POLITECNICO DI MILANO
INDUSTRIAL ENGINEERING

MASTER DEGREE IN
AERONAUTICAL ENGINEERING

AERODYNAMIC STUDY OF PERFORMANCE OF ACTIVE FLOW CONTROL IN PERIODIC FORCING CONDITION INSTALLED ON A NACA 2412 AIRFOIL

Advisor :

Prof. Ing. Federico Piscaglia

Co-Advisor:

Prof. Ing. Josep Maria Bergadà

Candidate :
Raffaele Fiore
Mat. 898079

ACADEMIC YEAR 2019 - 2020

Abstract

In this thesis work will be studied the active flow control (AFC) system on a NACA 2412 airfoil at a Reynolds number 30'000. The idea is to apply the flow control in periodic forcing condition in order to modify the lift and drag forces over the airfoil. Once fixed the angle of attack (AoA), 5 different frequencies will be considered (multiple or submultiples of main detach frequency of vortices) and 3 different angle of outflow from the groove will be considered. The simulation are performed in a two-dimensional case using the K-Omega SST turbulent model. The goal is to identify which combination of these parameters is the best one. The results give us back an increase of efficiency larger than 100%, increasing the angle versus the normal to the surface and increasing the frequency of AFC.

Acknowledgements

First of all I would like to thank my Co-Advisor Prof. Ing. Josep M. Bergadà because he allowed me to do my thesis work in Spain, he has gave me the chance to do my thesis abroad, with all his availability in helping me whenever I needed a help. I want to regard also my prof. Federico Piscaglia because he also helped me in doing this fantastic experience in an International contest.

Mi piacerebbe ringraziare i miei genitori, è solo grazie a loro se ho potuto vivere nuove esperienze che mi hanno portato a crescere e a vedere il mondo da un'altra prospettiva. Li ringrazio per avermi sostenuto tutte le volte in cui, per vari motivi, ero giù di morale e mi hanno dato quella forza necessaria per poter continuare sulla mia strada senza mai perdere di vista l'obiettivo. Ringrazio, ovviamente, mio fratello Giorgio, lo definirei come il faro per la mia crescita (solo perchè sei il fratello maggiore), una persona da cui prendere spunto in ogni situazione, che mi ha aiutato a superare i momenti più difficili rendendomi sempre più una persona più forte.

Ringrazio gli amici di via Schiaffino, Chicco e Mazza, due persone fantastiche (pezzi di pane) che fin da subito hanno dato vitto e alloggio ogni qual volta ce ne fosse bisogno, pronti ad ospitare chiunque solo per stare in compagnia, pronti ad aiutare con tutto quello che hanno e più di tutto sempre disposti a far serata (cazzata). Ringrazio il Delu, il primo collega e amico conosciuto a Milano, amico di tante bevute che alla fine ci sono servite per poter concludere questo percorso senza che perdessimo la fama di alcolizzati. Porterò sempre nel cuore la nostra esperienza vissuta con Mario a Barcellona, ho un po di malinconia se dovessi pensare a quante belle esperienze abbiamo vissuto in poco tempo.

Ringrazio Madda, la rubrica dei ristoranti (per modo di dire) di Milano, sempre pronto a organizzare tavolate. Poi c'è Lapo, personaggio fantastico, oserei definirlo l'onniscente, non avrei una parola migliore per definirti. Ringrazio i miei amici della triennale, o quasi, Dispo e Nicola, finiti insieme a Milano a causa di Nicola, dispensatore di consigli pessimi, Dispo mi ha seguito in tutto questo e direi che non potevo scegliere persona migliore con cui potermi confrontare in ogni momento e proseguire questo percorso. Ringrazio la quasi unica ragazza del gruppo Elena, l'ottimista per eccellenza e organizzatrice dei party @ Brescia di cui non ricordo nulla (per ovvi motivi).

Ringrazio tutti i miei amici di Altamura, i quali anche a distanza di mesi non mi hanno mai fatto pesare la mia lontananza da casa.

At least but not last ci sono gli amici della residenza, elencarvi tutti è impossibile, però vi ringrazio per essere stati la mia seconda famiglia. Tutt'ora penso che la residenza sia stata la mia scelta migliore per questa breve esperienza a Milano perchè mi ha fatto conoscere persone fantastiche con cui ogni sera bisognava cenare insieme perchè in famiglia si sta a tavola tutti insieme.

