The International Journal of Energy & Engineering Sciences, 2020, 5 (2) 146-155
ISSN: 2602-294X - Gaziantep University

AERODYNAMIC OPTIMIZATION OF NACA 0012 AIRFOIL

İbrahim Göv[1*] and M. Hanifi Doğru[2]

[1]Aerospace Engineering Dept., Aeronautics and Aerospace Faculty, Gaziantep, Turkey
[2] Pilotage Dept., Aeronautics and Aerospace Faculty, Gaziantep, Turkey

igov@gantep.edu.tr

ABSTRACT

Airfoils are used to produce the lift force to carry the weight of the aircraft. Aerodynamic forces are directly related to an airfoil shape. The force parallel to the direction of relative motion is defined as drag, and the force perpendicular to the direction of movement is defined as lift. Hence, the airfoil shape is directly affecting the aerodynamic performance of the wings. In this study, aerodynamic optimization of an airfoil is performed using computational fluid dynamic (CFD) analysis to increase the aerodynamic performance of the wings. The commonly used NACA 0012 Airfoil is optimized at a different angle of attack values using a commercial CFD program ANSYS. Approximately 5% and 10% improvement were achieved at 5^0, 10^0, and 15^0 AoA. Approximately 10% and 15% improvement were achieved at 20^0 AoA. There is almost no change in CD values at all AoA values.

Keywords: aerodynamic, optimization, CFD

1. INTRODUCTION

The use of the Computational Fluid Dynamics (CFD) tool in aviation has made it easier to understand fluid dynamics and aerodynamic phenomena in the last decade [1]. Also, numerical simulations have emerged as a critical and constantly evolving aspect of the aircraft design process. Thanks to CFD, dependency on wind tunnel investigations are reduced so consequently design cost is also reduced. As a result of these developments, wing performances are increased using CFD tools.

Wings are surfaces designed to move in the air and obtain an aerodynamic force perpendicular to the direction of movement. The most efficient devices designed to obtain lift force are wings. The Lift-drag ratio is one of the most important parameters for wings. This means that a significant amount of lift can be achieved with a smaller thrust [2].

The Angle of Attack (AoA) is defined as the angle formed between the air coming into the wing and the chord line of the wing (in figure 1).

The drag equation: $F_D = \frac{1}{2}\rho v^2 C_D A$ where ρ is density, A is the area and v is the velocity. So, the equation $C_D = \frac{F_D}{\frac{1}{2}\rho v^2 A}$, expresses the drag coefficient. In fluid dynamics, C_D is a dimensionless quantity used to measure the resistance of an object in a fluid medium.

The lift equation: $F_L = \frac{1}{2}\rho v^2 C_L S$ where S is surface area.
The lift coefficient is given as: $C_L = \frac{F_L}{\frac{1}{2}\rho v^2 S}$.
When fluid flows over the surface of an object, force is exerted on the object. This force that occurs perpendicular to the direction of the incoming flow is defined as the drag force parallel to the lifting force. If the environment is air, these forces are called aerodynamic forces.

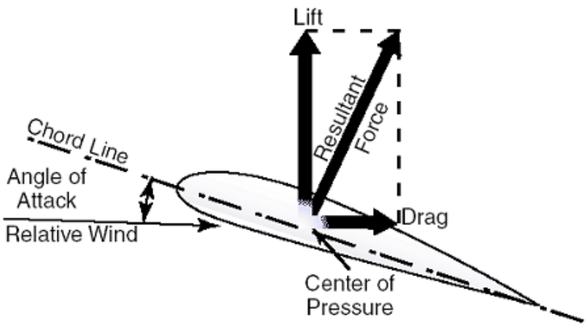

Figure 1. Lift force, drag force, and angle of attack [2]

In the literature, many different studies exist about the aerodynamic force analysis. NACA 0012 airfoil is chosen to model and simulate processes of computational fluid dynamics.

Analyzes to obtain velocity and pressure distribution on the wing surface were performed using ANSYS Fluent software. Also, the drag and lift coefficients were calculated according to variable relative velocities. It was observed that the results obtained coincide with the theoretical results [3].

