

Aerodynamic Numerical Testing of Megawatt Wind Turbine Blade to Find Optimum Angle of Attack

H. Sogukpinar^{a,*} I. Bozkurt^b M. Pala^c H. Turkmenler^d

^a Department of Energy Systems Engineering, Faculty of Technology, University of Adiyaman, Turkey

^b Department of Mechanical Engineering, Faculty of Engineering, University of Adiyaman, Turkey

^c Department of Civil Engineering, Faculty of Engineering, University of Adiyaman, Turkey

^d Department of Environmental Engineering, Faculty of Engineering, University of Adiyaman, Turkey

*E-mail address: hsogukpinar@adiyaman.edu.tr

Received date: December 2015

Accepted date: January 2016

Abstract

Wind energy is one of the oldest kinds of energy source used by mankind and it is dating back to thousand years. At the beginning of twentieth century, multi blades wind turbines were improved and used for charging in USA. After oil crisis in in the seventies, there was a surge of alternative energy sources in USA, Denmark and Germany. In terms of installed capacity, Germany had the leadership until 2007, then it passed to USA and finally China becomes the leader. Today, 6 MW capacity wind turbine are available in the market. The majority of the airfoils in use on horizontal-axis wind turbines today were originally developed for aircraft but it has been begun to design airfoil especially for wind turbine for 20 years. S series airfoil poses the most significant examples designed by NREL. In this study, S809 airfoil was simulated using the SST turbulence model and compared with experimental data to validate simulation accuracy of the computational fluid dynamics and this calculation. After verifying the consistency of the simulation model, S830 airfoil was simulated with different Reynolds numbers or velocities. Lift and drag coefficient, lift to drag ratio and pressure coefficient were calculated and compared.

Keywords: Renewable energy, wind power, wind energy, Airfoils, S830, NACA airfoil, optimum angle, lift and drag

1. Introduction

Wind energy is a type of energy has been used by mankind for thousands of years. With the start to produce electricity from wind power at the beginning of the 20th century, it was born a new application area for wind power. The electricity generated from wind energy had remained as small scale and met individual needs until 1960. However, research in this area continued rapidly and after that larger machines were designed and tested. It was turned to wind energy with the energy crisis which began in the 1970s. Especially after the oil crisis of the seventies in the USA, Denmark and Germany, studies have been initiated to bring development of wind turbines marketable condition. In the eighties, it was begun to establish large-scale wind turbines. With the effects of global warming has begun to occur, wind energy have become more popular.

To get the maximum output from the wind turbine, it must be designed for optimum airfoil shape. For the first design wind turbines NACA airfoils were used. Then some turbine manufacturers have used their own design airfoil. NREL has begun to design airfoil especially for wind turbine since 1984. Since that time seven airfoil families have been designed for various size rotors using the Eppler Airfoil Design and Analysis Code. The main objective of designing is to achieve maximum lift coefficient and lift to the drag ratio.

The results of aerodynamic simulations of the steady low-speed flow past two-dimensional S-series wind-turbine-blade profiles, developed by the National Renewable Energy Laboratory (NREL), are conducted and optimum blade profile for each wind speed was decided based on the maximum lift to drag ratio [1]. Effect of periodic excitation from individually checked synthetic jet actuators on the dynamics of the flow within the separation and re-attachment regions of the boundary layer over the suction surface of a 2D model wing was researched for S809 airfoil profile [2]. Results indicates that periodic stimulation from the synthetic jet actuators fulfills the laminar separation bubble shaped over the suction surface of the airfoil at Reynolds numbers of 2.3×10^5 at zero angle of attack. Numerical simulation of NACA 632-215 airfoil was performed to determine optimum angle of attack therefore lift, drag coefficient, lift to drag ratio and pressure coefficient around the airfoil were calculated and compared with different velocity [3]. Numerical calculation of horizontal axis wind turbine with untwisted blade was conducted to specify optimal angle of attack to produce highest power output for S809 airfoil and obtained result of 12° were compared with NREL experimental measurement. Then numerical analysis were performed by changing the pitch angles at different velocity and investigation shows that maximum mechanical power output were determined to be at $4.12^\circ, 5.28^\circ, 6.66^\circ$ and 8.76° for the wind speed 7.2, 8.0, 9.0 and 10 m/s respectively [4]. Two dimensional CFD calculations were conducted for S809 wind turbine airfoil by using commercial code CFD-ACE and obtained result compared with wind tunnel data from the Delft University. The result indicates that to get accurate simulations, transition from laminar to turbulent flow point must be modeled correctly [5].

