CFD Simulation on Aerodynamic Analysis of Wind Turbine Rotor Blade Airfoils

[1]Swarnkar Hemantkumar, [2]Prof. Patil Raghunath

[1]Reaserch Student, Mechanical Engineering, SGDCOE, Jalgaon, Maharashtra, India

[2]Head of Department, Associate professor, Mechanical Engineering, SGDCOE, Jalgaon, Maharashtra, India

ABSTRACT

One of the major challenges in the new century is the efficient use of energy resource as well as production of energy from the renewable energy resources. Although, scientist from around the world have shown that global warming has been caused in part by the greenhouse effect which is largely due to the use of fossil fuel for production of electricity and transportation facility. There are several alternative forms of energy that have already been developed such as wind, solar, geothermal, tidal, and hydroelectric power. The advancement in renewable energy technologies has been possible and the vast amount of research performed by engineer in order to make them more affordable, more efficient, most importantly and inexpensive. Wind energy is one of the most important sources of renewable energy. Wind turbine extract energy from the kinetic energy of the wind. At present many researches are concentrated on aerodynamic design of wind turbine rotor blade through wind tunnel test and blade element momentum. These conventional methods are difficult and time consuming. However, wind turbine blade simulation through computational fluid dynamics (CFD) offers less time consuming, easy and inexpensive way to aerodynamic blade design. In this study two dimensional airfoils (NACA 4424 and NREL S809) CFD models are present using ANSYS-FLUENT software. Using the Spalart-Allmaras viscosity the dimensionless lift and drag coefficient and forces are calculated using different angle of attack and different mach numbers. Viscosity is based on Sutherland model. One of airfoil is selected from the different two airfoils, which is responsible for maximum power and efficiency. Wind data is taken from the review paper based on wind speed of Bhopal bairagarh site.

KEYWORDS

The research aims to evaluate the aerodynamic performance of variable speed fixed –pitch horizontal axis wind turbine blades airfoils through two-dimensional computational fluid dynamics (CFD) analysis.

The objective of the research is to establish two dimensional CFD model of wind turbine blade airfoil-

1) To analyses the aerodynamic performance of different airfoils at two different wind velocity.
2) Compare different airfoil according to lift and drag coefficient.
3) Select one airfoil which is most efficient accordingly local site.

1 INTRODUCTION

1.1 Background

Energy is essential for civilisation development. India is developing country and with economic and socialisation progress, there is an expanding demand of renewable energy to secure the conventional energy source for long period. As a clean renewable energy source, wind energy plays very important role in modern life. Wind energy is an abundant resource as comparison to other renewable energy resource. Moreover unlike solar power usage does not depend on weather and climate condition. Wind turbine was invented to extract energy from the wind. Power in the wind turbine comes from transformation of air that is driven from the heat of the sun, which is abundant, clean and renewable. As one of the most popular energy source, wind energy exploitation is growing rapidly.

The development of wind power in India began in the 1986 with first wind farms being set up in the coastal areas of Maharashtra (Ratnagiri), Gujrat (Okha) andTamilnadu (Tuticorin) with 55 kW Vestas wind turbines. These demonstration projects were supported by MNRE. The capacity has significantly increased in the last few years. Although a relative newcomer to the wind industry compare to Denmark or the United State, India has the fourth largest installed wind power capacity in the world. In 2009-10 India's growth rate was highest among the other top four countries.

As of 31th march 2016 the installed capacity of wind power in India was 26,769 MW, mainly spreads across south, west and north region. East and north-east regions have no grid connected wind power plant as of march, 2015 end. In the year 2015 the MNRE set the target for wind power generation capacity by the year 2022 at 60,000 MW. The worldwide installed capacity of wind power reached 435 GW by the end of 2015. China (148,000 MW), US(74,347 MW) and Germany (45,192 MW) are ahead of India in fourth position. The short gestation periods for installing wind turbine, and the increasing reliability and performance of wind energy machines has made wind power a favourable choice for capacity addition in India. Wind power accounts nearly 8.6% of India's total installed power generation capacity and generated 28,604 million Kwh (MU) in the fiscal year 2015-16 which is nearly 2.5% of total electricity generation. The capacity utilisation factor is nearly 14% in the fiscal year 2015-16 (15% in 2014-15). 70% of wind generation is during the five months duration from May to September coinciding with southwest monsoon duration.

Installed Wind Power Capacity

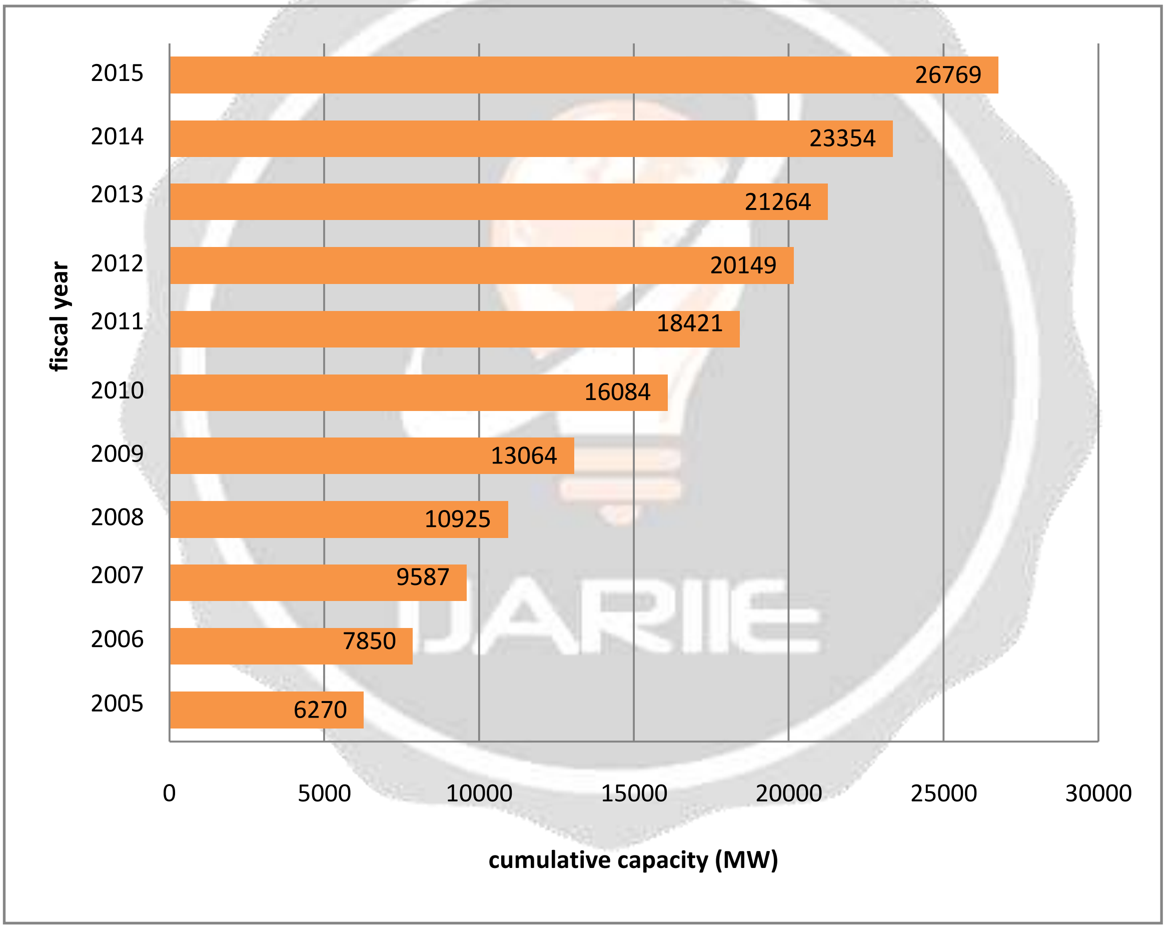


Figure 1.1 Installed wind power capacity in India

A wind turbine consist of several parts i.e. blade, rotor, generator, control system, driven chain etc. The rotor is driven by the wind and rotates at predefined speed by the virtue of the wind speed, so that the generator can produce electricity under the regulation of the control system. In order to get the maximum kinetic energy from the wind, engineers put much efforts on the design of effective blade geometry. In the early stage, the aerofoil of helicopters were used for wind turbine blade airfoil design but now many specialize aerofoil have been invented and used for wind turbine blade design. A rotor blade has different airfoil at different section to improve the efficiency, so the present time blades are more complicated and efficient than the early age wind turbine blades.

In the early stage, the research on wind turbine rotor blade was limited on theoretical only, field testing and wind tunnel testing which need a lot of effort and resource. Because of development of computer aided design codes, they provide another way to design and analyse the wind turbine blades. Aerodynamic design of wind turbine blade can be analyse using computational fluid dynamics (CFD), which is one of the branch of fluid mechanics that uses numerical method and algorithms to solve and analyze problem of fluid flow. Comparing to conventional theoretical and experimental method, numerical method saves money and time for the performance analysis and optimal design of wind turbine rotor blade.

