École Centrale Lyon

External aerodynamic report
Rapport

NACA 4412 airfoil : wind tunnel experiment and simulations

Élèves :
Harish REYYA

Enseignant :
Jean-François MONIER

24 novembre 2022

Table des matières

1 Introduction

In this document, we try to report experimental and numerical simulation results (using FLUENT) of the NACA 4412 airfoil design that we studied during the TP session. The main objective of this study is to measure lift and drag coefficients resulting from a number of different parameters :

— Geometry of the airfoil NACA 4412 (Chord and camber line, leading edge radius, thickness of the shape...)
— The angle of attack α
— Incoming air velocity U_0
— Air density and vescosity ρ and μ

As a rule of thumb, we want to increase lift and reduce drag as much as possible to limit fuel consumption of the airplane, and thus increase lift-to-drag ratio L/D.

2 Experimental bench test

During the experimental session, we have worked with a wind tunnel with adjustable air velocity including two airfoils designed as NACA 4412 profile. The first one allow to adjust its angle regarding the air flow and thus adjusting the angle of attack. This airfoil is related to a strain gauge allowing the direct measurements of drag and lift coefficients.It is also equipped with flap at its trailing edge which we will see the influence depending on its angle. The second airfoil, placed a little lower than the first one to avoid influencing the measurements, is related to a water level gauge which gives the velocity on the suction side of the foil with its 25 pressure holes.

From this experimental set, we can therefore measure drag D, lift L and the dynamic pressure depending on the influence of the angle of attack α, the air flow velocity U_0 and the angle of the flap i.

FIGURE 1 – Wind tunnel test bench with two airfoils NACA 4412

3 Theoretical Background

Lift, drag and pressure coefficients are non-dimensional coefficients used to compare the behavior of different objects on different scale levels under the influence of a flow. They are defined as follows :

$$C_l = \frac{L}{\frac{1}{2}\rho S U_0^2}$$

$$C_D = \frac{D}{\frac{1}{2}\rho S U_0^2}$$

$$C_P = \frac{P - P_{atm}}{\frac{1}{2}\rho U_0^2}$$

Where :
- L : The lift force
- D : The drag force
- U_0 : fluid velocity
- ρ : fluid density
- S : Wing surface
- $P - P_{atm}$: surface static pressure - pressure far from the wing

4 Experimental measurements and analysis

4.1 First set of experimental measurements

In the first instance, we fixed the angle of the flap at the trailing edge at $i = 0$ (and then $i = 10$) and the air flow velocity at $U_0 = 22m/s$ and we changed the angle of attack α from $[-5, 15]$ which allowed us to measure lift and drag forces as a function of α. We then calculated the coefficients associated with these forces.

