Scientific Journal of Impact Factor(SJIF): 3.134

e-ISSN(O): 2348-4470
p-ISSN(P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 2, Issue 10, October -2015

OPTIMIZATION OF A WIND TURBINE AIRFOIL AT ITS 50% SPAN UNDER COMPRESSIBLE FLOW

Shivasharana[1], Dr. K .V. Muralidharan[2]

[1]Mechanical Engineering Department, M V J College Bengaluru
[2] Professor Mechanical Engineering Department, M V J College Bengaluru.

Abstract: *This work presents shape optimization of S809 airfoil under compressible flow using CFD simulation software. The FLUENT software is used for calculation of the flow field. Optimal S809 airfoil shape is obtained which increases both C_L and C_L/C_D for a given, rotational speed and wind speed. The results showed that aerodynamics characteristics of optimized S809 are improved significantly.*

In this work compressible flow condition has considered. Day by day development of WT blade airfoil is going on and has got many modifications to improve the performance for wind conditions and special application. To get higher efficiency the blade should have correct aerodynamic shape. This aerodynamic characteristic should be to give the best possible energy capture for the site conditions and rotor speed. Length of the blade of a WT is increasing day by day. For a 100mblade rotating at 25 rpm the tip speed of WT will approximately reach a Mach no. of 0.78. Therefore an analysis and optimisation under compressible flow might be helpful for the designer to avoid unexpected incidents.

Keywords-*Airfoil; optimization; compressible flow; wind turbine; coefficient of lift; coefficient of drag; individuals;*

I. INTRODUCTION

Energy depletion is becoming one of the most significant concerns of society during the coming decades. The fossil fuels that are currently pushing human society to further prosperity were formed from the tissues of organisms that lived more than 100 million years ago. In order to keep up with the pace of modern civilization, fossil fuels, including coal, oil and natural gases, are being consumed at an incredibly fast rate. At this rate those valuable reserves will be depleted in the near future. Therefore, developing substitute energy sources is very important. Renewable energy sources, like wind, solar, tides and geothermal energy have nearly an unlimited supply and can be converted into useful power.

Wind power is one of the widely available energy sources on the earth. With recent significance on emission free renewable energy, wind energy has taken a centre stage in recent years with rapid growth in deployment of wind turbines (WT) worldwide. The first time people started to extract power from the wind can be traced back to the 1st century when the first wind power driven machine was invented. However, the application of wind power was very limited because of the structural complexity and high cost of wind turbines. The inherent capacity of wind energy as a power source was reinvigorated in the late 1960s because of the increasing concern for environment. As a consequence, pubic large scale wind turbines started emerging. A lot of effort and resources are being spent by researchers today in order to efficiently extract the power from wind. Compared with setting up a wind tunnel and performing a full scale test, the cost of numerical simulation is much less. Therefore, Computational Fluid Dynamics (CFD) analysis is becoming more desirable. With the advance of computational science it is possible to perform large CFD simulations on powerful computers. Among WT, HAWTs (horizontal-axis-wind-turbines) are mostly used for power production (Megawatt range). It is well known that the power generation of a HAWT depends on the number of blades, the C_L/C_D of airfoil, and the ratio tip speed. Thus, one of the goals in designing of a WT blade is to increase its C_L/C_D. Therefore significant effort has dedicated in recent years to optimization of shape of WT blade to achieve high C_L/C_D.

This work focused on the aerodynamic shape optimization of airfoil section used in wind turbine blade since they affect the aerodynamic performance of WT which in turn influences the power production of wind turbine. National Renewable Energy Laboratory (NREL) has developed airfoils for HAWT applications since 1984. This work focuses on the optimization of NREL S809 airfoil at its 50% span. This work presents shape optimization of S809 airfoil under compressible flow using CFD simulation software. The FLUENT software is used for calculation of the flow field. Optimal S809 airfoil shape is obtained which increases both C_L and C_L/C_D for a given, rotational speed and wind speed. The results showed that aerodynamics characteristics of optimized S809 are improved significantly. For the purpose of validation, the present results are compared with results of Ritlop and Nadarajah[6]. In this work optimisation of WT airfoil (S809) under compressible flow condition have been taken because, less work has been done in this field because of less availability of experimental data. Day by day development of WT blade airfoil is going on and has got many modifications to improve the performance for wind conditions and special application. To get higher efficiency the blade should have correct aerodynamic shape. This aerodynamic characteristic should be to give the best possible energy capture for the site conditions and rotor speed. Length of the blade of a WT is increasing day by day. In recent years NREL 5MW offshore wind turbine has blade length of 90 m. In future the blade length of the WT will increase more. For a 100mblade rotating at 25 rpm the tip speed of WT will approximately reach a Mach no. of 0.78[1]. Therefore a analysis and optimisation under compressible flow might be helpful for the designer to avoid unexpected incidents.

