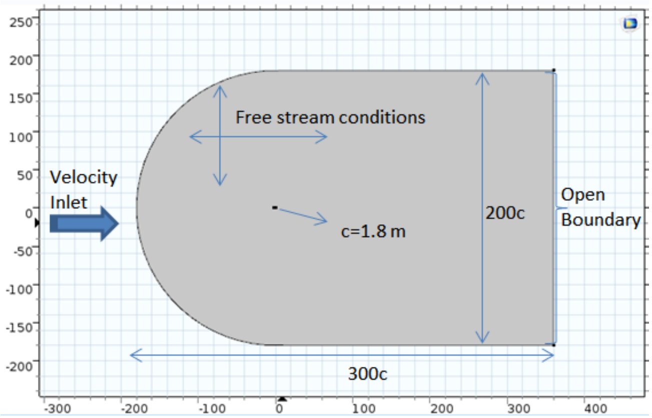


Fig. 2. Flow domain and boundary conditions

C-type mesh is adopted around the all airfoils to discrete the flow field in the simulation and as shown in Fig. 3. Dense grids are applied around the airfoil but sparse distributions are adopted in the region far away from airfoil. Model is divided into three part and 86800 quadrilateral elements are created. To verify the accuracy of the numerical analysis, the lift and the lift-to-drag ratio are compared with the wind tunnel experiment [10]. Because only if the grid number is in a certain range, the results are more agreement with experimental data otherwise too dense or too sparse mesh distribution may produce calculation error. For an accurate comparison, all three airfoils are meshed using the same methods. Because only S825 airfoil has experimental data and which constitute a reference for the other two airfoils.

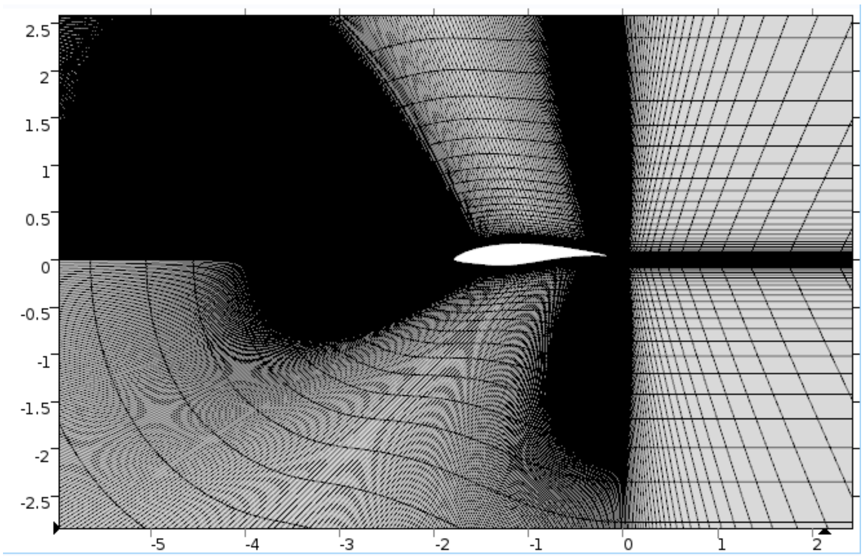


Fig. 3. Mesh distribution around the airfoil

4. Results and discussion

Currently, wind power companies are generally installing MW scale onshore wind turbine such as 1.5 MW, 2 MW, and 3 MW. Numerical design and simulation of a 20 to 40 m blade for 1-2 MW wind turbines is performed with the wind speed between 8.4 m/s and 25 m/s, using airfoils S814, S825, S826 respectively. Before the numerical calculation is attempted, the reliability and validity issue of CFD is investigated for S825 airfoil because it has reliable experimental data [12]. Fig.4 shows the comparison of SST turbulence computational lift coefficients with those of the experiment (NASA Langley) and also Eppler code (NREL) calculation results. The agreement for lift coefficient between the computational and experimental results is very good for operation angle from -5° to 8 degree. It is interesting that this SST turbulence calculation shows better agreement compare to agreement between Eppler code calculation and experimental data. For the lift coefficient Eppler code data are slightly higher than experimental results. According to Fig.4, with the increasing Reynolds numbers there is still good agreement for low angle side but at higher angle, agreement angle degrease from 8 to 6 degree. However, predicted results for higher than 8 degree is not in good agreement with the experiment for all conditions and lift coefficient increase with the increasing angle of attack but increment in experimental data starts to decrease at certain degree. Eppler code lifts calculation data slightly higher than experimental data for all condition. With the increasing angle of attack, flow separation and vortex formation begins as shown in Fig. 5.