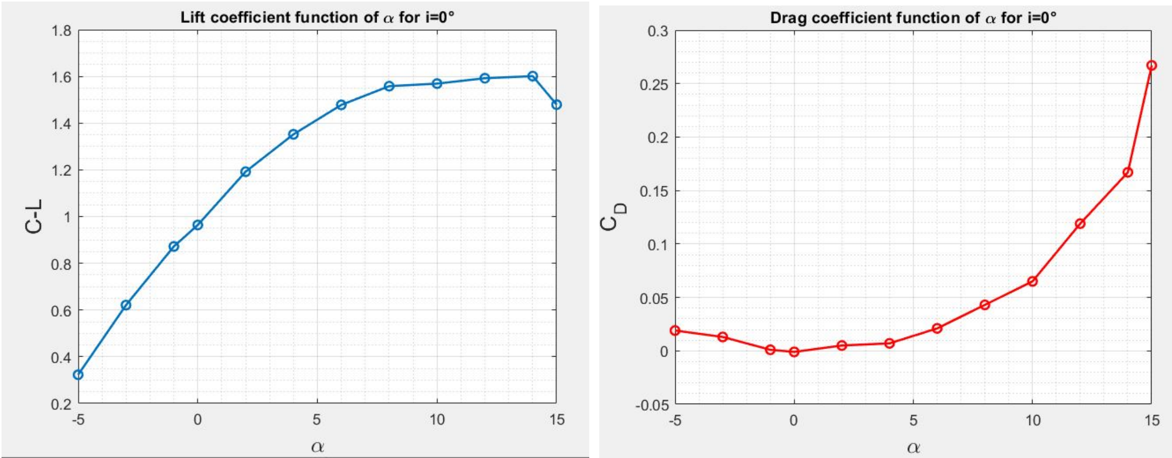

FIGURE 2 – Lift and drag coefficient measured for $i = 0$

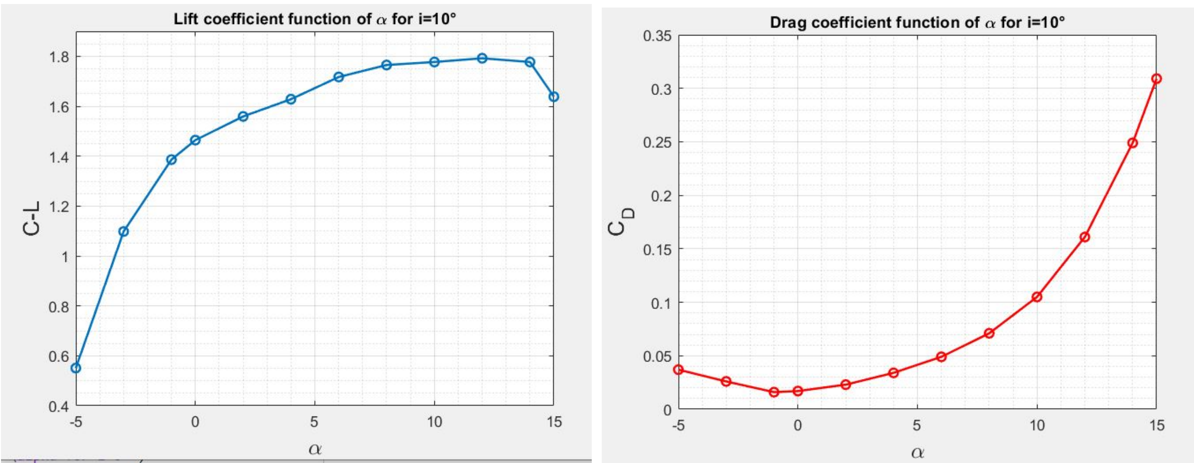

FIGURE 3 – Lift and drag coefficient measured for $i = 10$

4.2 Second set of experimental measurements

At a flap angle at $i = 0$, we then measured the pressure distribution at the suction side of the second airfoil for a velocity $U_0 = 22m/s$ and values of the angle of attack $\alpha = \{0, 5, 15\}$. The results are plotted in the following graphs :