II. METHODOLOGY

2.1 Meshing of airfoil

- Meshing has done in ANSYS ICEMCFD.
- 0.1million rectangular elements are used to generate the mesh.
- The maximum domain length is 125m and the width is 90 m.

To get nearly accurate solutions more numbers of elements are used near the airfoil wall. And also mesh is refined in order to capture the flow separation and boundary layer of the airfoil wall by using grid independency.

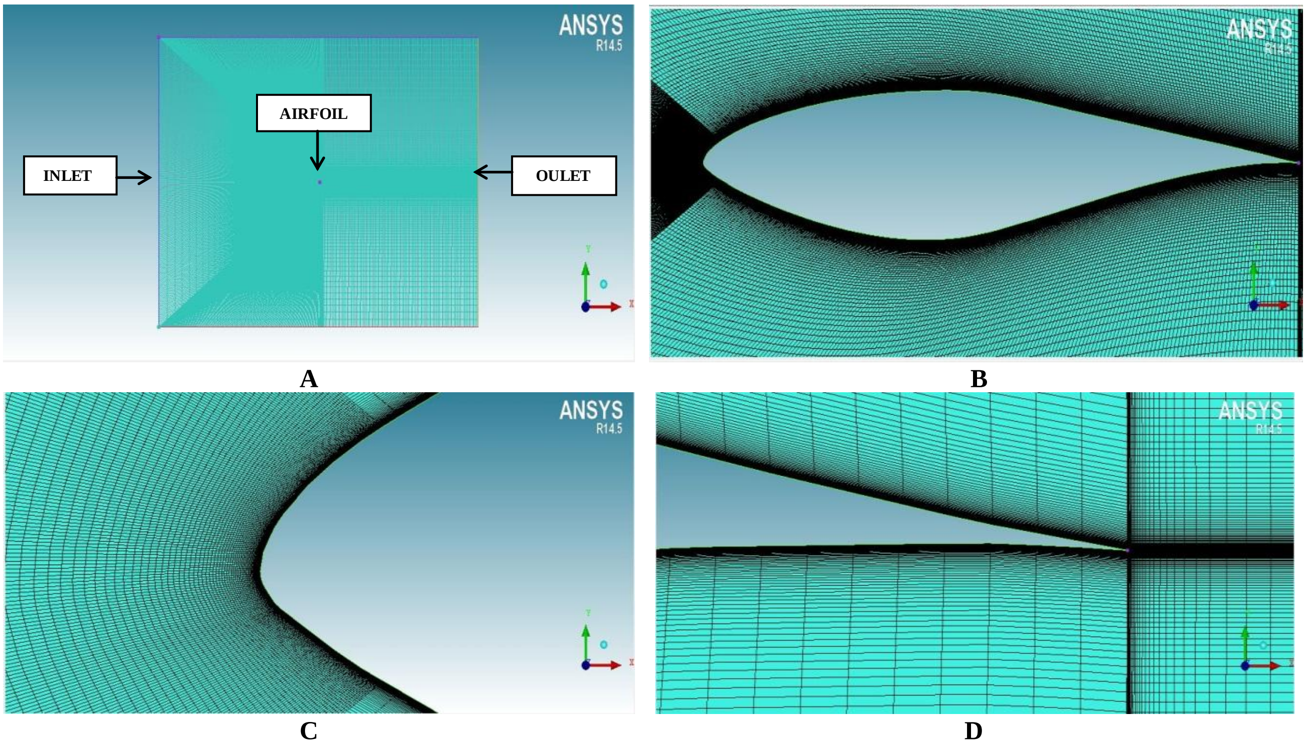

Figure 1. A) Mesh domain. B) Refined mesh near wall. C) Mesh near leading edge. D) Mesh near trailing edge.

