CFD ANALYSIS OF AIRFOIL SECTIONS

Vinayak Chumbre[1], T. Rushikesh[2], Sagar Umatar[3], Shirish M. Kerur[4]

[1,2,3] Student, Jain College of Engineering, Belagavi, Karnataka, INDIA
[4]Professor, Dept. of Mechanical Engineering, Jain College of Engineering, Karnataka, INDIA

Abstract - *Airfoil is a structure that provides required Lift force and Drag force depending on profile design i.e. curved surface when an Airfoil structure moved in a fluid medium, it provides the aerodynamic forces. Therefore it is quite important to understand the importance behind the shape of Airfoil. As CFD is a powerful tool used to carry out the simulation by considering the real-life behaviour of the fluid. In this study, the main objective is to carry out a detailed study on lift and drag coefficients of various Airfoil sections, five commonly used Airfoil sections were generated using SOLIDWORKS and the fluid flow simulation was carried out by using ANSYS Fluent and tried to study the variation in lift and drag coefficient.*

***Key Words*: CFD, Airfoil Sections, ANSYS Fluent, lift and drag coefficient.**

1. INTRODUCTION

An Airfoil is a structure with curved surfaces designed in such a way that it provides the favourable lift and drag forces. The Airfoil structure, when moved with a certain velocity in the medium of fluid, produces an aerodynamic force. The aerodynamic force is the force exerted by the fluid on the airfoil structure due to the relative motion between the fluid and the airfoil structure. This aerodynamic force arises due to two reasons. One due to the pressure exerted by the fluid on the surface of the fluid and second due to the shear force exerted by a fluid due to its viscosity on the surface of the body. The pressure exerted acts perpendicular to the surface of the body whereas the shear force acts parallel to the surface of the body. These pressure and shear force together create an aerodynamic force which acts in the direction opposite to the motion of the Airfoil structure. The aerodynamic force is resolved into two forces: Lift and Drag. Lift is the force which acts perpendicular to the direction of relative motion and Drag force that acts parallel to the direction of relative motion.

Airfoils work on Bernoulli's principle to produce lift force. When the fluid flows over the Airfoil structure, there will be curved streamlines of fluid created which results in the formation of lower pressure on one side while higher pressure on the other side. This pressure difference is responsible for creating the lift force. Jon Leary, [1] carried out CFD analysis on the blades of a wind turbine with the aim to analyze the lift and drag produced. He ended up with the conclusions that the lift keeps on increasing with the angle but starts decreasing after certain angle while the drag keeps on increasing continuously. The results suggested that the Airfoils with greater camber will give better lift. Chandrakant Sagat, [2] carried out experimental and CFD analysis of Airfoil at low Reynolds number in which he found that the coefficient of lift raised till 12^0 and then decreased while the value of the coefficient of drag is small but kept on increasing. Ankan Dash, [3] carried out CFD analysis of Airfoil NACA 0012 structured turbine at various angles of attack. He observed that the coefficient of lift increased rapidly but the coefficient increased but not as rapidly as the coefficient of lift. The coefficient of lift raised to 10^0 deg then started decreasing. Arvind et. al, [4] carried out CFD analysis of Airfoil which had flaps and slats structures. They compared these results with the Airfoil which had no flaps and slats. They concluded that the normal Airfoil had very low stall whereas the Airfoil with flaps and slats had very high stall angle. Mohamed et. al, [5] studied the aerodynamic performance of GOE 387 Airfoil at a various angle of attack with constant Reynolds number (3×10^5) using Transition k-kl-omega turbulence model. It was found that pressure co-eff at the upper surface of Airfoil was negative and lower surface was positive, thus indicating lift force of Airfoil was in an upward direction. Mr Mayurkymar Kevadiya, [6] studied the NACA 4412 Airfoil profile and recognized its importance for investigation of wind turbine edge. The geometry of the Airfoil is made utilizing GAMBIT 2.4.6. Also, CFD investigation is done utilizing FLUENT 6.3.26 at different approaches from $0^o$to 12^o. Shivananda et. al, [7] studied the effect of flow over NACA 2412 Airfoil at high Reynolds number, with respect to pressure distribution, velocity distribution, various aerodynamic forces while maintaining variation in its angle of attack.

