International Journal of Scientific Research & Engineering Trends
Volume 10, Issue 3, May-June-2024, ISSN (Online): 2395-566X

Numerical Study of Flow Over NACA 2412 Airfoil at Various AOA's

Nadirge Chandravadan
Department of Aeronautical Engineering,
Institute of Aeronautical Engineering, Hyderabad, Telangana, India

Abstract- An aerodynamic study was conducted to examine the effect of airflow around the NACA 2412 airfoil using CFD methods. The airfoil's coordinates were taken from the airfoil database and imported into a geometry modeler. A mesh was created, and simulations were performed using Fluid Flow (Fluent) software. The study involved solving the governing equations (Continuity, Reynolds Averaged Navier-Stokes, and Energy Equation) in 2-D using Fluent. Graphs were made to compare lift (CL) and drag (CD) coefficients at different angles of attack (α). Results showed that lift and drag forces increased with the angle of attack until reaching the stall point. The optimal angle of attack was found to be 8 degrees, where the NACA 2412 airfoil produced the highest lift-to- drag ratio. The critical stall angle was 16 degrees. Beyond this angle, the lift force decreased, indicating stall. The CFD simulation results closely matched the experimental results, indicating that CFD is a reliable alternative for determining drag and lift.

Index Terms- Computational Fluid Dynamics (CFD), Airfoil, Angle of Attack (AOA), Stall, Turbulence Model, National Advisory Committee for Aeronautics (NACA)

I. INTRODUCTION

The National Advisory Committee for Aeronautics (NACA) was a U.S. federal agency founded on March 3, 1915, to promote and conduct aeronautical research. The shape of NACA airfoils is identified by a series of digits following the word "NACA." A complete catalog of 78 airfoils was published in NACA's 1933 annual report. This study focuses on the airflow over the NACA 2412 airfoil using three different turbulence models and varying angles of attack. By employing Computational Fluid Dynamics (CFD) with ANSYS Fluent, the study evaluates the drag and lift characteristics of the NACA 2412 airfoil across angles of attack from -16 to 20 degrees.

For the simulations, the 2D geometry of the NACA 2412 airfoil is imported, and a computational domain is set up. A structured mesh is used to ensure accuracy and reliability. Real-world flow conditions are simulated by carefully defining boundary conditions. The study compares the lift (CL) and drag (CD) coefficients at different angles of attack. CFD results closely match experimental data, showing that CFD is a reliable alternative to experiments for determining drag and lift.

An airfoil is a shape that produces an aerodynamic force when placed in an airstream. This force is used in various applications, such as wing cross-sections, propeller blades, and turbine blades in jet engines. The basic geometry of an airfoil includes the leading edge (the front point with maximum curvature), the trailing edge (the rear point with maximum curvature), and the chord line (a straight line connecting the leading and trailing edges). The length of the chord line, known as the chord, is the main dimension of the airfoil section.

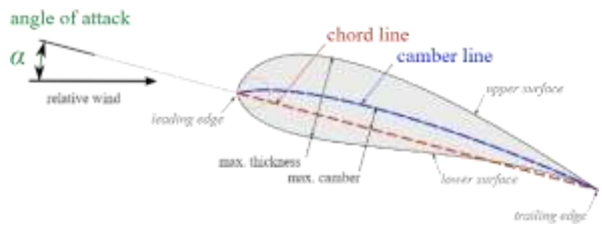


Fig 1: Basic nomenclature of an airfoil

II. METHODOLOGY

The setup used in the numerical analysis, which includes the airfoil model, flow domain, mesh generation and boundary condition.

1. SST k-ω Turbulence Model

The SST k-ω turbulence model is a popular two-equation model that combines the best features of both k-ω and k-ε models. It uses the k-ω formulation in the boundary layer's inner parts, allowing it to be used down to the wall without additional damping functions, making it suitable for low-Re turbulence modeling. In the free-stream, it switches to a k-ε behavior, reducing sensitivity to inlet free-stream turbulence. This model performs well in areas with adverse pressure gradients and separating flow but tends to produce slightly higher turbulence levels in regions with strong acceleration, though less so than the standard k-ε model.