2.2 Cfd simulation and validation

- CFD Simulations are carried out using ANSYS FLUENT software.
- A pressure-based solver had set and ideal gas approximation has considered for all the CFD simulations.
- In order to solve 2D Navier-stokes equation, correct boundary condition plays a very important role for appropriate results.
- k-ω SST turbulent model with no slip boundary condition at the wall has considered.
- 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 300 K. The operating condition is zero gage pressure.
- ANSYS FLUENT is used with Semi-Implicit Method for Pressure Linked Equation (SIMPLE) solution method. It is a steady state iterative technique.
- SIMPLE solver algorithm resolves the pressure-velocity coupling.

First thing is to validate the calculations by comparing them with the previous numerical results or with the experimental results. Therefore here the results are compared with numerical results of Ritlop and Nadarajah and the experimental data of Ramsay et al. [6]. The simulations are performed at Reynolds number of 1 million and Mach number of 0.044. Nine different angles of incidence are selected for validation, which are 0.0°, 2.1°, 4.1°, 6.1°, 8.2°, 10.1°, and 11.2°. The turbulence model employed is a two equation k-omega SST turbulence model. Tables 1 and 2 and Figures 2 and 3 show that present computed results for lift and drag coefficient agree well with the experimental values [6] as well as the computations of Ritlop and Nadarajah [6]. These comparisons validate the numerical methodology used in present calculations.

Boundary conditions considered for validation

- Mach No = 0.044
- Temperature = 288.16 K
- Atmospheric pressure = 101325 Pa
- Density of air = 1.225 kg/m^3
- Viscosity of air = 1.7894x10^{-5} kg/m-s