Contents

List of Figures

List of Tables

Chapter 1

Introduction

Airplane are one of the most important means of transport in the modern world, it is usually used for civil, commercial and military purposes. The first component at which someone thinks when imagine an aircraft probably will be the wing. Airplanes are the only vehicle that uses the wings for generating the lift (ailerons are used also for sport cars for generate downforce).
What is the lift?
The lift is a force perpendicular to the direction of the oncoming wind direction, that allows to the aircraft to get up from the ground (to counter the gravity force). It is generated thanks to the air that flows over the wing and, the difference of pressure between the higher and lower part of the wing, generates a force on the surface. The drag force, instead of the lift force, is parallel to the direction of flow. Lift and drag value increases with the increase of angle of attack, that is the angle between the chord of the wing and the flow direction.
One could think that thanks to the last assertion it is possible to increase the angle of attack (AoA) more and more and increase the value of lift and drag proportionally with the change of AoA, but in reality the situation is a little bit different. When the angle of attack is increased, increases the gradient of pressure that the flow have to win to go from the leading edge to the trailing edge and the deceleration of flow is proportional to the increase of this pressure gradient. The velocity of flow may become equal to 0 (in separation point), after its deceleration,

where it reaches the stall condition. Due to pressure gradient the flow reverts its behavior and generates vortices on the upper surface that reduces drastically the lift and drag forces.
In this thesis work will be studied the NACA 2412 profile with a Reynolds number equal to 30000 and an angle of attack of 11 degrees. The final purpose is to apply a system to reduce or eliminate, in a very unusual case, the birth of vortices. The system is called Active Flow Control (AFC) in periodic forcing mode.
What is this system? When one faces with separation problem usually needs to consider the systems passive or active that helps to control the separation. A very nice definition of surface where given by Flatt in 1961 where he told "Boundary layer control includes any mechanism or process through which the boundary layer of a fluid flow is caused to behave differently than it normally would were the flow develop normally along a smooth straight surface". But the history of boundary layer control is older, in 1904 Prandtl presents a document to a conference in Germany in which he explained the mechanisms of steady separation and used also the active flow control to show the great influence such a control exerted on the flow pattern (he used the suction to delay the separation of flow around a cylinder). In Germany after this research made by Prandtl, a lot of studies were made on aircraft across the Second World War in order to apply the AFC to obtain a low drag reduction to obtain a low-drag laminar state. These studies were made also in Europe and United States and the first aircraft were the X-21 in which the AFC were applied to a swept wing. An important era for AFC went from 1970 to 1990, the era of oil crisis, in which all the government agencies and private corporations want to find a way to conserve energy, and so drag reduction for civilian airplane. A lot of system controls were developed in this period for example the transition-delaying compliant coatings (Gad-el-Hak 1996) or large-eddy breakup devices (LEBUs) and riblets were systems to reduce skin-friction drag for turbolent boundary

layers.
The purpose of this thesis is to study the condition without implementing the AFC with the objective to find the separation point in which we want to insert the AFC groove. From the groove it is possible to suck the particles of fluid with lower energy (low velocity) or it is possible to energize the flow, increasing its velocity, blowing, with the AFC, pressurized air over the upper surface of airfoil. Firstly a constant sucking condition is considered, later simulations with periodic forcing with same frequency or multiplies of the main frequency of detach of vortices, with different angles of outflow from AFC groove, will be considered. The periodic forcing AFC is less energy-consuming w.r.t. a constant sucking system, i.e. if works well it could be mounted on an aircraft.

Chapter 2

Fluid Mechanics

2.1 Basic Concepts

Mechanics of fluid studies the behavior of fluids in quiet condition (static fluid) or when it is moving (fluid-dynamics), and the interaction between a fluid and a solid (the exchange forces).
How can we define the fluid? The fluid is a substance that we can find in liquid or gaseous form; however the fluid definition it is not a well defined concept because it is based on the substance capacity to resist to external solicitations rather than on the structure of matter. Indeed, on the contrary with respect to solids that have a atomic structure well defined and organized, thanks to the very strong intermolecular forces between atoms, for the liquids and gas the atoms haven't got a definite position and they are free to move without any restriction (thermal agitation) varying continuously direction thanks to the bumps between molecules and walls that delimit the system. The differences between is not only based on the shape. Indeed there exists a lot of different materials that when subjected to specific conditions, they can behave like solids or like fluids. A "simple" solid could be considered like a substance with a shape and the relative position of constitutive elements change slightly if subjected to a small change of forces that act on it. Correspondingly, a "simple" fluid could be defined as a substance where the relative position of constitutive elements changes by a quantity that

is not small if subjected to a particular type of forces (shear forces).
By definition, a fluid substance it isn't rigid. In other words, a small element of fluid is not able to keep its shape even when subjected to a very low shear force. This does not preclude the possibility, on the part of the fluid element, of offering resistance to the shear stress. However, this force of opposition to the cutting effort is not capable of preventing the change of shape. As mentioned, fluids are conventionally classified as liquids or gases. For reasons related to the nature of intermolecular forces, most substances can exist stably in two different phases which show a characteristic of fluidity or of easy deformability. The density of a substance in the liquid phase is normally much greater than in the gas phase, but this is not a useful characteristic to distinguish the two phases. The most important difference between the mechanical properties of liquids and gases lies in compressibility. Gases can be compressed much more easily than liquids, and as a consequence, any motion involving an appreciable change in pressure is accompanied by changes in the specific volume more importantly than liquids. Most of the properties of solids, liquids and gases are closely related to their molecular structure and the nature of the forces acting between the molecules. To clarify the phenomena of molecular interaction, it is possible to refer by way of example to the behavior between two molecules as a function of the separation distance between them. When the distance between the centers of the molecules is small, strong forces are generated. These forces could be of an attractive type or repulsive type, as shown in the figure below.