A wind turbine converts kinetic energy into mechanical power through a rotor, and then converts the mechanical power into electrical power through a generator which is linked to the rotor with gearbox. Various type of wind turbine are designed to get advantage of wind energy based on the principle of aerodynamics. Depending upon the wind turbine rotor orientation, there are two type of wind turbines, horizontal axis wind turbine (HAWT) and vertical axis wind turbine (VAWT). Generally according to wind turbine capacity, modern wind turbine can be classified as small wind turbine (below 50kW), medium wind turbine(50kW-250kW) and large wind turbine(above 250kW). When considering installation sites, there are onshore and offshore wind turbines. Based on operation scheme, wind turbine can be divided into fixed pitch wind turbine and variable pitch wind turbine. According to the relative flow direction of the wind turbine rotor, HAWT are either upwind or downwind turbines. Most modern horizontal axis wind turbine have three blade, however there are also turbine with two blade. For small wind turbine, there are turbine with 5 or 7 blade also. Three bladed horizontal axis wind turbine is the most common topology due to higher efficiency, better controlled performance and aesthetic appreciation.

2 Literature Review

In this some basic theories of wind turbine aerodynamics and computational fluid dynamics are introduced. Moreover the purpose and methods for wind turbine simulation are discussed.

2.1 Historical development of wind turbine

Wind turbine is a device, which converts the kinetic energy from the wind to electrical energy by a mechanical rotor, a drive train and a generator. One of the earliest wind turbine was designed by Poul La Cour, who was a professor at an adult education centre in Denmark in 1981. Nowadays Enercon E-126, the world biggest wind turbine can generate up to 7 MW of power under the rated wind speed. This capacity can provide the daily electricity for more than 4500 homes. Following the technology development of modern wind turbine, they can now be mounted either on the ground or on the seabed. A giant offshore wind turbine of 10 MW will be installed in 2011 by Enova SF in Norway. As the depletion of coal and fossil fuel, wind energy plays an important role in this century.

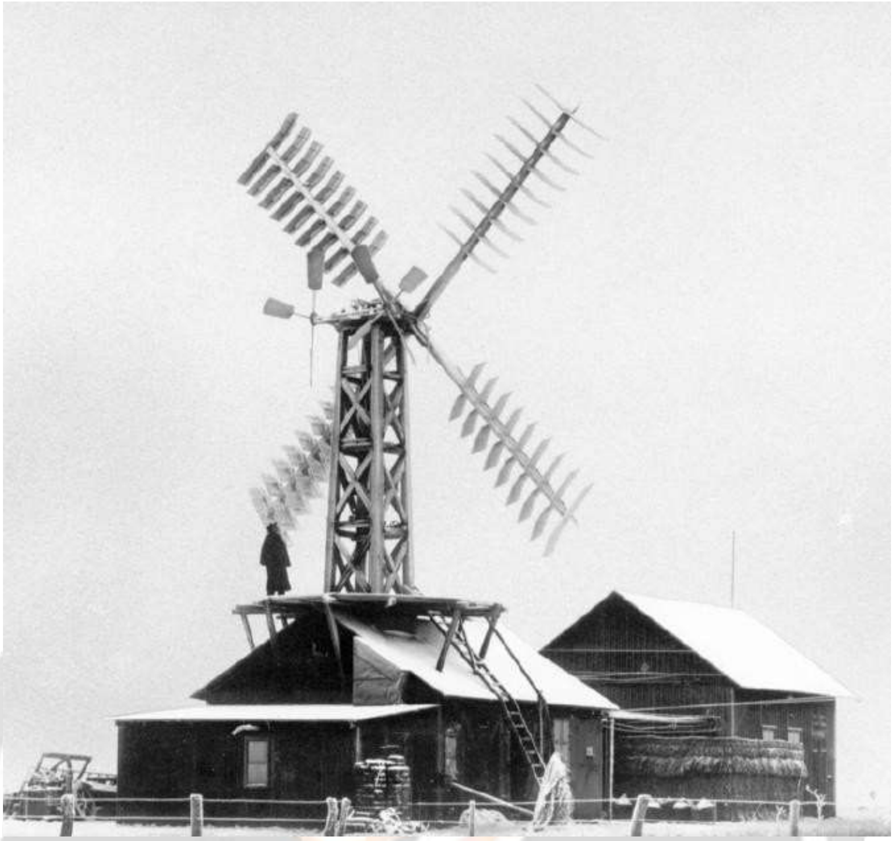


Figure 2.1 Pou La Cour's first electricity producing wind turbine in 1891 in Askov, Denmark(Golding 1977)

2.2 Wind turbine aerodynamics

According to the different rotational orientation, wind turbine can be categories as vertical axis wind turbine and horizontal axis wind turbine. The advantage of vertical axis wind turbine are-

1) Simple structure: vertical axis wind turbine can work without yaw system and most of them have a blade with constant chord and no twist, which is easy to construct.
2) Easy to install: because the drive trains can be located relative to the ground.

Comparing to horizontal axis wind turbine, stall control can only be used in vertical axis wind turbine as it is difficult to incorporate aerodynamics control such as variable pitch and aerodynamic brake, so the overall power efficiency is lower than horizontal axis wind turbine.

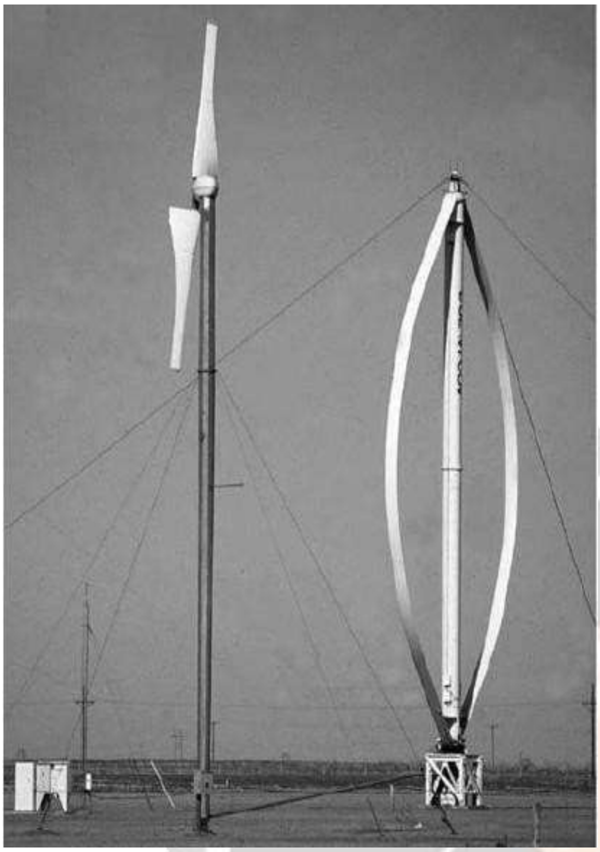


figure 2.2 Vertical axis wind turbine

figure 2.2 Horizontall axis wind turbine

2.3 Computational fluid dynamics (CFD)

There are many commercial CFD software used in engineering, such as PHOENICS (it is the first commercial CFD software), STAR-CD, ANSYS FLUENT/CFX and so on. All CFD software have three main structures which are Pre-Processer, Solver and Post-Processor.

2.4 Issue in wind turbine simulation using CFD software

A confident result of airfoil simulation was achieved in two dimensional simulation, but it was difficult to get a reliable result for three dimensional simulation. Initially the air flow passing through a rotating horizontal axis wind turbine blade is much more complicated than that of a two dimensional simulation because the changing angle of attack vary along the airfoil span. Moreover under high wind speed stall of the system can take place from the root section. There also centrifugal force along the blade due to rotor rotation. On the other hand accuracy of simulation is affected because of the limitation of CFD software. Firstly no matter what kind of turbulent model is used, it is extremely hard to simulate the turbulence in physical reality. Additionally fine mesh is a prerequisite in order to simulate full scale wind turbine, which are very memory restricted inside the computer meaning the simulation cannot be carried out using personal computer with low configuration. In order to reduce the mesh size normally, neither the simulation nor the ground are included into the model. Finally geometry of wind turbine blade is difficult to mesh with quality. Most wind turbine blade tip are designed using a thin airfoil for low induced drag and the root region is using a thick version for structural support, the size different between tip and root leads to mesh scales difficult to control.

3 Methods

GRAMBIT is the pre-processor of ANSYS-Fluent. A simulation model can be created directly in GRAMBIT or import from the other CAD software such as SolidWorks and Pro/Engineer®. In this section airfoils NACA 4424 and NREL S809 are modelled.

In this aerodynamic analysis of airfoils will be conducted in two-dimensional simulation using ANSYS Fluent. NACA 4424 and NREL S809 airfoils will be compare in order to find out which one has a better aerodynamic performance.

