RE-DESIGN AND OPTIMIZATION OF THE NACA 4412 AEROFOIL PROFILE

Author Name:- Aarush Goyal
Dept. of **Mechanical Engineering**
Indian Institute of Technology (BHU)
Varanasi, India
Email Id-
aarush.goyal.mec22@iitbhu.ac.in

Author Name:- Kush Aggarwal
Dept. of **Mechanical Engineering**
Indian Institute of Technology (BHU)
Varanasi, India
Email Id-
kush.aggarwal.mec22@iitbhu.ac.in

Author Name:-Ayansh Bhardwaj
Dept. of **Mechanical Engineering**
Indian Institute of Technology (BHU)
Varanasi,India
Email Id-
Ayansh.bhardwaj.mec22@itbhu.ac.in

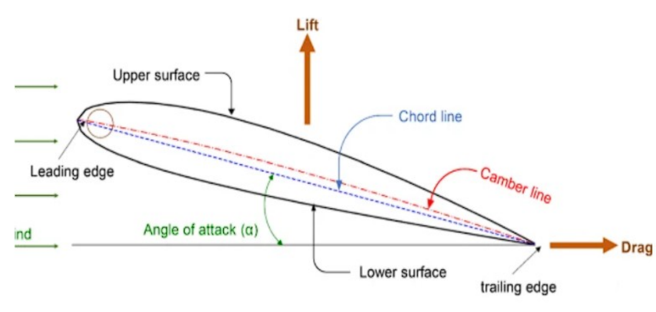
Author Name:- Kishan Soni
Dept. of **Mechanical Engineering**
Indian Institute of Technology (BHU)
Varanasi, India
Email Id-
kishan.soni.mec22@iitbhu.ac.in

Author Name:- Naman Kankariya
Dept. of **Mechanical Engineering**
Indian Institute of Technology (BHU)
Varanasi, India
Email Id-
naman.kankariya.mec22@iitbhu.ac.in

***Abstract*— In this paper , we have re-designed the NACA 4412 Aerofoil profile by changing the maximum thickness ,leading edge radius and the maximum camber for maximizing the lift to drag ratio of the aerofoil . The study was done using the XFLR5 Software and the commercial software ANSYS FLUENT 2022R1. The simulation was done using the Spalart Allmaras turbulence model .During the analysis on XFLR5 , it was observed that by reducing the thickness from 12% to 10% of the chord length , increasing the leading edge radius ratio from 1 to 0.6 and increasing the camber from 4% to 5% of the chord length maximized the lift to drag ratio . Four cases namely 1) 10% thickness , 2) 9% thickness along with 5%camber, 3) camber 5%, 4) thickness 9%, camber 5% and leading edge radius 0.6 which showed promising results were then verified and compared in ANSYS due to its higher credibility .The results showed that the airfoil with 10% thickness has a lift to drag ratio of 25.21 for an angle of attack of 3 degree.**

I. INTRODUCTION

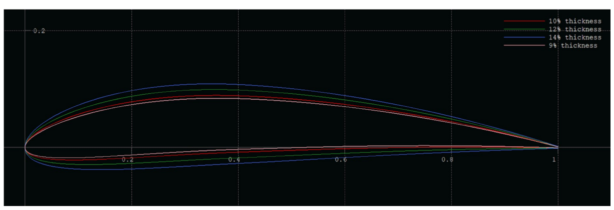
The NACA 4412 is a four digit aerofoil. The first digit expresses the camber in percent chord ,the second digit gives the location of the maximum camber point in tenths of chord, and the last two digit gives the thickness in the percent chord.Thus NACA 4412 has a maximum chamber of 4% of chord located at 40% chord back from the leading and is 12% thick . Its symmetrical shape makes it suitable for applications where lift needs to be generated equally above and below the airfoil, such as in symmetrically-loaded wings or control surfaces.The NACA 4412 airfoil has been used in various applications, including general aviation aircraft, wind turbines, and model aircraft. Its moderate thickness and camber make it versatile for a range of aerodynamic requirements. The shape of the aerofoil is defined using certain geometrical parameters such as, the leading-edge radius, maximum thickness,maximum camber. These play a very important role in the aerodynamics of the aerofoil. The leading-edge radius measures the airfoil's curvature at its most forward point. The ability of the airfoil to initiate a smooth flow across its surface, reducing turbulence and increasing lift, is influenced by this factor. The maximum difference between the upper and lower airfoil surfaces divided by the chord length is the maximum thickness percentage. It has an impact on both the lift and drag but its variation largely affects the drag force. Camber is the measure of the curvature of an aerofoil. Camber line is an imaginary line which lies halfway between the upper surface and lower surface of the airfoil and intersects the chord line at the leading and trailing edges. Aerofoil with higher camber can generate more lift at lower angle of attack but will also have a lower stalling angle.