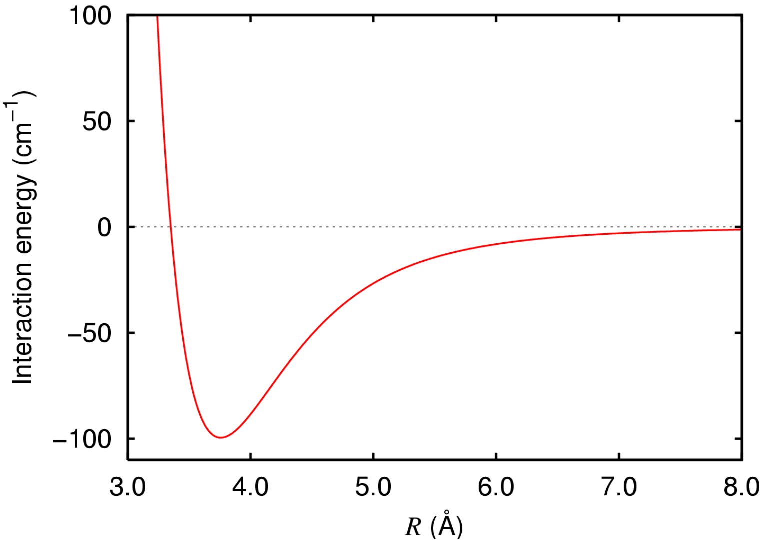

Figure 2.1: Diagram of intermolecular force [4]

At the separation distance in which the change of sign of the reaction occurs, the two molecules are in a condition of relative equilibrium. The relative equilibrium distance in gases is approximately ten times greater than the relative equilibrium distance of liquids and solids. For gases under ordinary conditions, the molecules are so far apart that only weak intermolecular forces act between them. However, in a substance in a gaseous form, during their chaotic path of thermal agitation it happens that the molecules approach or even collide with each other, developing great repulsive forces. In substances in the solid and liquid phase, on the other hand, the molecules have a smaller mutual distance. Therefore, each of them will be subject to the intermolecular forces of the adjacent molecules that limit their possible excursion. As said, the matter is composed of atoms which, in the gas phase, are quite distant from each other. From a practical point of view, however, it is better to ignore, when possible, the atomic nature of a substance and rather consider it as a continuous medium, that is, as a continuous, homogeneous matter, without voids. This abstraction allows to treat the properties of matter as functions of the generic point and to assume that they vary continuously in space without jumps of discontinuity. This approach is correct if the size of the system under consideration is large compared to the intermolecular space.

2.2 Properties

2.2.1 Density

The density of a fluid represents the mass contained within a certain volume. It is generally indicated with ρ and is measured in the International System in kg / m. Its inverse, which would be the volume per mass quantity, is called a specific volume. According to the state equation of perfect gases, the density depends on both temperature and pressure. Since we have that $\frac{p}{\rho} = RT$ the density is directly proportional to the pressure and inversely proportional to the temperature. Liquids and solids, on the other hand, are incompressible substances and the variations in density and volume are obviously negligible.

2.2.2 Bulk Modulus Elasticity

Another important property of fluids is the compressibility module. Generally fluids tend to increase their volume when they are subjected to an increase in temperature or a decrease in pressure, on the contrary, they tend to compress when they are subject to a decrease in temperature or an increase in pressure. The extent of the variation, different for each fluid, depends on some properties of the fluid such as the cubic compression modulus of elasticity and the cubic expansion coefficient, which relate the volume variations respectively with the pressure variations and with the variations of temperature.

$$E = \rho(\frac{\partial p}{\partial \rho})_T = -V(\frac{\partial p}{\partial V})_T$$

A high E value indicates that a high variation in pressure corresponds to a small variation in volume; therefore, a fluid with a very large E value is practically incompressible. In the case of liquids, the compressibility module assumes extremely high values ($E = 2.15 * 10^9 Pa$ for water, $E = 2.85 * 10^{10} Pa$ for mercury, $E = 1.3 * 10^9 Pa$ for gasoline). An important phenomenon that cannot be underestimated is the water hammer in general. This phenomenon can be generated when acting on the regulating elements and create sudden variations in flow rate inside a duct. The consequent abrupt variation in flow determines a considerable variation in pressure and a consequent small variation of fluid density. Because of all this, a pressure wave is produced, which is prepaid throughout the conduit and leads to a vibration of the whole complex.