2. Spalart–Allmaras Turbulence Model

The Spalart–Allmaras model is a one-equation model used to solve the kinematic eddy turbulent viscosity. Designed for aerospace applications involving wall-bounded flows, it performs well in boundary layers with adverse pressure gradients and is gaining popularity in turbo machinery.

It is a low-Reynolds number model that requires detailed resolution of the boundary layer. However, it is not calibrated for general industrial flows and has larger errors in free shear flows and predicting the decay of homogeneous, isotropic turbulence.

3. Realizable k-ε Model

The realizable k-ε model has become popular in the CFD community due to its better performance compared to the standard k-ε model, especially in flows with strong adverse pressure gradients, stream wise curvature, separation, and recirculation zones.

It also improves the predicted spreading rates of round jets. This two-equation high-Reynolds-number model differs from the standard k-ε model by using a different transport equation for the dissipation rate and a different eddy-viscosity formulation based on local flow parameters rather than a constant coefficient.

III. CFD MODELING AND MESHING

1. Geometry Description

The numerical study of the flow over the NACA 2412 airfoil begins with the acquisition of airfoil coordinates from Airfoil Tools, which are subsequently imported into Design Modeler. The specified airfoil coordinates serve as the foundation for creating a comprehensive geometry in the form of a C domain. The key points in the design involve meticulous consideration of the airfoil's shape and structure.

The NACA 2412 airfoil, characterized by its four-digit code, is known for its symmetrical profile and moderate thickness-to-chord ratio. The geometry description encompasses the chord length, camber distribution, and thickness distribution, providing a detailed overview of the airfoil's physical attributes. This geometric model lays the groundwork for a thorough numerical investigation into the aerodynamic performance of the NACA 2412 airfoil, allowing for insights into its flow characteristics and behavior under varying conditions.

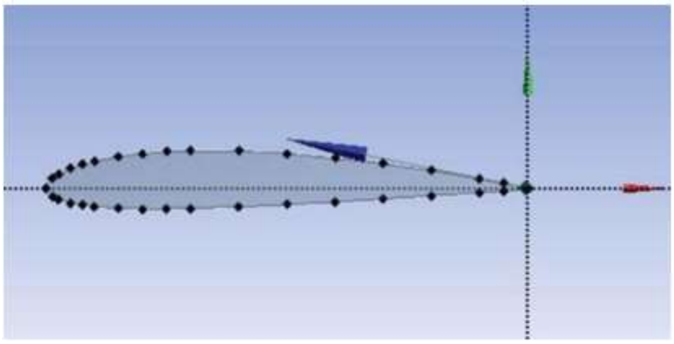


Fig 2: Geometry of NACA 2412

The flow field consists of a C-mesh topology that surrounds the airfoil. The C-mesh is preferred over other mesh topologies, such as the O-mesh, due to its ease of construction and better capture flow characteristics, such as the wake behind the leading edge. Grid points behind the trailing edge for C- mesh also have high grid density. The control volume is made with 20L Downstream and 10L in other three directions taking L as the chord length.

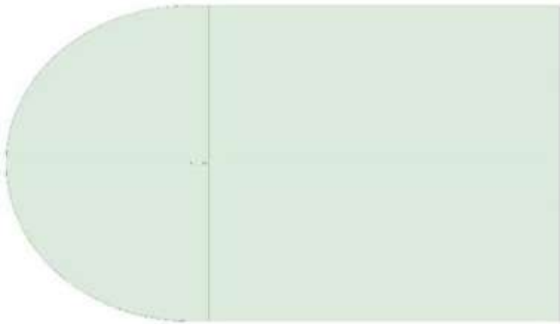


Fig 3: C-Domain Topology

2. Mesh Description

A domain consisting of a C-Domain 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 C-Domain 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.