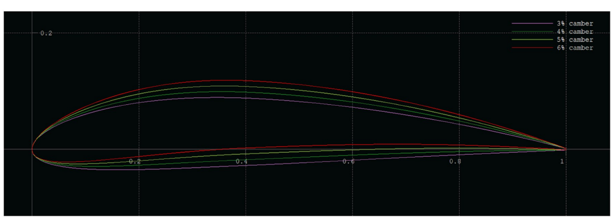


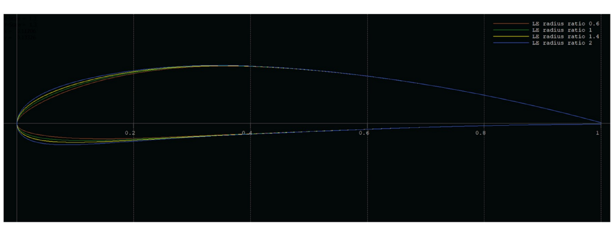
For aerofoil , a prominent tool is XFLR5 which is an open-sourced application that employs the XFOIL program from MIT to assess the parameters of the airfoil. Subsequently, it utilizes the Vortex Lattice Method, Lifting Line Theory, and 3D Panel Method to deduce the macroscopic behavior of airplanes. The analysis on XFLR 5 was carried out for different values of Mach number and corresponding Reynolds Number for standard atmosphere at the given altitude of 36,000 ft. The angle of attack was varied from 2 to 5 degrees. The final stage of analysis was carried out in ANSYS Fluent which is a Commercial fluid simulation software that uses RANS(Reynolds Averaged Navier-Stokes) equation to solve various CFD problems.The optimized aerofoil resulting from this study demonstrates the potential to significantly improve aerodynamics in various sectors. The findings contribute to the ongoing efforts in advancing aerodynamic design, providing valuable insights for future developments in aviation technology.

II. METHODOLOGY

The design process involves varying 3 main parameters- Maximum camber, Maximum thickness and Leading Edge Radius Ratio. The reason behind selecting these parameters is due to their key involvement in affecting the aerodynamic performance of an aerofoil. Some of the literatures also press upon varying these parameters to obtain suitable results(**give references**).The aerofoil optimisation process began with the selection of the base aerofoil NACA 4412 using it as a datum for further changes.







Aerofoil Optimization Process

III. AEROFOIL ANALYSIS

The XFLR 5 software has a large aerofoil database. The design process begins with the selection of our base aerofoil NACA 4412 from the Direct Foil Design option present in XFLR 5. Here multiple optimizations of the NACA 4412 aerofoil were generated by varying different geometrical parameters. The aerofoils were then analyzed using Direct Foil Analysis.

The analysis was defined using analysis of Type-2 with Mach number kept at 0.3 and 0.7 and the corresponding value of Reynolds number was taken for the given condition of altitude being 36000 ft. As no information about the forced trip site was mentioned the transition location was estimated using Ncrit value which predicts the disturbance in the air when it is allowed to flow freely. The Ncrit value was kept at 9 as the flow was assumed to have a free transition. The analysis was carried out for chord length 1 meter, angle of attack ranging from 2 to 5 degrees with a high iteration limit of 60.

.