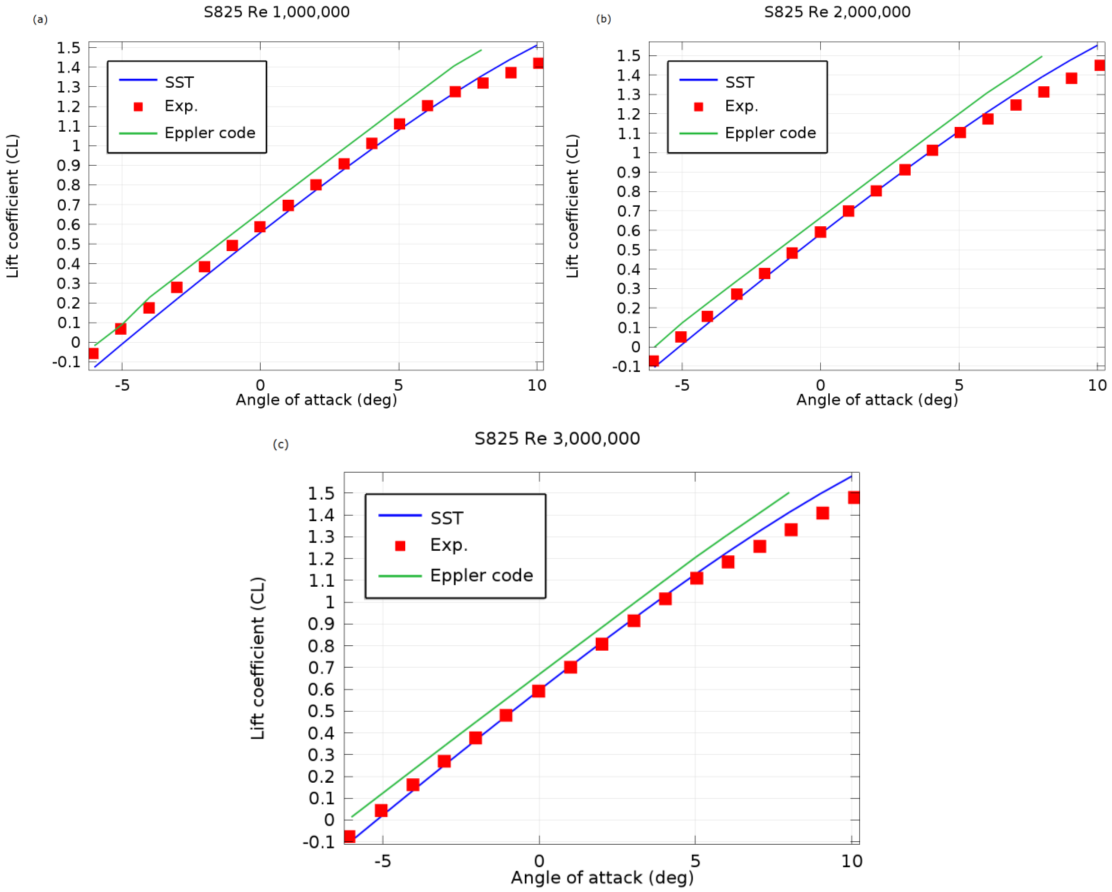


Fig. 4. Lift coefficient vs. angle of attack

As seen from the Fig.5, with the increasing angle of attack, wind speed increases on both pressure and suction side. Red (Dark) color indicates that wind speed is high in those area but blue (light) color indicate slower speed. Flow separations are clearly seen in all Fig.5 at the upper trailing edge side. With the increasing angle of attack, flow separation is growing and eddy current is evident. There are no vortex formations in Fig.5(a) and (b) but vortexes are clearly seen in in Fig.5(c) and (d). Fig.6 shows the comparison of SST turbulence model lift coefficient calculation results at the Reynolds numbers of 1x10^6, 2x10^6, 3x10^6, with the angle of attack from -2 to 12 degree. S826 airfoil intended to tip portion of the wind turbine blade in this modeling, has a maximum lift coefficient in each case compare to others. Because of higher lift coefficient S826 airfoil improve efficiency by producing higher torque at low wind speeds. S825 airfoil numeric data are calculated very close to S826's and they together create high efficiency. S814 airfoils are thicker than two others and create lower aerodynamic lift, but its physical structure is necessary for blade to be strong enough. The agreement between the computational lift coefficient and experimental results for S825 is very good at the angle of attack from -2 to 6 degree. As mention before S814 and S826 doesn't have experimental data so this calculation can be used to estimate aerodynamic properties of S814 and S826 until at angle of attack of 6 degree.

