AERODYNAMIC PERFORMANCE PREDICTION OF WIND TURBINE AIRFOIL THROUGH CFD

Tabinda Kanwal [1], Mukkarum Hussain [2], Samreen Ahmed [3], Madiha Imdad [4], Mirza Mehmood Baig [5]

[1,2,4]Department of Mathematics, NEDUET, Karachi, Pakistan

[3]Institute of Space Technology (IST), Karachi, Pakistan

Corresponding Author: tabindakanwal1@gmail.com

ABSTRACT: *Energy is considered as most vital instrument of socioeconomic development of a country. Unfortunately in our country enhancement in energy production is not as increased as its demand. Currently country is facing serious energy crises. This issue has not been addressed properly as well as could not find available economical and long term solutions. Wind turbines are one of the vital sources of energy and very much in progress worldwide. Research and development of airfoils for wind turbine have been largely focused by researchers for last three decades. In present study numerical computations is carried out for S809 airfoil which is specifically designed for horizontal axis wind turbine (HAWT). Aerodynamic performance data for S809 airfoil is generated through CFD and compared with available experimental data. Gridgen is used for pre-processing while computations are run on Fluent software. Post processing is done using Tecplot software. Results for high Reynolds number and low angles of attack are in good agreement with available experimental data. Differences for high angles of attack are due to large portion of separation and stall. Present study depicts that CFD is an appropriate choice for computing aerodynamic performance of airfoils designed for wind turbines. This study would be very helpful for future development of wind turbine airfoils.*

Keywords: Low Reynolds Number Flow, Angle of Attack, S809 Airfoil, Horizontal Axis Wind Turbine, Stall.

INTRODUCTION

Wind turbines are used to transform wind energy into electric power [1]. Wind turbine blade is the key component to generate output power. Blade geometry is complex 3D profile consist of a series of low Reynolds number airfoils. Airfoils developed by NASA for aircrafts [2] were initially used by wind turbine manufacturer for blade geometry. Thickness and performance required for wind turbine airfoils [3] are different than aircraft airfoils. Aircraft airfoils were best suited for high Reynolds numbers. Their performance at lower Reynolds number is very poor and suffers from significant laminar separation bubbles when use in wind turbine [4]. These airfoils are also very thin and not appropriate to address the structural requirement at the blade root region [5]. The progress of airfoils design and testing specifically for horizontal-axis wind-turbine applications has been a great interest of researchers for last three decades [3, 4]. National Renewable Energy Laboratory (NREL) and other agencies have developed several families of airfoils specifically for horizontal-axis wind-turbine [6, 7].

Aerodynamic performance data is necessary for airfoil before it is used in practical wind turbine systems [8, 9]. The first and most reliable source to generate aerodynamic performance data is wind tunnel testing but wind tunnel testing is costly and time consuming. Some empirical tools available for generating aerodynamic performance data but they are not much reliable. Computational Fluid Dynamics (CFD) is a methodology that enables to study the dynamics of fluid in motion [10, 11]. CFD as a computational technology is eminently suited to develop the concept of numerical test rig or virtual wind tunnel [12, 13] . In present study CFD computations are carried out for S809 [14] airfoil which is specifically designed for horizontal axis wind-turbine. Aerodynamic performance data is generated through CFD and compared with available experimental data [15, 16]. Geometrical modeling and grid generation is done in Gridgen software. Steady state simulations are carried out for all cases using ANSYS Fluent pressure-based coupled solver. Second order upwind spatial discretization is used for momentum and energy equations. Mesh is refined such that boundary layer is resolved properly.

TEST CASE

The experimental data that has been used is taken from technical report published by NREL in December 2001[15, 16]. Aerodynamic coefficients obtained at the CSU and OSU wind tunnel with a range of Reynolds number for various angles of attack. This data is chosen because of its authenticity. During literature study it has been found that this data has been used previously by a number of researchers. Geometrical description of test case used, grid generation and solver setting details are given in following sections.

Geometry Model and Grid Generation

The accuracy of the results depends on numerical scheme, convergence and grid quality Grid quality means how many points and how well they are allocated. In present study Gridgen software is used for grid generation. Structured mesh is generated around the airfoil which is shown in Fig.1a and Fig.1 b.