IV. GOVERING EQUATION

XFLR 5 uses the panel method which is a numerical technique used to model the flow around a two-dimensional aerofoil. The flow is approximated by dividing the airfoil surface into a series of flat panels. The aerofoil is assumed to be two-dimensional and both cases of compressible and incompressible flow were taken.

The basic equation used in the panel method is the integral equation of the potential flow. This equation states that the velocity potential, Φ, is equal to the sum of the source and doublet strengths along the airfoil surface

$$\Phi = \sum_{i} = N\,(Si + Ki)$$

where Si and Ki are the source and doublet strengths and N is the number of panels. The source and doublet strengths are calculated by integrating the velocity potential along the airfoil surface. The source strength is given by equation

$$Si = \int_{c} \Phi * Ni * ds$$

where ni is the normal vector to the i-th panel and ds is the differential arc length along the panel. The doublet strength is given by equation

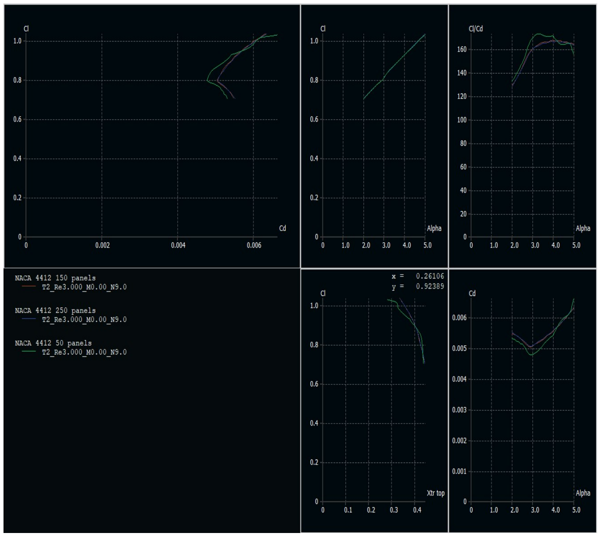
$$Ki = \int_{c} (\nabla 2\Phi) * Ni * ds$$

where ∇2 is the Laplacian operator.

After determining the source and doublet strengths, it becomes possible to compute the velocity potential at any given position inside the flow field. The calculation of velocity and pressure can be derived from the velocity potential through the utilization of the Bernoulli equation. The utilization of the panel approach enables the precise estimation of the flow characteristics encompassing a two-dimensional airfoil.

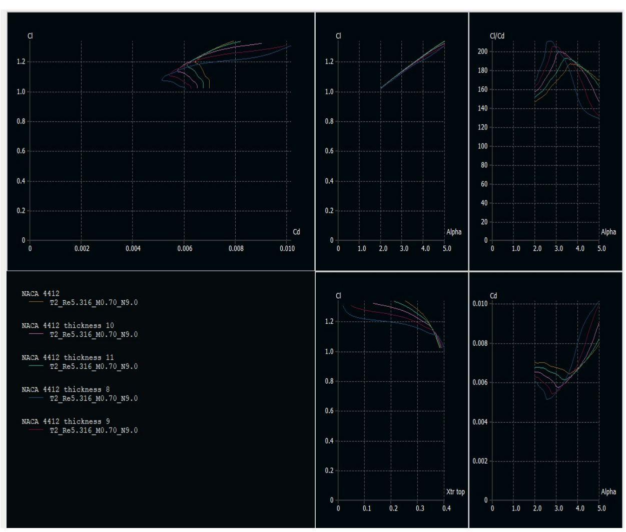
V. PANEL INDEPENDENCY

Generally higher number of panels give accurate results but with increase in the no.of panels comes the computational cost which is to be kept in vision. Hence the number of panels were chosen by analyzing the convergence of the results. It was observed that for the no.of panels being 50 the graphs deviated compared to 150 and 250 panel cases where the results converged very well. Hence the no.of panels were taken to be 200.