2.2.3 Viscosity

Another important characteristic of fluids is the viscosity. This physical measures the resistance of the fluid to the , that is the inertia of the fluid in the moment in which in which it modifies its form. We can consider for example a fluid element that initially is like a parallelepiped and we will apply on the surface S a force F that generates a shear force $\tau = \frac{F}{S}$. The fluid element will be deformed constantly under the action of the constant shear τ. Assuming that the upper surface is subjected to the force F is moving with constant velocity U, in a time Δt it will cover a distance equal to $U\Delta t$ producing an angular deformation equal to $tan(\Delta\gamma) = U\frac{\Delta t}{b} = \Delta\gamma$. For a velocity of angular deformation it is possible to write

$$\dot{\gamma} = \frac{\Delta\gamma}{\Delta t} = \frac{U}{b} = \frac{dU}{dy}.$$

The velocity of angular deformation $\dot{\gamma}$ is proportional to the shear applied through a constant μ that depends only from the type of fluid considered and from the temperature che si trova. It is possible to write

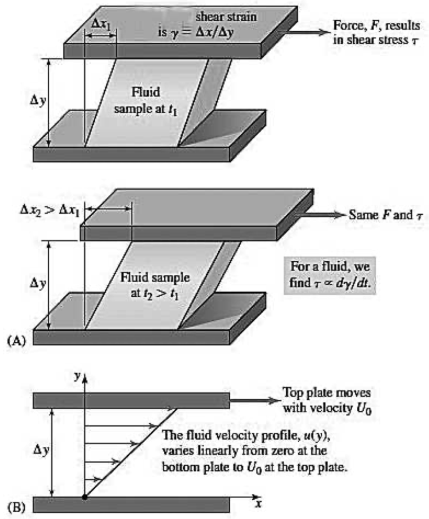

Figure 2.2: Shear stress applied to a fluid block

$$\tau = \mu\dot{\gamma} = \mu\frac{dU}{dy}$$

This formula permits to evaluate the shear generated inside the fluid considering the shear applied on it. The constant of proportionality μ is defined as the dynamic viscosity of the fluid. It has the dimensions of a shear on a velocity gradient and its measure is $Pa * s$. In a Newtonian fluid the velocity of angular deformation is directly proportional to the value of tangential shear, as for example the water, the air, gasoline and oils. They subsist, in general, different exceptions to the linear behavior that covers an important rule into life, as for example the blood, the muds e the paint. In these cases, the ratio between tangential shear and velocity of angular deformation, called apparent viscosity, is variable with the velocity of angular deformation. All the fluids described, where the viscosity change with the change of velocity of angular deformation are called dilatant, whereas the ones that have an opposite behavior are called pseudoplastic. When the shear force increases, the pseudoplastic fluid reduce their viscosity.

There are, then, some substances that are able to resist at final tangential shear and that, consequently, behaves like solids until the tangential shear become higher than a critical value. When this occurs, they starts to behave like newtonian fluid. These substances are called plastic fluids at Binghan. In mechanical fluids usually appears the ratio between the dynamic viscosity and density. This ratio takes the name of cinematic viscosity

$$\nu = \frac{\mu}{\rho}$$

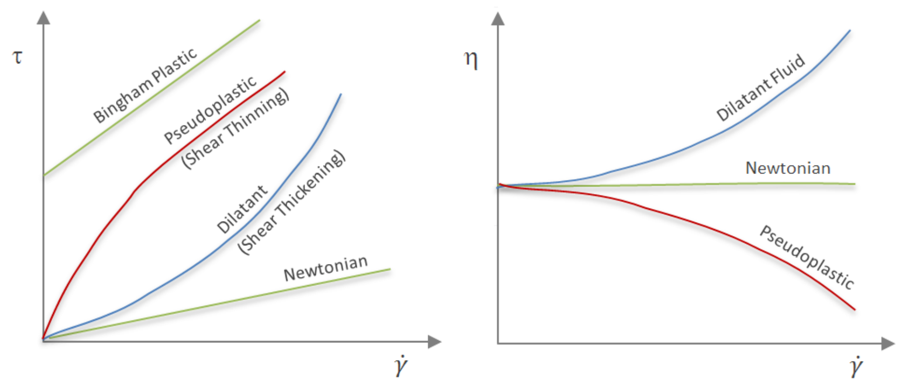

Figure 2.3: Apparent viscosity for different types of fluid

where the dimensions are $\frac{m^2}{s}$. When you want define ν it is possible to notice that it depends by pressure. Indeed if a fluid is compressed its density will increase and consequently the dynamic viscosity will be reduced. This effect is very important for gases whereas it is negligible in case of liquids.