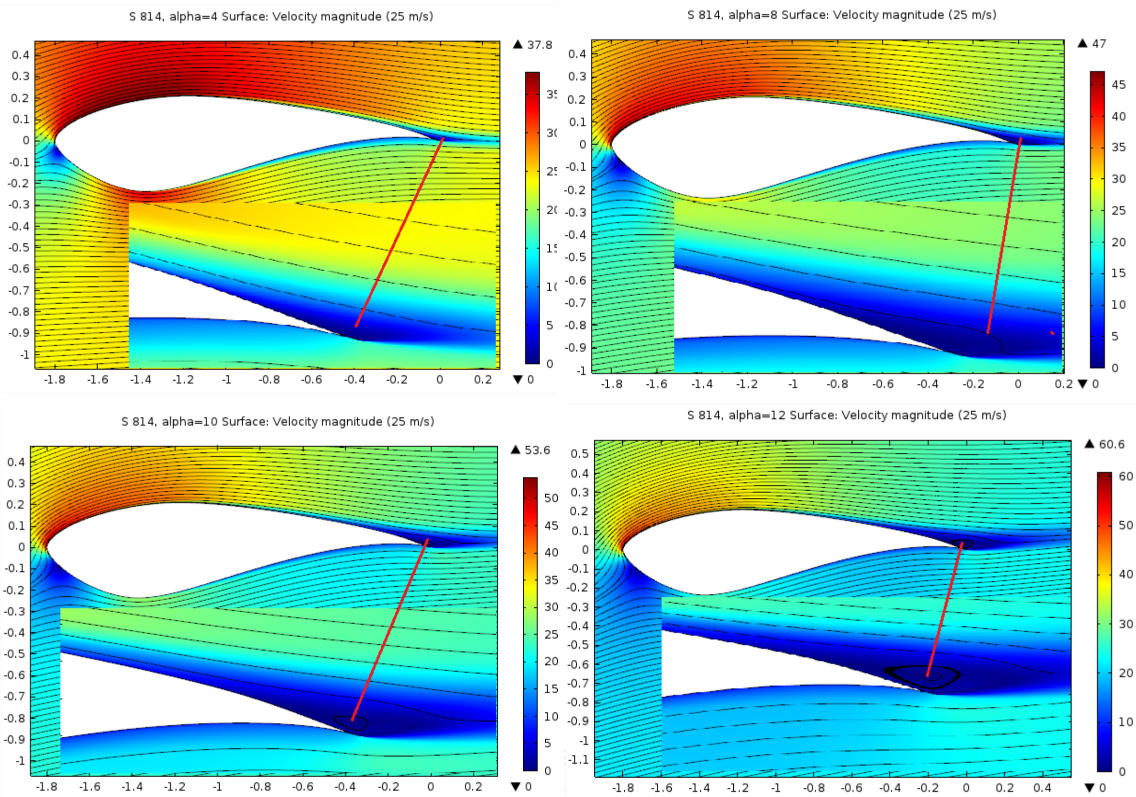


Fig. 5. Velocity magnitude (m/s) and streamline vs. angle of attack (Deg.)

