Vedant Chaudhari, 2024, 12:4
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

International Journal of Science,
Engineering and Technology

An Open Access Journal

Study the Effects of Trailing Edge Geometry on Airfoil Performance

Vedant Chaudhari
Department of Aerospace Engineering,
School of Engineering and Sciences, MIT Art, Design and Technology University, Pune

Abstract- The primary objective of this paper is to investigate the impact of trailing edge geometry on airfoil performance. The chosen airfoil, such as NACA 652415, serves as the basis for four distinct cases, each involving variations in the trailing edge. This leads to alterations in the airfoil geometry and NACA series. In the first case, the trailing edge is configured to be sharp. In the second case, the trailing edge is rounded by inscribing a circle with a specific radius. The third case involves considering a circumcircle at the trailing edge point and drawing tangents connecting the airfoil. The fourth case introduces changes to the dimensions of the circle. The project utilizes IMIME software for meshing and CFD++ for CFD analysis, both developed by Metacomp Technologies. The analysis is centred around examining key aerodynamic parameters, including the coefficient of lift, coefficient of drag, pitching moment, drag polar, coefficient of pressure, and the location of the Centre of pressure. These investigations are conducted across a range of angles of attack to comprehensively understand the airfoil's performance under varying conditions. This investigation utilizes IMIME software for meshing and CFD++ for CFD analysis, both developed by Metacomp Technologies.

Keywords- NACA 652415, Trailing edge, Cl vs alpha, L/D ratio, Edge thickness

I. INTRODUCTION

In the realm of aerodynamics, the pursuit of accurate predictions for aerodynamic characteristics, namely lift coefficient (c_l), drag coefficient (c_d), and pitching moment coefficient (c_m), is a fundamental goal. Theoretical aerodynamics aims to derive these values through the application of basic physical science equations. However, the complexity of aerodynamic flows often necessitates the use of simplifying assumptions, introducing a degree of approximation into theoretical results.

Recent advancements in computational capabilities, specifically the utilization of high-speed digital computers, have brought us closer to achieving precise calculations of aerodynamic characteristics. Nevertheless, challenges persist due to limitations imposed by numerical methods and the current capacity of computing systems. As a consequence, practical aerodynamics frequently relies on direct experimental measurements of c_l, c_d, and c_m for specific aerodynamic bodies.

Central to understanding and optimizing aircraft performance is the concept of the lift-to-drag ratio (L/D), a key efficiency factor. Achieving a high L/D is paramount for enhancing the gliding capabilities of aircraft, influencing parameters such as glide angle and horizontal travel distance per vertical unit of descent. At subsonic speeds, L/D is predominantly influenced by wing span and wetted area. This research endeavours to analyze and optimize airfoil performance, with a particular focus on the trailing edge geometry's impact. The selected airfoil, exemplified by the NACA 652415, serves as the baseline for four distinct cases, each involving variations in the trailing edge.

Employing computational fluid dynamics (CFD) analysis through IMIME software and CFD++, developed by Metacomp Technologies, this project delves into crucial aerodynamic parameters, including lift and drag coefficients, pitching moment, drag polar, coefficient of pressure, and the location of the centre of pressure. By exploring these parameters across various angles of attack, the study aims to comprehensively understand the airfoil's performance under diverse conditions, offering valuable insights into optimizing aerodynamic efficiency.

II. PROBLEM DEFINITION

Generally, the airfoil data is calculated on a sharp tailing edge. But in a practical scenario during manufacturing, it is not possible to make a sharp trailing edge airfoil.

In this project, we have considered the NACA 652415 airfoil. In this project, the geometry of the standard airfoil is varied to change the trailing edge thickness of the airfoil. Hence the aerodynamic characteristics of the airfoil are calculated.

III. SOLUTION / APPROACH

1. Case One

In the first case, the sharp-edge airfoil is considered. The first is practically not possible.

Figure 1- Case One Airfoil

2. Case Two

In case two the trailing edge is made round by inscribing a circle of a particular radius.

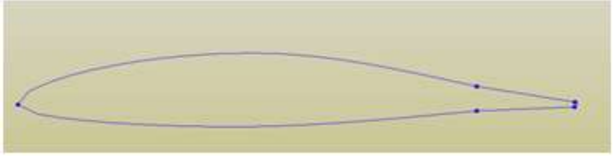


Figure 2- Case Two Airfoil

3. Case Three

In case three a circumcircle at the trailing edge point is considered and tangents are drowned connecting the airfoil.