In this study, S809 airfoil was simulated using the SST turbulence model and compared with experimental data to validate simulation accuracy of the computational fluid dynamics and this calculation. After verifying the consistency of the simulation model, S830 airfoil was simulated with different Reynolds numbers or velocities. Lift and drag coefficient, lift to drag ratio and power coefficient were calculated and compared.

2. Numerical Method

The commercial CFD software COMSOL based on finite volume method is employed. Turbulence is a property of the flow field and it is characterized by a wide range of flow scales. For an isothermal flow to become turbulent is measured by the Reynolds number with the equation (1)

$$Re = \frac{\rho UL}{\mu} \quad (1)$$

where μ is the dynamic viscosity, ρ the density, and U and L are velocity and length scales of the flow, respectively. High Reynolds number flow creates turbulence flow. Navier-Stokes equations can be used for simulations of turbulent flow. Numeric results can be obtained by using different modeling. One of these models is the Reynolds-averaged Navier-Stokes (RANS) and which is the model type most commonly used in industrial flow applications and represent with the equation (2) [6].

$$\rho \frac{\partial u}{\partial t} + \rho(u \cdot \nabla)u = \nabla \cdot [-pI + \mu(\nabla u + (\nabla u)^T)] + F \quad (2)$$

where u is the velocity of the fluid, ρ the fluid density, and μ the dynamic viscosity, and F is the sum of all forces. According to Reynolds-Averaged Navier-Stokes' theorem, flow is assumed incompressible and Newtonian.

SST turbulence equations are formulated in terms k and ω with the equation (3) and (4).

$$\rho \frac{\partial k}{\partial t} + \rho u \cdot \nabla k = P - \rho \beta_0^* k \omega + \nabla \cdot ((\mu + \sigma_k \mu_T) \nabla k) \quad (3)$$

$$\rho \frac{\partial \omega}{\partial t} + \rho u \cdot \nabla \omega = \frac{\rho \gamma}{\mu_T} P - \rho \beta \omega^2 + \nabla \cdot ((\mu + \sigma_\omega \mu_T) \nabla \omega) + 2(1 - f_{v1}) \frac{\rho \sigma_\omega^2}{\omega} \nabla \omega \cdot \nabla k \quad (4)$$

Where k is turbulence kinetic energy, ω specific dissipation rate, f_{v1} interpolation functions and default model constants are given as,

$$\beta_1 = 0.075, \gamma_1 = \frac{5}{9}, \sigma_{k1} = 0.85, \sigma_{\omega1} = 0.5, \beta_2 = 0.0828, \gamma_2 = 0.44, \sigma_{k2} = 1.0, \sigma_{\omega2} = 0.856, \beta_0^* = 0.09, \sigma_1 = 0.31$$

To eliminate domain size effect on the results, computational domain was extended 100 and 200 chord length of the airfoils. Computational domain is shown in Figure 1. Computational domain is consist of semicircle (100c) and rectangular in shapes. Inlet port is set as velocity inlet and outlet port is set as open boundary. No slip condition is applied for the airfoil surface.