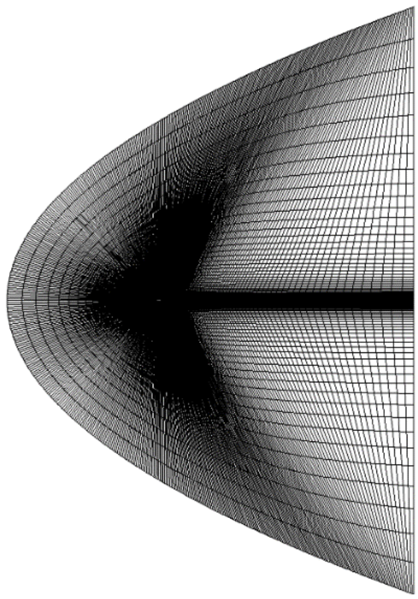

Figure 1a: Grid Generated for S809 Airfoil

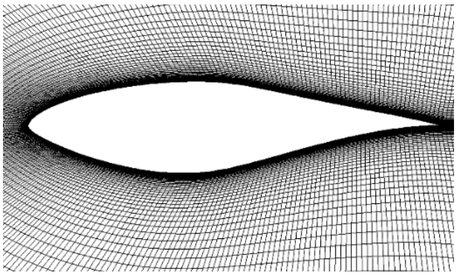

Figure 1b: Grid Generated for S809 Airfoil

Boundary Conditions

Boundary conditions applied in present computation are shown in Fig.2. Specifications and solver setting used in Fluent solver is given in TABLE I and TABLE II.

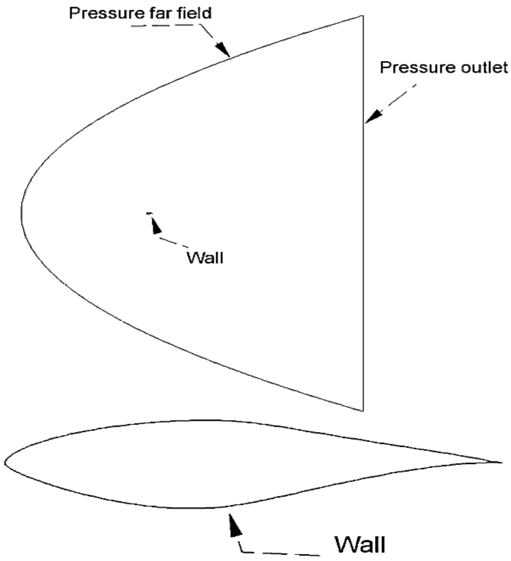

Figure 1: Boundary Conditions Description

TABLE I: Ansys Fluent Specification

Material	Air
Cell Zone Conditions	Fluid
Density (kg/m^3)	1.225
Depth (m)	1
Enthalpy (j/kg)	0
Length (m)	1
Pressure (Pascal)	101324.9

TABLE II: Ansys Fluent Solver Setting

ANSYS FLUENT	
Numerical Method	
Algorithm	Coupled
Solver	Density Based
Spatial Discretization	
Gradient	Least Square Cell Based
Pressure	Standard
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Specific Dissipation Rate	Second Order Upwind

RESULTS AND DISCUSSION

Flow around S809 airfoil is computed through Fluent pressure-based coupled solver. Results are compared with available experimental data for validation purpose. Lift and drag coefficients are compared for a range of Reynolds number and angles of attack. CFD software Fluent is used as a solver while post processing is done using Tecplot and Excel.

CSU wind tunnel data is used for validation of numerical computations carried out at Reynolds numbers 0.3, 0.5, and 0.6 million. Experimental data for approximately '0' to '90' degree angles of attack is available for validation.

Figure 3 and Figure 4 shows the behavior of computed aerodynamic coefficients (C_l and C_{dp}) as a function of angle of attack, for Re = 3×10^5.

Results depict that the qualitative behavior of computed result is in good agreement with experimental data. As far as quantities are concerned computed results are over predicted at low angles of attack while under predicted at higher values of angles of attack.

