Numeric Investigation of Compressible Flow Over NREL Phase VI Airfoil

Mohammad A. Hossain^{1,2}, Ziaul Huque^{1,2}, Raghava R. Kommalapati^{2,3}, Shubarna Khan^{1,2}

¹Department of Mechanical Engineering ²Center for Energy and Environmental Sustainability ³Department of Civil and Environmental Engineering Prairie View A&M University TX 77446, USA

Abstarct

This work deals with the numeric analysis of compressible flow around National Renewable Energy Laboratory (NREL) phase VI wind turbine blade airfoil S809. Although wind turbine airfoils are low Reynolds number airfoil, a reasonable investigation might be helpful for compressible flow under extreme condition. We considered a subsonic flow (mach no. 0.8) and determined the impact of this flow under seven different angle of attacks. The results show that shock takes place just after the mid span at the top surface and just before the mid span at the bottom surface. Slowly this transforms their position as angle of attack increases. A K- ω SST turbulent model is considered and the commercial CFD code ANSYS FLUENT is used to find the pressure coefficient (Cp) as well as the lift (C_L) and drag coefficient (C_D) . A graphical comparison of shock propagation has been shown with different angle of attack. Flow separation is also calculated along the airfoil.

Keywords— Compressible flow, Wind turbine airfoil, CFD, shock, flow separation

1. Intruduction

In near future wind will be the most reliable green energy in the history of mankind. According to the US Department of Energy the combustion of fossil fuels results in a net increase of 10.65 billion ton of atmospheric carbon dioxide every year [1] which have an enormous impact on environmental imbalance. So more focus on conversion of energy from alternate source has been given for the last few decades. The field of wind energy started to develop in 1970s after the oil crisis, with a large infusion of research money

in the United States, Denmark and Germany to find alternative resource of energy especially wind energy [2]. To design the blade of a wind turbine proper assessment of aerodynamic characteristics of airfoil plays the most important role. The most effective way to design the blade is to have accurate experimental data set for the correct airfoil. But such data set are not always available and the designer must rely on calculated data such as simulated data generated by large scale CFD code. Recent applications of CFD to solve the Navier Stokes equations for wind-turbine airfoils are reflected in the works of Yang, et al and Chang, et al [3]. They used their in-house code to solve the 2-D flow field about the S805 and S809 airfoils in attached flow (Yang, et al, 1994; Chang, et al, 1996) and the S809 airfoil in separated flow (Yang, et al, 1995). Computations were made with the Baldwin-Lomax (1978), Chein's low-Reynoldsnumber k- ε [4], and Wilcox's low-Reynolds-number k- ω (1994) turbulence models [5].

In recent years development of wind turbine blade airfoil has been ongoing and have many modifications in order to improve performance for special application and wind conditions. To gain efficiency the blade should have both twist and taper. The taper, twist and airfoil characteristics should all be combined in order to give the best possible energy capture for the rotor speed and site conditions [6]. In this paper we tried to find out the aerodynamic characteristics in compressible flow condition because to the best the author's knowledge very little work has been done in this field due to lack of available experimental data.

2. K-w SST Turbulent Model

The SST k-ω turbulence model (Menter 1993) is a two equation eddy-viscosity model. SST K-w model can be used as a turbulent model without considering any extra damping function [7]. This model can produce a large turbulence levels with regions of large normal strain like stagnation region and regions with strong acceleration [8]. The original K- ω model can be defined as-

$$\frac{\partial\rho k}{\partial t} + \frac{\partial\rho u_j k}{\partial x_j} = P_k - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_{k1} \mu_t) \frac{\partial k}{\partial x_j} \right]$$
$$\frac{\partial\rho \omega}{\partial t} + \frac{\partial\rho u_j \omega}{\partial x_j} = \gamma_1 P_\omega - \beta_1 \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_{\omega1} \mu_t) \frac{\partial\omega}{\partial x_j} \right]$$

The Shear Stress Transport (SST) formulation combines the two equations. The shear stress boundary layer and kinematic eddy viscosity can be defined as:

$$\tau = \rho \sqrt{\frac{Production_k}{Dissipation_k}} a_1 k$$
$$\nu_t = \frac{a_1 k}{max(a_1 \omega; \Omega)}$$

3. Airfoil Selection

National Renewable Energy Laboratory (NREL) has developed different airfoil specially for horizontal axis wind turbine [9]. Some of the airfoils are S801, S805, S809, S8012 etc. Among them we considered S809 as this airfoil was used in NREL phase VI wind turbine experiments.



Figure 1. S809 Airfoil profile

The airfoil for simulation is created from the set of vertices obtained from the University of Illinois at Urbana Champagne (UIUC) airfoil database [10]. These vertices are connected with a smooth curve creating the surface of the airfoil.

4. CFD Simulation

4.1. CFD Modeling

We considered a subsonic flow (mach 0.8) and a range of 0^0 to 10^0 angle of attack (α). Grid generation is done by ANSYS ICEM CFD algorithm. In this work approximately 0.2 million unstructured triangular elements were used to generate the mesh. Computational domain consists of a smooth parabola for better resolution of results.



Figure 2. Mesh Domain

In order to have a stable and reliable solution, the mesh has minimum number of elements in the airfoil wall and grid points are clustered near the leading edge and trailing edge Fig. 3 in order to capture the flow separation and boundary layer of the airfoil wall.