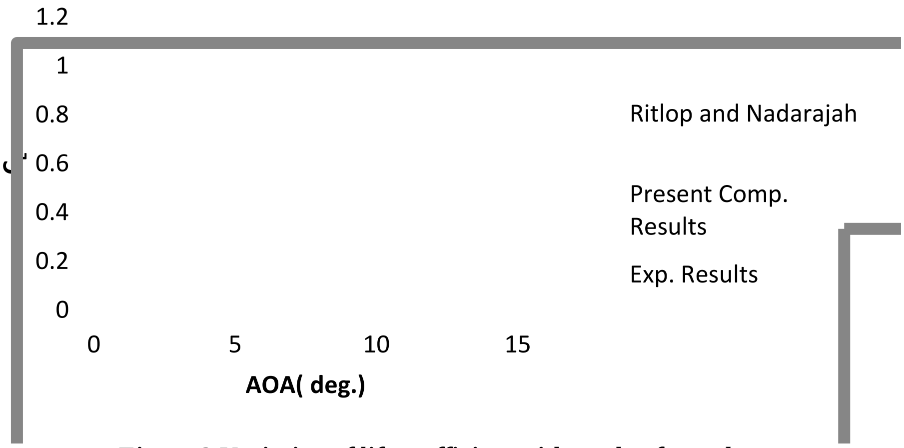

Figure 2.Variation of lift coefficient with angle of attack

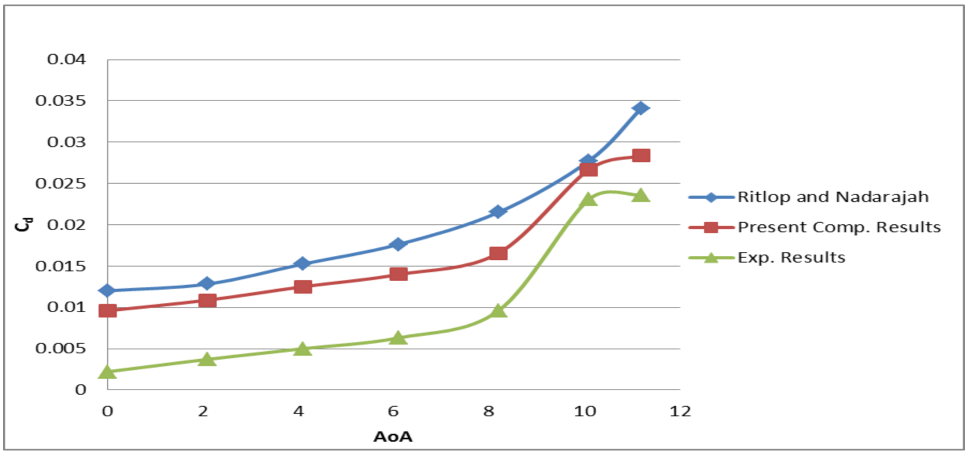

Figure 3.Variation of drag coefficient with angle of attack

Table 1.Results for lift coefficient with angle of attack

AOA (deg.)	Ritlop and Nadarajah C_L	Present Comp. Results C_L	Exp. Results C_L
0	0.12529	0.113835	0.07
2.1	0.35428	0.318293	0.3
4.1	0.55472	0.565594	0.55
6.1	0.75412	0.780504	0.79
8.2	0.94169	0.92154	0.9
10.1	1.0678	1.01004	0.94
11.2	1.1046	1.02892	0.93

Table 2.Results for drag coefficient with angle of attack

AOA (deg.)	Ritlop and Nadarajah C_D	Present Comp. Results C_D	Exp. Results C_D

0	0.012006	0.012595	0.0022
2.1	0.012824	0.0128758	0.0037
4.1	0.015248	0.0136029	0.005
6.1	0.017615	0.0149734	0.0063
8.2	0.021507	0.0185649	0.0096
10.1	0.027757	0.0267029	0.0231
11.2	0.03413	0.0283791	0.0236

III. RESULTS AND DISCUSSION

As mentioned before, we optimize the S809 airfoil at its 50% span of the blade under compressible flow, 50% location from the Centre of the rotor which correspond to the mid-section of the blade. We consider the free stream wind velocity of 130m/s and 0^o angle of attack. We set two objectives: maximize C_L and C_L/C_D. The airfoil shape that results in the best possible highest values of both objectives from the Pareto front approximation is considered as the shape of the optimized airfoil. Figures 4, 5, 6, 7 and 8 shows the evolution process of the airfoil shape and figures 10 and 11 shows the variation of C_L, C_D and C_L/C_D for all individuals. Figure 8 shows the dominant solutions in the Pareto front for the airfoil sections at 50% span location. Figures 9 show the comparison of airfoil shapes between the individual airfoils and the original airfoil at 50% span location.

A comparison between the present optimized airfoil shape and original airfoil is shown in Figure 9. Table 3 shows the comparison of C_L and C_L/C_D for the present optimized airfoil with other individuals. When compared to original airfoil, present results gave higher value for the C_L and C_L/C_D. These results gave the optimized airfoils that meet the objective of maximizing C_L and C_L/C_D. Boundary conditions considered for this compressible flow analysis are,

- Velocity of air = 130 m/s(M=0.3744)
- Temperature = 300 K
- Atmospheric pressure = 101325 Pa
- Density of air = 1.225 kg/m^3
- Viscosity of air = 1.7894x10^{-5} kg/m-s

Table 3. Comparison of C_L, C_D and C_L/C_D for the present optimized airfoil with other individuals.

Name	C_L	C_D	C_L/C_D
Original	1.46900e-01	9.10867e-03	16.127491
A1	1.48678e-01	9.20037e-03	16.160002
A2	1.58656e-01	9.14661e-03	17.345880
A3	1.59219e-01	9.19970e-03	17.306977
A4	2.22635e-01	9.28193e-03	23.985852
Optimal	2.77860e-01	9.24823e-03	30.044668