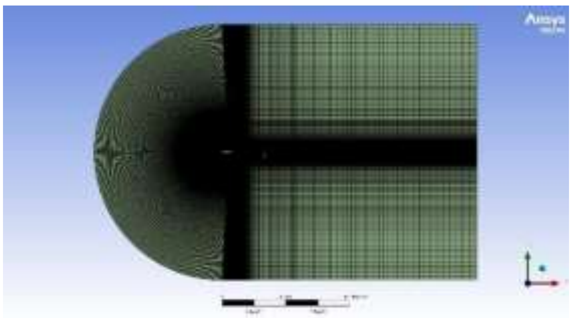


Fig 4: Meshed Airfoil Domain

A fine mesh implies a higher number of calculations which in turn makes the simulation take 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 C domain shown in Fig4, consists of nodes and elements. Each of the airfoil sections used are meshed and contains 2,50,000 elements on an average.

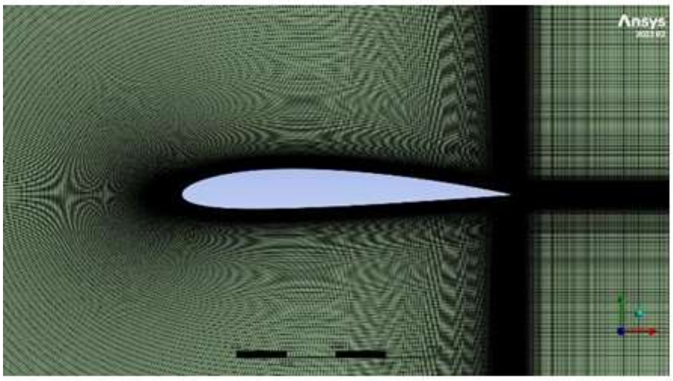


Fig 5: Focus around the airfoil

3. Solver Settings

The boundaries conditions to the mesh were set as velocity inlet at the entry and as the end boundary or outlet.

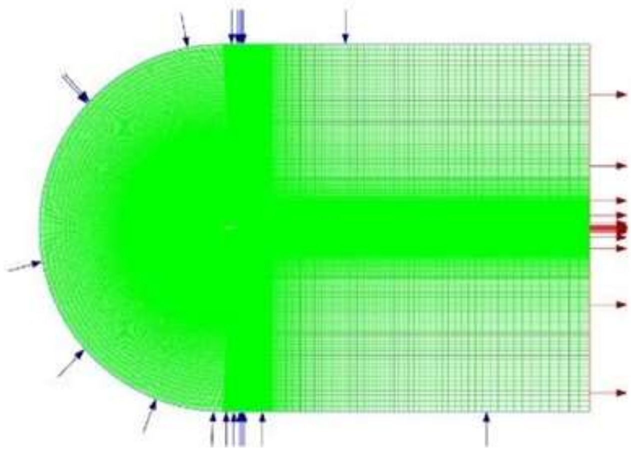


Fig 6: Boundary conditions

Table1: Boundary conditions of fluid domain

Name	Condition Applied
Inlet	Velocity
Outlet	Pressure

These settings provide valid results in this study. The convergence of flow is determined by the order of magnitude of the residuals. The decrease of all scaled residuals below 1×10−6 is used as the convergence criterion. The maximum number of iterations is set as 1000, which is sufficient for the residuals to attain convergence in the simulation.

Table 2: Solver settings applied for simulation

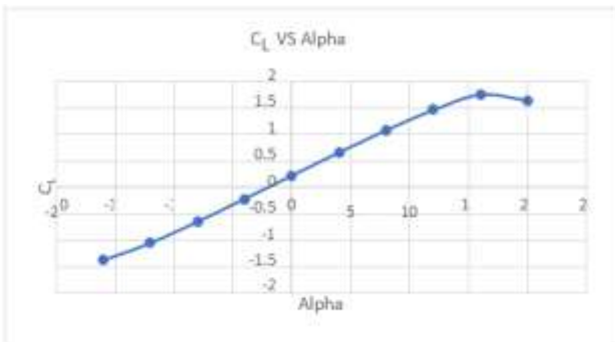
Description	Setting used
Pressure–velocity coupling scheme	Coupled
Gradient algorithm	Least squares cell based
Pressure interpolation scheme	Standard
Momentum interpolation scheme	Second Order Upwind
Turbulent kinetic energy interpolation scheme	Second Order Upwind
Turbulent dissipation rate interpolation scheme	Second Order Upwind