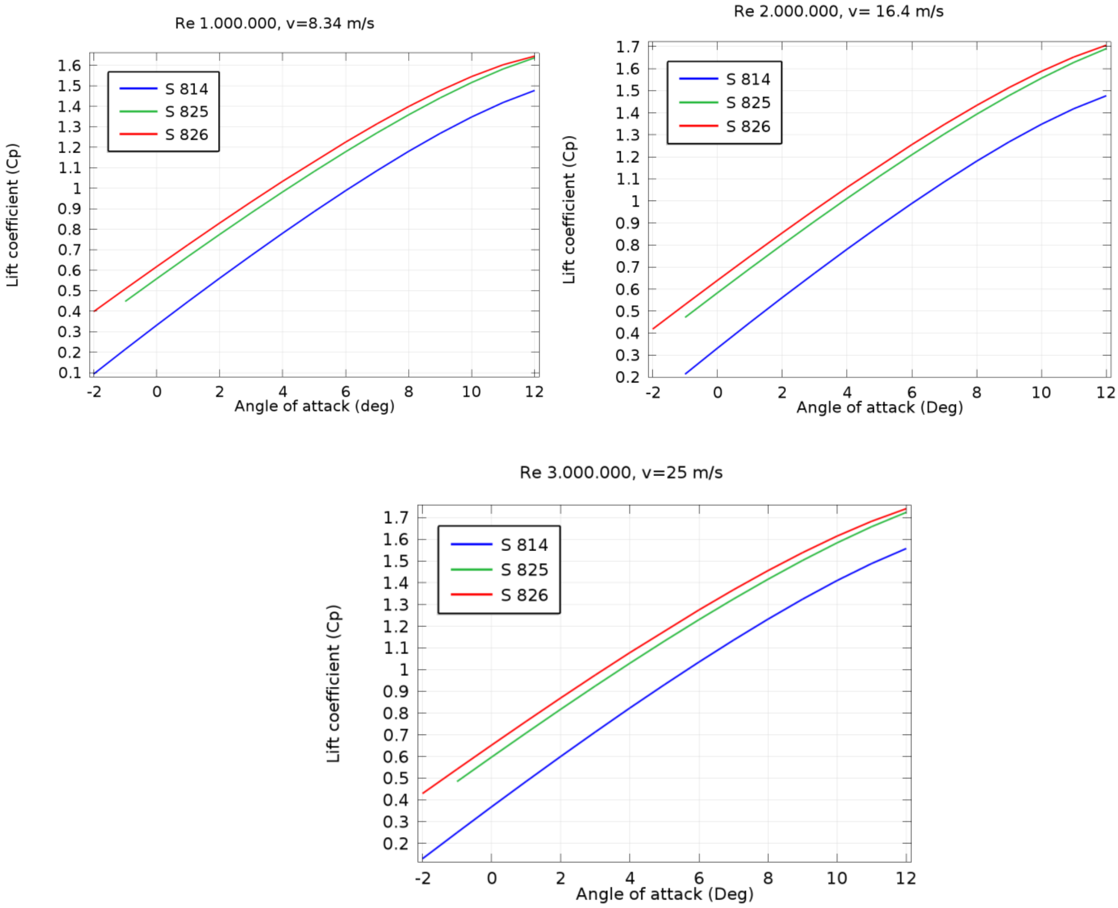


Fig. 6. Computational lift coefficients vs. angle of attack (Deg.)

Lift to drag ratio versus angles of attack for experimental and both Eppler code and this calculation are shown in Fig. 7. For experimental measurement, lift to drag ratio increase until 5° and become highest at this point and after that it starts to degrease again. Maximum lifts to drag ratio in experimental studies vary between 110 and 120. In calculations made by using Eppler code, despite poor compliance between lift coefficient with experimental data, lift to drag calculation results show very good agreement with experiments for each case. Lift to drag ratio calculated by using SST turbulence model reach maximum lift to drag ratio at the angle of attack 3° in each case. In this numerical calculation, although the lift coefficients show full compliance with experiment at certain interval, the lift to drag ratio doesn't correlate with experiment. Drag coefficient calculated with SST turbulence model are lower than experimental data. The measurements of higher drag coefficient in real conditions are normal. The reason why many effects are neglected or approximations are used to solve out complex numerical equations. Dust particles deposited on the blade and surface roughness are some of the parameters in real condition affect the drag force.