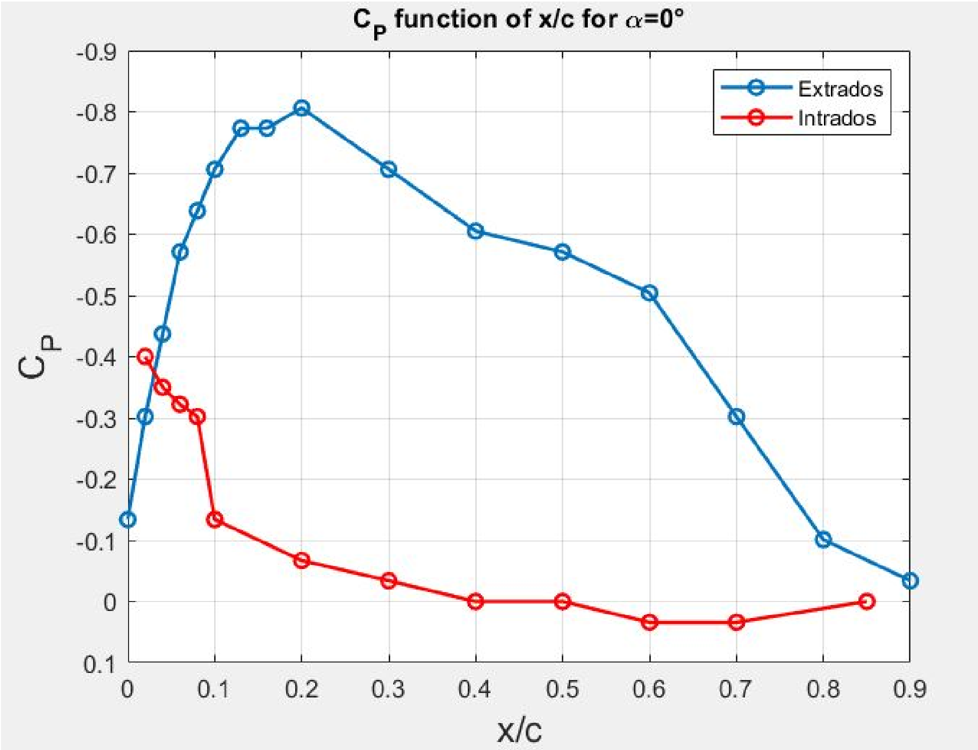

FIGURE 4 – C_p for $\alpha = 0$

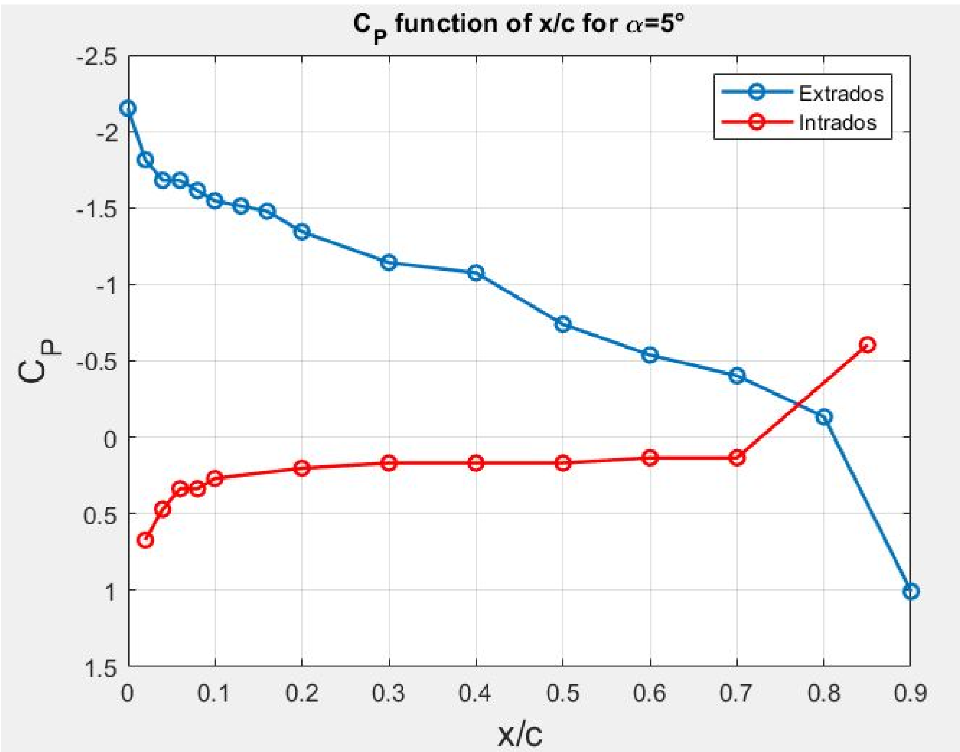

FIGURE 5 – C_p for $\alpha = 5$

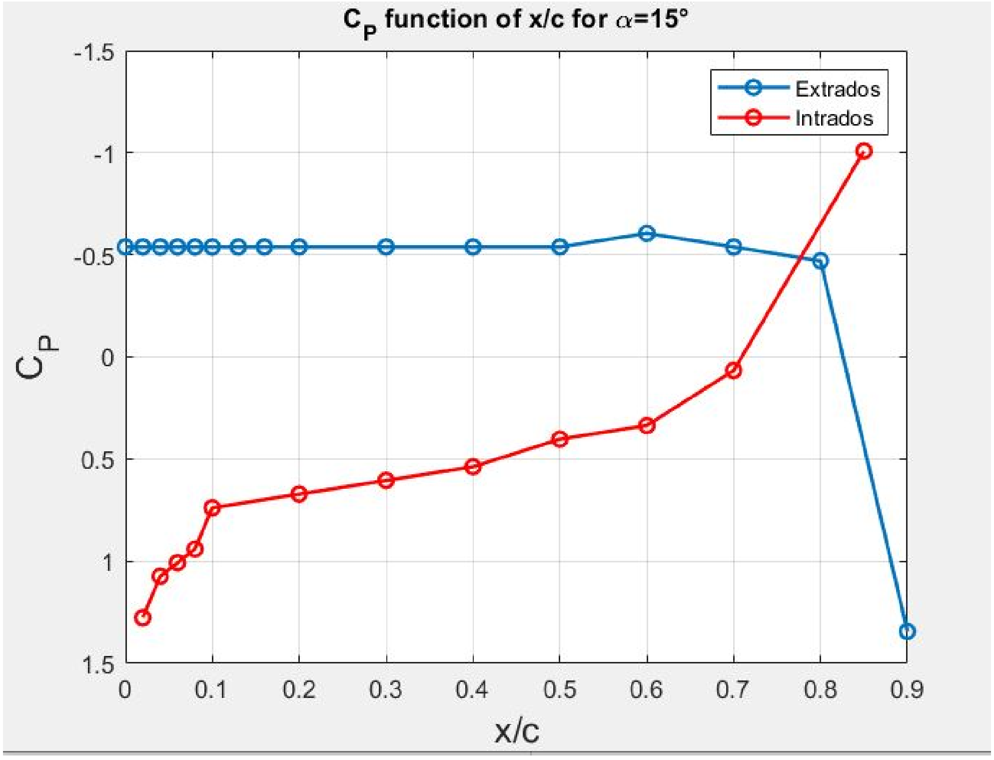

FIGURE 6 – C_p for $\alpha = 15$

4.3 Third set of experimental measurements

At a flap angle of $i = 0$ and an angle of attack $\alpha = 5$, we measured the lift and the drag forces at various velocities so we can plot the figure below :

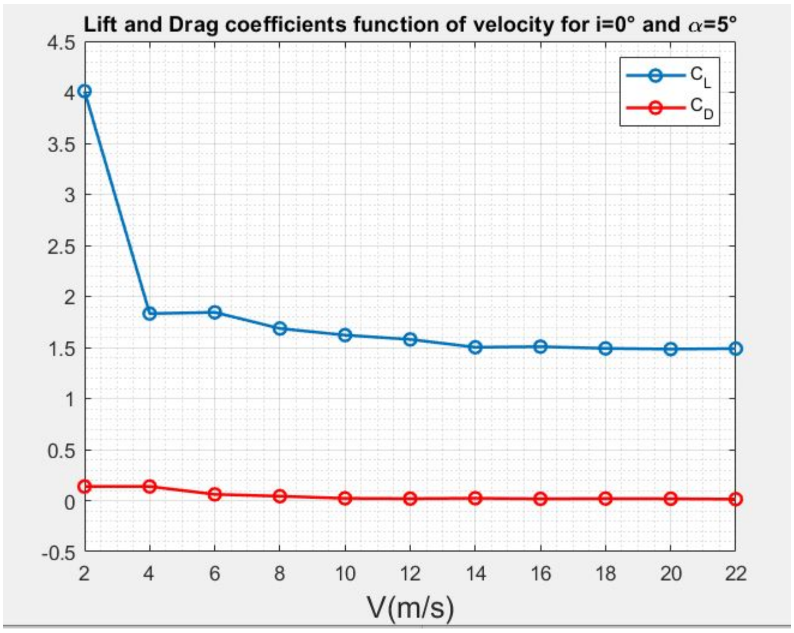

FIGURE 7 – C_L and C_D as a function of velocity

5 Numerical simulation and analysis using Ansys Fluent

In this section we will compare the experimental results with the Simulated results. We used Ansys Fluent as the Computational Fluid Dynamics (CFD) solver. The NACA airfoil model mesh file was imported and used 2D Reynolds-averaged Navier-Stokes equation (RANS) simulation to solve the problem.

Upon setting all Operating conditions and running the calculation for varying α from -5° to 15° we observerd the shift of static pressure as we go from negative attack angle to 0 to stall conditions. these can be seen in the graphs below.