They concluded that the lift force and drag force increase respectively as the angle of attack increases until it reaches stall, 5^o angle of attack was found optimum where the Airfoil model produced max. Lift to drag ratio and 15^o angle was found to be the critical angle of attack beyond which amount of lift generated dropped drastically. A L Wensuslaus et. al, [8] developed 3D Airfoil wing model to study flutter in the wind tunnel. The existence of flutter at high velocity of aircraft was demonstrated, as flutter was observed at 40m/s and as velocity reduced to 30m/s, the amplitude of vibration decreased until wing stopped vibrating. Flutter was observed only at a high angle of attack (15^o). K. A. Ahmad et. al, [9] used RANS code to predict the flow field around a pitching NACA 0012 Airfoil. They concluded that the results demonstrated the best turbulence model that gives the best agreement with the experimental data is SST k-□ with y+ value is set to 1.

They also found that the c_L and c_D were not affected by varying time step and mesh parameters. Yan Wang et. al, [10] studied the effect of leading-edge pitting erosion on the aerodynamic performance of a S809 Airfoil. It was found that effect of pitting erosion depth has the greatest influence at an angle of attack 8.1^o, while having little influence when the angle of attack is smaller than 2.1^o. They concluded that the aerodynamic coefficients are mostly affected when erosion area is located at the first 15% of the Airfoil in chordwise. K. K. Koay et. al, [11] studied the effect of boundary conditions used in CFD software which is used to generate the flow pattern. They concluded that the boundary condition has a great effect on CFD analysis. It was found that the boundary condition of 600×300×100 mm^3 when Z-axis of the boundary condition is set as same as chord length of NACA 0012, is the best boundary condition.

2. CFD ANALYSIS

In this study, ANSYS Fluent is used as solver domain, the coordinates required to model the Airfoil are imported from NACA website and by using those coordinates, model was generated using SOLIDWORKS, and further this model was imported in ANSYS Fluent. The cross-section is developed to be fine at areas near to the Airfoil and with high vitality and coarser more remote far from the Airfoil. For these Airfoils, an organized Quadratic Dominant lattice was utilized. A fine work infers a higher number of counts which thus makes the simulation long. The Max Face size of 0.05m was generated. And at the boundaries of Airfoil, those edges were refined with level 3 as shown in Fig -1.

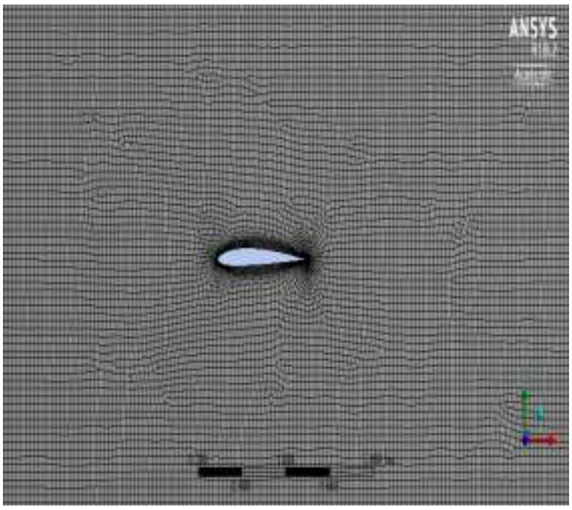

Fig -1: Mesh structure

The most fundamental part of any CFD problem is the definition of its boundary conditions. To conduct the simulation, boundary conditions for the problem and initial inputs are taken, and it is shown in the table-1.

Table -1: Input boundary conditions

Sl. No.	Input	Value
1	Velocity of Flow	51 m/s
2	Operating pressure	101325 Pa
3	Model	Inviscid
4	Density of fluid	1.225 kg/m^3
5	Kinematic viscosity	1.7894e-05 kg s/m^2
6	Fluid	Air as ideal

Table -2: Mesh details

Number of Nodes	31078
Number of Elements	31222

3. RESULTS AND DISCUSSION

Initially, in our study, results were obtained by considering the same model NACA 4412 which was the one studied by Paul et al [12]. By considering the same boundary conditions and input values, results were obtained in ANSYS Fluent. The results were found to be in good agreement with the published results at two angles of attacks i.e. at 0^0 and 15^0. While the result at 6^0 was found to be bit different from the published results, the comparison of validated results and published is shown in the chart -1:

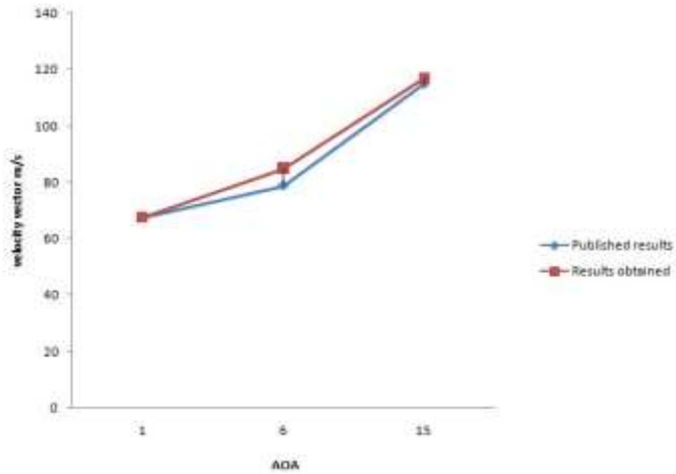

Chart -1: Validation of results

Usually, with the variation of nodes and elements formed after meshing the body, the results obtained will also change. So it is very important to make a grid independence study in which we obtain the values of nodes and elements at which the results obtained become independent. In our study too we carried out the grid independence study at 1627, 5669, 9568, 23388 and 30000 nodes and found out that the results remained independent of elements and nodes formed after approximately 10000 nodes. The below chart shows the variation of results for corresponding nodes:

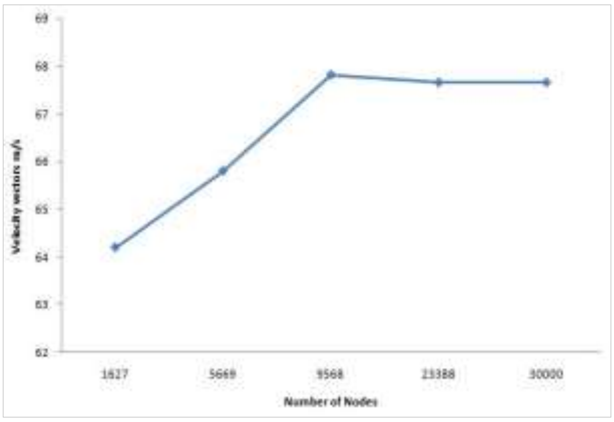

Chart -2: Grid independence study

Further, our study was extended to carry out a detailed study on various airfoil sections, we have modelled five commonly used airfoil sections namely NACA 0012, NACA 2412, NACA 6409, NACA 4412, NACA E387 and tried to study their variation in lift and drag coefficient.

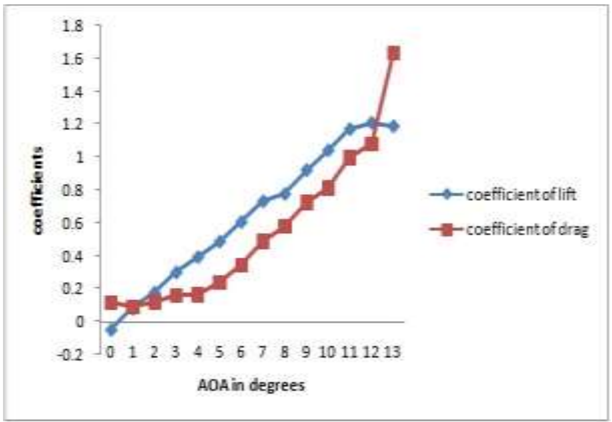

Chart -3: Coefficient vs AOA for NACA 0012

For a better understanding of variation in coefficient of lift and drag of the airfoil sections for a different angle of attack of airfoil sections that obtained results were superimposed. Since the coefficients of lift values are comparatively higher than the values of coefficient of drag, therefore the values of coefficient of drag are magnified and then superimposed with a coefficient. As we can see that in the chart -3,4,5,6,7 for different airfoil sections where the blue line represents the coefficient of lift and the red line represents the magnified value coefficient of drag.

We can see that coefficient of lift valves increases till it reaches the stall angle. The angle at which, the coefficient of lift attends maximum values such angle is known as stall angle.

And also we can see that coefficient of drag will initially be almost same with no much variation, but after a certain angle of attack, the coefficient of drag will continuously increase. At stall angle, we can see that coefficient of drag will suddenly increase and due to this, it is evidence that the coefficient of lift is decreasing.