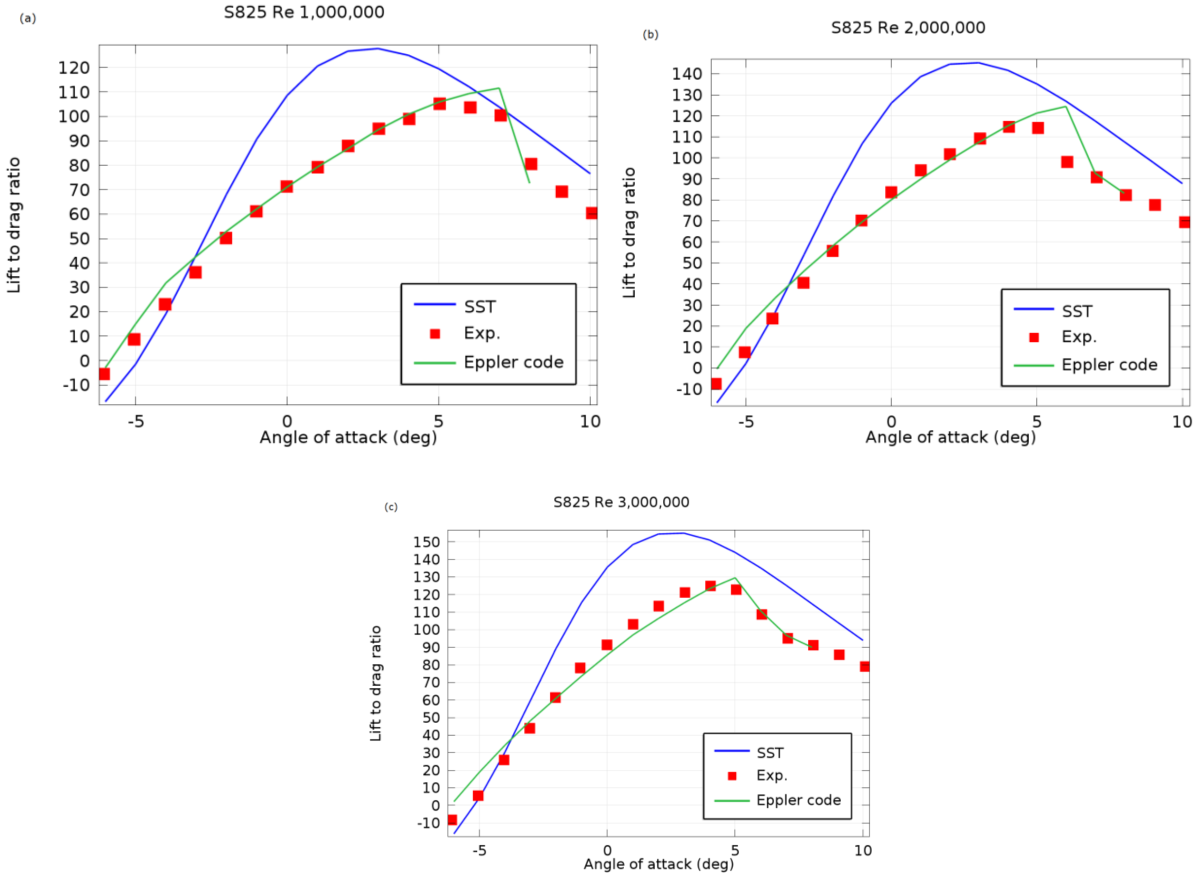
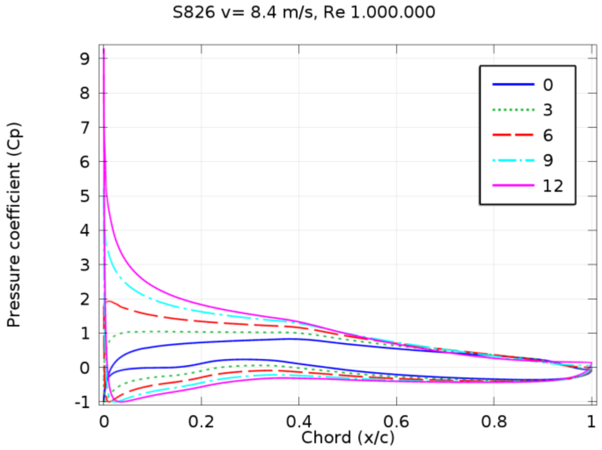
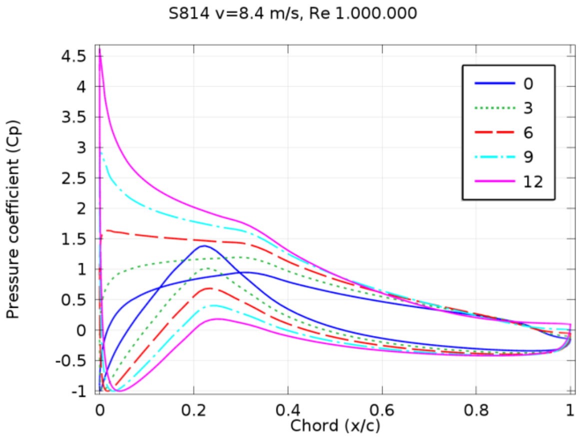
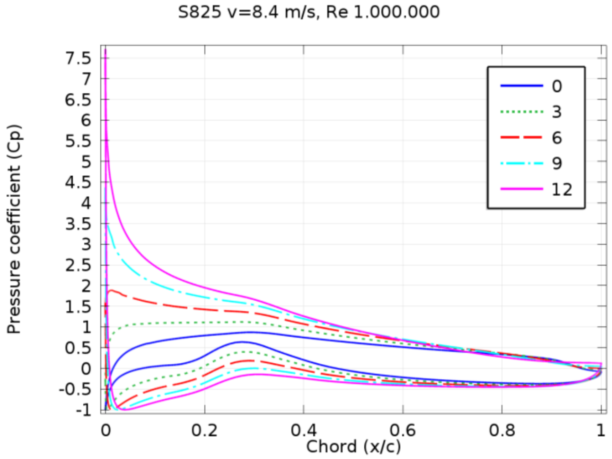


