CFD simulation of NACA airfoils at various angles of attack

Y Seetha Rama Rao¹, M V N SrujanManohar² and S V V SivaPraveen³

¹Associate Professor, Dept of Mechanical Engineering, Gayatri Vidya Parishad College of Engineering (A), Visakhapatnam, Andhra Pradesh, India
²Research scholar, Pondicherry Engineering College, Pondicherry, India
³Bachelor of Technology in Mechanical, Gayatri Vidya Parishad College of Engineering(A), Visakhapatnam, Andhra Pradesh, India

Abstract. The need for more efficient design has become a main concern to work with a good range for angle of attack (AOA) to reduce the stall formation. In this project, two types of NACA series airfoils i.e., symmetrical and asymmetrical profiles depending on their usage in various sectors are compared. The profile of the airfoils is imported into ANSYS Fluent from the data files. The 2D geometry of the airfoils is simulated under two different wind velocities i.e., at 3m/s and 15m/s respectively. The graphs are plotted for comparing coefficients of lift (C_L) and drag (C_D) with respect to the angle of attack(α) for symmetrical and asymmetrical airfoils. The stall formation with change of geometry and angle of attack is also observed.

Keywords: NACA airfoil, Angle of Attack, Stall, Ansys Fluent, CFD Simulation

1. INTRODUCTION

An airfoil is the cross-sectional shape of a wing, blade (of a propeller, rotor, or turbine) or sail. It is the most fundamental and prime element in building a wind turbine blade. An aerodynamic force is produced when an airfoil-shaped body is moving through a fluid. The component of this force perpendicular to the direction of motion is called Lift. The component parallel to the direction of motion is called Drag. The lift developed on an airfoil is basically the outcome of its angle of attack[1]. The onrushing air gets deflected by the airfoil, when aligned at a suitable angle. This results in creating a force on the airfoil that is in the direction opposite to the deflection. The key factors altering both of these coefficients are the angle of attack (α) between the incoming wind direction and the airfoil chord, the geometry of the airfoil and the Reynolds’s number. Behaviour of the airfoil due to movement of air is divided into three phases: (1) the attached flow phase, (2) the high lift/stall development phase and (3) the flat plate/fully stalled phase.

The NACA airfoils are the shapes of different airfoils designed for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA). The shape of the these airfoils is designated using a series of numbers followed by “NACA”. With the help of equations by entering these digits, the properties of airfoil can be calculated. The cross-section of the airfoil can also be generated easily.

The NACA series 4-digit airfoil section defines its profile as follows:

- First digit indicates maximum camber as percentage of the chord.
- Second digit provides the distance of maximum camber from the leading edge of the airfoil in tenths of the chord.
- Last two digits indicates maximum thickness of the airfoil as a percentage of the chord.

Airfoils of this series have max. thickness at 30 percent of chord from the leading edge.

A. Meana-Fernandezet.al [2] proposed an airfoil shape used in the applications of vertical-axis wind turbine. Various airfoil geometries have been examined using Java Foil, a panel method software. The performance of the airfoil generated is optimized using the results obtained from the analysis. At different angles of attack, the CFD simulations were ran in order to give detailed perception regarding field of fluid and other mechanisms useful to enhance the performance of the presented airfoil. The two designs related to VAWT are compared with the primary and presented airfoil to verify the improvement in performance. Ultimately, the feasibility of Java Foil had been tested by comparing geometries of various airfoils because it is convenient and has the ability to obtain results for a large range of flow.
environments in a relatively less amount of time. However, in case of higher angles of attack i.e., beyond stall, the results always deviate from the original outcome.

Anagha S Gowda [3] performed aerodynamic performance characteristics comparison between two airfoils NACA 2412 and NACA 4412 under similar flow conditions at a Reynolds Number of 2 million. The creation of geometry and meshing were done using ANSYS software and FLUENT fulfilled the purpose as solver. The performance of every airfoil with aerodynamic parameters was compared with respect to angle of attack and from that set, a particular airfoil which is optimal for precise aerodynamic characteristics were selected.