Static analysis is a type of analysis applied to determine the stresses resulting from the applied loads. Random vibration analysis and modal analysis were used to determine stresses due to frequencies. The buckling analysis was applied to determine the load factor and deformation. CFD analysis was performed to determine drag and lift forces. According to the analysis results, it was concluded that the drag force was modified better than the original model [4].

Lift and drag forces can be calculated by CFD analysis as well as by wind tunnel tests. A two-dimensional subsonic flow analysis study was carried out for the NACA 0012 airfoil in different AoAs and 3 × E + 06 Reynolds number. Analysis and simulation results match each other. Therefore, the analyzes proved their accuracy as an alternative to experimental methods [5].

The distribution of velocity over the shell is a critical specification for the bullet's range and penetration concept. Three different bullet tip shapes were examined. As a result of the study, velocity and pressure distribution on bullets with different tip geometries were obtained [6].

CFD analysis was performed for NACA 23024 airfoil using Ansys Fluent solver. In the analysis, the K-omega SST turbulence model, Spalart Allmaras turbulence model, and Standard K-Epsilon Turbulence model were used as the turbulence model. When the results obtained from the analysis were compared with the experimental wind tunnel results, it has been observed that the results were in good agreement with each other [7].

Flow separation on the airfoils NACA 4412 and S809 were compared to each other. The aerodynamic performance of these two-airfoil profiles was compared at a different angle of attack (between 0^0 and 20^0) values. Drag coefficient (CD), lift coefficient (CL), moment coefficient (CM), and flow separation were used as a performance parameter [8].

The shape of the wing profile (by aerodynamic design requirements) was changed during the flight to achieve maximum performance at different angles of attack. In the study, 2-dimensional computational fluid dynamics analysis was used based on NACA 4412 airfoil. To achieve higher aerodynamic performance at different angles of attack during flight, two different wing profiles (NACA 4412_1 and NACA 4412_2) were obtained by changing the NACA 4412 profile and the aerodynamic performances of these two profiles were compared with the original NACA 4412 profile. The lift coefficient, drag coefficient, and attachment loss performance parameters were examined in these analyzes [9].

2. NUMERICAL STUDY

Description of the geometry model

NACA 0012 airfoil (in figure 2) which is symmetric is used in the analyses.

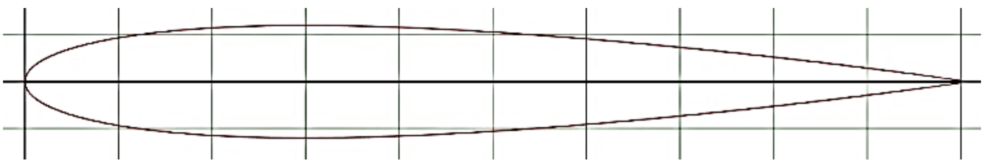

Figure 2. NACA 0012 airfoil profile

Mesh Generation

The analysis is performed using the CFD tool of ANSYS which is a commercial finite element analysis program. Complete mesh distribution can be seen in figure 3, an enlarged view of NACA 0012 mesh is given in figure 4. Analysis parameters are listed in table 1.

Table 1. Analysis Parameters

Node	11279
Elements	11029
Solver	Pressure based steady-state
Viscous model	k-epsilon
Density (kg/m3)	1.225
Turbulent viscosity ratio	10
Inlet velocity (m/s)	1
Chord-length (m)	1
Momentum	Second-order upwind