1. Maximum Thickness

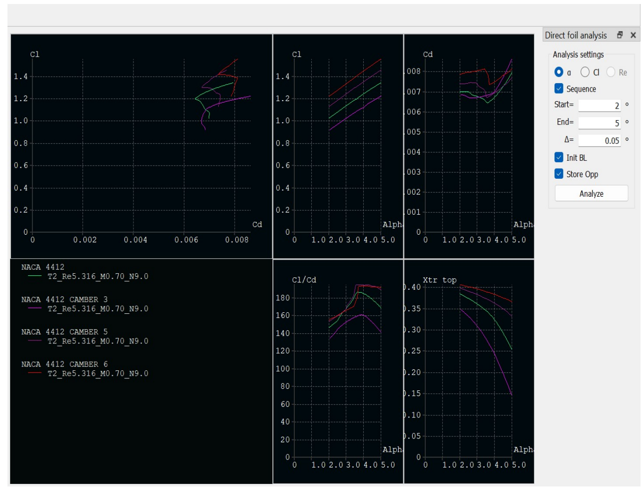
Firstly the impact of variation of maximum thickness was observed. It was found that for a Mach number 0.7 and Corresponding Reynolds Number at an altitude of 36000 ft being 5.3161e+6 1/m on decreasing the thickness the coefficient of drag for small range of angle of attack(2-3 degrees) was lower for aerofoil with lesser thickness but drastically increased especially in the 8% and 9% thickness case which resulted in sharp decrease in the value of Cl/Cd just before 3 degree angle of attack. The lift coefficient was almost unaffected.



2. Maximum Chamber

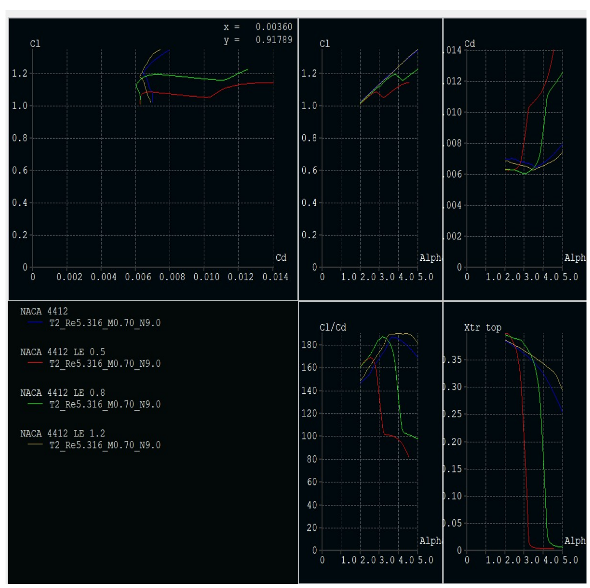
On increasing the camber both the lift and drag coefficient increased considerably but the increase in lift was more significant which resulted in an overall increase in the value of CL/CD.

Despite giving maximum value of CL/CD the aerofoil with maximum camber 6% of chord length had to be dropped from the analysis because the minimum value of CL was found to be more than 1.2 which exceeds the allowed range for the minimum CL i.e 0.6 to 1.2



3. Leading Edge Radius Ratio

Leading edge is one of the important parameters which affects the aerodynamics performance of an aerofoil. After performing analysis on XFLR5 it was observed that on decreasing the LE radius ratio the Lift coefficient as well as the Drag coefficient increased but the increase in drag was very high which resulted in a very sharp decrease in CL/CD Ratio which could even be predicted by the fact that due to the increase in steepness the drag increased considerably.



After carrying out multiple batch analysis on XFLR5 4 aerofoils were selected for further analysis in ANSYS. The properties of these aerofoils are mentioned in the table:

Name of the Aerofoil	Maximum Thickness(%)	Maximum Camber(%)	LE radius ratio
Design 1	10	4	1
Design 2	9	5	1
Design 3	12	5	1
Design 4	9	5	0.6