Low angle of attack results are very much precise and accurate. For higher values of angles of attack deviation between computed results and experimental data is observed. This deviation is due to presence of separation. Separated flows are complex and very challenging to compute through numerical method. At low Reynolds and high angles of attack large portion of airfoil leeward side separated from wall and create vortex. Figure 13 to Figure 15 show separation formation in S809 airfoil for high angles of attack. It is evident from above mentioned figures that separation bubble becomes larger as angle of attack increases. Figure 5 and Figure 6 shows comparison of aerodynamics performance for Re = 5×10^5. Results depict same behavior as discussed for Re = 3×10^5 previously.

For Re = 6.5×10^5computed results are much closer to experimental data as compare to low Reynolds number values as shown in Figure 7 and Figure 8. Prediction of lift and drag coefficients before separation are excellent. At higher values of angles of attack some deviation is due to presence of separation.

OSU wind tunnel data is used for validation of numerical computation carried out at Re = 7.5×10^5 and Re = 1×10^6. Although experiments at higher angles of attack for this Reynolds number is not carried out however for negative angles of attack experimental data of lift and drag coefficients are available. Since S809 airfoil is non-symmetric airfoil therefore experimental data at negative angle of attack has its significance. Figure 9 and Figure 10 shown results of drag and lift coefficients for Re = 7.5×10^5 respectively. Results depict that qualitative behavior of drag and lift coefficients are predicted remarkably well. Difference between results are minor and within acceptable range. Results for Reynolds number '1' million is presented in Figure 11 and Figure 12. Drag is little over predicted at zero angle of attack while under predicted at larger values. Behavior of drag coefficient is predicted well. Results for lift coefficient are in very good agreement with experimental data.

Overall results for lower values of angle of attack are excellent. Drag coefficient is slightly over predicted while lift coefficient is computed very well. Discrepancies between experimental data and numerical computation at higher angle of attack are observed. The main reason for this discrepancy is the presence of separation and formation of vortex. This issue can be resolved by using higher order low dissipative schemes. Results computed at higher Reynolds number are more accurate as compare to low Reynolds number computations.