NACA 4424modelling

To draw any airfoil we need x and y co-ordinate of the airfoil. Below given table shows the co-ordinate of the airfoil-

Number	X/C	Y/C
1	1.0000	0
2	0.95196	0.02240
3	0.90320	0.04099
4	0.80464	0.07447
5	0.70487	0.10312
6	0.60405	0.12674
7	0.50235	0.14474
8	0.40000	0.15606
9	0.29401	0.15738
10	0.24111	0.15287
11	0.18858	0.14416
12	0.13674	0.13045
13	0.08611	0.11012
14	0.06153	0.09651
15	0.03775	0.07942
16	0.01536	0.05624
17	0.00530	0.03964
18	0.00000	0.00000
19	0.01970	-0.03472
20	0.03464	-0.04656
21	0.06225	-0.06066
22	0.08847	-0.06931
23	0.11389	-0.07512
24	0.16326	-0.08169
25	0.21142	-0.08416
26	0.25889	-0.08411
27	0.30599	-0.08238
28	0.40000	-0.07606
29	0.49765	-0.06698
30	0.59595	-0.05562
31	0.69513	-0.04312
32	0.79536	-0.03003
33	0.89680	-0.01655
34	0.94804	-0.00964
35	1.00000	0.00000

Table - NACA 4424 airfoil coordinates

NACA 4424 profile

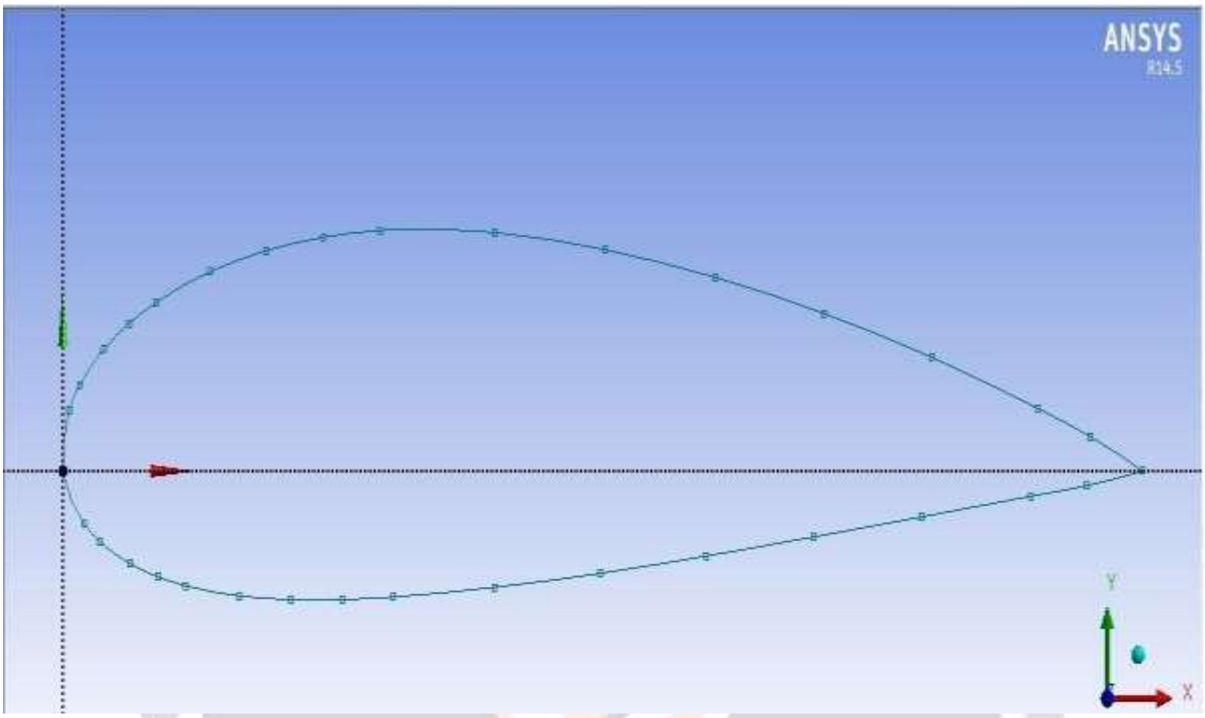


Figure - NACA 4424 airfoil

NACA 4424 airfoil is made by ANSYS-Fluent software of Workbench 14.5 . In ANSYS-Fluent there are four steps to perform CFD analysis on airfoils i.e. Geometry, Mesh, Setup and Solution. With the help of coordinates NACA 4424 airfoil is made in the geometry section.

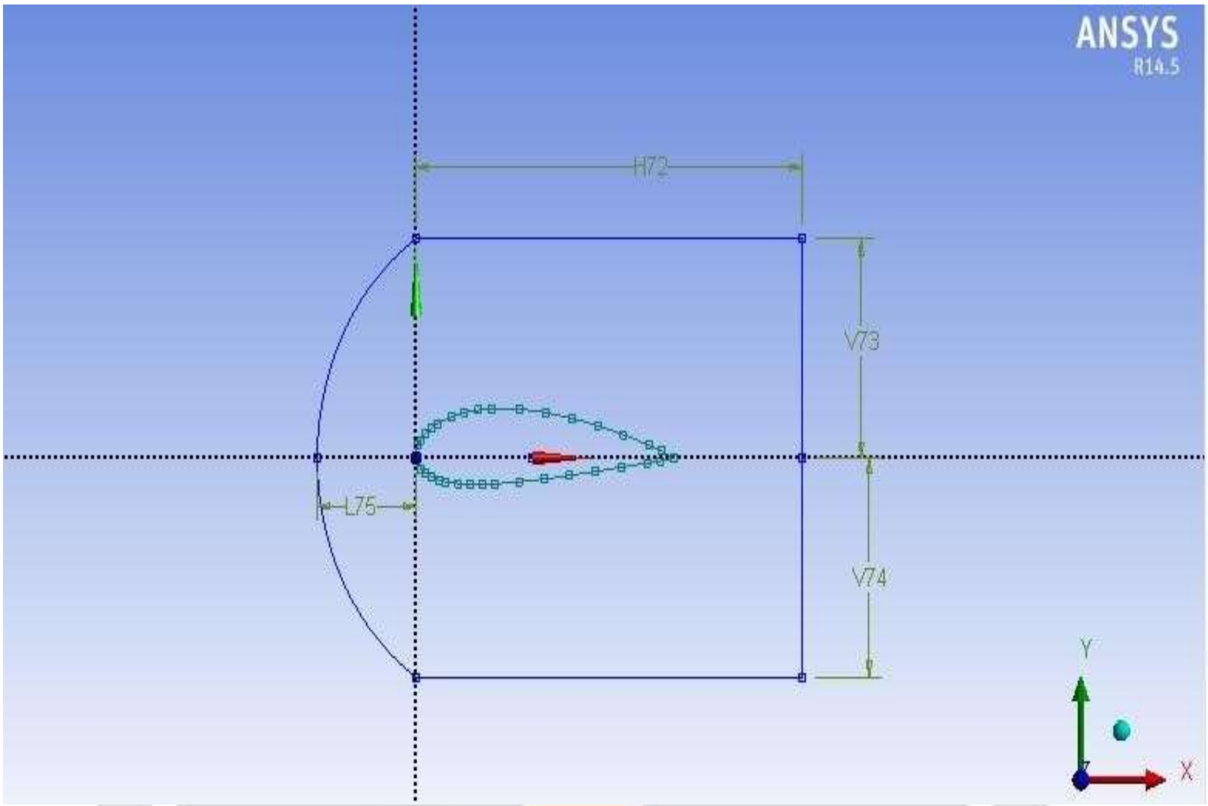


Figure - NACA 4424 airfoil with far-field

H72=

1.5C

V73=

V74=

0.7C

L75=.

3837

C

Where C= chord length

Far-field is made for analysis around the airfoil. There are many methods to draw far-field. Far-field may be structured grid or unstructured grid far-field. In this analysis we use structured grid far-field.