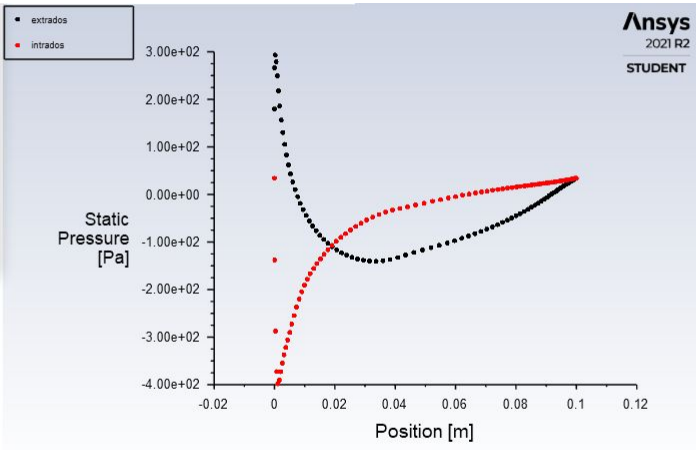

FIGURE 8 – Static Pressure as a function of position for $\alpha = -3$

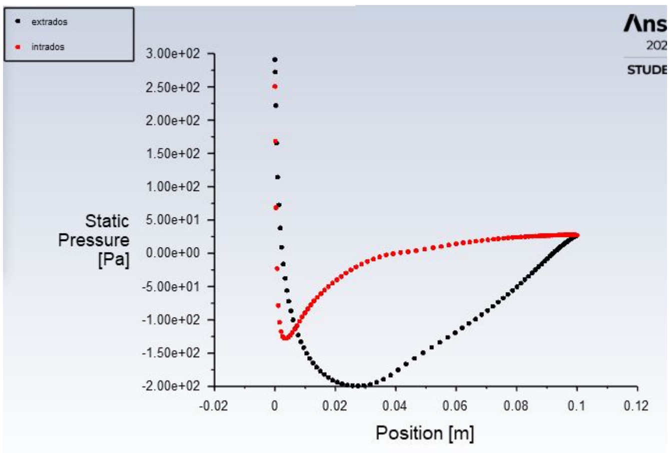

FIGURE 9 – Static Pressure as a function of position for $\alpha = 0$

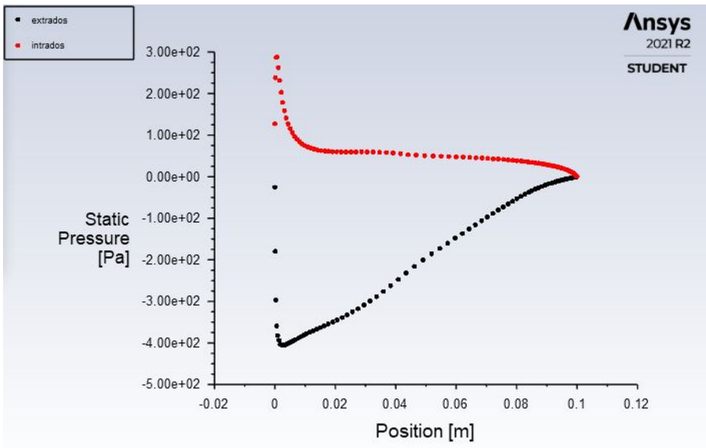

FIGURE 10 – Static Pressure as a function of position for $\alpha = 6$

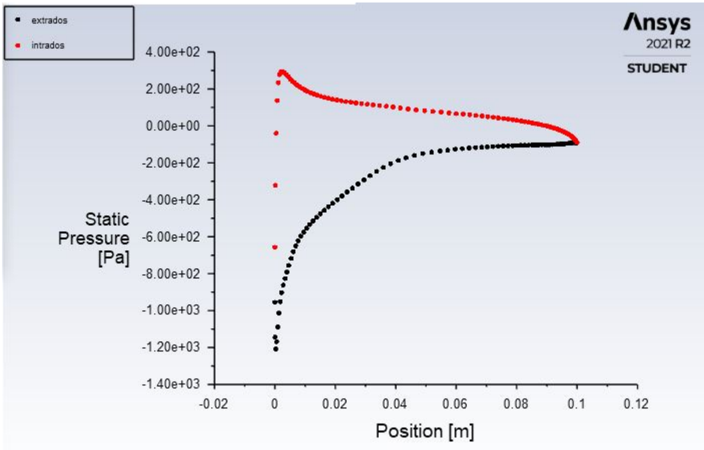

FIGURE 11 – Static Pressure as a function of position for $\alpha = 14$

The contours of static pressure below clearly shows the generation of negative pressure gradient in stall condition vs the static pressure condition in steady condition