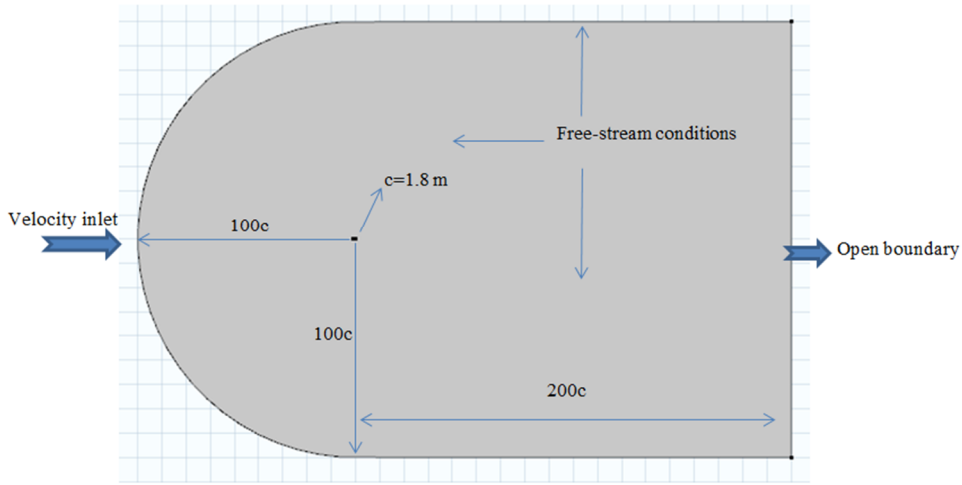


Fig. 1. Flow domain and boundary condition

For the mesh distribution C-grid mesh is adopted to simulate flow field around airfoil. High size-ratio mesh is applied to the domain surface. Mesh distribution around airfoil is shown in Figure 2.

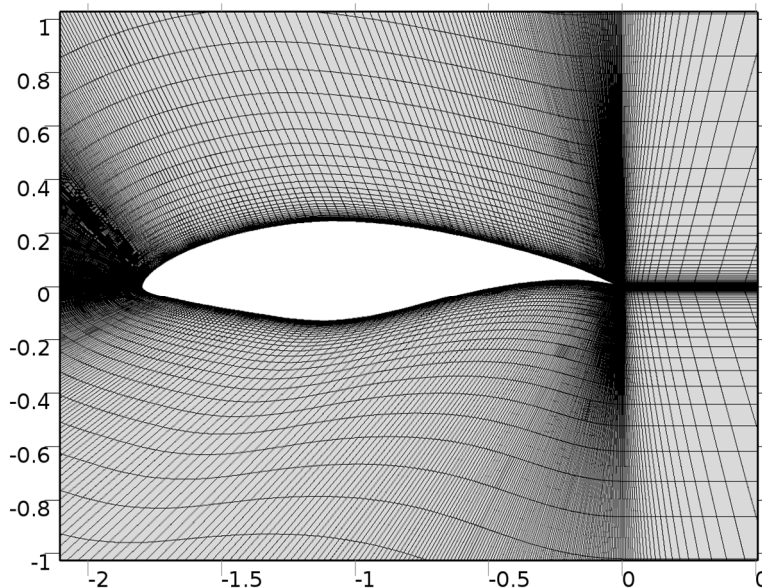


Fig. 2. Mesh distribution around airfoil

3. Results and discussion

S 830 airfoil is designed for mega-watt scale wind turbines. There is no wind tunnel test for S 830 airfoil therefore first of all numerical calculations were conducted for S 809 airfoil and obtain results were compared with experimental data using the SST turbulence model to confirm the validation of this calculation. After observing the harmony between theoretical calculations and experimental data, numerical calculation was made for the S 830 airfoil using the same method of S 809. Velocity magnitude and streamline for the flow around S 830 airfoil with the angle of attack 10° is shown in Figure 3. Red color over the airfoil indicates that speed increases and there is low pressure in there. At the trailing side eddy currents are formed. This situation negatively affects the performance of the wing.