The viscosity depends from the temperature and from the pressure. It is connected with the friction internal force due to the different layers of fluid when the moves one with respect to the other one. Into li quids, the viscosity is due to the cohesion force of molecules, whereas for the gas it is due to molecular collision and so the influence of temperature is fundamental. Increasing the temperature the viscosity will be reduced for the liquids and will be increased for the gas. This is due to because inside the liquid, increasing the temperature, molecules increase more and more their energy and they can win the intermolecular force of cohesion. As consequence, when they are heated up, liquid molecules will

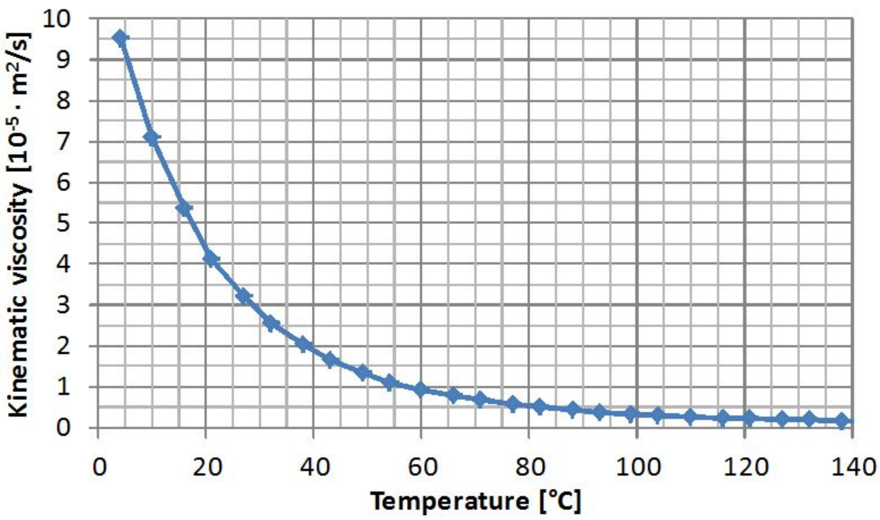

Figure 2.4: Viscosity in function of temperature, [1]

move with more freedom. On the contrary, inside gas, where the intermolecular forces are negligible, increasing the temperature the molecules will move casually with an higher velocity. This implies more molecular collisions for unit volume and for unit of time and so more resistance to the motion.

2.2.4 Mach number

Strictly related to the concept of compressibility is the velocity of sound, or rather the velocity at with a infinitesimal pressure disturbance will propagate inside a fluid in quiet condition. Indeed the density could be written as function of pressure or entropy (beyond as function of pressure and temperature) like:

$$\rho = \rho(p, s)$$

And so it is possible to define the coefficient of compressibility at constant entropy:

$$E_S = -V(\frac{\partial p}{\partial V})_S = \rho(\frac{\partial p}{\partial \rho})_S$$

It is possible to demonstrate that for small perturbation of pressure, such as to be considered isentropic, the velocity of sound can be evaluate using following way:

$$a^2 = (\frac{\partial p}{\partial \rho})_S = \frac{E}{\rho}$$

Therefore the square of the velocity at which a isentropic pressure wave will propagate results to be inversely proportional to the coefficient of compressibility of fluid matter. It is important to understand the condition in which the fluid conditions in which the effects of compressibility are important, i.e. when the variations of pressure inducted from variations of velocity will produce very large variations of density and temperature. For this reason will be defined the number of Mach $M = u/a$, where u represents the velocity of fluid (or of an object immersed inside a fluid in quiet condition). For low value of number of Mach the effect of compressibility could be negligible. For mechanical of fluid, varying the number of Mach M, it is possible to define some types of fluid :

- Incompressible flow $M <= 0.3$;
- Subsonic flow $M <= 0.3$;
- Transonic flow $0.3 <= M <= 1.2$;
- Supersonic flow $1.2 <= M <= 5$;
- Ipersonic flow $M > 5$;

2.3 Separation of flow

An important role in this analysis is due to the number of Reynolds, that is a dimensionless coefficient proportional to the ratio between the inertial forces and the viscous forces. It is usually used to understand how much important are the viscous forces. The Reynolds number is defined as $Re = \frac{\rho v d}{\mu}$ (where ρ is the density, v is the velocity of flow, d is the diameter and μ is the viscosity). The threshold that separates the laminar flow from the turbulent flow is $Re = 2300$, a turbulent flow is a flow in with the viscous forces are not able to win the inertial forces and usually this flow is disarranged and we cannot know in precise manner how it develops and behaves.