VII. ANSYS AEROFOIL ANALYSIS

In this study, the aerodynamic performance of the 4 selected aerofoil profiles was obtained by using the commercial software ANSYS FLUENT 2022R1.
We have used the Spalart Allmaras turbulence model because it is known to give accurate results for boundary layers subjected to adverse pressure gradients and also because it is less memory-intensive. The coordinates of the airfoil were downloaded from the (Airfoil Tools) database[7] , and the geometry of the airfoil was created in ANSYS Design Modeler with a chord length equal to 1 m. The C-type fluid domain which surrounds the airfoil is created in ANSYS DESIGN Modeler, and it consists of a semicircle with a radius of 5C (where C is the chord length) and a rectangular domain with a size of 10Cx10C.The particular aspect behind the C-type domain construction is to capture the curvature effect properly from the leading edge . The boundary conditions and the dimensions are shown in Figure 1; the blue arrows represent the inlet boundary condition while the red arrows represent the outlet boundary condition.

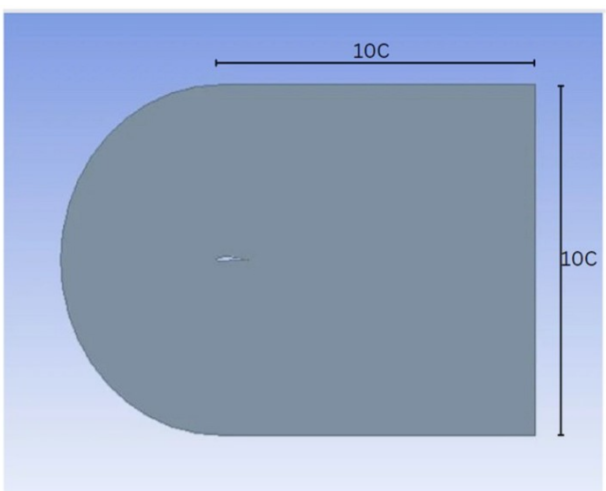
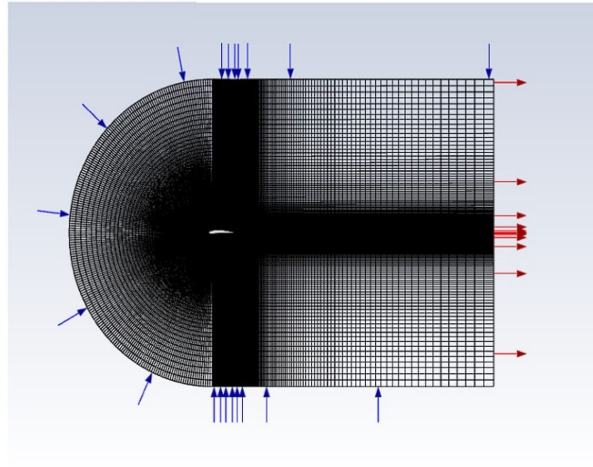


Fig1 a)C-type fluid domain



b)Fluid Domain Mesh

1. Governing Equation

In this study, the flow is assumed to be steady, two-dimensional, and compressible. The simulation is done for a Re number of 5.3161e+6 1/m, and the governing equations are the continuity equation and the Navier- Stokes equations. They can be written as follows:

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0$$

$$\frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

$$\frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)$$

Since the flow is assumed to be compressible, the air density and kinematic viscosity are variable . The equations that are used to calculate the lift and drag coefficients are :

$$C_d = \frac{2F_d}{\rho u^2 A} \qquad C_l = \frac{2F_l}{\rho V^2 A}$$

2. Turbulence model

The turbulence model chosen for the simulation is the Spalart-Allmaras turbulence model, specifically designed for aerospace applications . It showed promising results for boundary layers exposed to an adverse pressure gradient. It is a comparatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. In its primary form, it is constructively a low-Reynolds number model, demanding the viscous-affected region of the boundary layer to be appropriately resolved [8] . The transport equation is given as [9]

Name of the Aerofoil	Optimum angle of attack(degree)
Design 1(10% thickness)	3
Design 2(9% thickness and 5% camber)	4
Design 3(5% camber)	4
Design 4(9% thickness, 5% camber and 0.6 LE radius ratio)	5