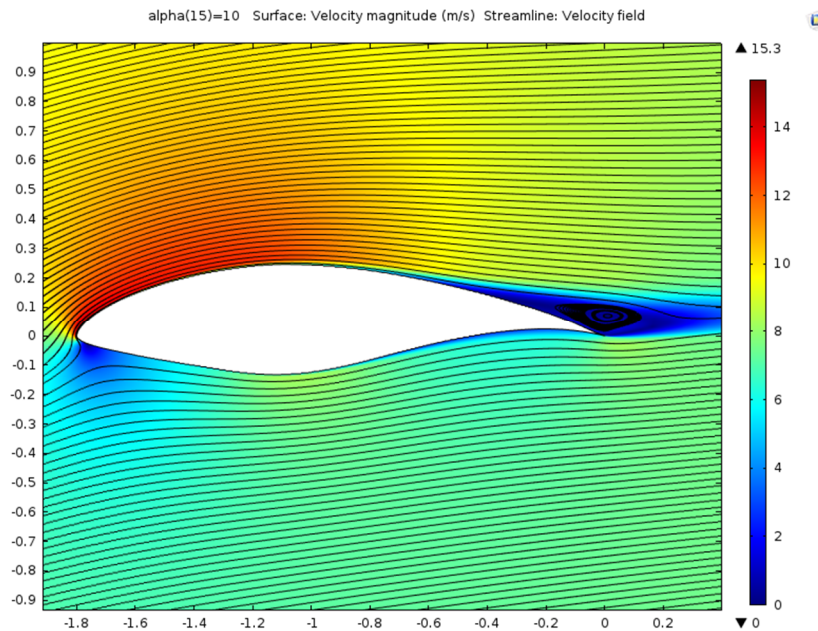


Fig. 3. Velocity magnitude and streamline for the flow around S 830 airfoil

Lift coefficient vs. angle of attack for S 809 airfoil with experimental results is shown in Figure 4. The agreement between the computational and experimental results is almost very good. There is little difference between experimental data and theoretical calculation from 4 to 8 degrees. This graph also indicates that theoretical calculations performed in this study is compatible with experiment.

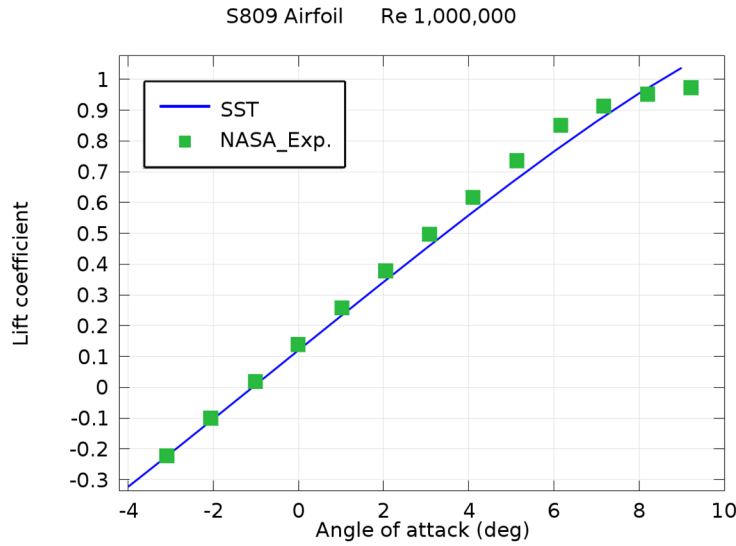


Fig. 4. Computational and experimental results for the lift coefficient vs. angle of attack for S809 airfoil

Lift coefficient vs. angle of attack for different velocity or Reynolds numbers is as shown in Figure 5. With the increasing wind speed, lift coefficient increases linearly but lift coefficient increment decreases. Drag coefficient vs. angle of attack is shown in Figure 6. Minimum drag coefficient is observed at the angle of -1° and maximum at 10° . With the increasing wind speed, drag coefficient increment increases sharply.