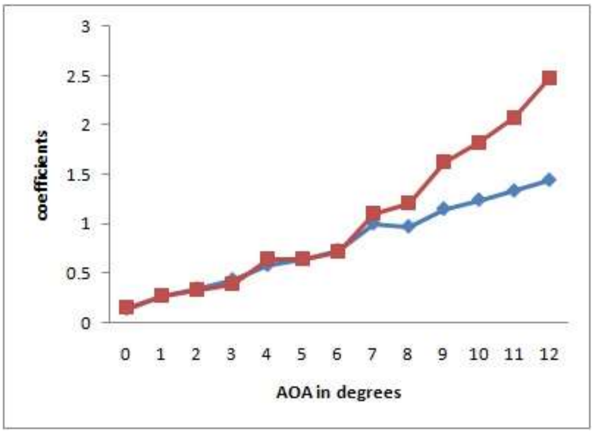

Chart -4: Coefficient vs AOA for NACA 2412

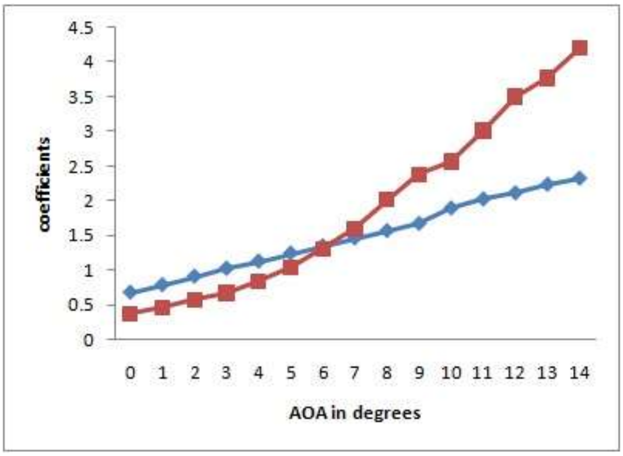

Chart -5: Coefficient vs AOA for NACA 6409

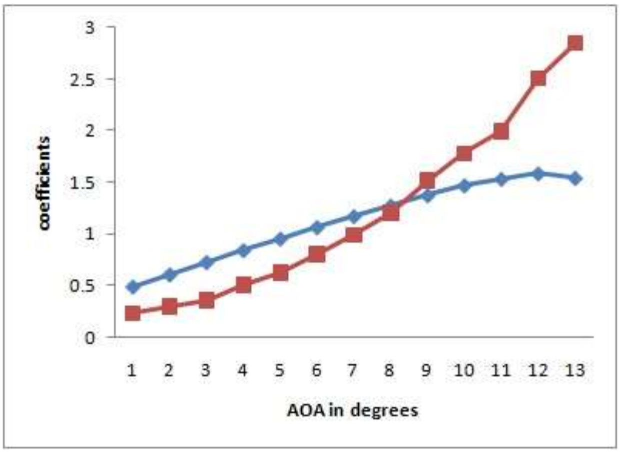

Chart -6: Coefficient vs AOA for NACA 4412

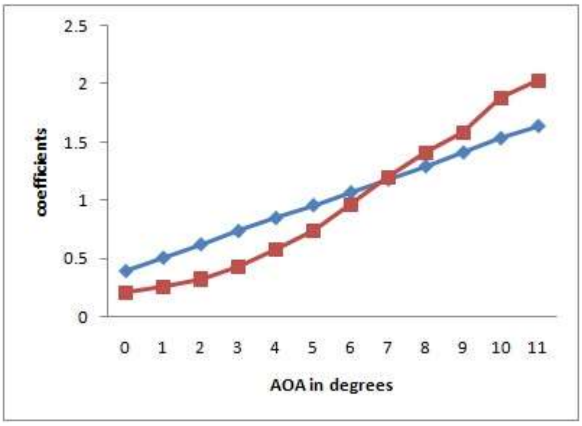

Chart -7: Coefficient vs AOA for NACA E387

Further, the lift force and drag force per unit area were compared for five airfoils for the corresponding angle of attack. This comparison is carried out by plotting the values of lift force and drag force per unit area as shown in below chart -8 & 9.