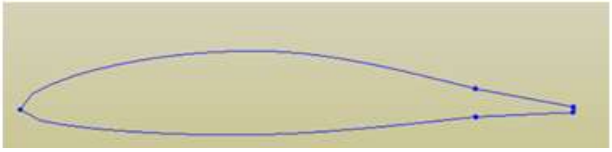


Figure 3- Case Three Airfoil

4. Case four

In case four the dimensions of the circle is changed.

Figure 4- Case Four Airfoil

IV. SIMULATION

1. Geometry Designing

We have designed the NACA 652415 from the standard geometry. There are different types of series NACA airfoils. In this project, we have used NACA 6-digit series airfoil. The digits in the NACA airfoils are denoted as follows:

- The first digit denotes the series.
- The second digit denotes the minimum in tenths of the chord
- 2 is the subscript
- The third digit denotes the design lift coefficient
- The fourth and fifth digit denotes the thickness of the airfoil in per cent of the chord

2. Meshing

IMIME (Integral Multi-purpose Intelligent Meshing Environment) is a versatile software for generating high-quality 2D and 3D meshes for various engineering simulations. It offers flexibility in element types and mesh refinement, and leverages a user-friendly interface with an intelligent engine to guide even novice users through the mesh creation process. This intuitive software, built on the ICMP framework, simplifies mesh generation for various applications like CFD, CAA, CSM, and EDD.

Figure 5- Meshing of Airfoil

3. Flow Conditions

The general fluid dynamics problem under consideration involves the simulation of compressible gas flow, specifically a single- species perfect gas. The flow is characterized as being mostly below Mach 0.5, making it subsonic. The fluid is viscous and turbulent, with the chosen simulation type being steady-state.

The governing equations for the simulation include the Preconditioned/Pressure-Based Compressible PG NS/Euler equations, and a 2-equation Realizable k-epsilon turbulence model is employed. The solution methodology incorporates preconditioning, and the simulation will focus on the behavior of the flow using specified physics source terms.

The fluid properties are defined for air, with an initial condition set at a base pressure level of 101325.0 Pa. The initialization employs temperature-based primitive variables, including pressure, temperature, and velocity components in the x and y directions. The boundary conditions are established for different regions, with an airfoil designated as a wall with viscous characteristics, utilizing a wall function and adiabatic wall heat transfer. The far-field condition is open, falling under the "Inflow and Outflow" category, and is characterized by primitive variables specified through characteristics-based conditions.

Numerical parameters are set based on the physics regimes, considering the flow as subsonic, external, and with an approximate system velocity of 32.0 m/s. The simulation involves 3000 global steps, with a convergence criterion of 10.0 for run termination. The local time step is ramped from 1 to 50 based on the Courant number. The turbulence length scale is known, with a specified free stream turbulence level of 0.002 and a turbulence length scale of 0.1.

The chosen output options for the simulation include parameters related to turbulence, such as turbulent eddy viscosity and the ratio of turbulent to laminar viscosity. Overall, this comprehensive set of conditions and specifications provides a detailed framework for the computational fluid dynamics problem at hand. Subsequent Pages

For pages other than the first page, start at the top, and continue in double-column format. The two columns on the last page should be as close to equal length as possible.

V. SOLVER

CFD++ is a highly flexible Computational Fluid Dynamics software suite, guided by three key principles: unified-grid, unified- physics, and unified-computing. It seamlessly handles diverse cell shapes and grid topologies, surpassing conventional grid transparency to manage arbitrarily non-aligned or overset meshes.

The unified-physics treatment enables the software to solve various governing equation sets for flows ranging from incompressible to hypersonic, with optional modules for reactions, turbulence, and acoustics. Its unified-computing capability ensures portability across different platforms, including multi-CPU machines.

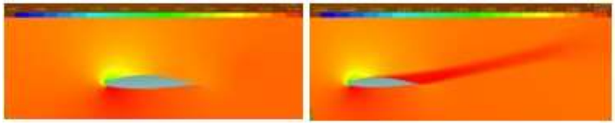
Major features include solving compressible and incompressible Navier- Stokes equations, unifying Cartesian, curvilinear, and unstructured grids, diverse turbulence models, robust boundary conditions, multi-grid relaxation, and automatic binary-file conversion for platform versatility. CFD++ offers a comprehensive and adaptable framework for fluid dynamics simulations. In the CFD simulation angle of attack(α) is provided from the range of - 4 degrees to 18 degrees and beta is considered as 0 for all cases.