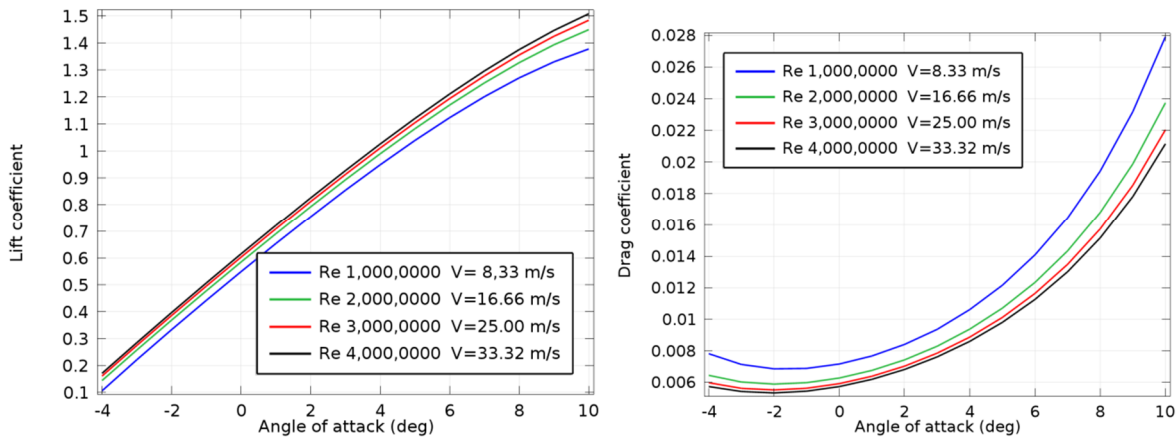


Fig. 5. Lift (a) and drag (b) coefficient vs. angle of attack.

Lift to drag ratio with respect to angle of attack for S 830 airfoil is shown in Figure 6. With the increasing angle of attack, lift to drag ratio increases until 3° then starts to decrease again. This figure indicates that optimum angle of attack for S 830 airfoil is 3° . With the increasing airflow, maximum lift to drag ratio increases but optimum angle of attack stays constant for each case.