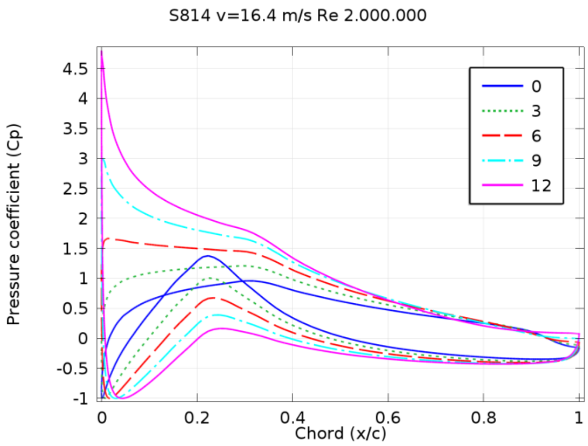
Fig. 7. Lift to drag ratio vs. angle of attack (Deg.)

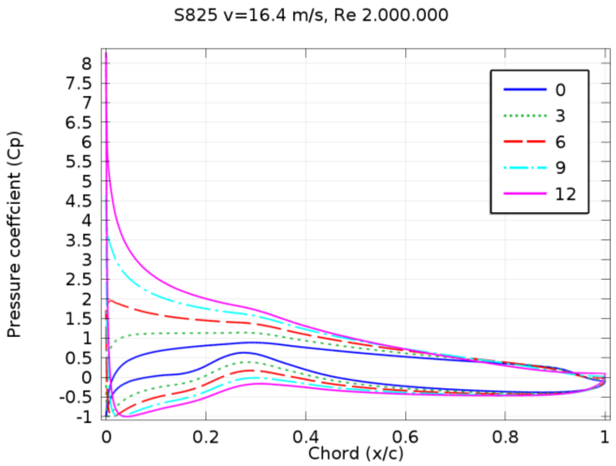
Fig. 8 shows pressure coefficient along airfoil at the angle of attack 0°, 3°, 6°, 9° and 12° respectively. As seen in the Fig.8, with the increasing angle of attack pressure increase sharply at the zero point of the chord line then immediately decrease again. For S814 airfoil, chord distance (x/c) between 0.1 and 0.3, the pressure coefficient on the upper surface of the airfoil are calculated lower than pressure side of airfoil and this reduces the lift coefficient. With the increasing angle of attack the pressure difference between upper and lower surfaces increases which results in the lift coefficient increase. Pressure sides of S825 and S826 airfoil have smooth pressure distribution but S814 airfoil has a fluctuating pressure distribution on the same side. All three show similar pressure distribution profile on suction side. The uniform pressure distribution on lower surfaces is shown to improve the aerodynamic efficiency of airfoil for this modeling.