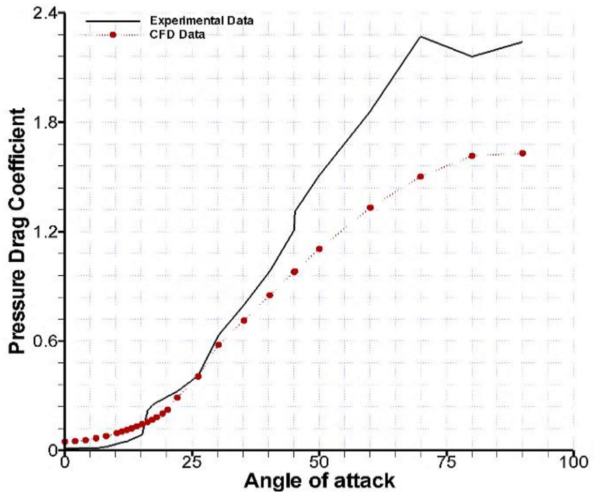

Figure 3: CFD v/s CSU Wind Tunnel Pressure Drag Coefficient for Re = 300, 000

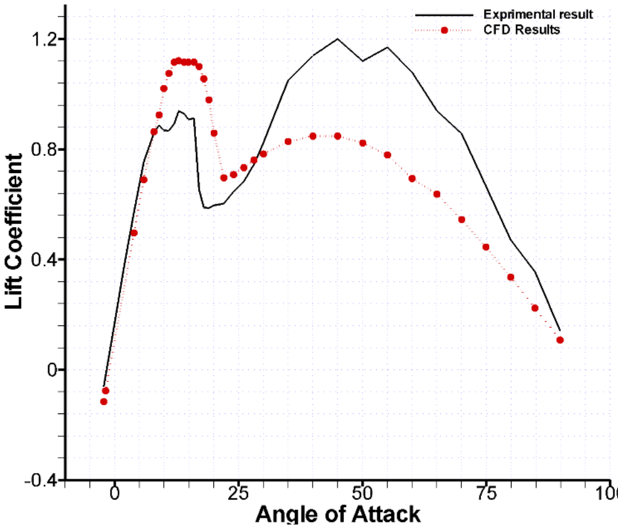

Figure 6: CFD v/s CSU Wind Tunnel Lift Coefficient for Re = 500, 000

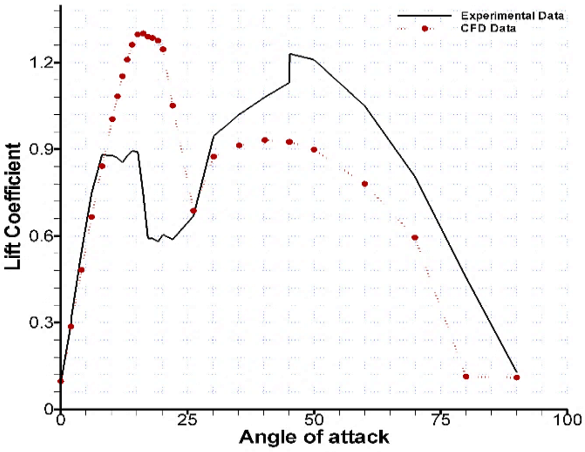

Figure 4: CFD v/s CSU Wind Tunnel Lift Coefficient for Re = 300, 000

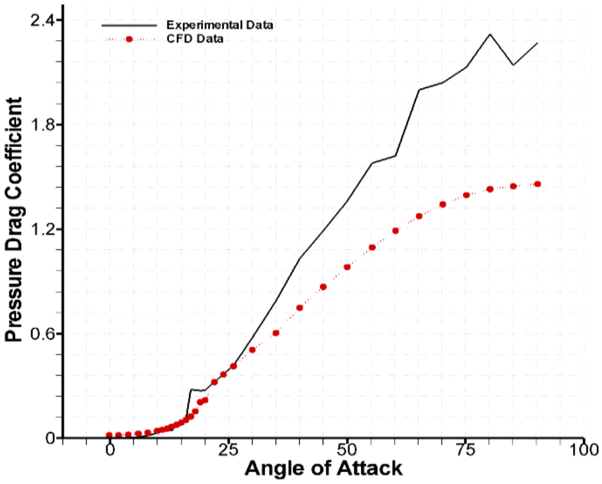

Figure 7: CFD v/s CSU Wind Tunnel Pressure Drag Coefficient for Re = 650, 000