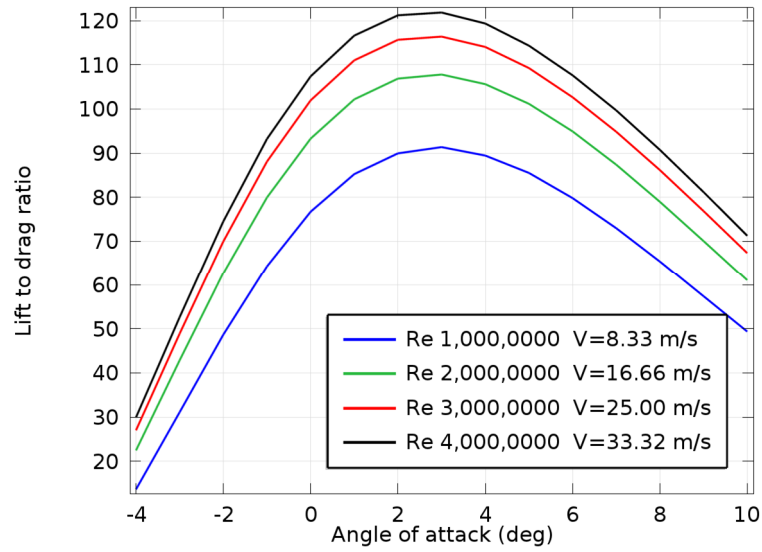
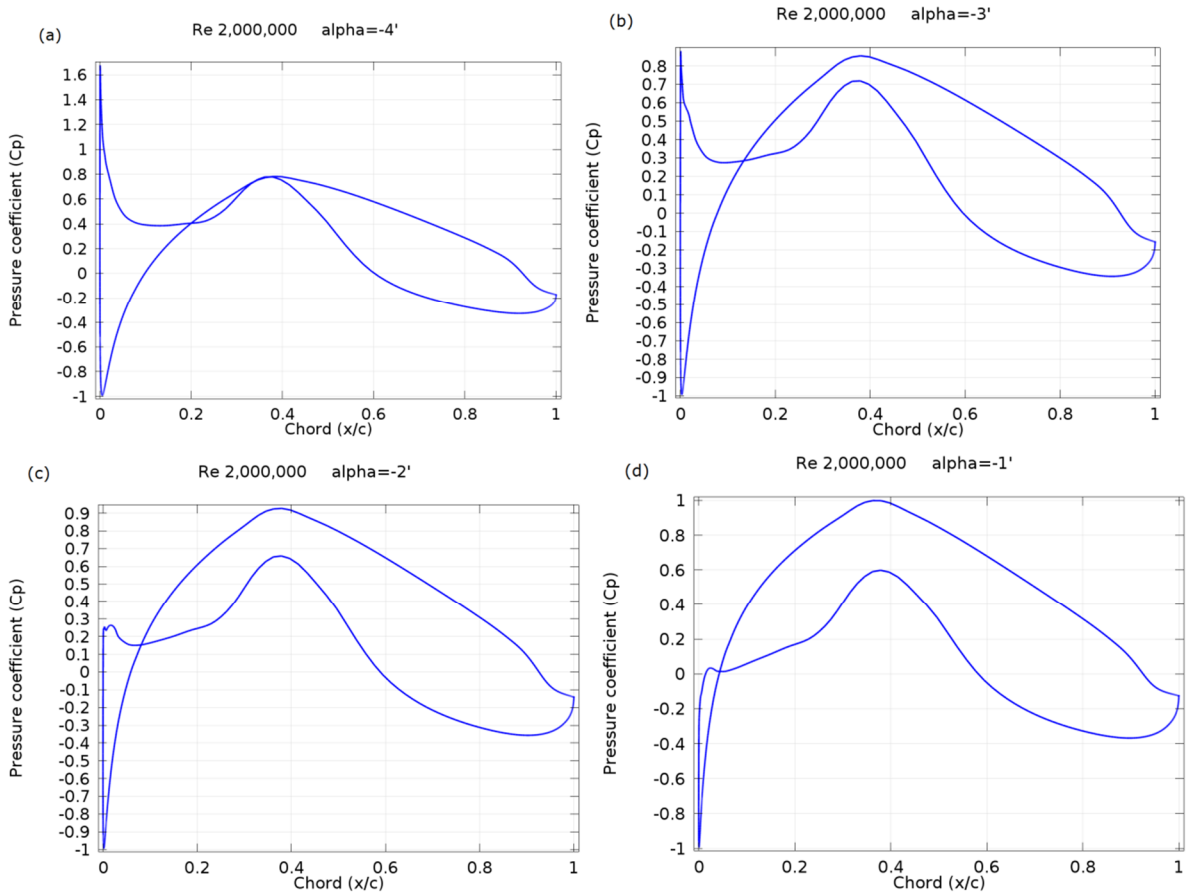