VI. POST-PROCESSING

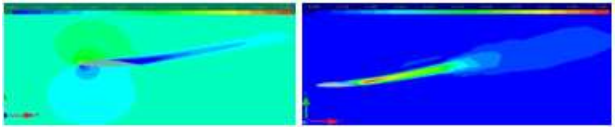
Post-processing in CFD distils actionable insights from simulation data, employing visualization tools like contour plots and flow animations. It validates results against experimental data, ensuring accuracy and reliability in predicting fluid dynamics. Engineers delve into detailed analyses, scrutinizing parameters such as turbulence and heat transfer, guiding optimization efforts for improved design performance. This iterative process, fuelled by post-processing, enhances understanding and aids in making informed decisions for more effective engineering solutions.

1. Case One

1. pressure distribution 2.temperature distribution

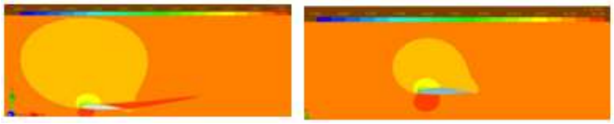


3. VelMag distribution 4. mut_ovr_mu

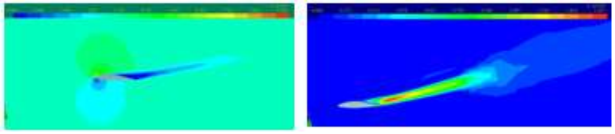


2. Case Two

1. pressure distribution 2. temperature distribution

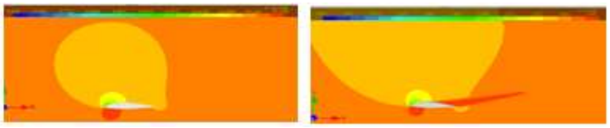


3. VelMag distribution 4. mut_ovr_mu

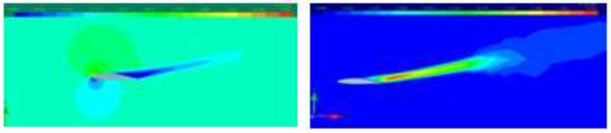


3. Case Three

1. pressure distribution 2. temperature distribution

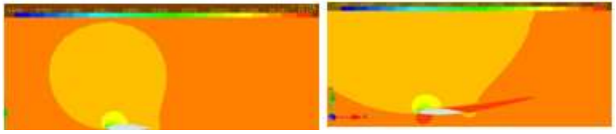


3. VelMag distribution 4. mut_ovr_mu



4. Case Four

1.pressure distribution 2. temperature distribution



3. VelMag distribution 4. mut_ovr_mu

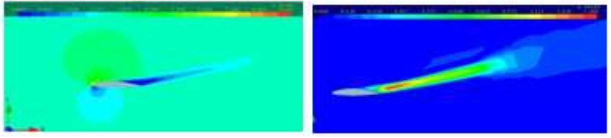


Figure 6 – Contours

VII. RESULT AND DISCUSSION

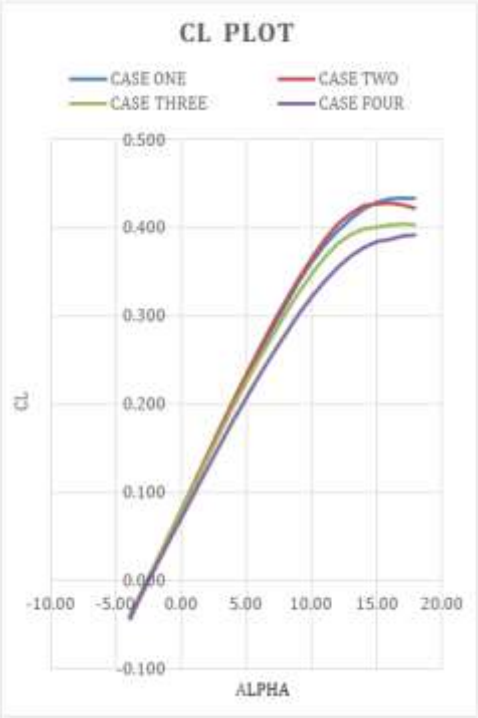


Figure 7: Coefficient of lift VS Angle of Attack

The graphical representation reveals that the second airfoil configuration exhibits superior performance compared to cases three and four. Notably, the first case, featuring a sharp trailing edge airfoil, and the second case both demonstrate higher lift generation in relation to alpha when compared to the remaining cases. It is evident that the curves for cases one and two closely parallel each other, showcasing only marginal variations.