Table 3: Working Parameters in Ansys Fluent

Working Fluid	Air
Density of Air	1.225 kg/m3
Viscosity of Air	□.□□□□□*10-5 kg/m-s
Solver Type	Pressure -Based
Viscous Model	SST k-ω, Realizable k-ε, Spalart–Allmaras
Pressure -Velocity Coupling	Coupled
Momentum	Second Order Upwind
Analysis Type	Steady State
Fluid Velocity	65.45/s

IV. RESULTS AND DISCUSSION

The influence of turbulence model selection was investigated based on the prediction of flow and aerodynamic performance to validate the method used in the study. Accurate turbulence modelling is necessary to predict the performance of airfoil. The selection of the proper turbulence model is an important problem to be solved in CFD. This method is a matter of deliberation to find the right turbulence model for specific problems. All three turbulence models were selected for the analysis to assess the best suitable turbulence model for predicting the correct aerodynamic performance.



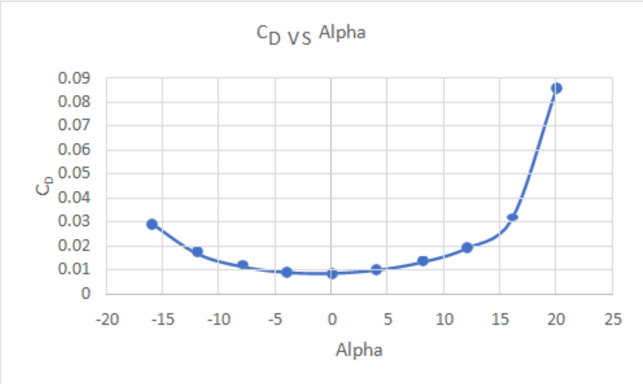
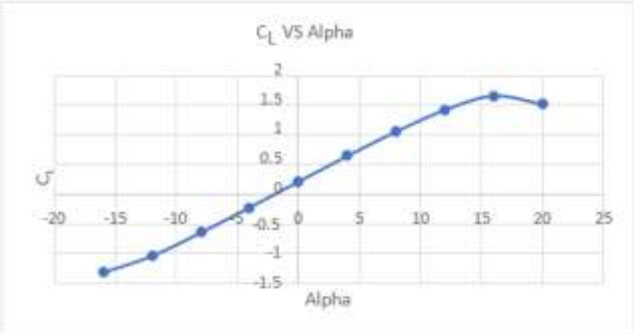


Fig 7: SA (Spalart–Allmaras) Turbulence model comparison of CL, CD vs. Alpha



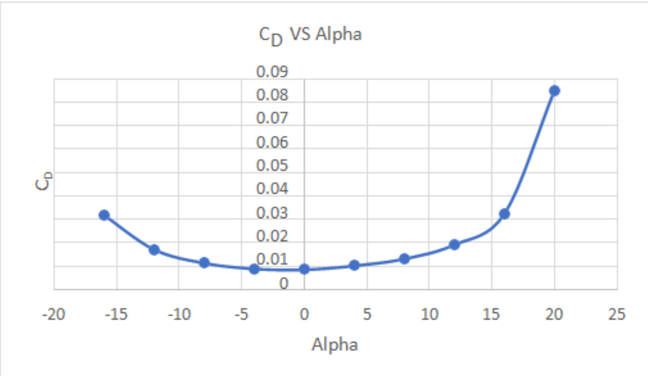
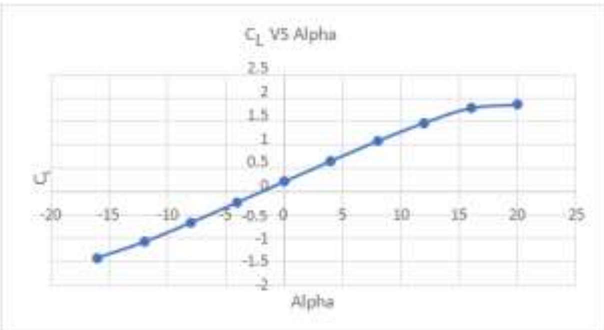


Fig 8: SST k-ω Turbulence model comparison of CL, CD vs. Alpha.