Figure 3. Mesh around airfoil





Figure 4. Mesh around (a) Leading edge, (b) Trailing edge

A pressure based solver is set and ideal gas approximation is considered. In order to solve 2D Navier-stokes equation, correct boundary condition plays very important role for appropriate results. We considered K- ω SST turbulent model with no slip boundary condition at the wall. Outlet pressure is considered as atmospheric pressure. Coupled second order upwind method is used as a solving method. The turbulent viscosity ratio is considered 10% and operating temperature is assumed 300K. The operating condition is zero gage pressure or 101325 Pa absolute pressure. Sutherland's viscosity law which is the relation between the dynamic viscosity (μ) and the absolute temperature (T) is considered. Sutherland's law is based on kinetic theory of ideal gases and an idealized intermolecular-force potential [11] which is being used for many advanced CFD simulation.

4.2. CFD Result

Our objective was to find out the flow behavior around the airfoil in compressible flow condition. In order to do that we calculated the static pressure the mach number the turbulent viscosity and the temperature variation around the airfoil. We have also calculated the coefficient of pressure (Cp) distribution around the airfoil and the lift (C_{I}) and drag (C_D) coefficient at different angle of attack. In order to validate the model we need to compare the results with the experimental data. But we have experimental data for low Reynolds number. So we ran our simulation code considering Re = 300,000at different angles of attack (α) which gives a good agreement with the experimental data. After that we ran our code with actual compressible flow boundary conditions.

Figure.5(a)-5(f) show a static pressure contour of S809 airfoil at various angles of attack and 0.8 mach number. Fig.7(a)-7(f) show the velocity

distribution of the same condition as the previous.



Figure 5(a). Static Pressure distribution of S809 airfoil at $\alpha=0^{0}$



Figure 5(b). Static Pressure distribution of S809 airfoil at $\alpha=2^{0}$



Figure 5(c). Static Pressure distribution of S809 airfoil at $\alpha=4^{\circ}$



Figure 5(d). Static Pressure distribution of S809 airfoil at $\alpha=6^{0}$



Figure 5(e). Static Pressure distribution of S809 airfoil at α =8⁰



Figure 5(f). Static Pressure distribution of S809 airfoil at $\alpha=9^{0}$

The pressure contour shows that there is a shock at both top and bottom wall of the airfoil. As angle of attack increases shock shifts their positions and at above 8 degree the shock has a remarkable change at the lower surface. Fig.6 shows the pressure distribution along the chord for the low velocity flow over the airfoil which is a typical pressure distribution curve. And Figure.7(a)-7(f) show the variation due to shock. It is observed that at compressible flow condition pressure suddenly changes both in upper face and lower face of the airfoil and its position changes with the change of angle of attack.



Figure 6. Coefficient of pressure (Cp) along airfoil at V=7.54ms⁻¹ and $\alpha = 0$



airfoil at $\alpha = 2^0$



airfoil at $\alpha = 4^0$







Figure 7(e). Coefficient of pressure (Cp) along airfoil at $\alpha = 8^0$



Figure 7(f). Coefficient of pressure (Cp) along airfoil at $\alpha = 9^0$

Figure.8 and Figure.9 show the change of integrated lift and drag coefficients ($C_L \& C_D$) as a function of angle of attack (α). It has been observed that the stall condition occurred above 8 degree of angle of attack.



Figure 8. Integrated Lift Coefficient (C_L) with respect to angle of attack (α)



Figure 9. Integrated Drag Coefficient (C_d) with respect to angle of attack (α)

Flow separation is also observed during the simulation. We found that after the shock the flow separation starts and as angle of attack increases flow separation also occurs more rapidly.



Figure 10. Separation of Flow just after the shock at angle of attack $\alpha = 9^0$

5. Conclusion

Observation of shock generation and flow behavior of compressible flow over S809 airfoil was the primary objective of our work. It also shows the pressure distribution and effect of shock around airfoil. Flow separation shows that it must be taken into account during design of wind turbine blade.

Acknowledgment

This work is supported by the National Science Foundation (NSF) through the Center for Energy and Environmental Sustainability (CEES), a CREST Center, award no. 1036593

Reference

[1] US Government, Department of Energy, US Department of Energy on Green House Gases, 2009 http://www.eia.doe.gov

[2] Promod jain, "*Wind Energy Engineering*", McGraw Hill 2011, Chaptern 1, pp 1,

[3] Chang, Y. L., S. L. Yang, and O. Arici, 1996, "Flow Field Computation of the NREL S809 Airfoil Using Various Turbulence Models," *ASME, Energy Week-96*, Book VIII, vol. I-Wind Energy, pp. 172-178

[4] Chien, K.-Y., 1982, "Predictions of Channel and Boundary-Layer Flows with a Low-Reynolds-Number Turbulence Model," *AIAA J.*, vol. 20, pp. 33

[5] Wilcox, D. C., 1994, Turbulence Modeling for *CFD*, DCW Industries, Inc., La Cañada, CA.

[6] Anders Ahlstrom, " Aeroelastic Simulation of Wind Turbine Dynamics", PhD Thesis, Royal Institute of Technology, Department of Mechanics, Sweden, April 2005

[7] Menter, F. R. (1993), "Zonal Two Equation $k-\omega$ Turbulence Models for Aerodynamic Flows", *AIAA* Paper 93-2906

[8] Menter, F. R. (1994), "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", *AIAA Journal*, vol. 32, no 8. pp.1598-1605

[9] J. L. Tangler, D. M. Somers, 'NREL Airfoil Families for HAWTs', AWEA 1995 doc

[10]"UIUC Airfoil Coordinates Database," http://www.ae.illinois.edu/m-selig/ads/coord database.html.

[11] Sutherland, W. (1893), "The viscosity of gases and molecular force", *Philosophical Magazine*, S. 5, 36, pp. 507-531 (1893