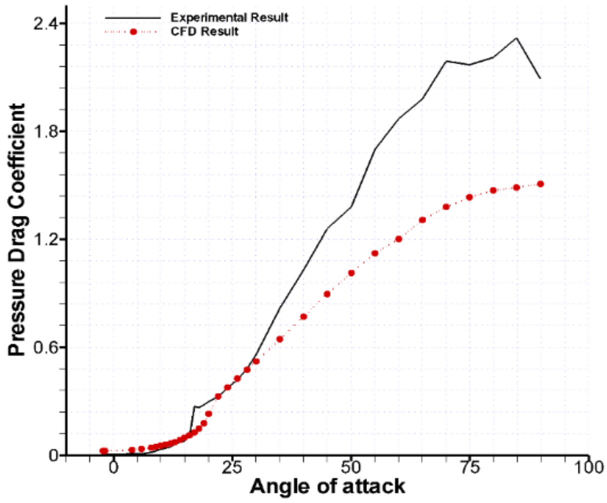

Figure 5: CFD v/s CSU Wind Tunnel Pressure Drag Coefficient for Re = 500, 000

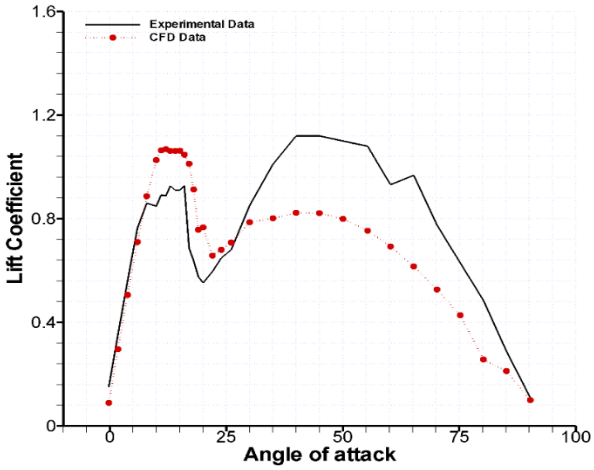

Figure 8: CFD v/s CSU Wind Tunnel Lift Coefficient for Re = 650, 000

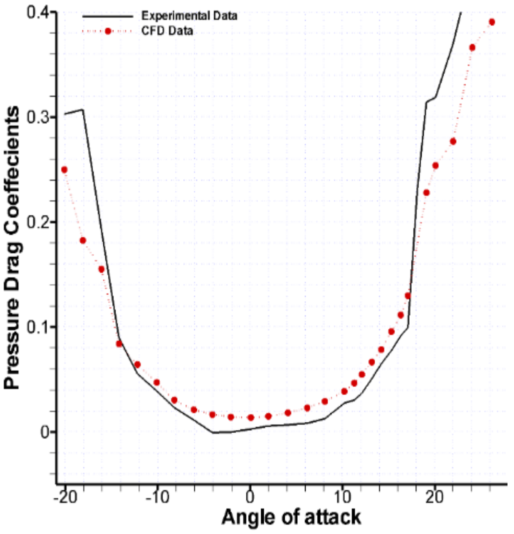

Figure 9: CFD v/s OSU Wind Tunnel Pressure Drag Coefficient for Re = 750, 000

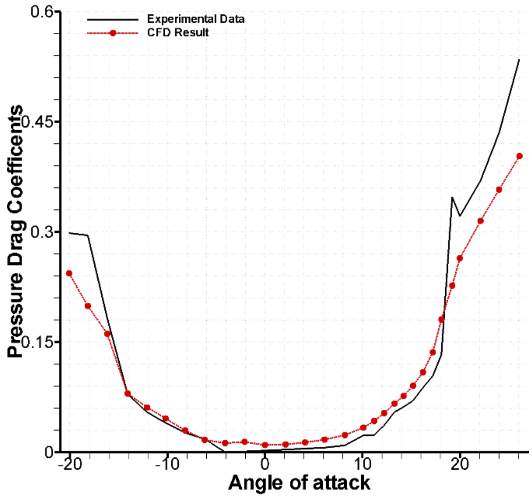

Figure 11: CFD v/s OSU Wind Tunnel Pressure Drag Coefficient for Re = 1, 000, 000