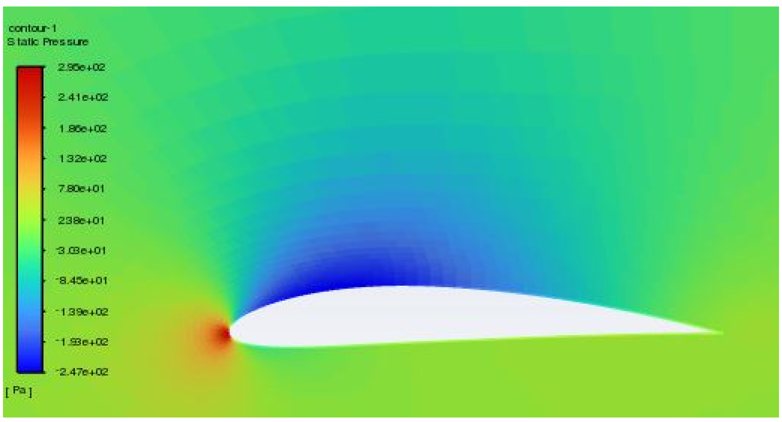

FIGURE 12 – Static Pressure contour for $\alpha = 2$

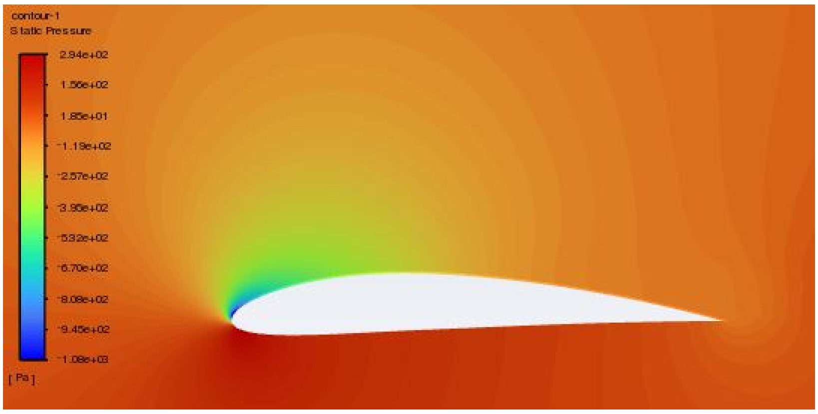

FIGURE 13 – Static Pressure contour for $\alpha = 12$

The lift and drag force were also calculated during the simulation, which allows us to plot the graph between α and Forces (as shown in the figure below). We could clearly see the increment of drag force with increase of the angle of attack. Similarly we can observe that by increasing the angle of attack we increase the lift force, but around 13° stall condition is achieved where the lift force does not increase but significant increase of drag force is observed :