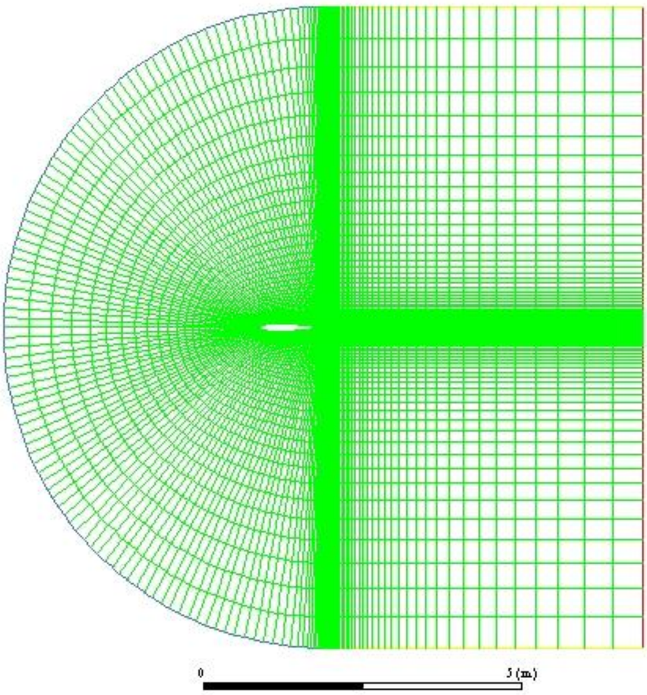

Figure 3. Complete mesh distribution

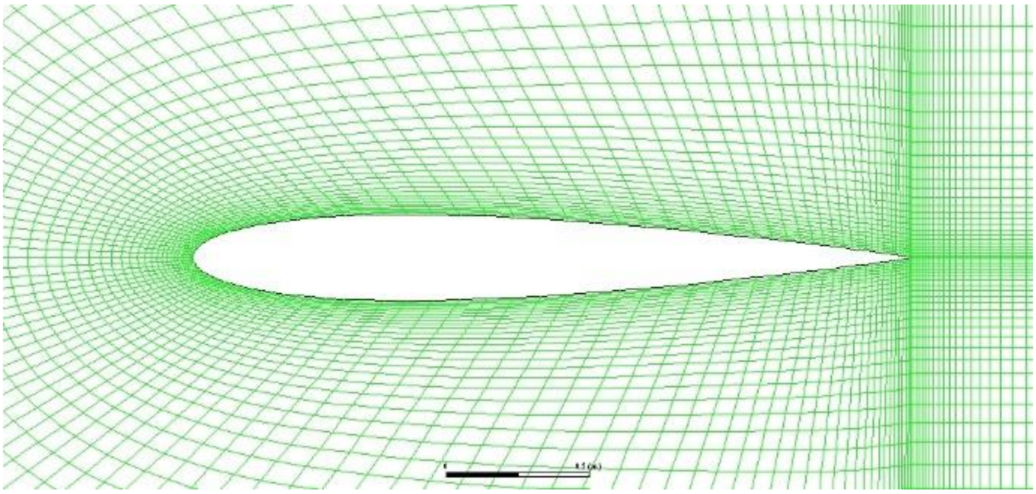

Figure 4. Enlarged view of the mesh

Numerical Method and Boundary Conditions

ANSYS FLUENT software which is based on the finite volume method was used to perform numerical calculations. Gravity is to be neglected. Calculations were set as pressure-based, and steady-state solution. Standard k-e, Standard Wall Func. is chosen as the turbulence model.

Momentum modeling is done by using second-order functions. The use of higher-order functions increases the accuracy of the results. But the processing time is higher than for low order function solutions. Different boundary conditions can be determined for different regions. The density of the air is 1.225kg/m3, and the kinematics viscosity of the air is 1.7894e-5. The input velocity of 1 m/s was assumed. The defined parameters above are used to perform simulations.

For the optimization, ANSYS Adjoint Based Optimization tool is used. The adjoint equation is a linear differential equation. They are usually derived from the derivation of primary equations. Additional equation solutions are used to calculate gradient values efficiently. Methods based on this equation solution are used in fluid flow control, uncertainty measurement, and blade shape optimization. For example:

$$dX_t = a(X_t)dt + b(X_t)dW \tag{1}$$

this is an Itō stochastic differential equation. Now by using the Euler scheme, we integrate the parts of this equation and get another equation,

$$X_{n+1} = X_n + a\Delta t + \zeta b\sqrt{\Delta t} \tag{2}$$

here ζ is a random variable, later one is an adjoint equation [10].