NACA 4424 airfoil with meshing:

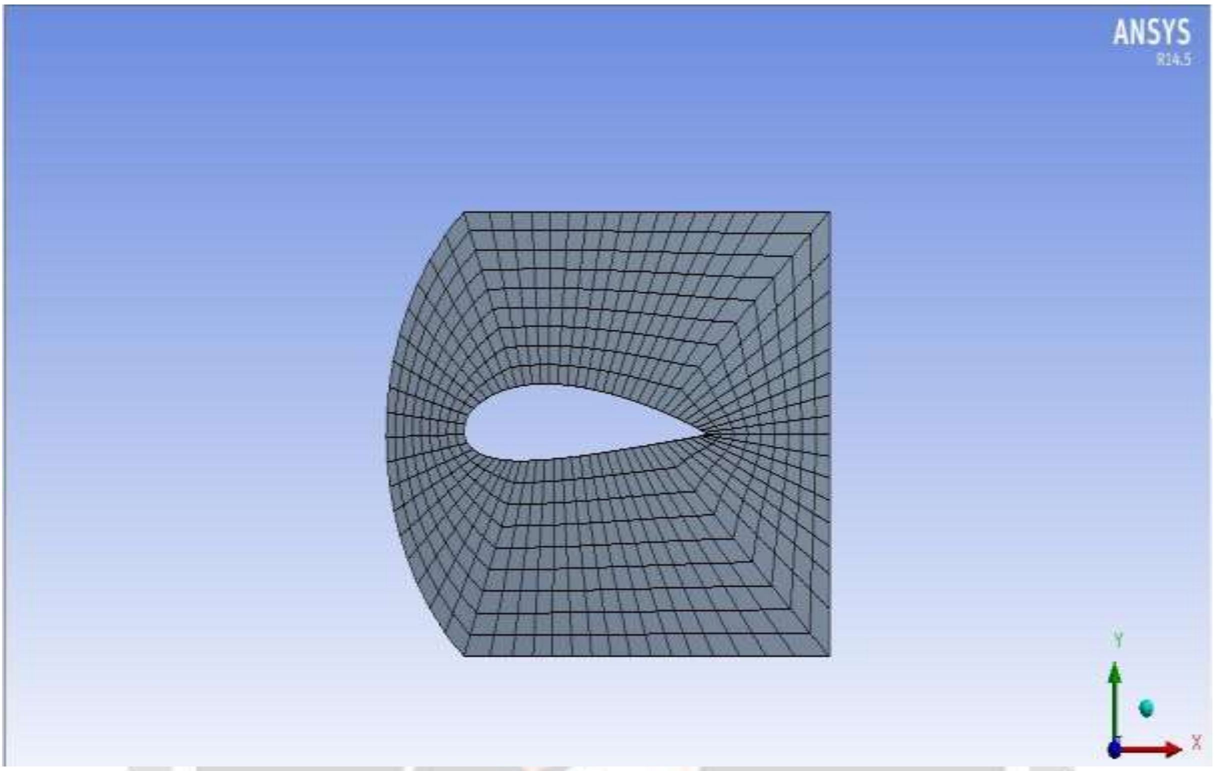


Figure -NACA 4424 airfoil with meshing

The mesh consists of 670 quadrilateral cells. A large number of grids around the airfoil surface is used to capture the pressure gradient accurately at the boundary layer. This is because the adverse pressure gradient induces flow separation. Stall will occur when separation region extends. In the far field area the mesh resolution can become progressively coarser since the flow gradient approach zero.

In the figure close to the airfoil surface, the most grids should be located near the leading and trailing edges since these are critical areas with the steepest gradient. It is better to transit the mesh size smoothly because the large and discontinuous transition may decrease the numerical accuracy.

- Pressure far-field boundary condition was used in meshing method as the computation domain is large enough. Airfoil is treated as stationary wall condition with no slip shear condition. The computational condition is-

Airfoil	NACA 4424
Simulation Type	Steady Simulation
Fluid Material	Air
Temperature	300 K
Kinematic Viscosity	1.716 ×10-5 m2/s (Sutherland)
Mach Number	0.1147 and0.1382
Density	1.2 kg/m3
Pressure	101325 pa
Wind Speed	39 m/s and 47 m/s
CFD algorithm	SIMPLE (default option)
Interpolating scheme	Pressure (Standard) Density (second order upwind) Momentum (second order upwind) Modified Turbulent Viscosity (second order upwind)
Turbulent model	Spalart-Allmaras
Boundary condition	Pressure far-field Stationary wall with no slip shear condition

Table - Computational condition of NACA 4424 airfoil simulation

Spalart-Allmaras model will be used for turbulent modelling because it is designed specifically for aerospace application, which involves wall-bounded flow and has been shown good results for boundary layers subjected to adverse pressure gradients. Before running the simulation, lift, drag and pitching moment coefficients need to be monitored in ANSYS- Fluent and will be used to estimate the convergence of calculation. Lift coefficient is defined to be perpendicular to the direction of oncoming airflow; drag coefficient is defined to be parallel to the direction of oncoming airflow and the pitching moment centre is set at 1/4 chord length from the leading edge.

4 Modeling and Analysis

NREL S809 modelling

To draw any airfoil we need x and y co-ordinate of the airfoil. Below given table shows the co-ordinate of the airfoil-

Number	X/C	Y/C
1	1.000000	0.000000
2	0.996203	0.000487
3	0.985190	0.002373
4	0.967844	0.005960
5	0.945073	0.011024
6	0.917488	0.017033
7	0.885293	0.023458
8	0.848455	0.030280
9	0.807470	0.037766
10	0.763042	0.045974
11	0.715952	0.054872
12	0.667064	0.064353
13	0.617331	0.074214
14	0.567830	0.084095
15	0.519832	0.093268
16	0.474243	0.099392
17	0.428461	0.101760
18	0.382612	0.101840
19	0.337260	0.100070
20	0.292970	0.096703
21	0.250247	0.091908
22	0.209576	0.085851
23	0.171409	0.078687
24	0.136174	0.070580
25	0.104263	0.061697
26	0.076035	0.052224
27	0.051823	0.042352
28	0.031910	0.032299
29	0.016590	0.022290
30	0.006026	0.012615
31	0.000658	0.003723
32	0.000204	0.001942
33	0.000000	0.000000
34	0.000213	-0.001794
35	0.001045	-0.003477
36	0.001208	-0.003724
37	0.002398	-0.005266
38	0.009313	-0.011499
39	0.023230	-0.020399
40	0.042320	-0.030269

41	0.065877	-0.040821
42	0.093426	-0.051923
43	0.124111	-0.063082
44	0.157653	-0.073730
45	0.193738	-0.083567
46	0.231914	-0.092442
47	0.271438	-0.099905
48	0.311968	-0.105281
49	0.353370	-0.108181
50	0.395329	-0.108011
51	0.438273	-0.104552
52	0.481920	-0.097347
53	0.527928	-0.086571
54	0.576211	-0.073979
55	0.626092	-0.060644
56	0.676744	-0.047441
57	0.727211	-0.035100
58	0.776432	-0.024204
59	0.823285	-0.015163
60	0.866630	-0.008204
61	0.905365	-0.003363
62	0.938474	-0.000487
63	0.965086	-0.000743
64	0.984478	-0.000775
65	0.996141	-0.000290
66	1.000000	0.000000

Table -NREL S809 airfoil coordinates

NREL S809 profile:

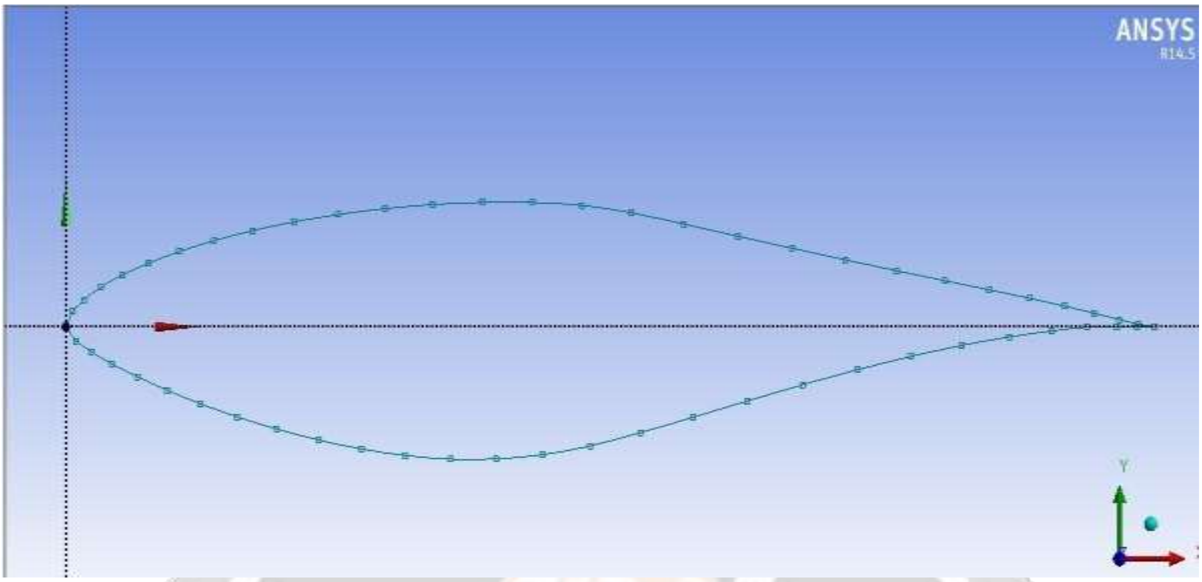


Figure -NREL S809 airfoil

NREL S809 airfoil with meshing:

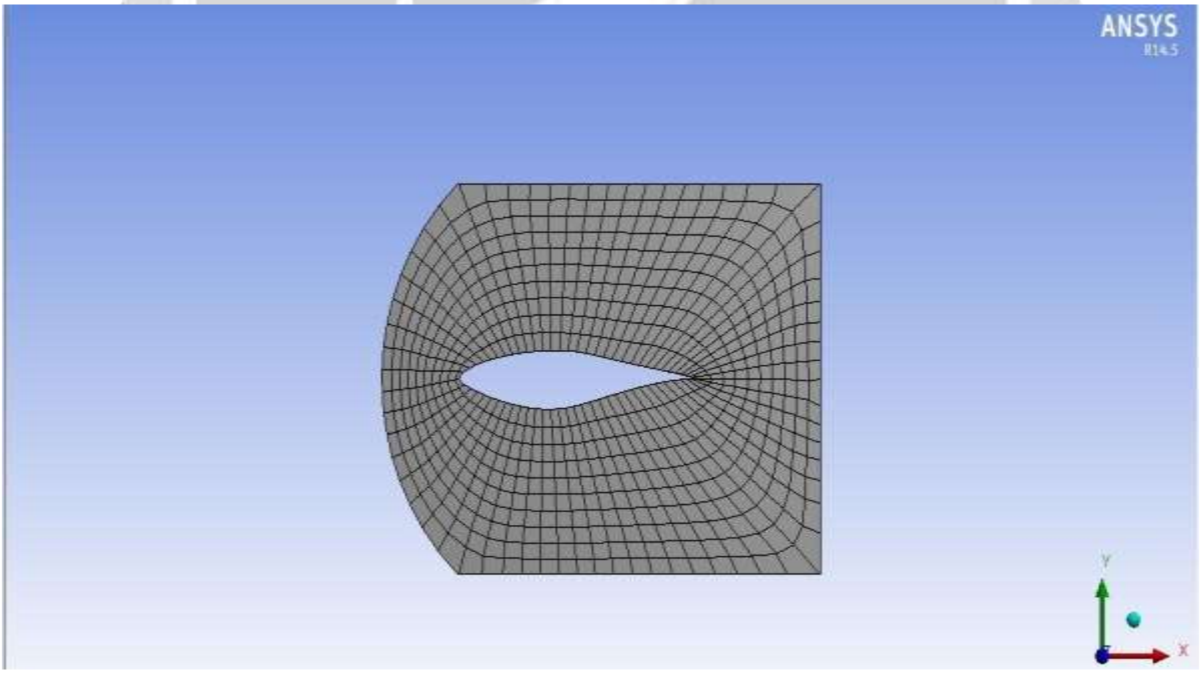


Figure -NREL S809 airfoil with meshing

- Pressure far field boundary condition was used in meshing method as the computational domain is large enough. Airfoil is treated as stationary wall condition with no slip shear condition. The computational conditionis-

Aerofoil	NREL S809
Simulation Type	Steady Simulation
Fluid Material	Air
Temperature	300 K
Kinematic Viscosity	1.716 ×10-5 m2/s (Sutherland)
Mach Number	0.1147 and 0.1382
Density	1.2 kg/m3
Pressure	101325 pa
Wind speed	39 m/s and 47 m/s
CFD algorithm	SIMPLE (default option)
Turbulent model	Spalart-Allmaras
Interpolating scheme	Pressure (Standard) Density (second order upwind) Momentum (second order upwind) Modified Turbulent Viscosity (second order upwind)
Boundarycondition	Pressure far-field Stationary wall with no slip shear condition

Table -Computational condition of NREL S809 airfoil simulation

5 Result and Discussions

In this aerodynamic analysis of airfoils will be conducted in two dimensional simulation using ANSYS Fluent. NACA 4424 and NREL S809 airfoils will be compare in order to find out which one has a better aerodynamic performance.

Aerodynamic performance evaluation of NACA 4424 and NREL S809airfoils

Lift and drag coefficient are the crucial values for aerodynamic performance evaluation. The critical and optimum angle of attack can be calculated by plotting the lift and drag coefficient polar curves.

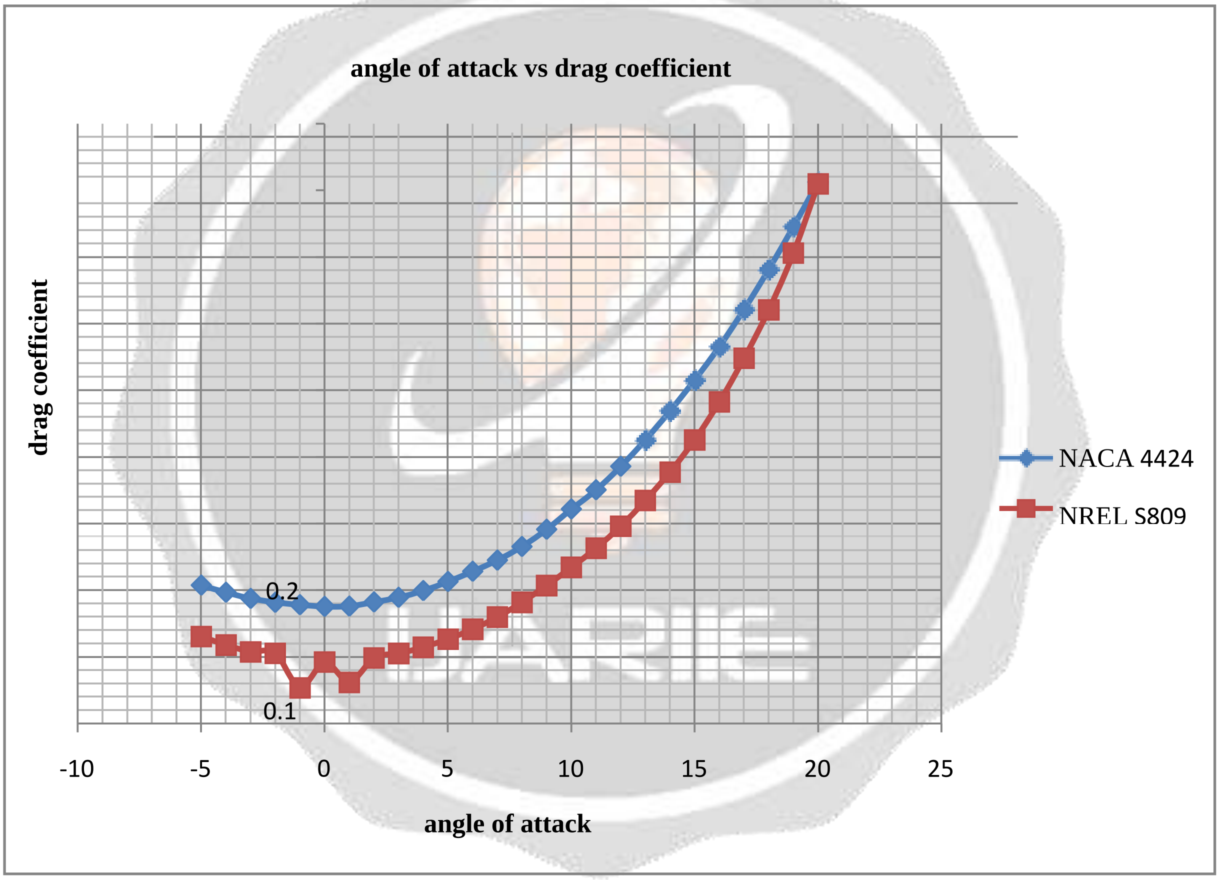
Aerodynamic performance evaluation of NACA 4424airfoil

At different wind velocity i.e. 39 m/s and 47 m/s aerodynamic analysis of NACA 4424 are done at different angle of attack i.e. -5° to 20°. Drag force, lift force, drag coefficient and lift coefficient are calculated for different conditions and results.