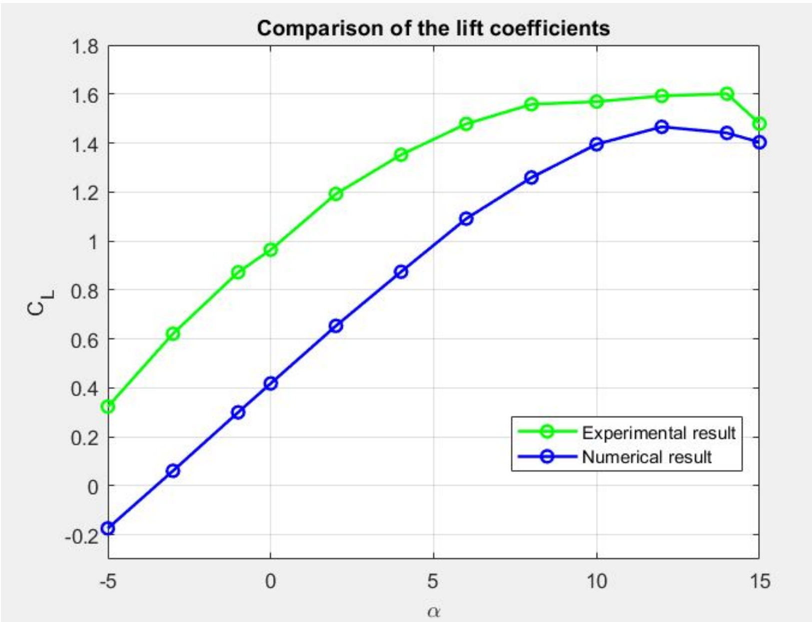

FIGURE 14 – Experimental and numerical C_L

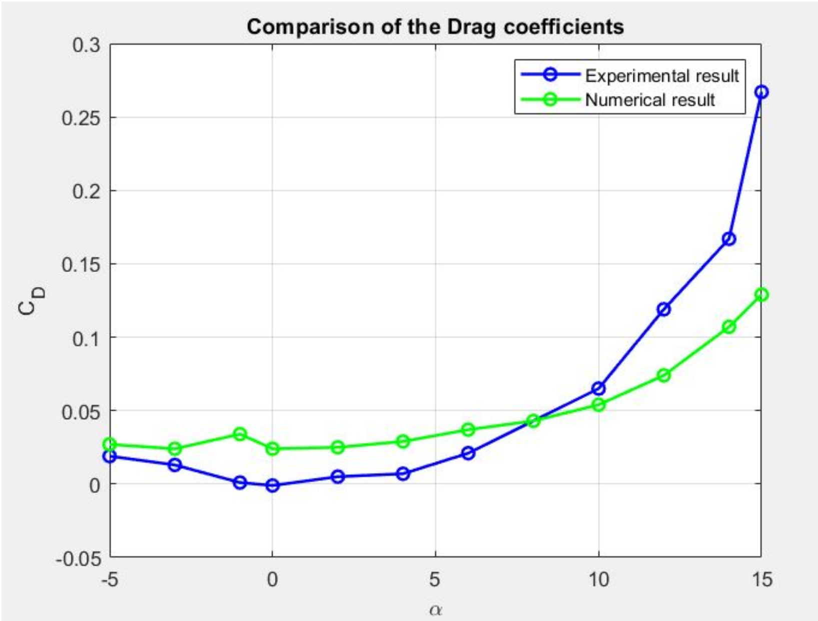

FIGURE 15 – Experimental and numerical C_D

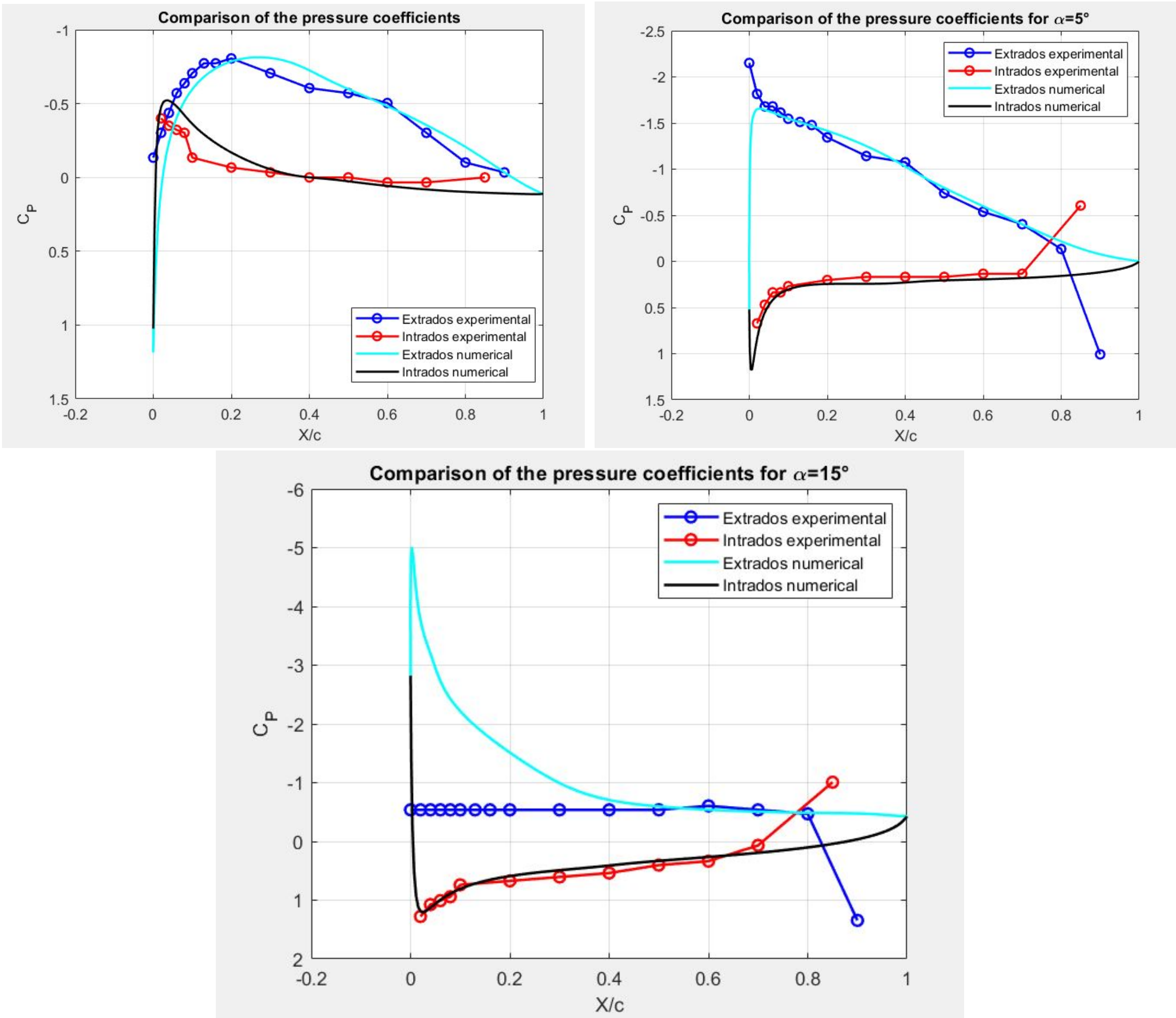

FIGURE 16 – Experimental and numerical C_P for $\alpha = 0$, $\alpha = 5$ and $\alpha = 15$

As a comparison between the experimental and numerical graphs, we can see that for angles 0 and 10 the graphs are nearly the same indicating that our numerical method used was the corresponding one. But increasing the angle of attack to 15 degrees we can see that the area under the numerical curve is larger than the experimental one. This indicates that the numerical simulation predicts a late separation of flow.

We can also assume that for the experimental results, we have the influence of the first airfoil on the second, and also the influence of the wall of the wind tunnel on the overall performances.

Furthermore, the numerical simulations could imply some limits. As we assumed that the fluid is incompressible, we neglected its density. This assumption is weakened as the flow velocity increases. Also, depending on the chosen mesh and other parameters(Temperature, ambiant pressure...), the result could be different and the simulation could include errors regarding the experience.

Near the stalling angle of attack, we see that the experimental pressure coefficient for the Extrados is separated from the numerical one. This could be the result of some inaccuracies on the bench level.

It can also be observed from Figure that the higher pressure in the lower side of the wing than the upper side is correctly predicted by the numerical simulation.

6 Conclusion

From the experiment we could observe how the variation of lift, drag and pressure coefficient vary with respect to the angle of attack of the wing, angle of attack of the flap, etc.) which gives an idea about how lift is generated and how an aircraft fly. When compared to the experimental work, numerical simulation neglected few phenomena such as turbulence,it also have great importance on cost reduction since it gives an optimum parameter to design and perform to obtain a required performance. The stall conditions during flight should be avoided since the lift is drastically reduced and drag is increased hence the flight won't be stable. Also the airfoil should be designed in such a way to have a large value of angles of attack resulting to high lift during take-off and acceleration regimes as well as cruise flight which needs a very high L/D to reduce the cost of the thrust to overcome the drag.

The implementation of high lift devices such as flaps at a certain angle indeed increases lift we increase the area under the pressure curve, but comes with a cost of drag increment.

There were few challenges while performing the experiment such as accuracy of the experimental data which was caused due to the vibration created by the airflow due to the higher angle of attack. One must also consider the wall effects as it will influence the velocity field of the incoming air. There were few limitations observed in the simulation as well. During simulation we did not consider the energy equation as well as the compressibility effect.