Castineira-Martinez Set.al [4] performed a large number of simulations to arrive at for a typical profile for a wind turbine. The solutions of these simulations were calculated over various sizes of mesh by a 2-dimensional Reynolds-Average Navier-Stokes (2D-RANS) approach. Three other turbulence models like k-ε and k-Ω were also used in the simulations. For each and every model and size of mesh, over four angles between -2.5 and 12.5, coefficients of lift, drag and momentum were calculated and compared them with the experimental values. As a result of all these attempts, a workable model that holds good standards (i.e., accord with both numerical & experimental data) and can be used for further investigation as a basic model was obtained.

2. TURBULENCE MODEL & GEOMETRY

Turbulence modelling is the construction and use of a mathematical model to predict the effects of turbulence. K-epsilon (k-ε) turbulence model is the most common model used in computational fluid dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. It is a two-equation model which gives a general description of turbulence by means of two transport equations (PDEs).

The original momentum for the K-epsilon model was to upgrade the mixing-length model, and to find an auxiliary to algebraically prescribe turbulent length scales in moderate to high complexity flows.

Realizable k- ε Model: The benefit of the realizable k-ε model is that it provides advanced predictions for the diffusion rate of both planar as well as round jets. It also shows superior performance for flows involving boundary layers and rotation under strong adverse pressure gradients, recirculation and separation. In every measure of comparison, Realizable k-ε signifies a superior potential to express the mean flow of the complex structures.

Geometry of NACA airfoils were acquired as co-ordinate vertices i.e., texts file and imported into the CATIA. Few modifications were made to those files to adjust the geometry and prepared a valid surface model for further steps.

![Figure 1. Geometry of airfoil](image)
FLUENT is essential in performing of CFD analysis as it creates the working environment where the object is simulated. The crucial part of this is to create the mesh throughout the object. This has to be extended in every direction possible to apply the physical properties of the surrounding fluid (air). In order to apply the boundary conditions that are required to carry out analysis, mesh and edges must be grouped together for the effective output. At first, import the coordinates of the airfoil and create the curve using 2D analysis and launch the design model. Now, draw the required size of the domain for the airfoil section. The coordinate system can be created either at the leading edge or at the trailing edge of the airfoil which would be helpful in creating the rectangular mesh domain geometry using dimension tool and sketcher toolbox.

3. MESHING & FLUENT SETUP

A domain consisting of a rectangular profile of adequate dimensions is to be designed with airfoil as an interior surface. The mesh that is generated has to be very fine at regions close to the cross section of the airfoil. For this airfoil, a structured rectangular mesh was used. But depending upon the accuracy and limitations in the software, the mesh at certain areas of the domain can be varied accordingly.

A fine mesh implies a higher number of calculations which in turn makes the simulation use longer time to finish. For the NACA airfoils, starting from the leading edge, the distance between the nodes increases. An even number of points are distributed and analysed from the point of maximum thickness on the airfoil to the trailing edge. The meshed rectangular domain shown in Fig 2. consists of nodes and elements. Each of the airfoil sections used are meshed and contains 82,833 nodes and 81,017 elements on an average.

Assigning properties to the respective geometries is a crucial part of the simulation to make it work. In this case, the airfoil itself was treated as interior surface. The boundaries conditions to the mesh were set as velocity as inlet at the entry and as the end boundary or outlet, pressure at zero-gauge had been simulated. All the other surfaces are treated as walls that are stationary and having no slip conditions. The boundary conditions that are provided for this simulation is tabulated referring to Figure 1 as shown below.
Table 1. Boundary conditions of fluid domain.

| EDGE | NAME     | CONDITION APPLIED |
|------|----------|-------------------|
| A    | INLET    | VELOCITY          |
| B    | OUTLET   | PRESSURE          |
| C    | TOP WALL | STATIONARY&NO SLIP |
| D    | BOTTOM WALL | STATIONARY&NO SLIP |

Figure 3. Flow of the air in the domain

The meshed component was imported into FLUENT, and the parameters of the system were set to Standard and Double precision to make certain that it provides ample accuracy. Generally, as default it possesses single precision but as the solution has to be meticulous, double precision is advisable. The residuals for the various turbulence model variables were set to 10e and the maximum iteration count to 100. If the values of the coefficients are stabilized then also the process can be terminated.