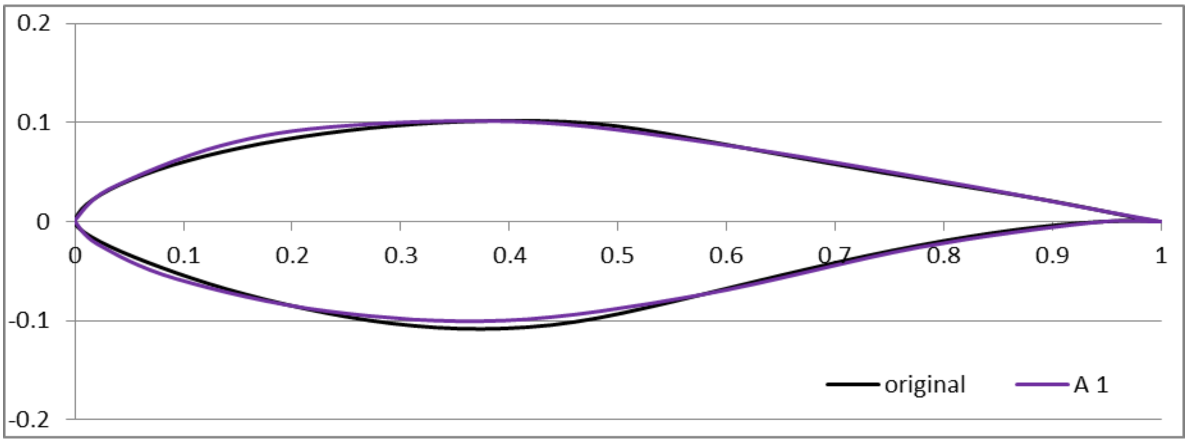

Figure 4.Comparison of shape of A1 individual with original airfoil.

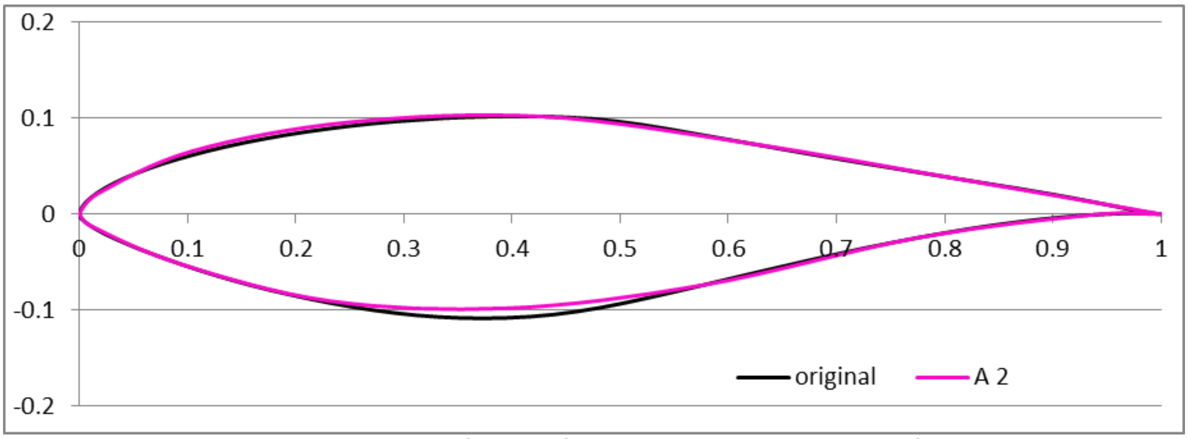

Figure 5.Comparison of shape of A2 individual with original airfoil.

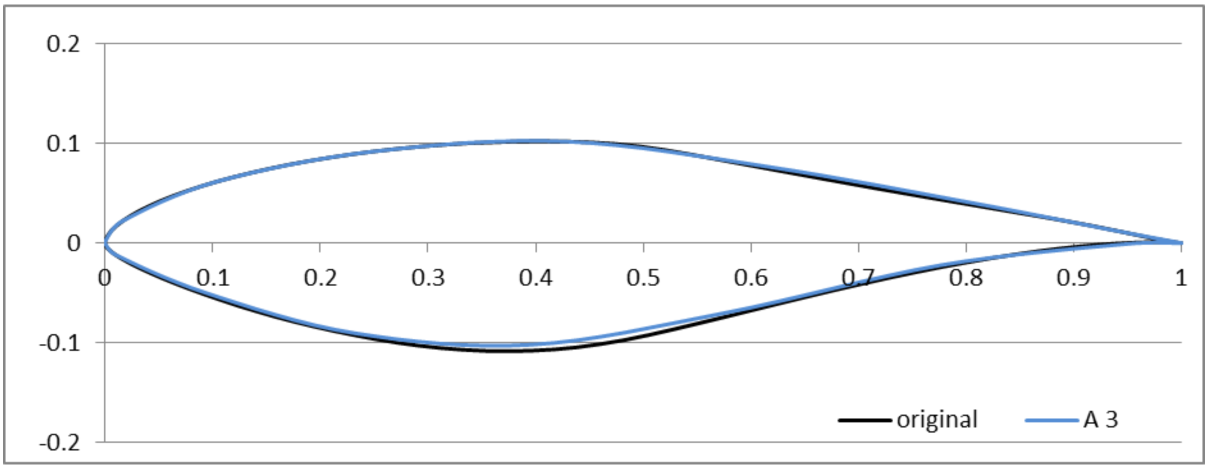

Figure 6.Comparison of shape of A3 individual with original airfoil.