3. ANALYSIS RESULTS

In this part, lift and drag coefficients of NACA 0012 (figure 2) airfoil are calculated using CFD analysis at different AoA values.

Static pressure results at 0^0 and 20^0 AoA values are given in figures 5 and 6 respectively. Velocity results at 0^0 and 20^0 AoA values are given in figures 7 and 8 respectively. Also, optimization of NACA 0012 as 10%, 20%, 30%, and 40% improvement is applied to the airfoil. And their CL, CD, and CL/CD results are given in figures 9, 10, and 11 respectively.

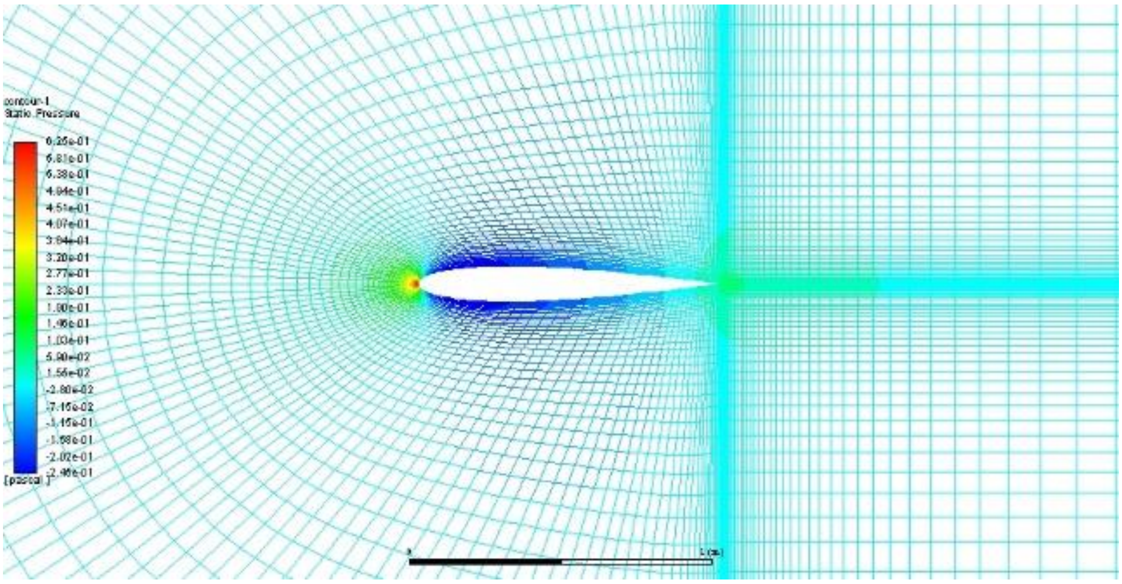

Figure 5. Static pressure at 0^0 AoA

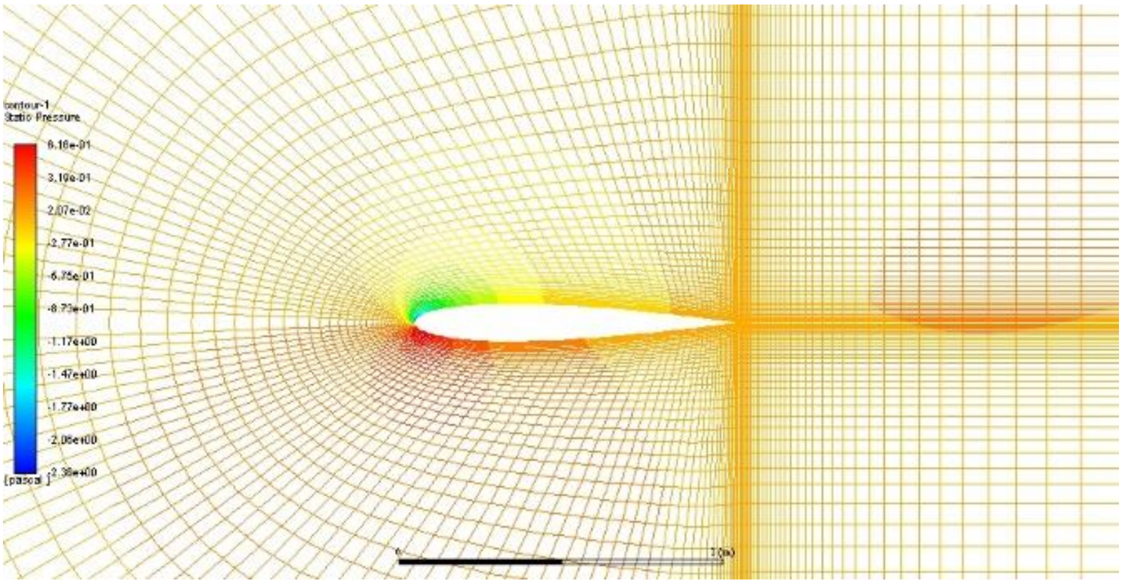

Figure 6. Static pressure at 20^0 AoA

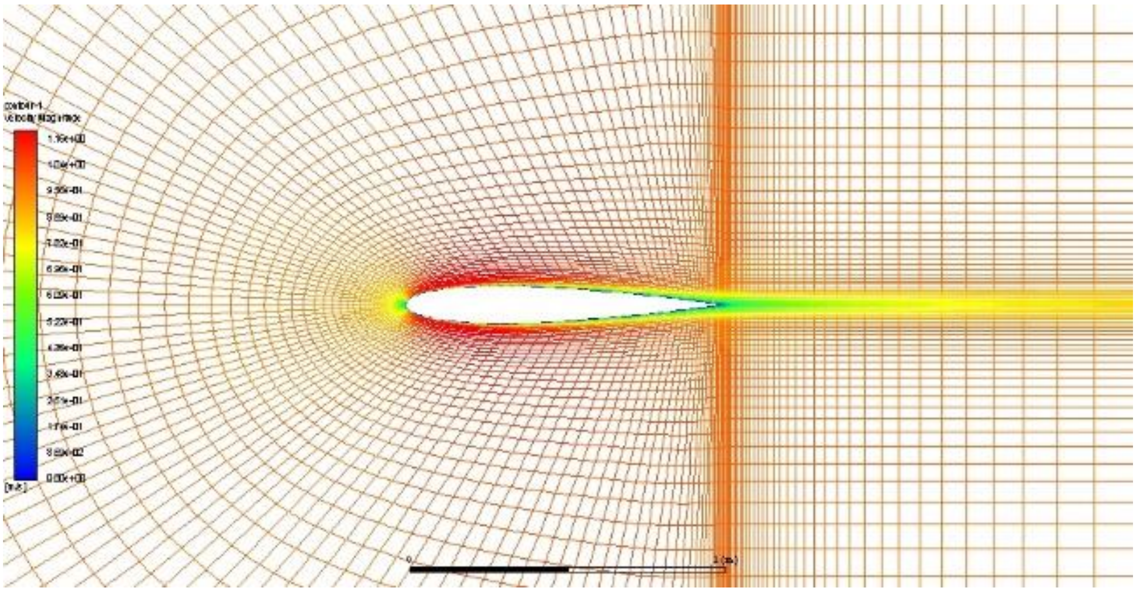

Figure 7. Velocity at 0^0 AoA

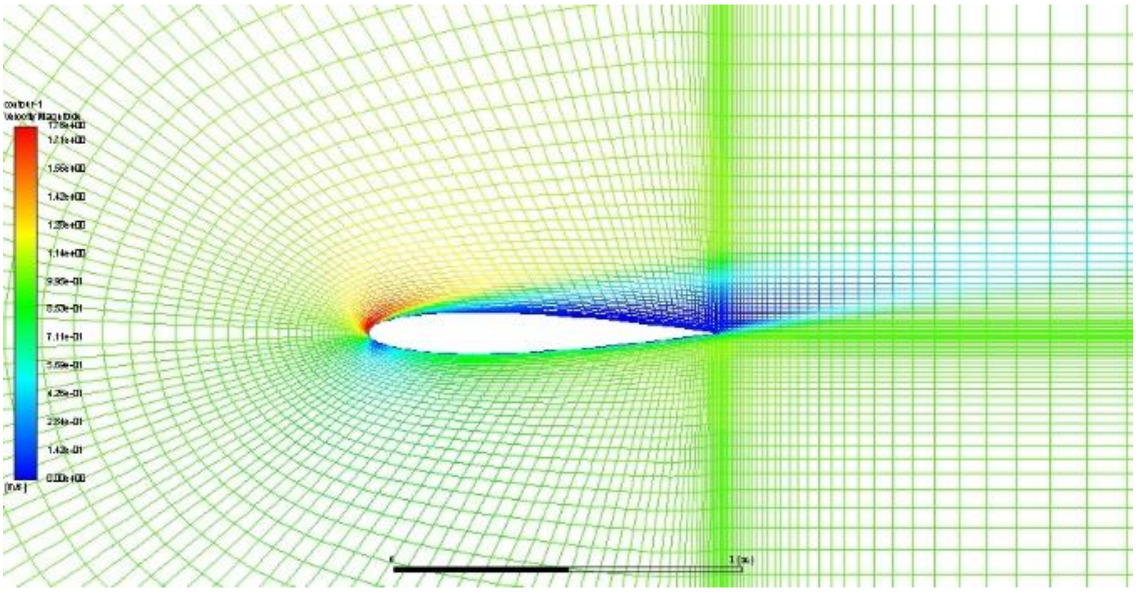

Figure 8. Velocity at 20^0 AoA

The International Journal of Energy & Engineering Sciences, 2020, 5 (2) 146-155
ISSN: 2602-294X - Gaziantep University