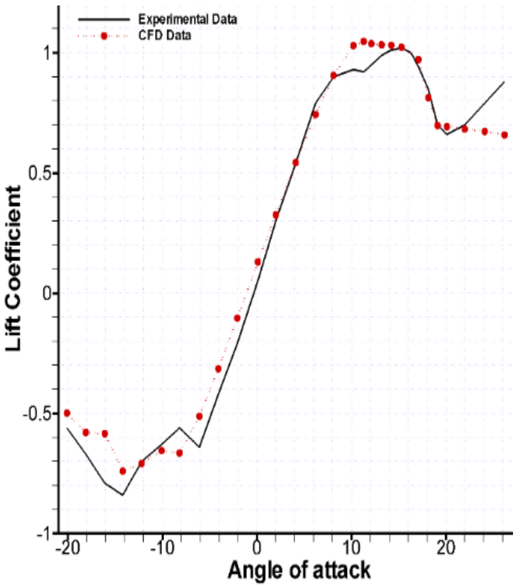

Figure 10: CFD v/s OSU Wind Tunnel Lift Coefficient for Re = 750, 000

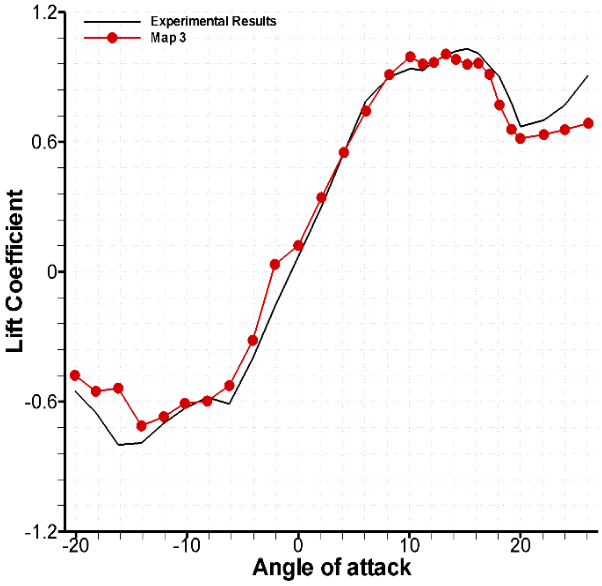

Figure 12: CFD v/s OSU Wind Tunnel Lift Coefficient for Re = 1, 000, 000

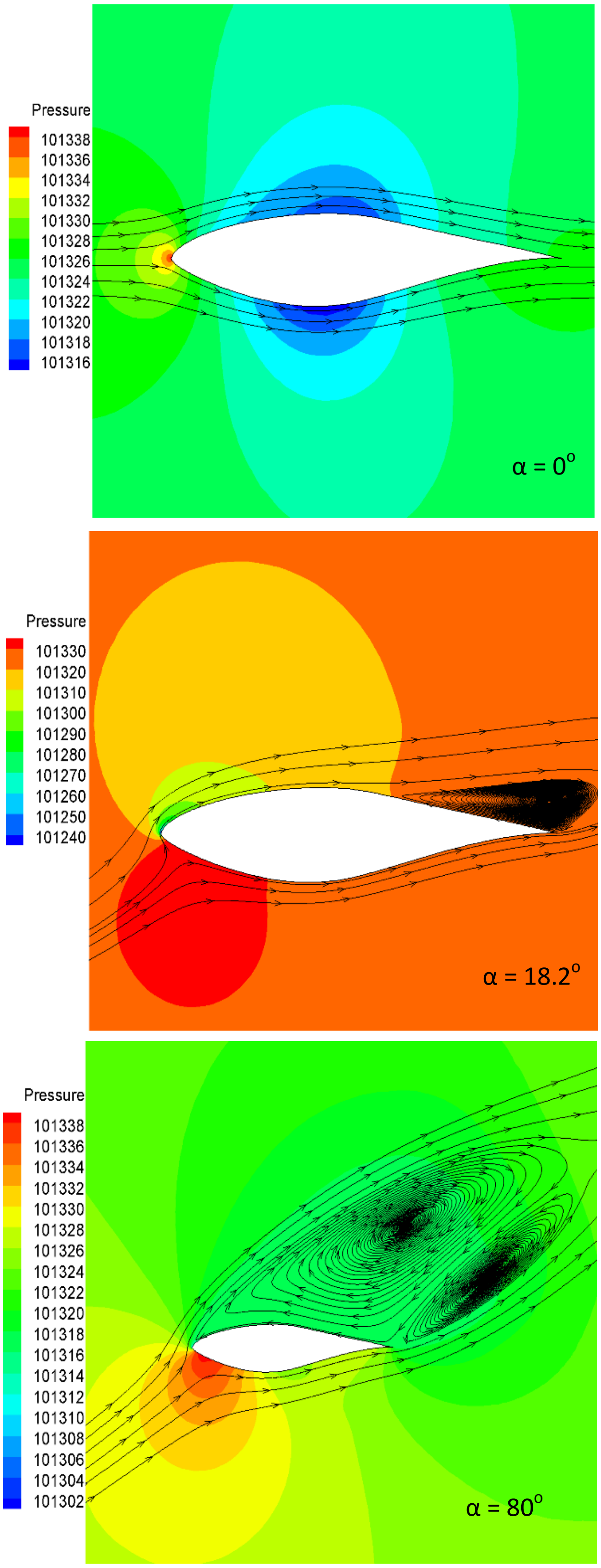

Figure 13: Pressure Contour around S809 Airfoil Re = 300, 000

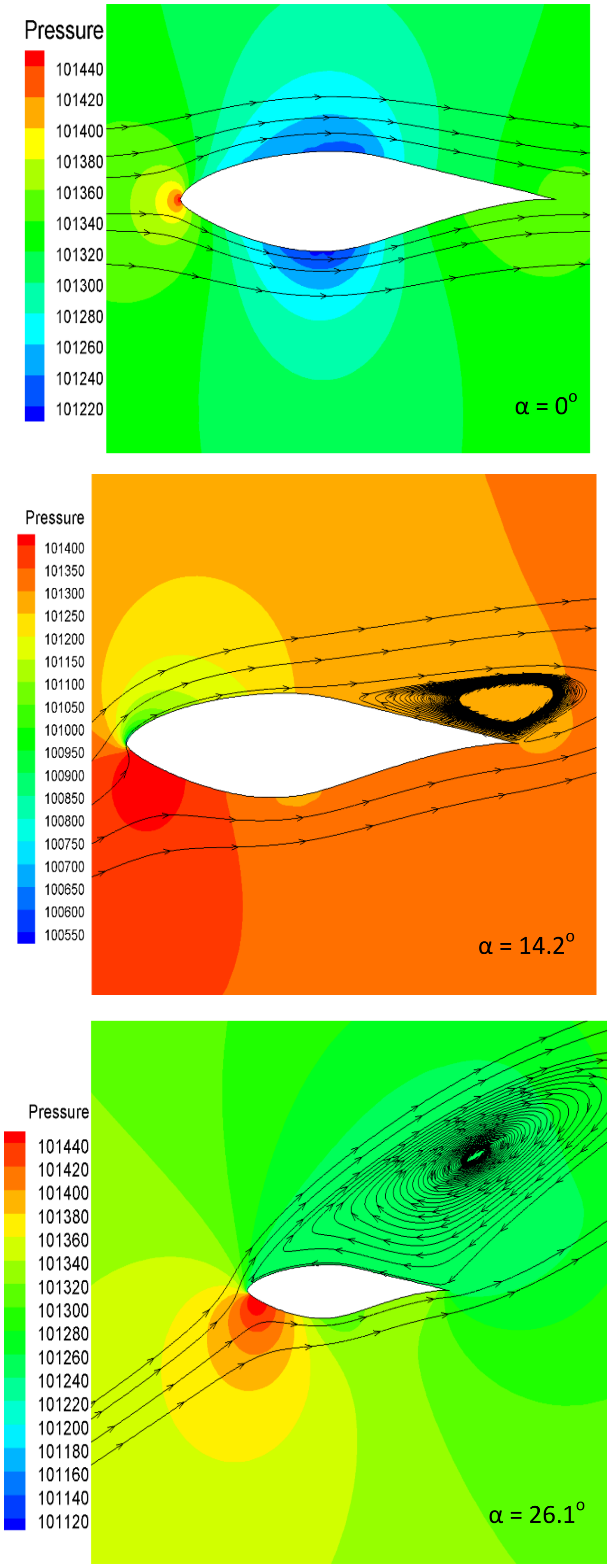

Figure 14: Pressure Contour around S809 Airfoil Re = 650, 000

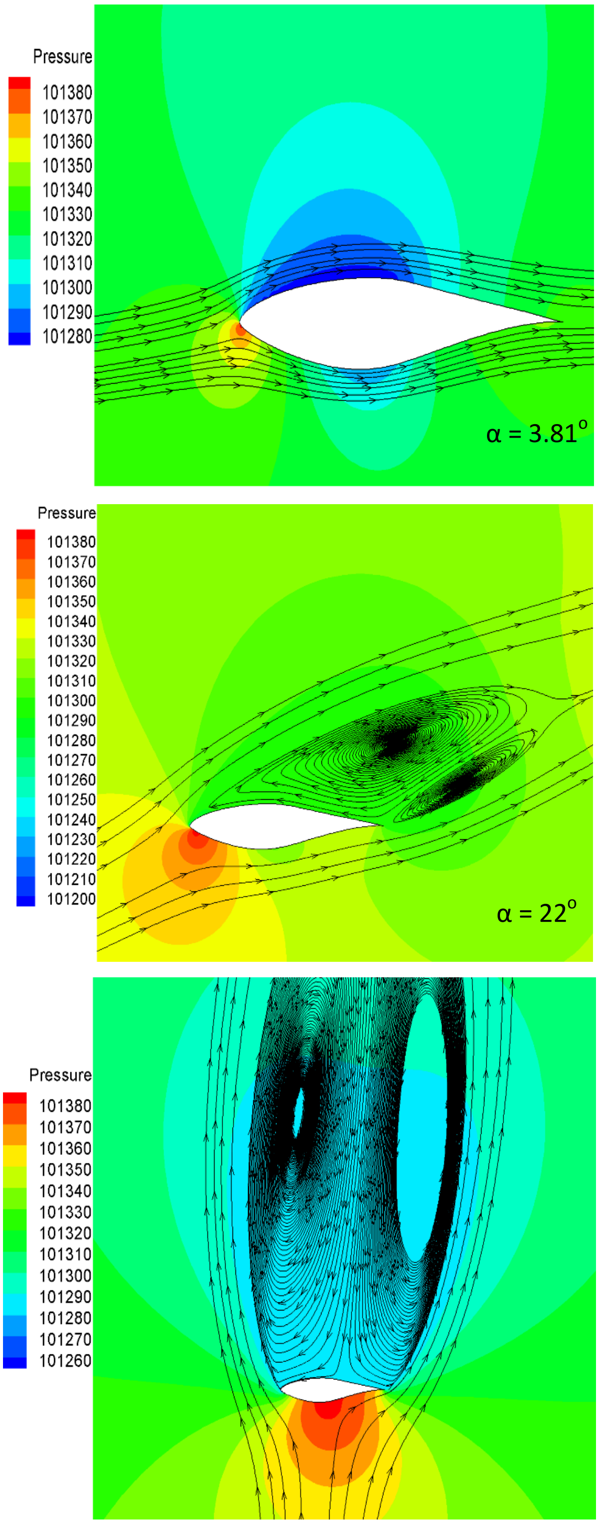

Figure 15: Pressure Contour around S809 Airfoil Re = 1,000, 000

CONCLUSION

Overall results are satisfactory and very promising. Present study highlights the bright future of CFD for research and design of low Reynolds number airfoils particularly suitable for wind turbines. Use of CFD in design and analysis phase of wind turbine in general and airfoil in particular not only reduce total cost but also speed up the whole process.

ACKNOWLEDGMENT

Authors are thankful to Department of Mathematics, NEDUET, Karachi, Pakistan for their support throughout the work.

REFERENCES