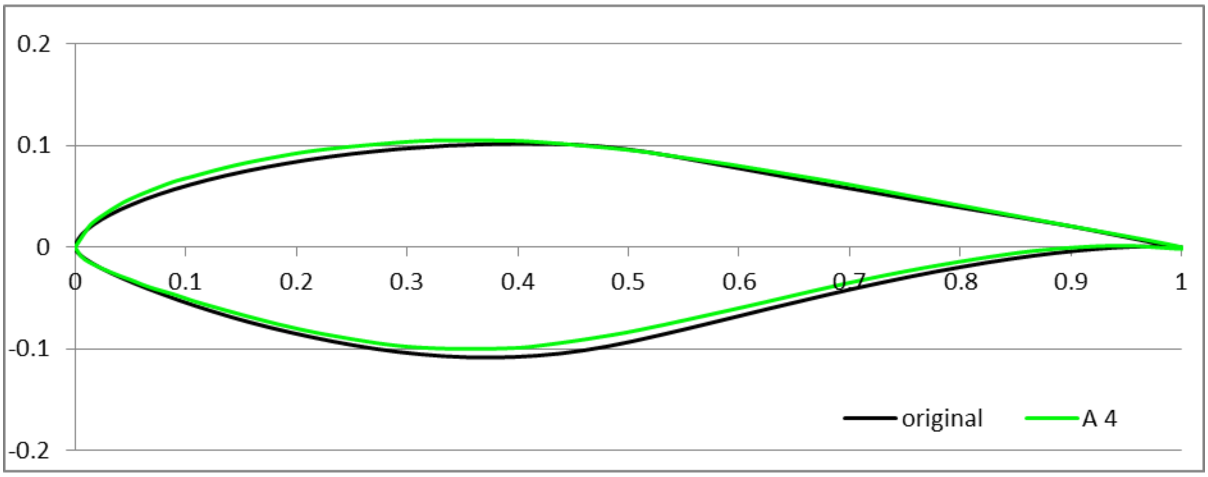

Figure 7.Comparison of shape of A4 individual with original airfoil.

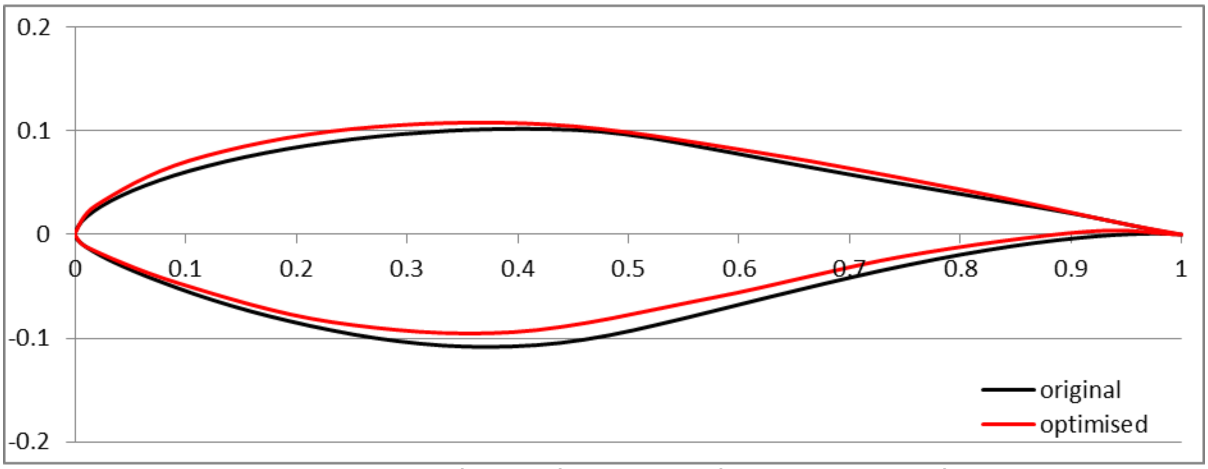

Figure 8.Comparison of shape of optimized airfoil with original airfoil.