:

$$\frac{D\tilde{v}}{Dt}= c_{b1}\left(1-f_{t2}\right)\tilde{v}\tilde{s}+\frac{1}{\sigma}\left[\nabla.\left(\left(v+\tilde{v}\right)\nabla\tilde{v}\right)+c_{b2}(\nabla\tilde{v})^2\right]-\left(c_{w1}f_w-\frac{c_{b1}}{k^2}f_{t2}\right)\left(\frac{\tilde{v}}{d}\right)^2+f_{t1}\Delta U^2$$

3. Boundary Conditions

The aerofoil surface has been named selected as wall which would automatically considers the no-slip condition on the aerofoil surface. The inlet velocity is taken to be 206.61 m/s after considering Mach number value as 0.7 and altitude 36000 ft. The turbulent viscosity ratio for the inlet was taken as 1 while for the outlet the value was taken to be 10. Keeping a higher value of turbulent viscosity ratio considers the flow to be turbulent from the starting itself which is not the for the inlet where the flow is initially laminar and then gradually transitions to turbulent flow.

4. Methods:

The pressure-based coupled algorithm is used to solve the compressible RANS (Reynolds-Averaged Navier-Stokes) equations in which the momentum and pressure-based continuity equations are solved in a nearly coupled manner. The pressure-based coupled algorithm remarkably enhances the convergence rate. Second-order pressure is used for the discretization, and the momentum and the modified turbulent viscosity are set to second-order upwind.

VIII. Results and Discussion

1) Mesh Validation

The meshing was validated through results from an experimental study performed by Abbott aerospace(reference). For the validation the inlet velocity was taken to be 43.9 m/s for a Reynolds number of 3,000,000 at 0 angle of attack. The Plot of the CL and CD values versus iterations is shown in the figure

.

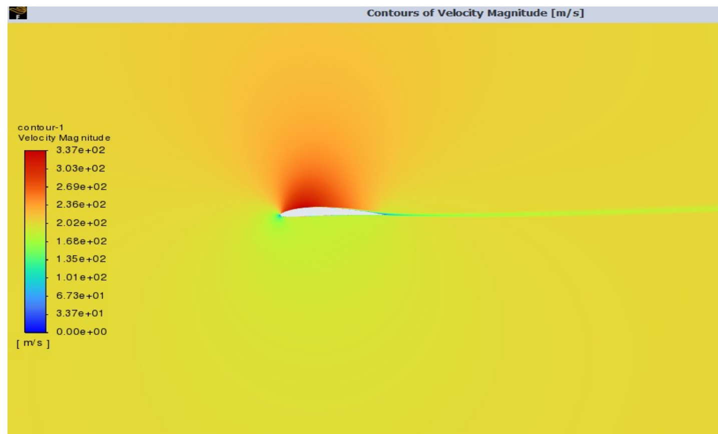
2) Required Data for CFD simulation

After the analysis on XFLR5 The maximum CL/CD value was observed and correspondingly the optimum angle of attack for the 4 aerofoils was chosen as shown in the table

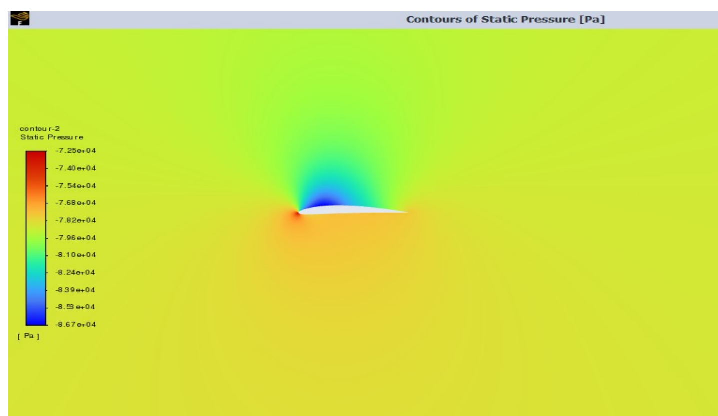
3) CFD simulation Result

In this study, the aerodynamic characteristics, which are the lift and drag coefficients , lift to drag ratio , velocity contour and pressure contours for the different designs of NACA 4412 Aerofoil profile, are extracted by using (ANSYS FLUENT 22.0 R1). The simulation is done for a wind speed of (206.61m/s or mach number 0.7 at an altitude of 36000ft) The results show that lift to drag ratio for the four cases shortlisted above were higher than the original aerofoil . Out of those four cases , 2 cases thickness 10 % and camber5%,thickness9%, performed better than the other two cases in terms of lift to drag ratio . the lift coefficient ,drag coefficient ,velocity contour and pressure contour of these four re-designed aerofoil profiles at their optimum angle of attack are :