[1] Tony Burton, David Sharpe, Nick Jenkins and Ervin Bossanyi, Wind Energy Hand Book, Wiley & Sons, Ltd., 2001.

[2] Ira H. Abbott and Albert E. Von Doenhoff, Theory of Wing Section:Including a Summary of Airfoil Data, Dover Publication Inc, 1958.

[3] S.J.Miley, "A Catalog of Low Reynolds Number Airfoil data For Wind Turbine," Texas, February 1992.

[4] M. S. Selig, "Low Reynolds Number Airfoil Design," in NATO Research and Technology Organization (RTO), USA, 24-28 November 2003.

[5] D. A. Sepra, Wind Turbine Technalogy Fundamental Concept of Wind Turbine Engineering, Fifth ed., New York: ASME Press, 2009.

[6] D.M.Somers, "Effects of Airfoil Thickness and Maximum Lift Coefficient on Roughness Sensitivity," NREL, January 2005.

[7] D. Somers, "Design and Experiment Results for the S809 Airfoil," NREL/SR-440-6918, 1997.

[8] A.Gross and H.F.Fasel, "Numerical Investigation of Different Wind Turbines Airfoils," AIAA, January 2011.

[9] John J. Bertain and Russell M. Cummings, Aerodynamics of Engineers, Canada: Pearson Educational International, 2009.