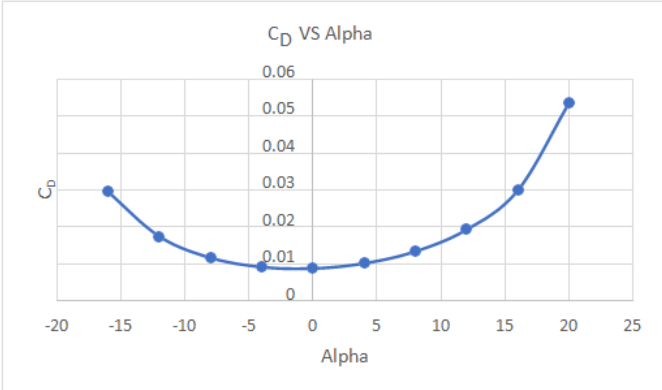
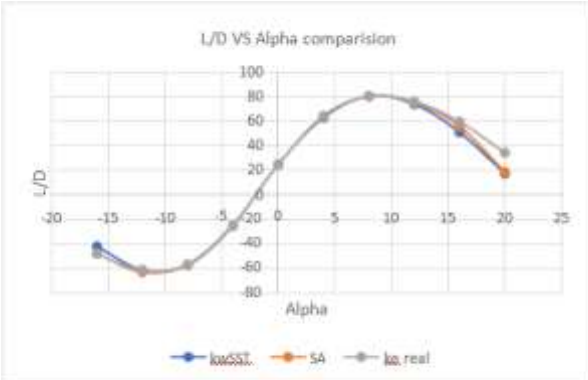


Fig 9: Realizable k-ε Turbulence model comparison of CL, CD vs. Alpha.

The lift-to-drag ratio is a useful measure of an airfoil's performance. A high lift-to-drag ratio at any angle of attack means lower drag and higher efficiency, resulting in better fuel economy and climb performance for the aircraft. Figures 8 and 9 show the values and ratios of the lift and drag coefficients. Increasing the angle of attack increases lift, but this only applies up to the stall or critical angle of attack. Beyond this point is the stalled region. The 2412 downward tail airfoil has a higher lift-to-drag ratio compared to other designs, as shown in Figure 10. The best angle of attack for the 2412 downward tail is 8 degrees.



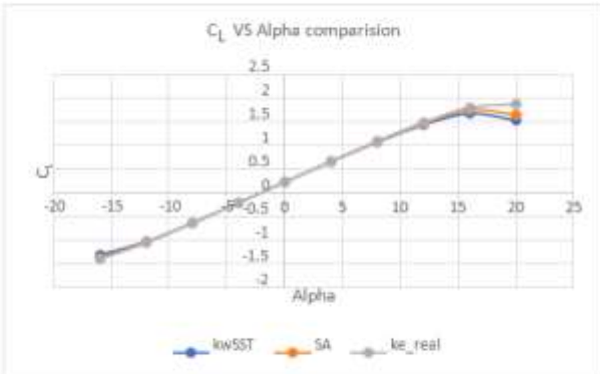


Fig 11: CL vs. Alpha comparison for all Turbulence models.

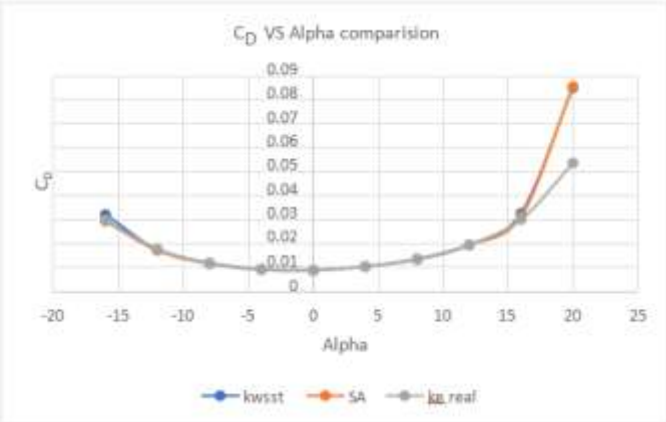


Fig12: CD vs. Alpha comparison for all Turbulence models.