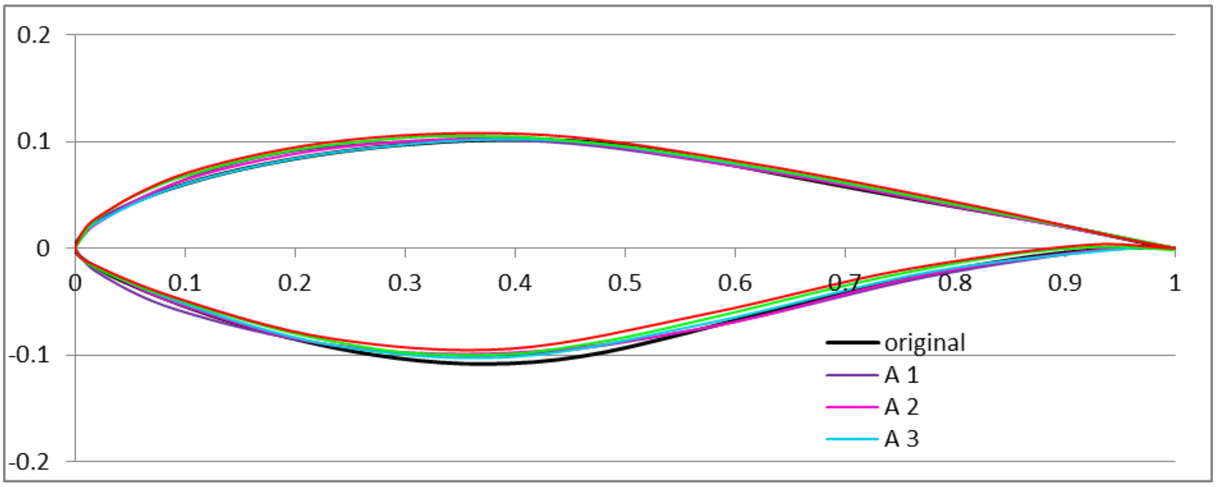

Figure 9. Comparison of shape of all individuals with original airfoil.

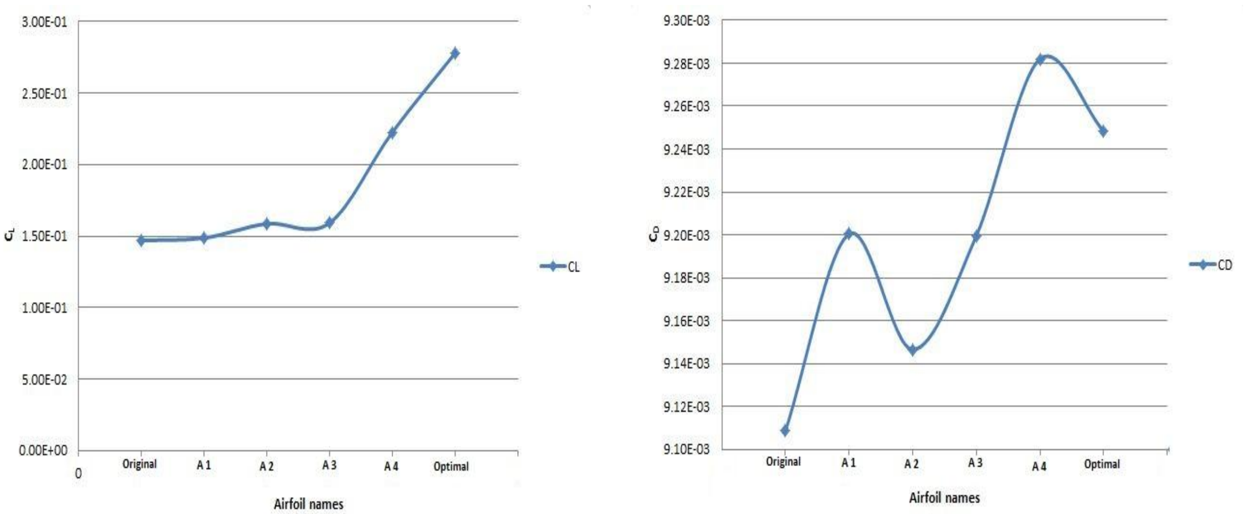

Figure 10. Comparison of Lift (C_L) and Drag (C_D) coefficients.