[10] John C Tannehill, Dale A. Anderson and Richard H. Pletcher, Computational fluid mechanics and heat transfer, Washington DC, United State Of America.: Taylor & Francis, 1997.

[11] J. D. Anderson, Computational Fluid Dynamics, the Basic Application, USA: McGraw-Hill, Inc., 1995.

[12] K. A. Hoffmann and S. T. Chiang, Computational Fluid Dynamics,Fourth Edition,Volume I, Wichita,Kansas,USA.: Engineering Education System(EES), August 2000.

[13] Hoffman, Klaus A., J.Chiang and Steve T, Computational fluid dynamics, 4th edition, Volume III, August 2000.

[14] Walter P. and Wolfe, "CFD Calculations of S809 Aerdynamic Characteristics," AIAA, 1997.

[15] M.M.Hand, D.A.Simms, L.J.Fingersh, D.W.Jager, J.R.Cotrell, S.Schreck and S.M.Larwood, "Unsteady Aerodynamics Experiment Phase VI:Wind Tunnel Test Configurations and Avaliable Data Campaigns," NREL Technical Report, December 2001.

[16] Simms D., Schreck S., Hand M. and Fingersh L.J., "Unsteady Aerodynamics Experiment in the NASA Ames Wind Tunnel: A Comparision of Predections to Measurements," NREL Technical Report, 2001.