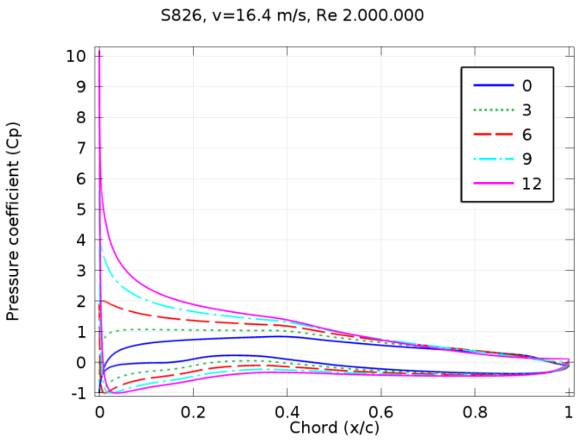

S826 v= 8.4 m/s, Re 1.000.000
Pressure coefficient (Cp)
Chord (x/c)
0
3
6
9
12


S814 v=8.4 m/s, Re 1.000.000
Pressure coefficient (Cp)
Chord (x/c)
0
3
6
9
12


S825 v=8.4 m/s, Re 1.000.000
Pressure coefficient (Cp)
Chord (x/c)
0
3
6
9
12


S814 v=16.4 m/s Re 2.000.000
Pressure coefficient (Cp)
Chord (x/c)
0
3
6
9
12


S825 v=16.4 m/s, Re 2.000.000
Pressure coefficient (Cp)
Chord (x/c)
0
3
6
9
12


S826, v=16.4 m/s, Re 2.000.000
Pressure coefficient (Cp)
Chord (x/c)
0
3
6
9
12

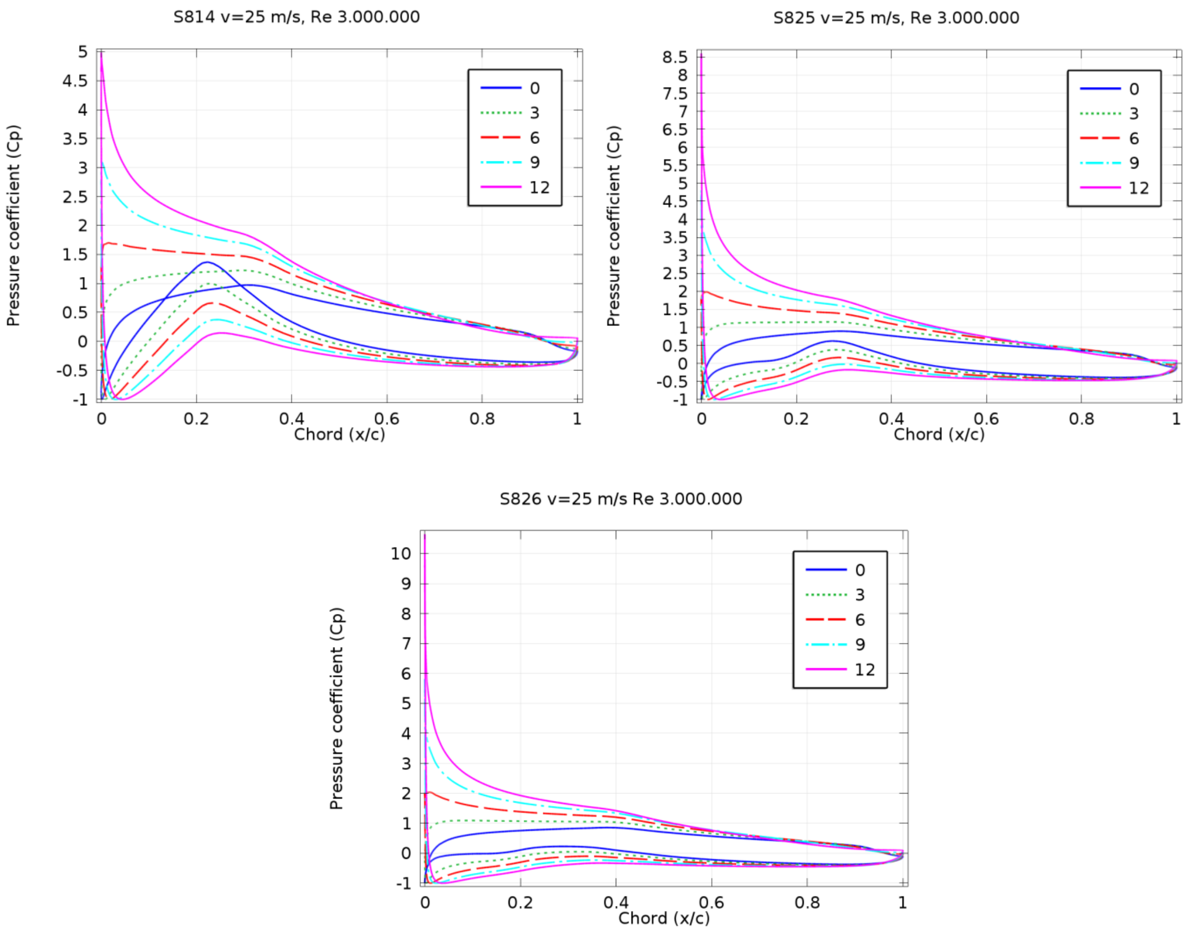


Fig. 8. Pressure coefficient along airfoil

5. Conclusion

This paper numerically investigates influence of multi airfoil effect on the aerodynamic performance of wind turbine blade by CFD with the SST turbulence model. Initially, S825 airfoil is simulated, and obtained results are compared with experimental data to validate the simulation accuracy of this modeling. The comparisons show good agreement for the numerical approach with experiment in the lift coefficients at the angle of attack from -2 to 6 degree. This interval is the normal operating range of the wind turbine. Then numerical calculations are conducted for S825 and S826 airfoil at the angle of attack from -2 to 12 degree. Lift coefficients, lift to drag ratios and pressure coefficient along the surface for S814, S825 and S826 airfoil are calculated, analyzed and presented. The objective of the simulation is also to compare aerodynamics properties of S814 and S826 airfoil with S825 in the same figure. All simulated airfoils were designed by NREL and are shown in Fig.1. With the increasing angle of attack, lift coefficient increase. While the S826 airfoil create maximum lift coefficient, S814 has the lowest value at all angle. Next, lift to drag ratio are calculated and the result are compared those obtained experimental study and Eppler code calculation data. Maximum lift to drag ratio are found at the angle of attack 3° but in experimental study maximum lift to drag ratios were obtained at the angle of attack around 5°.Finally, pressure coefficient around airfoil are calculated and compared at the angle of attack from 0° to 12°. At the pressure side of S825 and S826 airfoil has more uniform pressure distribution with respect to S814 airfoil, therefore those two airfoils indicate better aerodynamic efficiency. As a result, to get maximum efficiency from proposed three-airfoils system S814 should be at the root, S825 is at primary and S826 should be at the tip of blades.

Acknowledgment

Many thanks to Middle East Technical and Adiyaman University to conduct this study with their facility.

Nomenclature