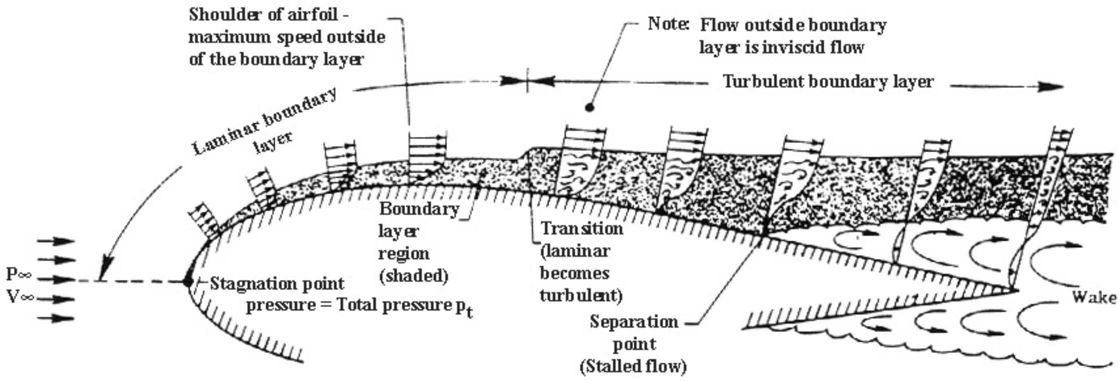

Figure 2.5: Laminar-turbolent transition over an airfoil

The transition process could be sensible to some parameters as for example the pressure gradient (increasing the pressure gradient the transition will be helped), active flow control, roughness of the surface. The problem of a turbulent flow over an airfoil is that increases the drag over the surface and for this reason an important part of research want to improve the characteristic of flow reducing the drag and increasing the lift.

2.4 Control of separation

The separation control is one of the most important theme of research during the last decade because applying an active system on a wing in some cases it is possible to control the stall condition. Usually an active flow control (AFC) system could be implemented using plasma actuators, with steady suction (or periodic forcing), using fluid actuators and moving mass actuator (it is possible to move surfaces over the wing to move the mass without injecting or sucking fluid over the wing). The active systems could be open-loop or closed-loop, in the first case

the actuation command is pre-determine, there will be no changes of its working conditions, instead of the closed-loop control in which its command is due to a monitoring of the state of the system.
Why is the control of separation used? The stall condition (i.e. phenomenon due to separation of flow), when we are facing with a wing of an aircraft, could be a bad condition because it reduces the aerodynamics characteristic of the aircraft. This means that bad aerodynamic condition produce more drag (more fuel consumption), less lift (increasing this phenomenon the situation could become critical and the aircraft could fall down) and noise. The control of separation is important to reduce these problems in order of lift increasing, drag reduction and for a noise mitigation. These characteristics are important because increasing the performance it is possible to increase the climb rate, reduce the runway length needed for take off, speed of approach during landing.

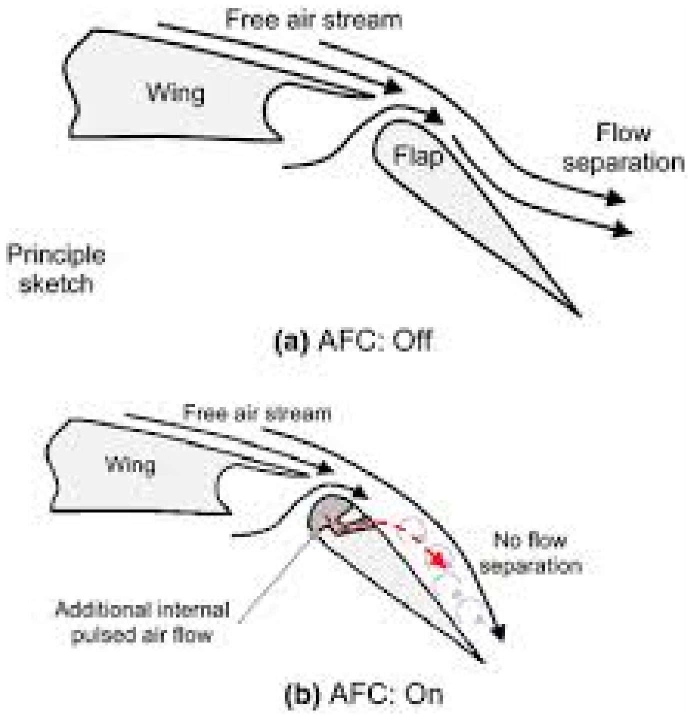

Figure 2.6: Wing without AFC and with AFC

In the figure on the top we can see how without AFC control we have separation of flow, applying the AFC it is possible to eliminate the separation with a fluid more attached at the surface of airfoil.

2.5 Applied aerodynamics: The aerodynamic coefficients

Applied aerodynamics means that the aerodynamics is applied in real cases to calculate the aerodynamic performance of airplanes and space vehicles. In the thesis work will be used some aerodynamic coefficient in order to evaluate the performance of the wing that will be presented later. These coefficients are: C_L, C_D, C_P, C_f and C_μ.