Aerodynamic performance comparison between NACA 4424 and NREL S809airfoils

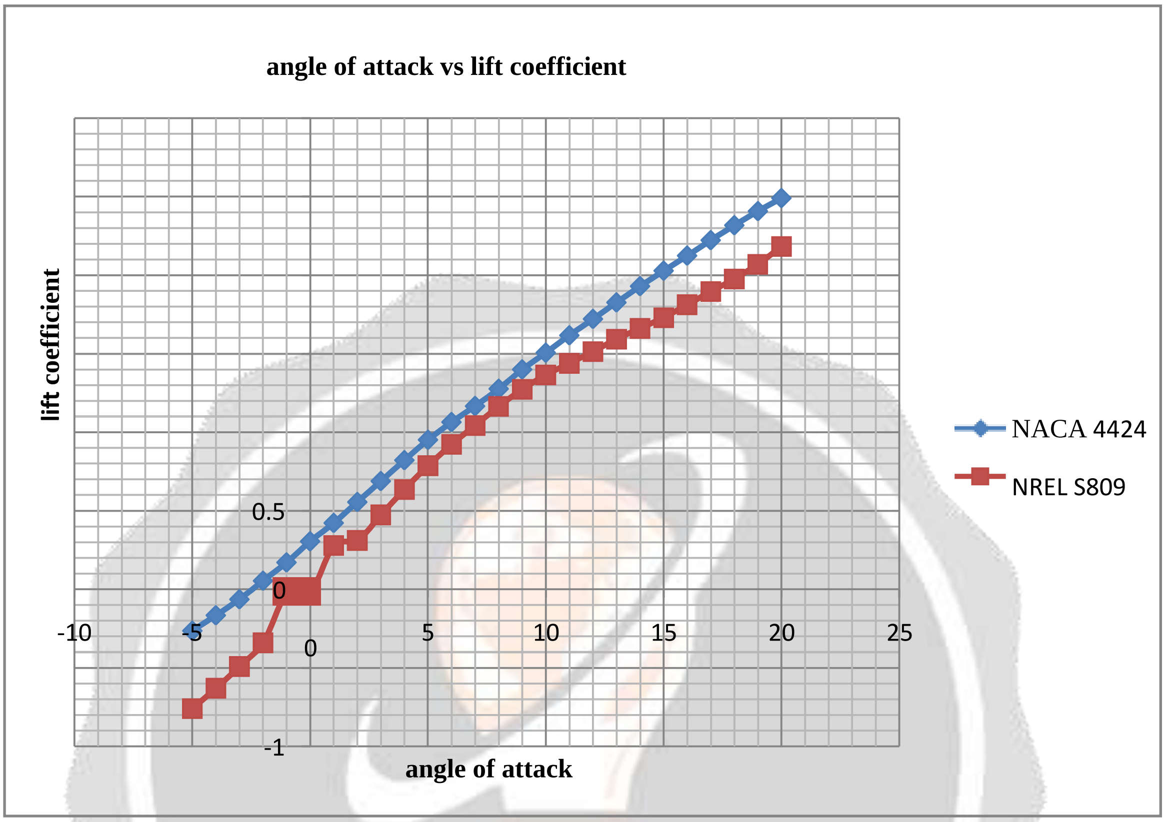
Based on previous results airfoils were simulated between range of angle of attack (-5° to 20°) to observe the optimum angle of attack.

At 39 m/s comparison of NACA 4424 and NREL S809 airfoil for drag coefficient



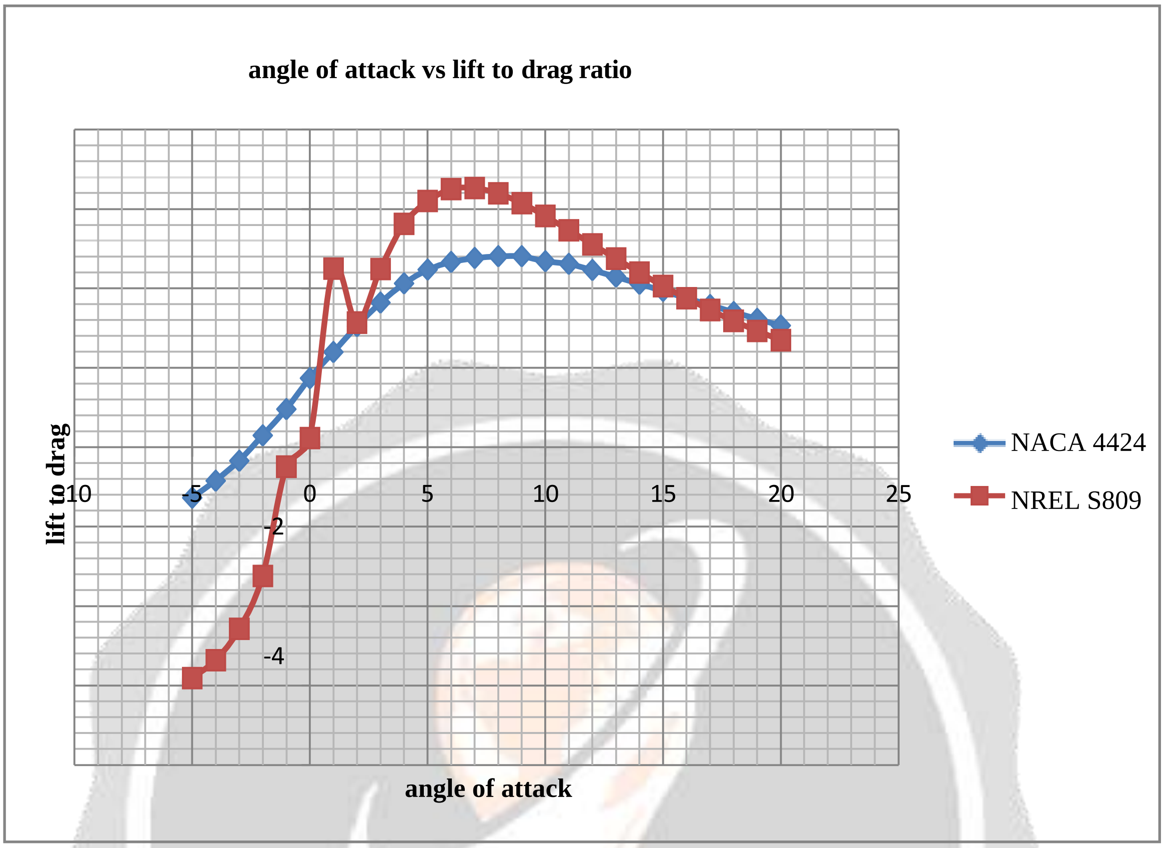
From the graph it is clear that increase in angle of attack first drag coefficient decrease from negative angle of attack to zero angle of attack for both the airfoils. Further increase in angle of attack drag coefficient increase for both NACA 4424 and NREL S809 airfoils. For the same conditions drag coefficient for NACA 4424 is more than for the NREL S809 airfoil but at 20° angle of attack drag coefficient same for both theairfoils.

At 39 m/s comparison of NACA 4424 and NREL S809 airfoil for lift coefficient



From the graph it is clear that increase in angle of attack increases lift coefficient for both NACA 4424 and NREL S809 airfoils. For the same conditions lift coefficient for NACA 4424 is more than for the NREL S809 airfoil. At the condition of angle of attack between 7° to 9° lift coefficient is approximately same for both the airfoils. For NREL S809 airfoil lift coefficient is constant between -1° to 0° and 1° to 2° angle of attack.