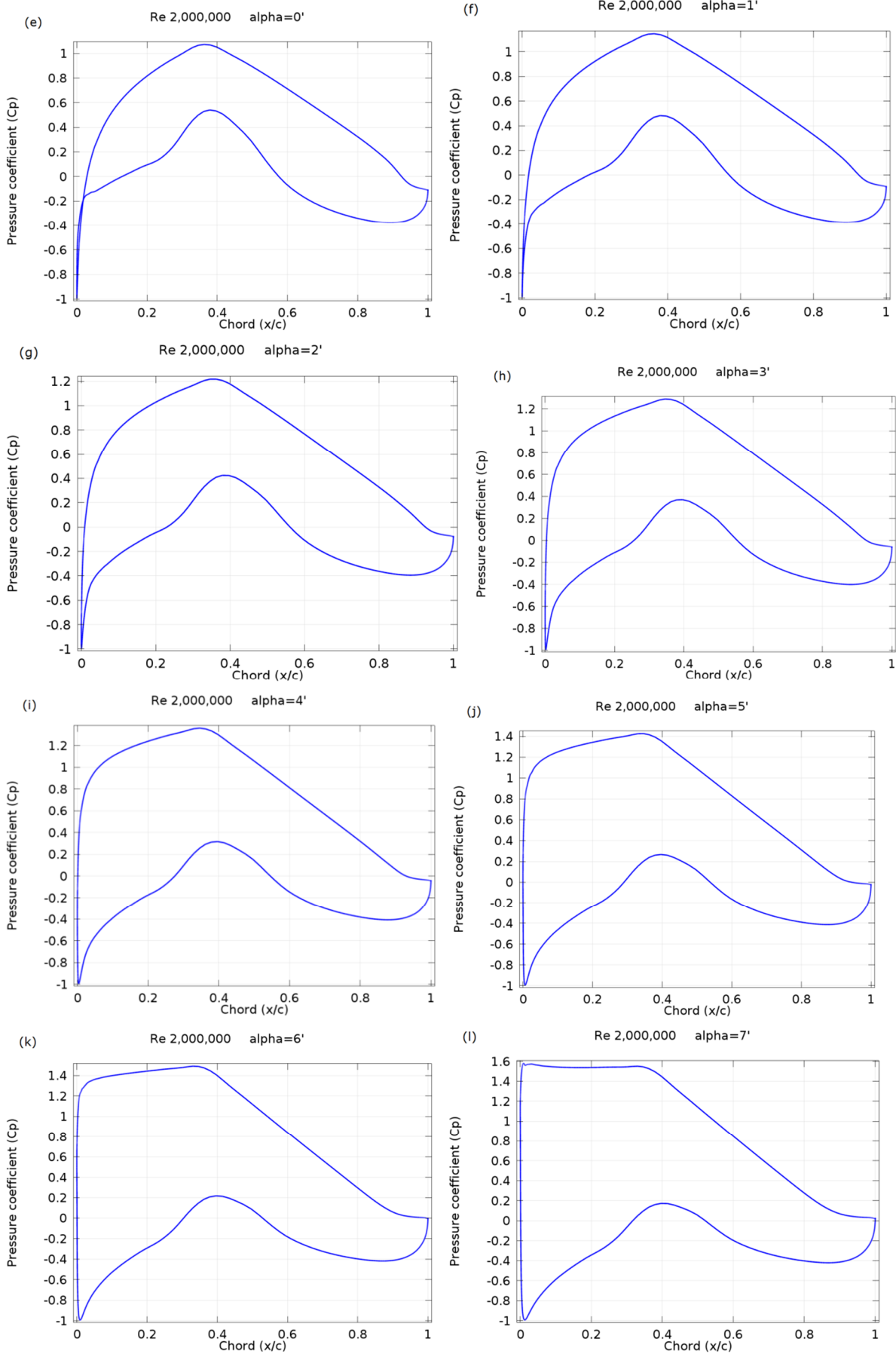