- The first two coefficient are called coefficient of lift and coefficient of drag, dimensionless coefficient proportional to the lift or to the drag and related to the dynamic pressure (function of fluid density and of fluid velocity) around the body and a reference area, i.e. one can write:

  $$C_L = \frac{L}{qS} = \frac{L}{\frac{1}{2}\rho u^2 S}$$
  $$C_D = \frac{D}{qS} = \frac{D}{\frac{1}{2}\rho u^2 S}$$

  A typical trend of C_L and C_D in function of α (angle of attack) is shown in the figure below:

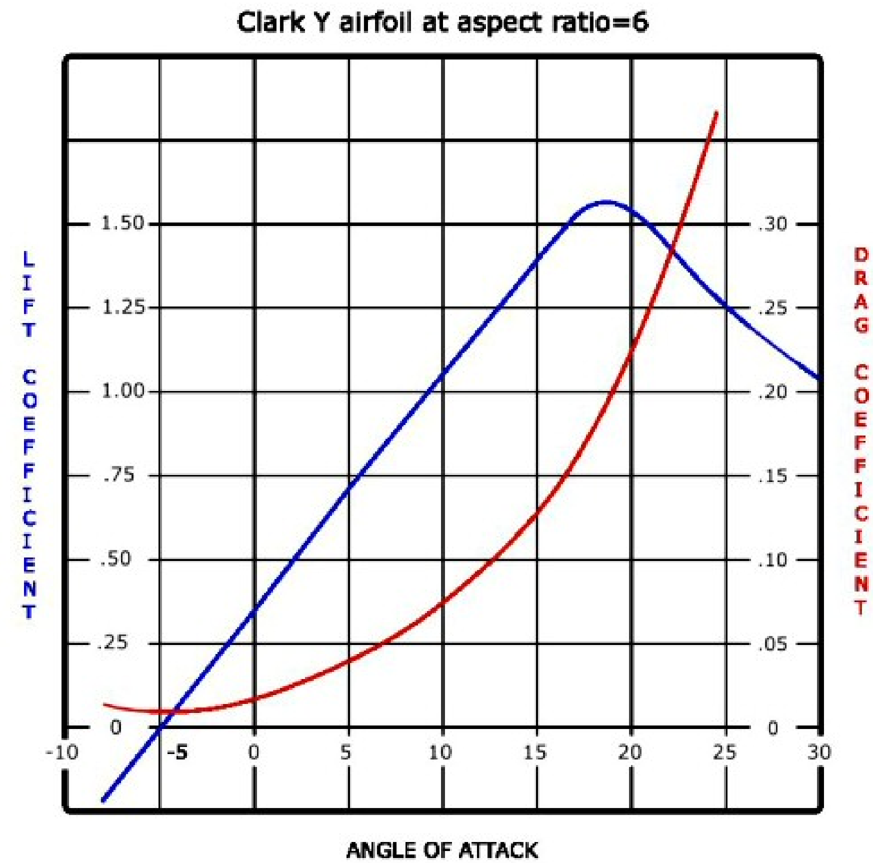

Figure 2.7: C_L and C_D vs α

- The third coefficient is the pressure aerodynamic coefficient. Considering a system like an airfoil, every point over the surface has got a different value of pressure coefficient. Usually the pressure coefficient is used for comparing simulation in different scales because it is independent from the dimensions of the body. The pressure coefficient is defined as

$$C_P = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty V_\infty^2} = \frac{p - p_\infty}{p_0 - p_\infty}$$

- The fourth coefficient is called skin friction coefficient is a dimensionless skin shear stress that is non dimensionless using the dynamic pressure of the flow. The definition is :

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho v^2}$$

where the denominator is the dynamic pressure and the numerator τ_w is the shear stress over the surface.

- The last coefficient is called momentum coefficient and it is a characteristic parameter of the AFC, indeed it measures the strength of the actuator. It is possible to write the momentum coefficient as

$$C_\mu = \frac{\rho V_{jet}^2 A_{jet} sin(\alpha_{jet})}{\frac{1}{2}\rho_\infty V_\infty^2 S} = \frac{V_{jet}^2 A_{jet} sin(\alpha_{jet})}{\frac{1}{2}V_\infty^2 S}$$

On the numerator we have the velocity of jet (the mean value of half cycle) and the area of groove (in the two-dimensional case this correspond to the length of the groove), the denominator is the infinite dynamic pressure multiplied by the surface of airfoil and the sine of angle of velocity of jet w.r.t. the tangent to the surface in the center of the groove.