At 39 m/s comparison of NACA 4424 and NREL S809 airfoil for CL/CD

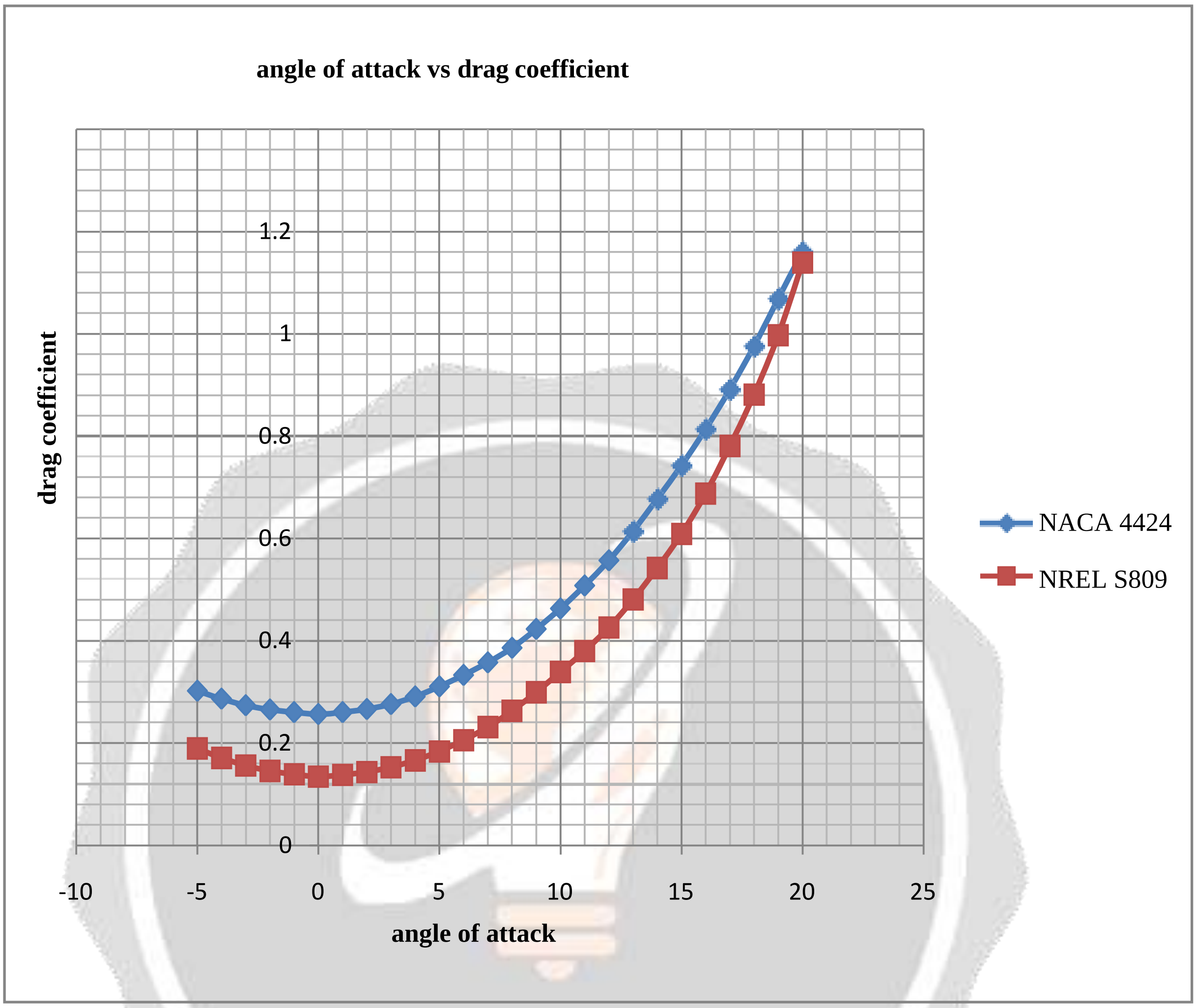


Lift to drag ratio for NREL S809 is less than the NACA 4424 airfoil for the negative angle of attack to zero angle of attack but after the 1° lift to drag ratio of NREL S809 is more than the NACA 4424 airfoil. Initially when angle of attack increase lift to drag ratio increases up to certain angle of attack and then lift to drag ratio start decreasing continuously. From the graph it is clear that for NREL S809 airfoil maximum lift to drag ratio is achieved at 7° angle of attack and for NACA 4424 airfoil maximum lift to drag ratio is achieved at 8° angle ofattack.

For an airfoil lift to drag ratio should be maximized as a result it can improve the efficiency when wind turbine generates electricity. From the above results we can say that for 39 m/s wind velocity NREL S809 airfoil is better than the NACA 4424 airfoil, because lift to drag ratio is more than the NACA 4424airfoil.

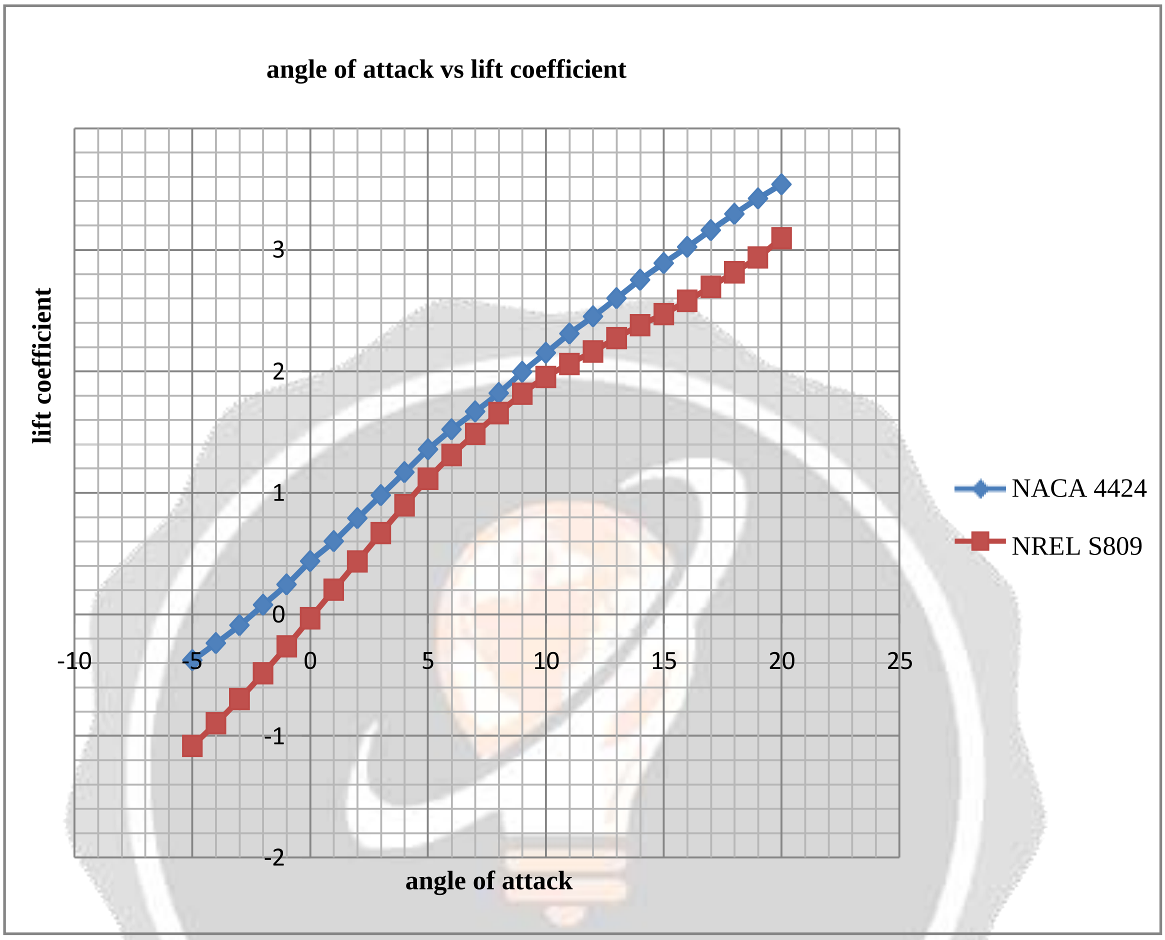
Optimum angle of attack can be defined as the angle of attack where maximum lift to drag ratio should be found. For NACA 4424 airfoil optimum angle of attack is 8° and for NREL S809 airfoil optimum angle of attack is 7°at 39 m/s windspeed.

At 47 m/s comparison of NACA 4424 and NREL S809 airfoil for drag coefficient



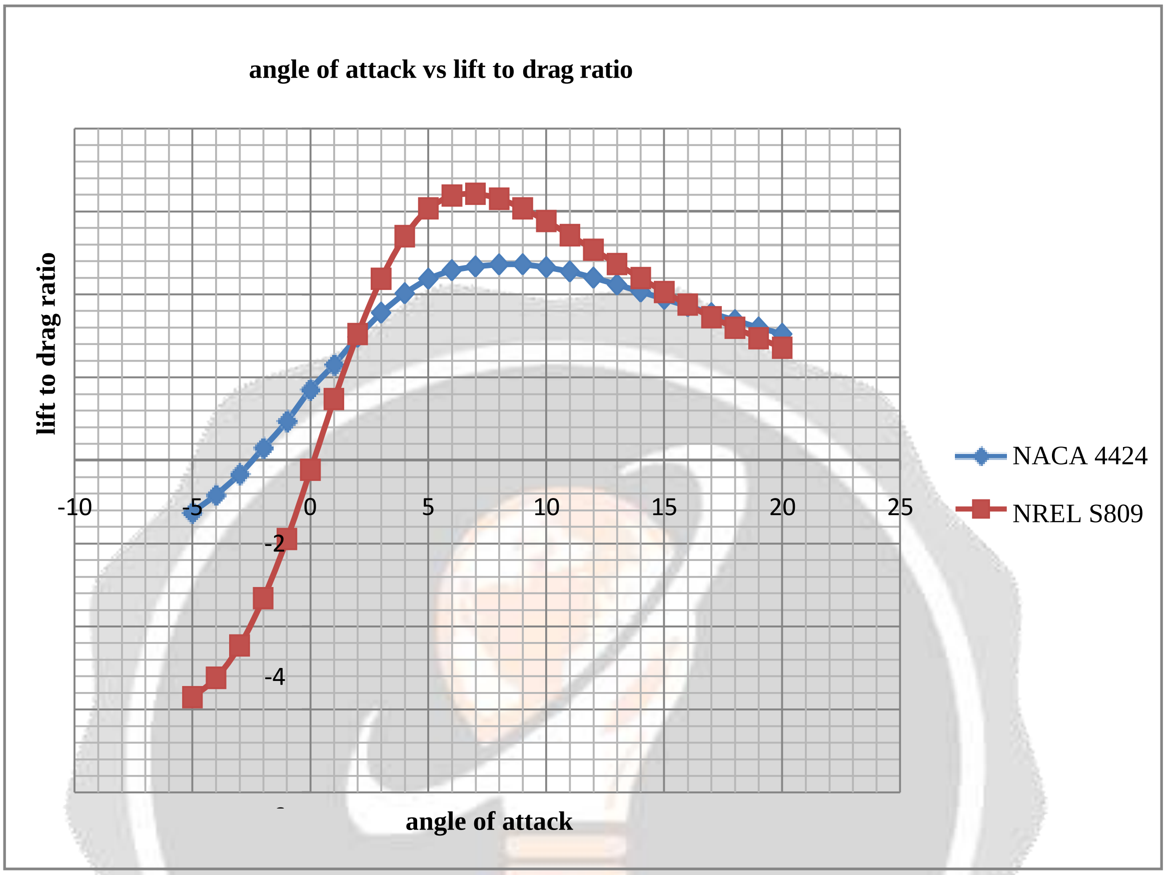
From the graph it is clear that increase in angle of attack first drag coefficient for both NACA 4424 and NREL S809 airfoils decrease up to the 0° angle of attack. After 0° angle of attack drag coefficient increase with increase in angle of attack. For the same conditions drag coefficient for NACA 4424 is more than for the NREL S809 airfoil but at 20° drag coefficient is same for both theairfoils.