Fig. 6. Lift to drag ratio vs. angle of attack.

Pressure coefficient along the airfoil is shown in Figure 7 for each angle of attack. With the increasing angle of attack, pressure difference between upper and lower surface decreases first then increases.





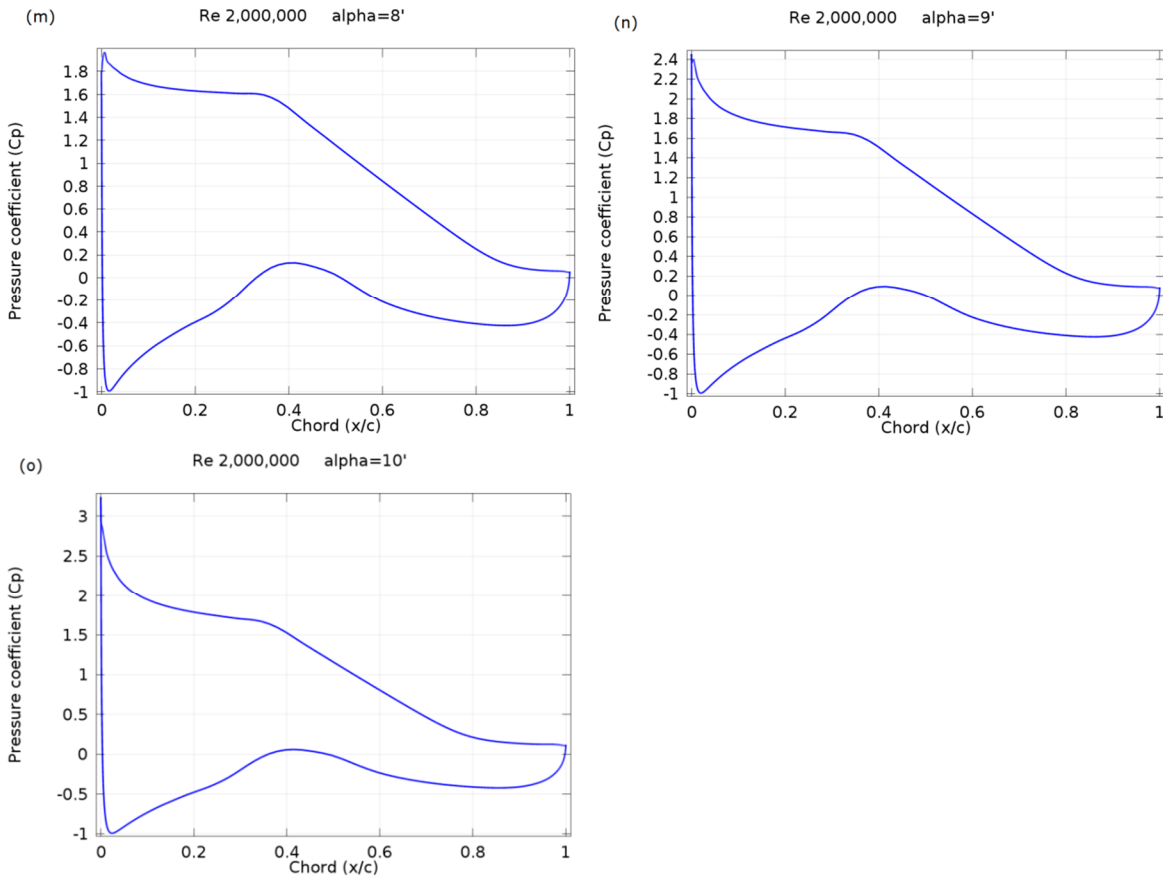


Fig. 7. Pressure coefficient along the airfoil from -4° to 10°

4. Conclusion

This paper numerically investigates the effect of angle of attack on the dynamic performance of S830 airfoil with SST turbulence model. Initially S809 airfoil is simulated and the results are compared with wind tunnel experiment. The comparison indicates good agreement and after validating the simulation accuracy of this calculation lifts and drag coefficient, lift to drag ratio, pressure coefficient are calculated for S830 airfoil. With the increasing angle of attack, lift and drag coefficient increase. Lift coefficient increment decreases with angle but drag coefficient increment increases. Next lift to drag ratios are calculated and compared. Maximum lift to drag ratio is obtained at the angle of attack 3 degree. With the increasing angle, maximum lift to drag ratio increases but maximum lift to drag ratio angle doesn't change. Finally pressure coefficient around airfoil is calculated and compared with different angle of attack.

Acknowledgment

I thank Middle East Technical University, allowing me to this work in there with their facility.

References

- [1] 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.

[2] Gul, M., Uzol, O., Akmandor, I.S. An Experimental Study on Active Flow Control Using Synthetic Jet Actuators over S809 Airfoil, *Journal of Physics: Conference Series* 524-012101, 2014.

[3] Sogukpinar, H., Bozkurt, I. 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, 1103-1108, 2015.

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

[5] Wolfe, P.W., Ochs, S.S. CFD Calculations of S809 Aerodynamic Characteristics, *AIAA*, 97-0973, 2015.

[6] COMSOL CFD, Module user guide. <http://www.comsol.com>, 2015.