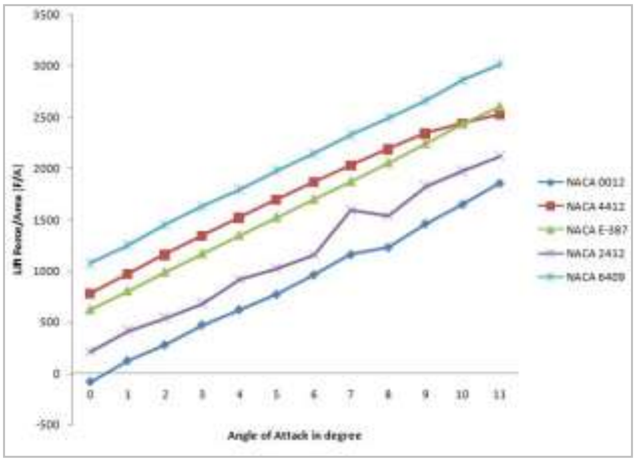

Chart -8: Lift force/Area (N/m²) vs AOA

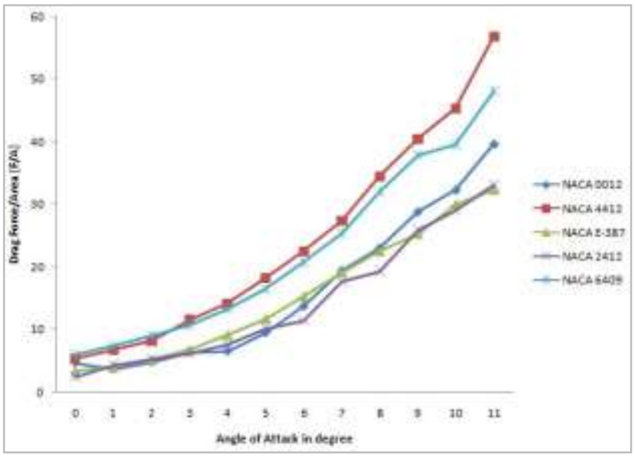

Chart -9: Drag force/Area (N/m²) vs AOA

The above figures show the variation of Lift force/Area and Drag force/Area with the change in Angle of Attack. It can be seen from the chart -8 that NACA 6409 gives the maximum lift force while NACA 0012 gives the least lift force among the five airfoil sections studied. From the chart -9, it can be seen that NACA 4412 gives the maximum drag force while NACA 2412 gives the minimum drag force among the five airfoil sections studied.

4. CONCLUSIONS

- NACA 6409 not only gives the maximum lift force but also gives greater drag force. The better lift can be obtained by using this airfoil but a lot of energy is needed to overcome the drag force generated. Greater energy means more fuel is needed.
- NACA 2412 gives the minimum drag force among all the studied airfoil sections but the lift force that it generates is lower. Hence we can say that it has a better fuel efficiency among all 5 airfoils but cannot give a better lift force.
- We can see that NACA E-387 gives an intermediate value lift force among the five airfoils while the drag force experienced by it is also very low and in fact at certain angles, the drag force is the least among the 5 airfoils.
- So our study on the five airfoil sections recommends that NACA E387 is better as it gives good lift force with least drag force. So the fuel efficiency will be better.

REFERENCES

[1] Jon Leary, “Computational Fluid Dynamics Analysis of a Low-Cost Wind Turbine”, EPSRC (2010).

[2] Chandrakant Sagat, “Experimental and CFD Analysis of Airfoil at Low Reynolds Number”, Vol 1, No3, (2012) 2278-0149.

[3] Ankan Dash, “CFD Analysis of Wind Turbine Airfoil at Various Angles of Attack’, Vol 13 (2016) 2278-1684.

[4] Arvind Prabhakar, Ayush Ohri, “CFD Analysis on MAV NACA 2412 Wing in high Lift Take-Off Configuration for Enhanced Lift Generation”, (2013) 2168-9792.

[5] Mohamed A. Fouad Kandil, Abdelrady Okasha Elnady “Performance of GOE-387 Airfoil Using CFD”, IJAS 2017, 5(1): 1-7

[6] Mr Mayurkymar kevadiya, “CFD Analysis of Pressure Coefficient for NACA 4412”, IJETT-Volume 4 Issue 5- May 2013.

[7] Shivananda Sarkar, Shaheen Beg Mughal, “CFD Analysis on effect of flow over NACA 2412 Airfoil through the shear stress transport turbulence model”, ISSN: 2320-2092, Volume- 5, Issue-7, Jul.-2017.

[8] A L Wensuslaus, A J McMillan, "Aerofoil flutter: fluid-mechanical analysis and wind tunnel testing", MPSVA 2012.

[9] K.A. Ahmad, M.Z. Abdullah, J.K. Watterson, "Numerical Modelling of Pitching Airfoil", June 2010.

[10] Yan Wang, Ruifeng Hu, Xiaojing Zheng, "Aerodynamic Analysis of an Airfoil with Leading Edge Pitting Erosion", JSEE- July 2017

[11] K.K. Koay, W.C. Tan, "Computational Fluid Dynamics Study For Aerofoil", 2nd NCMER 2010.

[12] Jeyo Paul, Ajo Alexander, Manikandan, "CFD Analysis of Velocity Profiles of NACA 4412 Aerofoil", Vol-2, 2016, ISSN:2454-1362.