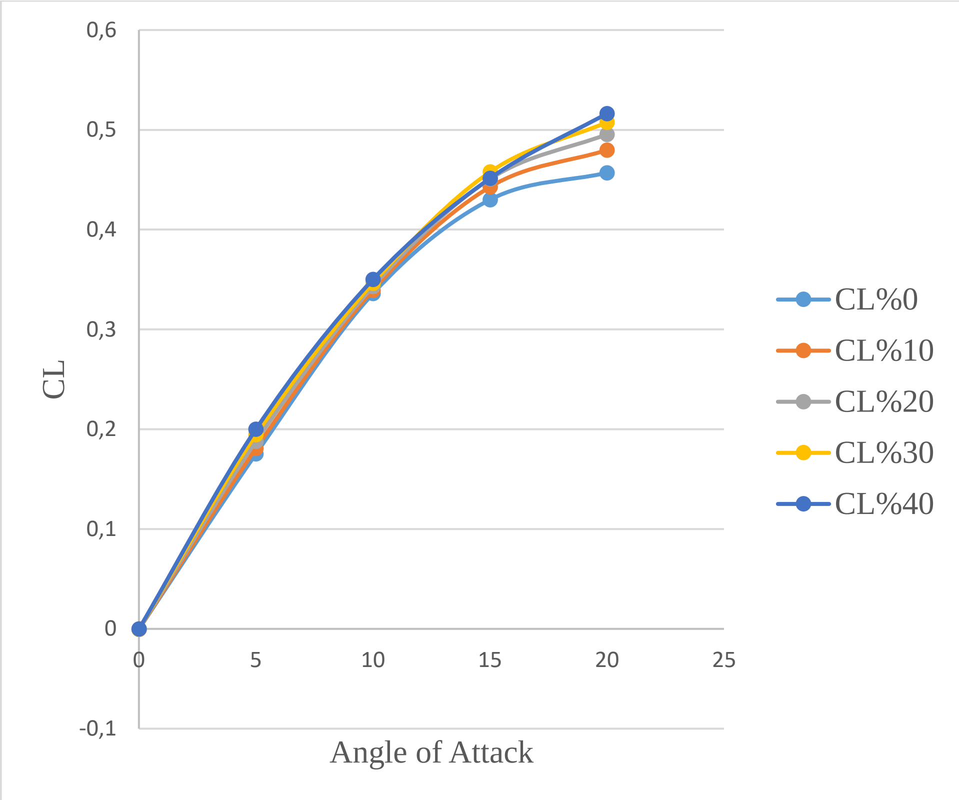

Figure 9. CL Results of Optimized NACA 0012 at different AoA values

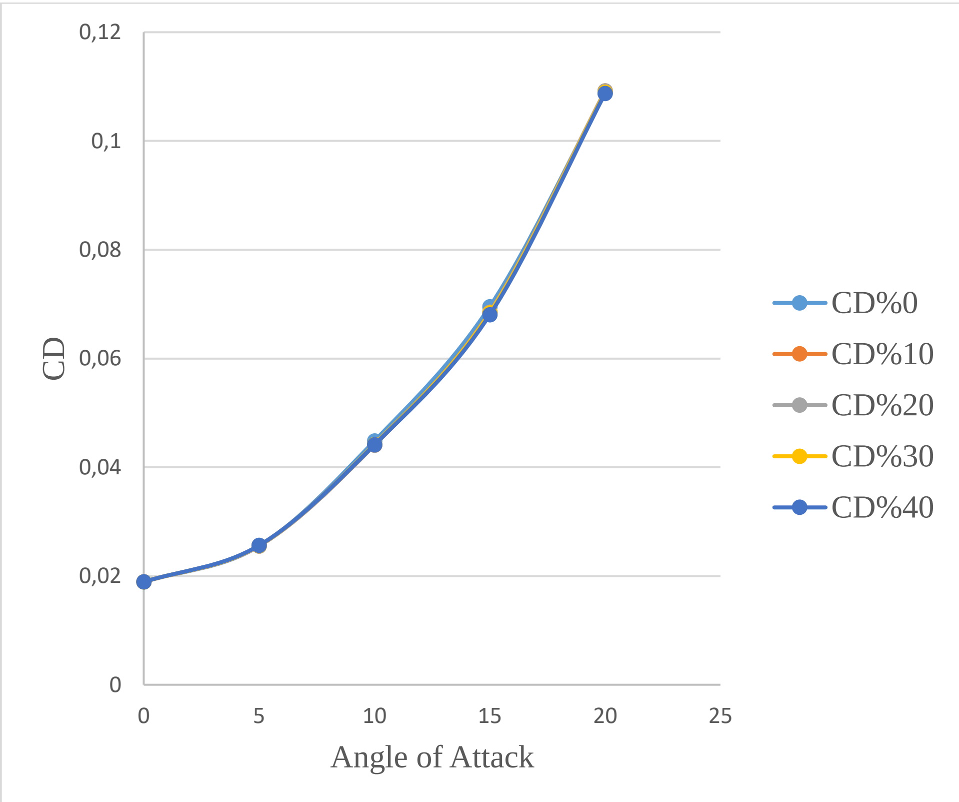

Figure 10. CD Results of Optimized NACA 0012 at different AoA values

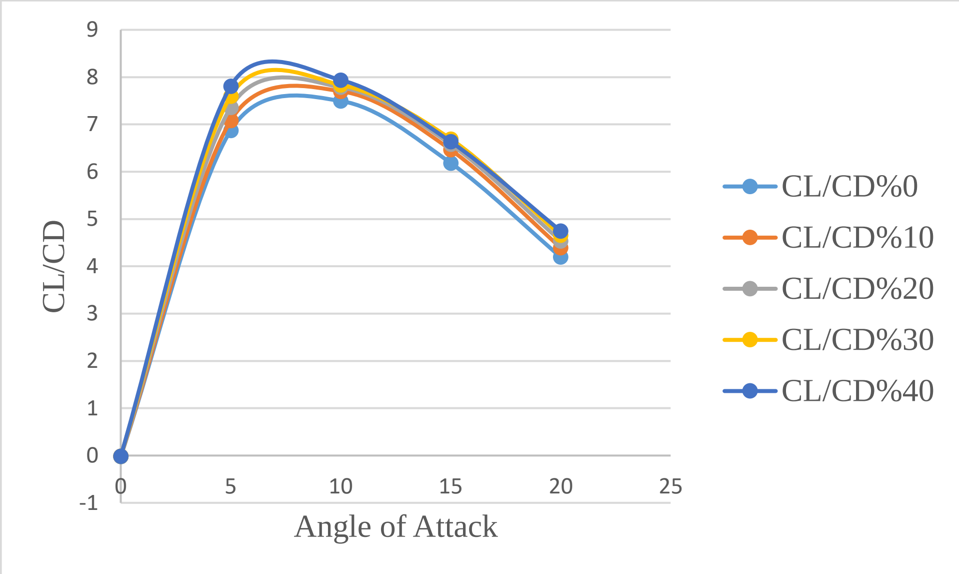

Figure 11. CL/CD Results of Optimized NACA 0012 at different AoA values

4. CONCLUSION

In the design stage of an aircraft wing airfoil shape is the most critical parameter for the aerodynamic performance of the aircraft. Hence in this study, aerodynamic optimization of an airfoil is performed using computational fluid dynamic (CFD) analysis to increase the aerodynamic performance of the wings. The commonly used NACA 0012 Airfoil is optimized at a different angle of attack values using a commercial CFD program ANSYS.

When the results were investigated, approximately 5% and 10% improvement were achieved at 5^0, $10^{0,}$ and 15^0 AoA. Approximately 10% and 15% improvement were achieved at 20^0 AoA. There is almost no