An important part is selection of suitable viscous model for the simulation to proceed and in case of FLUENT, many choices are available. From the available viscous models, Realizable k-epsilon model is selected which uses two set of equation to arrive at the solution. Further in the selection of solution methods, the pressure-velocity coupling is SIMPLE. Then the initialisation of the solution is done in standard method from the inlet and the calculation is initiated.
Table 2. Working Parameters in Ansys Fluent.

| Working Fluid | Air |
|----------------|-----|
| Density of Air | 1.225 kg/m³ |
| Viscosity of Air | 1.7894*10⁻⁵ kg/m-s |
| Solver Type | Pressure-Based |
| Viscous Model | Realizable k-epsilon |
| Pressure-Velocity Coupling | SIMPLE |
| Momentum | Second Order Upwind |

4. GRAPHS AND OBSERVATION

The coefficients of lift and drag are used to plot graphs with respect to angle of attack at two different velocities namely at 3 m/s and 15 m/s i.e., at Reynold’s number of 20000 and 100000 respectively to compare the effect of variation in flow over each of the airfoil profiles.

4.1. Comparison of Symmetrical airfoils

The above-mentioned procedure is applied to a set of symmetrical airfoils namely NACA 0012, 0015, 0018 and 0021. These four profiles are simulated under two velocities and the respective graphs are plotted with the results obtained for comparison with one another.

![Graph of C_D vs \(\alpha\) at 3 m/s](Image)

From the graph shown in Figure 4, at a velocity of 3 m/s, the lift coefficient for all the four symmetric airfoils varies linearly till an angle of attack of 15° and then attains a peak angle (stall) and gradually decreases thereafter.
Similarly, for the plot between drag coefficient and angle of attack in Figure 5, the variation of drag is almost same for a certain angle and then changes for each of the profiles.

![Figure 5. CD Vs \( \alpha \) at 3 m/s](image)

However due to low velocity acting on the airfoil, the variation of the forces may not be as accurate as required. So, for proper outputs the same set of airfoils are analysed for a velocity of 15 m/s i.e., at a Reynold’s number of 100000 and the respective graphs are plotted as shown below.

![Figure 6. CL Vs \( \alpha \) at 15 m/s](image)

From the Figure 6, it is observed that the curves obtained at this velocity of 15 m/s are more linear when compared with 3 m/s. The variation of lift coefficient with respect to angle of attack is minimum but airfoil NACA 0021 shows the more linear behaviour compared to the other profiles.
Figure 7. $C_D$ Vs $\alpha$ at 15 m/s

This graph in Figure 7. shows the drag coefficient variation, also depicts that the airfoil profile NACA 0021 gives the lower drag with increase in angle of attack compared to the other three profiles in the set.

4.2. Comparison of non-symmetrical airfoils

Similar to the symmetrical airfoils, a set of non-symmetrical airfoils are taken with same thickness and chord length. The airfoils considered are NACA 0012, 2412, 3412 and 4412 and using lift and drag coefficients graphs are plotted against angle of attack.

Figure 8. $C_L$ Vs $\alpha$ at 3 m/s
**Figure 9.** $C_D$ Vs $\alpha$ at 3 m/s

Figures 8 & 9. shows the graphs plotted between lift and drag coefficients of non-symmetrical airfoils with respect to angle of attack. Compared to symmetrical airfoils, these profiles provide less drag forces.

**Figure 10.** $C_L$ Vs $\alpha$ at 15 m/s

At a velocity of 15 m/s, the variation of lift with the angle of attacks seems to be stabilised as it gives curves with less fluctuations. From the graph in figure 7, NACA 2412 shows the highest value at an angle of 20° approximately and falls thereafter. For the profiles NACA 3412 and 4412 it keeps on increasing.

**Figure 11.** $C_D$ Vs $\alpha$ at 15 m/s

The following are the pressure and velocity contours of two airfoils namely NACA 0021 and NACA 2412. With the help of these contours, the variation of pressure and velocity over the cross-section of the airfoils can be clearly observed and understood. The difference caused by the application of camber is distinctly visible from these contours of 4-digit airfoils. This also helps in making changes to airfoil geometry if necessary, to alter the pressure or velocity on its cross-section.
Figure 12. Pressure Contour of NACA 0021 airfoil

Figure 13. Pressure Contour of NACA 2412 airfoil

Figure 14. Velocity Contour of NACA 0021 airfoil
5. CONCLUSION

From the CFD analysis conducted on the symmetrical and non-symmetrical airfoils having same chord length at Reynold’s number of 20000 and 100000 respectively with angles of attack of 0° to 30°, the following are the observations drawn from the results obtained.