1)Design 1 (10%thickness,angle of attack=3 degree):



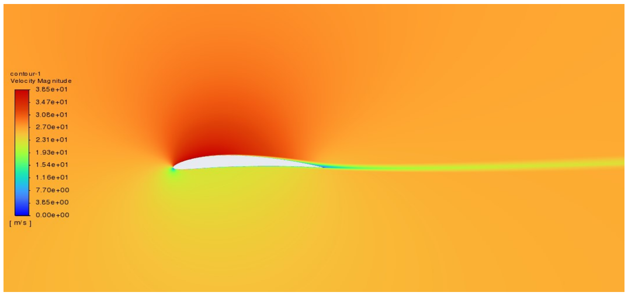
velocity contour



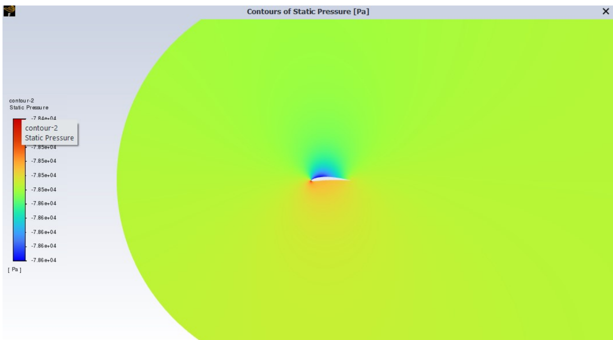
Pressure contour

Lift coefficient =0.97518
Drag coefficient = -0.038682
Lift to drag ratio = 25.21017

2) Design 2 (9% thickness and 5% camber, angle of attack= 4 degree):



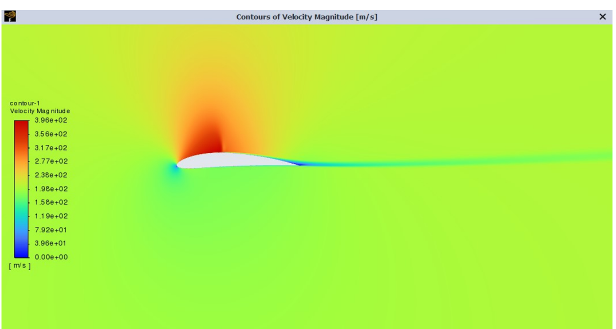
Velocity contour



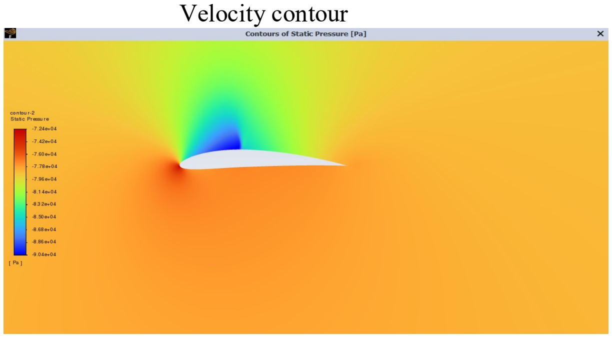
Pressure contour

Lift coefficient = 0.95067
Drag coefficient = -0.049987
Lift to drag ratio = 19.0183448

3) Design 3(5% camber,angle of attack= 4 degree):