C_p	Pressure coefficient
C_L	Lift coefficient
C_D	Drag coefficient
p	Static pressure
p_∞	Free stream pressure
U_r	Relative velocity
U_∞	Free stream velocity (wind velocity)
u	Velocity field x component
v	Velocity field y component
c	Airfoil chord
t	Percentage of the maximum thickness
k	Turbulence kinetic energy
ε	Turbulence dissipation rate
ω	Rotational velocity
ρ	Density
ρ_∞	Free stream density
μ	Dynamic viscosity
α	Angle of attack
∅	Scalar quantity of the flow
NACA	National Advisory Committee for Aeronautics
NASA	National Aeronautics and Space Administration

References

[1] Zhang, X., Li, W., Liu, H., Numerical simulation of the effect of relative thickness on aerodynamic performance improvement of asymmetrical blunt trailing-edge modification, *Renewable Energy*, 80, 489-497,2015.

[2] Sogukpinar, H., Bozkurt, I., Pala, M., Turkmenler, H., Aerodynamic Numerical Testing of Megawatt Wind Turbine Blade to Find Optimum Angle of Attack, *International Journal of Engineering & Applied Sciences (IJEAS)*, 7, 1-9, 2015.

[3] Sogukpinar, H., Bozkurt, I.,Calculation of Aerodynamic Performance Characteristics of Airplane Wing and Comparing with the Experimental Measurement," *International Journal Of Engineering Technologies*,1, 83-87,2015.

[4] Sogukpinar, H, Numerical Simulation Of 4-Digit Inclined NACA 00xx Airfoils To Find Optimum Angle Of Attack For Airplane Wing, Uludag University Journal of The Faculty of Engineering, 22(1), 169-178,2017.

[5] Sogukpinar, H., Bozkurt, I., Calculation of Optimum Angle of Attack to Determine Maximum Lift to Drag Ratio of NACA 63_2-215 Airfoil, *Journal of Multidisciplinary Engineering Science and Technology (JMEST)*, 2, 1103-1108, 2015.

[6] Yoo, H.S., Lee, J.C., Numerical Analysis of NACA64-418 Airfoil with Blunt Trailing Edge, *Int'l J. of Aeronautical & Space Sci.,*16(4), 493–499,2015.

[7] Thumthae, C., Chitsomboon, T., Optimal angle of attack for untwisted blade wind turbine, *Renewable Energy*,34, 1279–1284, 2009.

[8] Sayed, M.A., Kandil, H.A., Shaltot, A., Aerodynamic analysis of different wind-turbine-blade profiles using finite-volume method, *Energy Conversion and Management*, 64, 541–550,2012.

[9] Wolfe, W.P., Ochs, S.S., CFD Calculations of S809 Aerodynamic Characteristics, *AIAA*, 97-0973, 1997.

[10] Lanzafame, R., Messina, M., Fluid dynamics wind turbine design: Critical analysis, optimization and application of BEM theory, *Renewable Energy* , 32, 2291–2305,2007.

[11] Sogukpinar H., Bozkurt I., Finding optimum airfoil shape to get maximum aerodynamic efficiency for a wind turbine, AIP Conference Proceedings, 1815(1), 1-4,2017.

[12] National Renewable Energy Laboratory, *http://wind.nrel.gov. 2015*