1. In symmetrical airfoils, the only variant being the thickness of airfoil as it increases, the coefficient of lift as well as stall formation angle also widens. Additionally, with the increase in Reynolds number, increase and decrease in coefficients of lift and drag is observed respectively. This also in turn improves the performance of the airfoil. Also, due to increase in stall angle, it provides a wide range of angles to be used for the airfoil alignment.

2. In case of cambered airfoils, the performance of the airfoil gets improved as the value of Reynolds number increases. Compared to symmetrical airfoils, the lift coefficient values of cambered airfoils are a cut above for same thickness and also gives almost same or slightly low values of stall angle.

3. With arise in thickness, the increase in coefficient of lift and angle of stall formation is about 10%.

4. Increasing camber value to 2% in airfoils results in increase of lift and decrease of drag compared to the symmetrical airfoil. A slight reduction in the stall angle can be observed but still cambered airfoils perform better compared to symmetric airfoils. Eventually, there is an increment in stall angle and $C_L$, whereas decrement in $C_D$ as Reynolds number increases.

5. In order to enhance lift and delay stall formation, thickness of airfoil can be increased but doing this could result in some limitations like increase in cost and weight of the airfoil and also put excess load on the turbine.

REFERENCES

[1] Hurt, H. H., Jr, “Aerodynamics for Naval Aviators”. U.S. Navy Aviation Training Division. January 1965. NAVWEPS 00-80T-80. pp.21-22.

[2] A. Meana-Fernandez, L. Diaz-Artos, J.M. Fernandez Oro & S. Velarde-Suarez., “An optimized airfoil geometry for vertical-axis wind turbine applications”, International Journal of Green Energy, 2020.

[3]. Anagha S Gowda, Comparison of Aerodynamic Performance of NACA 4412 and 2412 using Computational Approach, International Journal of Engineering Trends and Technology (IJETT) - Volume 67 Issue 4 - April 2019.
[4] Castineira-Martinez, E., I. Solis-Gallego, J. Gonzalez, J. Fernandez Oro, K. Arguelles Diaz, and S. Velarde-Suarez., “Application of computational fluid dynamics models to aerodynamic design and optimization of wind turbine airfoils”. Renewable Energy and Power Quality Journal, 2014. Volume 1, pp 370–75.

[5] Vad Mathiesen, Brian., "Smart Energy Systems for coherent 100% renewable energy and transport solutions”. Applied Energy. 2015. Volume 145, pp 139–154.

[6] MH Mohamed. “Performance investigation of Darrieus turbine with new airfoil shapes”. Renewable Energy Lab. of Mechanical Power Engineering. Energy 47(2012), pp 522-530.

[7] Mohammad Jafari, Alireza Razavi, Mujtaba Mir Hosseini,. “Effect of airfoil profile on aerodynamic performance and economic assessment of H-rotor vertical axis wind turbines”, ELSEVIER Journal, 15 December 2018, Volume 165 Part A, pp. 792-810.

[8] Bertin, John J.; Cummings, Russel M., Pearson Prentice Hall (ed.). Aerodynamics for Engineers (5th ed.).2009. p. 199.

[9] Jianlong Ma, Yafan Duan, Ming Zhao, WenchunLv, Jianwen Wang, Qilao Meng Ke and Yongfeng Ren., “Effect of Airfoil Concavity on Wind Turbine Blade Performances”, Hindawi Shock and Vibration, Volume 2019, pp 1-11.

[10] Houghton, E. L.; Carpenter, P.W. Butterworth Heinmann (ed.). Aerodynamics for Engineering Students (5th ed.).2003. p. 17.

[11] Sorensen. N and J. Michelsen.,“Drag prediction for blades at high angle of attack using CFD”. Journal of Solar Energy Engineering,2004.126, pp 1011–16.

[12]. Argyropoulos, C., and N. Markatos., “Recent advances on the numerical modelling of turbulent flows”. Applied Mathematical Modelling,2015. Volume 39, pp 693–732.

[13]. Hidesada Kanda, KenshuuShimomukai,, “Numerical study of pressure distribution in entrance pipe flow”. ELSEVIER Journal of Complexity, June 2009, Volume 25, Issue 3, pp 253-267.