change in CD values at all AoA values. Similar changes were obtained for CL/CD with CL values. There were no sufficient changes were obtained for all 0^0 AoA values because at 0^0 AoA values lift force was not produced for symmetric airfoils.

REFERENCES

[1] Holst TL. Computational Fluid Dynamics Uses in Fluid Dynamics/Aerodynamics Education. NASA Technical Memorandum 1994; 108834.

[2] Chitte P, Jadhav PK, Bansode SS. Statistic and dynamic analysis of typical wing structure of aircraft using Nastran. International Journal of Application or Innovation in Engineering & Management (IJAIEM) 2013; 2-7.

[3] Triet NM, Viet NN, Thang PM. Aerodynamic Analysis of Aircraft Wing. VNU Journal of Science: Mathematics – Physics 2015; 31, 2.

[4] Kavya G, Reddy BCR. Design and Finite Element Analysis of Aircraft Wing Using Ribs and Spars. International Journal & Magazine of Engineering Technology, Management and Research 2015; 2-11.
[5] Patel KS, Patel SB, Patel UB, Ahuja A P. CFD Analysis of an Aerofoil. International Journal of Engineering Research 2014; 3, 3.

[6] Doğru MH. Investigation of Velocity Distribution and Turbulent Energy for the Different Tip Shaped Projectiles. Çukurova University Journal of the Faculty of Engineering and Architecture 2017; 32: (3), 39-46.

[7] Kumar BSA, Ramalingaiah, Manjunath S, Ganganna R. Computational Investigation of Flow Separation over Naca 23024 Airfoil at 6 Million Free Stream Reynolds Number. International Journal of Science, Technology and Society 2015; 3-6.

[8] Göv İ, Korkmaz Ü. Comparison of Aerodynamic Performance of NACA 4412 and S809 Airfoil Profile. International Mechanical Engineering and Technologies Conference 2016.

[9] Göv I, Dogru MH, Korkmaz U. Improvement of the aerodynamic performance of NACA 4412 using the adjustable airfoil profile during the flight. Journal of The Faculty of Engineering and Architecture of Gazi University 2019; 34: (2), 1110-1125.

[10] URL: https://en.wikipedia.org/wiki/Adjoint_equation.