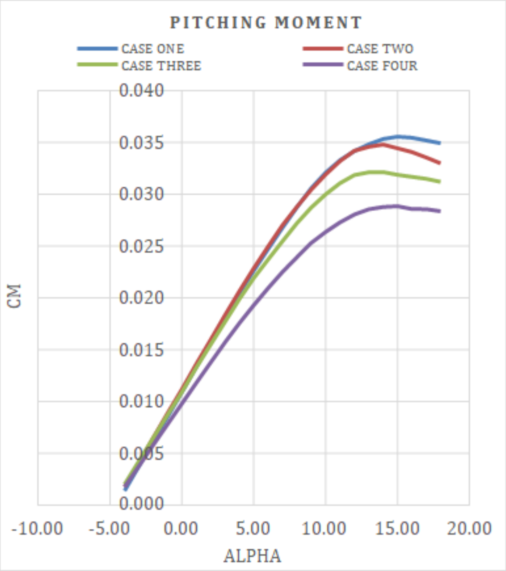


Figure 8: Pitching Moment VS Angle of Attack

This plot shows the Coefficient of pitching moment with respect to alpha. Here we see that the stability of case two in better than other airfoils.

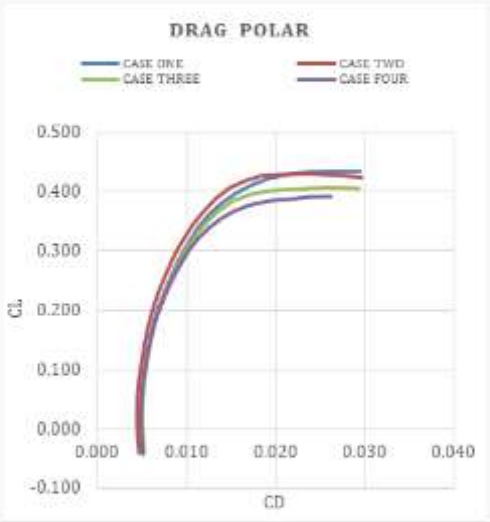


Figure 9: Coefficient of lift VS Coefficient of Drag

Here also case one and two are performing better generating greater lift than other cases. After the Cd reaches 0.0020 the graph is in steady state.

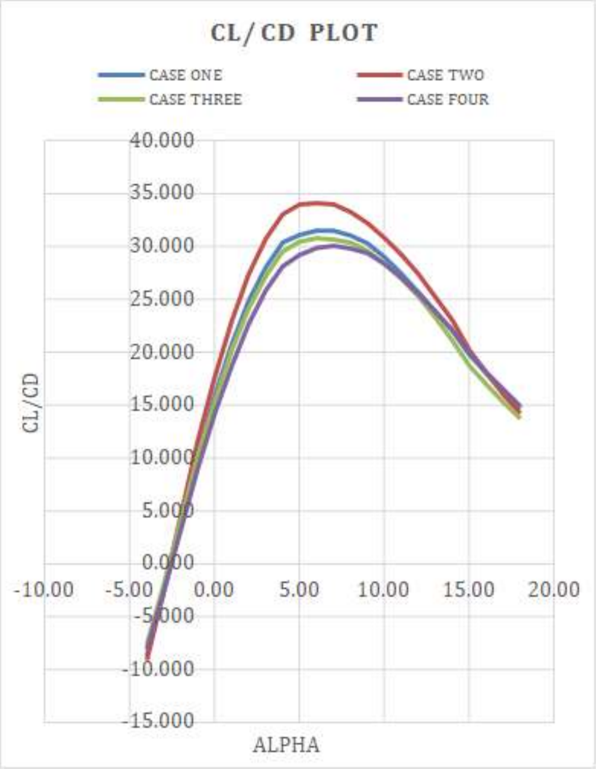


Figure 10: L/D ratio VS Angle of Attack

This graph clearly shows the best airfoil is case two. Higher the L/D ratio higher the efficiency and performance. The airfoil is generating less drag w.r.t. alpha.

V. CONCLUSION

Hence, this project studies the effects of trailing edge geometry on airfoil performance. This project studies the aerodynamic characters of four cases of different geometry. From the graphs, the results can be predicted that as the tailing edge thickness of airfoil increases the efficiency of the airfoil decreases. Hence the sharp trailing edge airfoil performs best; but it is impossible to manufacture sharp tailing edge airfoil. So the airfoil is made round at the trailing edge. The airfoil which has highest Cl vs alpha curve is good for generating high lift which is necessary for quick take off. The airfoil which has highest L/D ratio is most efficient for cruise performance. There are many criteria for airfoil selection like mission profile, speed range, design constraint, etc. In this project the performance of case two airfoil is the best.

REFERENCES

1. Fundamentals of Aerodynamics" by John D. Anderson
2. Introduction to Flight" by John D. Anderson.
3. Theory of wing sections by Ira H. Abbott, Albert e. Von Doenhoff