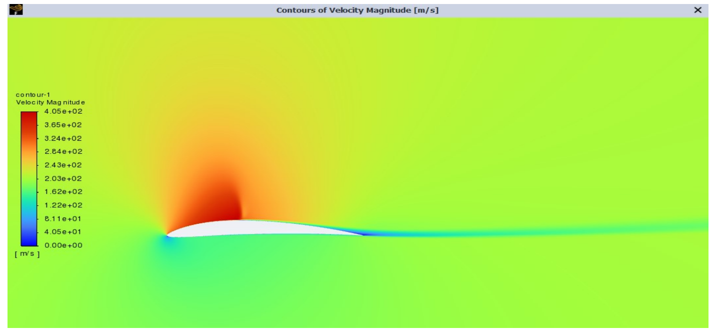
Velocity contour



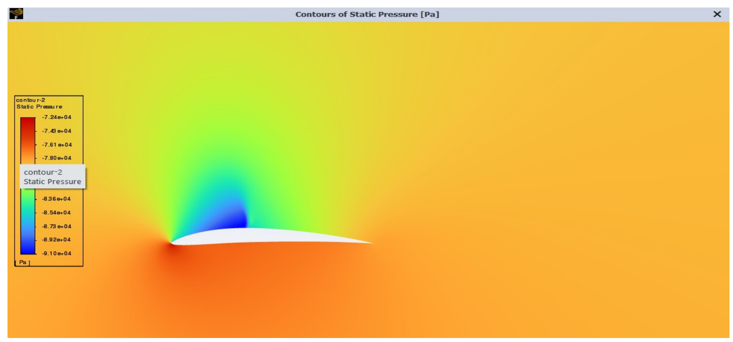
Pressure contour

Lift coefficient = 1.1903
Drag coefficient = -0.064822
Lift to drag ratio = 18.36259

4)Design 4 (9% thickness, 5% camber and 0.6 LE radius ratio,angle of attack = 5 degree):



Velocity contour



Pressure contour

Lift coefficient = 1.3541
Drag coefficient = -0.097570
Lift to drag ratio = 13.8782

Conclusions :
The best profile for the NACA 4412 aerofoil is the one with 10 % thickness because it has the greatest lift to drag ratio. We have also selected it because it offers less drag than the other profiles for comparable lift to drag ratios . Therefore , it will have to withstand less loads which is better for it even structurally .

REFERENCES

1]https://iopscience.iop.org/article/10.1088/1757-899X/1094/1/012078/pdf
2]https://link.springer.com/article/10.1007/s13369-024-08837-6#Fig7
3]https://www.abbottaerospace.com/downloads/naca-report-563-calculated-and-measured-pressure-distributions-over-the-midspan-section-of-the-naca-4412-airfoil/
4]https://www.researchgate.net/publication/281814485_Effect_of_Leading_Edge_Radius_and_Blending_Distance_from_Leading_Edge_on_the_Aerodynamic_Performance_of_Small_Wind_Turbine_Blade_Airfoils
5]https://iopscience.iop.org/article/10.1088/1742-6596/2276/1/012027/pdf
6]https://ntrs.nasa.gov/api/citations/19930084610/downloads/19930084610.pdf
7]http://airfoiltools.com/airfoil/naca4digit?MNaca4DigitForm%5Bcamber%5D=4&MNaca4DigitForm%5Bposition%5D=40&MNaca4DigitForm%5Bthick%5D=12&MNaca4DigitForm%5BnumPoints%5D=200&MNaca4DigitForm%5BcosSpace%5D=0&MNaca4DigitForm%5BcosSpace%5D=1&MNaca4DigitForm%5BcloseTe%5D=0&MNaca4DigitForm%5BcloseTe%5D=1&yt0=Plot
8]K. Oukassou, S El Mouhsine, A El Hajjaji and B Kharbouch 2019 Comparison of the Power, lift and Drag Coefficients of Wind Turbine Blade from Aerodynamics Characteristics of Naca0012 and Naca2412 (Procedia Manuf. (Morocco)) vol 32
9]Spalart P R and Allmaras S R 1992 A One Equation Turbulence Model for Aerodynamic Flows (AIAA 30th Aerospace Sciences Meeting and Exhibit (USA))
10] https://www.pdas.com/flowcalc.html