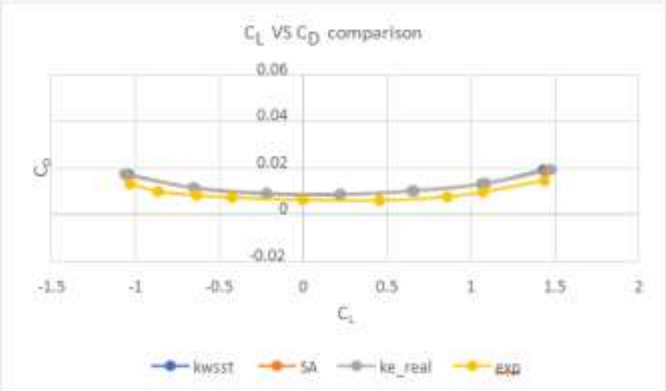


Fig 13: CL vs. CD for Realizable k-ε, SA, SST k-ω Turbulence models and Experimental values.

Table 4: CL and CD Experimental values

CL Exp	CD Exp
-1.03158	0.013197
-0.86316	0.009929
-0.63509	0.00834
-0.42456	0.007319
-0.00351	0.006298
-0.452633	0.006128
0.856142	0.00766
1.066667	0.009702
1.435088	0.014638
1.592982	0.018383

V. CONCLUSION

The numerical results obtained closely match the experimental data for most angles of attack, demonstrating that the CFD method can replicate experimental results with acceptable accuracy.

The following conclusions were drawn:

- Lift and drag forces increase with the angle of attack until reaching the stall point.
- The optimal angle of attack is 8 degrees. At this angle, the NACA 2412 airfoil with a downward tail achieves the highest lift-to-drag ratio, indicating ideal performance compared to other angles.
- The stall angle of attack for this study is 16 degrees. Beyond this angle, the lift force decreases, indicating the stall region.

This study used three turbulence models (Spalart-Allmaras, Realizable k-ε, and SST k-ω) to simulate the aerodynamic performance of the NACA 2412 airfoil. The lift and drag coefficients from the simulations were compared with wind tunnel experimental data. The simulations covered angles of attack ranging from -16 to 20 degrees. The optimal angle of attack was confirmed to be 8 degrees, where the airfoil achieved the maximum lift-to-drag ratio. The critical stall angle was found to be 16 degrees.

REFERENCES

1. "NACA Overview". history.nasa.gov. Retrieved April 20, 2022.
2. http://airfoiltools.com/polar/details?polar=xf-naca2412-il-1000000.
3. The Element of Aerofoil and Airscrew theory By H. Glauert
4. S.Shivananda, and B. Shaheen , "CFD analysis of effect of flow over naca 2412 airfoil through the shear stress transport turbulence model,", (International Journal of Mechanical And Production Engineering, 2017),Vol.5, pp 58-62.
5. Alex Sullivan Cleveland, F. A., "Aerodynamic forces acting on an airfoil"
6. https://courses.ansys.com/index.php/courses/flow-over-an-airfoil/
7. 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.
8. Argyropoulos, C., and N. Markatos., "Recent advances on the numerical modelling of turbulent flows". Applied Mathematical Modelling,2015. Volume 39,pp 693–732.
9. https://www.researchgate.net/publication/353244238_Numerical_Validation_of_NACA_0009_Ai rfoil_in_Ultra-Low_Reynolds_Number_Flows
10. P. S. Kapsalis, S. Voutsinas, and N. S. Vlachos, Comparing the effect of three transition models on the CFD predictions of a NACA0012 airfoil aerodynamics,

Jnl. Wind Eng. Ind. Aerodyn., vol. 157, pp. 158–170, 2016.
11. H. Yang, W. Shen, H. Xu, Z. Hong, and C. Liu, Prediction of the wind turbine performance by using BEM with airfoil data extracted from CFD, Renew. Energy, vol. 70, pp. 107–115, 2014
12. CFD simulation of NACA airfoilsat various angles of attack - IOPscience
13. Houghton, E. L.; Carpenter, P.W. Butterworth Heinmann (ed.). Aerodynamics for Engineering Students (5th ed.).2003. p. 17.