At 47 m/s comparison of NACA 4424 and NREL S809 airfoil for lift coefficient



From the graph it is clear that increase in angle of attack increases lift coefficient for both NACA 4424 and NREL S809 airfoils. For the same conditions lift coefficient for NACA 4424 is more than compare to the NREL S809 airfoil. At 7° angle of attack lift coefficient is approximately same for both the airfoils.

At 47 m/s comparison of NACA 4424 and NREL S809 airfoil for CL/CD



Lift to drag ratio for NREL S809 is less than the NACA 4424 airfoil for the negative angle of attack to 2° angle of attack. After 2° angle of attack lift to drag ratio for NREL S809 is more than the NACA 4424 airfoil. Initially when angle of attack increase lift to drag ratio increases up to certain angle of attack and then lift to drag ratio start decreasing continuously. From the graph it is clear that for NREL S809 airfoil maximum lift to drag ratio is achieved at 6.5° angle of attack and for NACA 4424 airfoil maximum lift to drag ratio is achieved at 9° angle ofattack.

For an airfoil lift to drag ratio should be maximized as a result it can improve the efficiency when wind turbine generates electricity. From the above results we can say that for 47 m/s wind velocity NREL S809 airfoil is better than the NACA 4424 airfoil, because lift to drag ratio is more than the NACA 4424airfoil.

Optimum angle of attack can be defined as the angle of attack where maximum lift to drag ratio should be found. For NACA 4424 airfoil optimum angle of attack is 9° and for NREL S809 airfoil optimum angle of attack is 6.5° at 47 m/s windspeed.

6 Conclusion

I. ANSYS-Fluent shows a good performance in calculating the lift and drag coefficient of airfoils when compare to the experimental data. Especially for low angles of attack. So this software has good ability to predict the optimum angle ofattack.

II. Using the second order SST k-ω turbulent model, ANSYS-Fluent shows a good agreement with the measure data for a variety of windspeeds.

III. NREL S809 airfoil is gives always best performances compare to NACA 4424 airfoil for both 39 m/s and 47 m/s windspeed.

IV. Both NACA 4424 and NREL S809 airfoils are low optimum angle of attack airfoils for good aerodynamicperformance.

7 Future Work

For CFD method analysis, software like ANSYS with CFX should be linked with PRO/E. So an actual lifts and drag reduction, vortex effects for the respective winglet can be calculated by varying the parameters. Optimization of the geometry for each type of winglet could be done, by targeting drag as minimizing factor.

8 Acknowledgement

I express my deep sense of gratitude and indebtedness t my Project Guide Prof. R.Y. Patil Associate professor and H.O.D., Department of Mechanical Engineering, S.G.D.C.O.E., Jalgaon for providing precious guidance, inspiring discussions and constant supervision throughout the course of thos work being carried out. His timely help, constructive criticism and conscientious effort made it possible to present the work contained in this thesis.

9 References

1 Siddharth Joshi, “Design, Simulation And Analysis Of Grid Connected Wind Energy Conversion System”, A Thesis Submitted to GTU the Degree of Master of Engineering in Power System, June 2011PP14.

2 G.M. Joselin Herbert et al, “A review of wind energy technologies”, Renewable and Sustainable Energy Reviews 11 (2007) PP1117–1145.

3 Wikipedia, the free encyclopedia,http://en.wikipedia.org/wiki/wind_power

4 Metin Ozen, Ashok Das, Kim Parnell, “CFD Fundamental and Applications”, PP7-8.

5 Prashant Bhatt et al, “Computational Fluid Dynamics Analysis of Wind Turbine Rotor Blades- A Review”,IJCRR, Nov 2012 / Vol 04 (21) Page163.

6 H. V. Mahawadiwar, V.D. Dhopte, P.S.Thakare, Dr. R. D. Askhedkar, “CFD Analysis of Wind Turbine Blade”, International Journal of Engineering Research and Applications, May-Jun 2012, PP-3188-3194.

7 Ji Yao, Jianliang Wang, Weibin-Yuan, Huimin Wang, Liang Cao, “Analysis on the Influence of Turbulence Model Changes to Aerodynamic Performance of Vertical Axis Wind Turbine”, ELSEVIER, International Conference on Advances in Computational Modelling and Simulation, Procedia Engineering 31 (2012)274-281.

8 Chris Kaminsky, Austin Filush, Paul Kasprzak and Wael Mokhtar, “A CFD Study of Wind Turbine Aerodynamics”, Proceedings of the 2012 ASEE North Central Section Conference.

9 C. Rajendran, G. Madhu, P.S. Tide, K. Kanthavel, “Aerodynamic Performance Analysis of HAWT Using CFD Technique”, European Journal of Scientific Research, ISSN 1450-216X Vol. 65, No. 1 (2011), PP28-37.

10 R.S. Amano, R.J. Malloy, “CFD Analysis on Aerodynamic Design Optimization of wind Turbine Rotor Blades” , World Academy of Science and Technology602009.

11 Le Pape A and Lecanu J, “3d navier-stokes computations of a stall-regulated wind turbine”, Wind Energy,7(4):309–324, October-December2004.

12 Piggott H. “Small Wind Turbine Design Notes”http://users.aber.ac.uk/iri/WIND/TECH/WPcourse/index.html

13 Larwood, S. and Zuteck, M., “Swept Wind Turbine Blade Aeroelastic Modelling for Loads and Dynamic Behavior”2006.
14 Mandas, N., Cambuliand, F., and Carcangiu, C., 2006, “Numerical Prediction of Horizontal Axis Wind Turbine Flow,” University of Caglairi, EWEC 2006, Athens, Business, Science, andTechnology.
15 Ferrer, E. and Munduante, W., “Wind Turbine Blade Tip Comparisons Using CFD.” 2007, Journal of Physics Conference series 75,012005.
16 Bhargav Patel “Aerodynamic analysis of wind turbine blade using CFD technique”
17 Carlo Enrico Carcangiu, “wind turbine functioning and aerodynamics”,2008.
18 Erich Hau, “Wind Turbine Fundamentals Technologies” , Application German Springer,2006.
19 J.F. Manwell, J.G. McGowan and A.L. Rogers, “Aerodynamic of wind turbine”, Wind Energy Explained- Theory Design and Application, John Wiley and Sons limited, London,2002.
20 Maalawi KY, Badawy MTS, “A direct method for evaluating performance of horizontal axis wind turbine”. Renew Sustain Energy Rev2001;5:175-90.
21 Martin O.L. Hansen, “Aerodynamics of wind turbine” Earthscan UK and USA, 2008.
22 R. Lanzafame, M. Messina, “Fluid dynamics wind turbine design: critical analysis, optimization and application of BEMtheory”.
23 Vaughn Nelson, “Wind Energy: Renewable Energy and the Environment”,2009.
24 Han Cao, “Aerodynamic analysis of small horizontal axis wind turbine blade by using 2D and 3D CFD modelling”,2011.
25 Xinzi Tang , “Aerodynamic and analysis of small horizontal axis wind turbine blade”, 2012.
26 C.A. Ramirez Gutierrez, “Aerodynamic and aeroelastic design of low windspeed wind turbine blade”,2011.
27 N.Manikandan, B. Stalin, “Design of Naca63215 airfoil for a windturbine”.
28 S. Rajakumar, Dr.D.Ravindran “computational fluid dynamics of wind turbine blade at various angles of attack and low Reynolds number” ,2010.
29 Gómez-Iradi, S., Steijl, R., and Barakos, G. N., "Development and Validation of a CFD Technique for the Aerodynamic Analysis of HAWT," Journal of Solar Energy Engineering, 131,(3).
30 C. Rajendran, G. Madhu, P.S. Tide, K. Kanthavel, “Aerodynamic Performance Analysis of HAWT Using CFD Technique”, European Journal of Scientific Research, ISSN 1450-216X Vol. 65, No. 1 (2011), PP28-37.