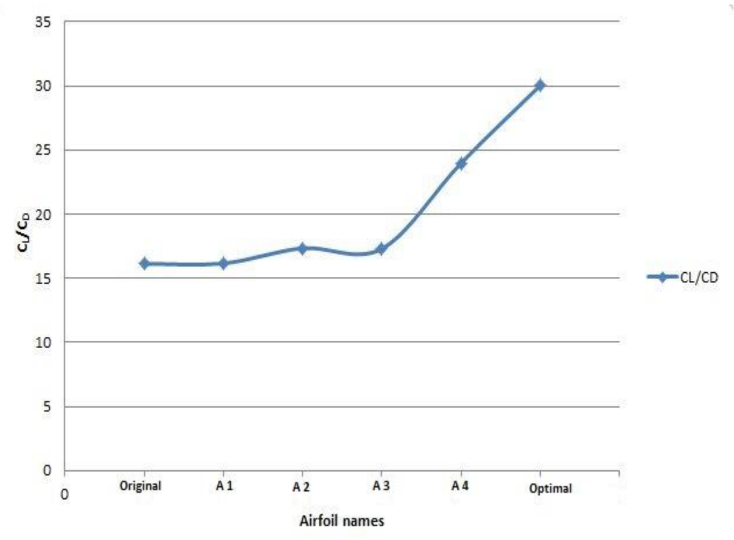

Figure 11. Comparison of C_L/C_D.

IV. CONCLUSIONS

In this work optimization of WT airfoil(S809) at its 50% span under compressible flow condition has been done to improve its lift and drag characteristics. For this two objectives have been considered, those are, to increase its C_L and C_L/C_D. Using FLUENT software flow field has calculated. A comparison between the present optimized airfoil shape and original airfoil has been done. When compared to original airfoil, present results gave higher value for the C_L and C_L/C_D. These results gave the optimized airfoils that meet the objective of maximizing C_L and C_L/C_D.

REFERENCE

1. Mohammad A. Hussain,1,2 Ziaul Huque,1,2 and Raghava R. Kamalapati1,2 "PROPAGATION OF SHOCK ON NREL PHASE VI WIND TURBINE AIRFOIL UNDER COMPRESSIBLE FLOW", Hindawi Publishing Corporation, Journal of Renewable Energy, Volume 2013, Article ID 653103, 9 pages.
   1) Department of Mechanical Engineering, Prairie View A&M University, Prairie View, TX 77446, USA.
   2) Center for Energy and Environmental Sustainability, Prairie View A&M University, Prairie View, TX 77446, USA.
2. Yilei He and Ramesh K. Agarwal "SHAPE OPTIMIZATION OF NREL S809 AIRFOIL FOR WIND TURBINE BLADES USING A MULTI OBJECTIVE GENETIC ALGORITHM", Hindawi Publishing Corporation, International Journal of Aerospace Engineering, Volume 2014, Article ID 864210, 13 pages. Department of Mechanical Engineering and Materials Science,Washington University in St. Louis, St. Louis, MO 63130, USA.
3. Emrah Kulunk "AERODYNAMICS OF WIND TURBINES", New Mexico Institute of Mining and Technology, USA.
4. Xiaomin Chen. "OPTIMIZATION OF WIND TURBINE AIRFOILS/BLADES AND WIND FARM LAYOUTS" Spring 1-20-2014 All Theses and Dissertations (ETDs), Paper 1228. Washington University in St. Louis.
5. Dan M. Somers. "DESIGN AND EXPERIMENTAL RESULTS FOR THE S809AIRFOIL", NRELlSR-440-6918 • UC Category: 1213 • DE97000206 , Airfoils, Incorporated, State College, Pennsylvania
6. Robert M. Ritlop. "TOWARD THE AERODYNAMIC SHAPE OPTIMIZATION OF WIND TURBINE PROFILES", Master of Engineering in Mechanical Engineering, McGill University Montreal, Quebec.
7. Yen-Pin Chen. "A STUDY OF THE AERODYNAMIC BEHAVIOR OF A NREL PHASE VI WIND TURBINE USING THE CFD METHODOLOGY", July 27, 2011. B.S., Wright State University, 2009.
8. Y.Song and J. B.Perot. "CFD SIMULATION OF THE NREL PHASE VI ROTOR", Theoretical & Computational Fluid Dynamics Laboratory, Department of Mechanical & Industrial Engineering, University of Massachusetts Amherst, Amherst, MA01003, USA.