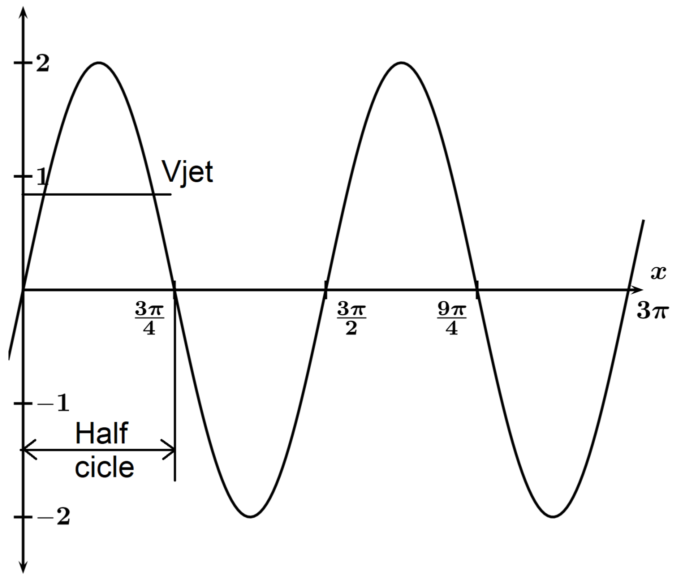

Figure 2.8: Value of V_{jet}

Chapter 3

Modeling

3.1 Overview

In this chapter will be presented both the part that regards the generation of the mesh and the part of simulation. For the former the choice fell back on the Pointwise software whereas for the latter will be used the OpenFoam opensource software. All the data obtained from the simulation will be elaborated through the Matlab software. The purpose of this thesis is to study the behavior of the flow around an airfoil and the figure below shows the most important parts of the airfoil.

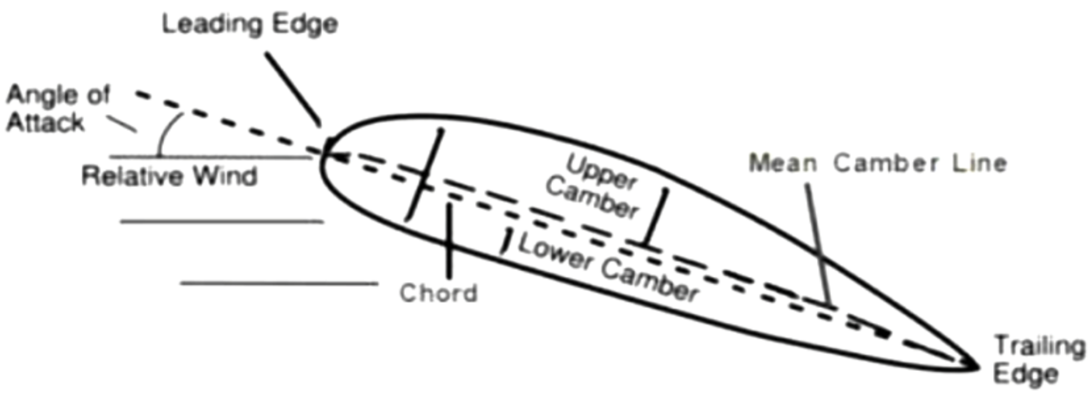

Figure 3.1: Airfoil Description

3.2 Model description

Every-time one facing with an aerodynamic or fluid-dynamic problem he has to consider a body and a computing domain around this body, made with the purpose of obtaining the best results in terms of accuracy. The purpose of this thesis work is to study the aerodynamic characteristic of the airfoil NACA 2412 firstly in not disturbed conditions, because it is important to understand how the flow behaves around the airfoil for a fixed angle of attack, the data will be analyzed to obtain the point of separation of flow and the frequency of detach of vortex from the upper surface of the airfoil. In the figure below is shown the NACA 2412:

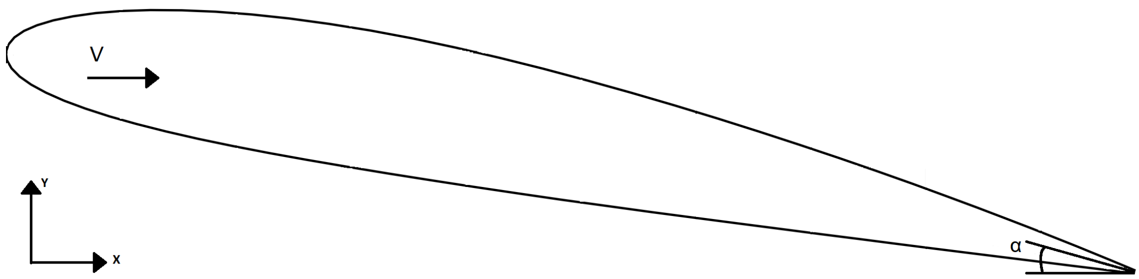

Figure 3.2: NACA 2412 profile

Referring to the upper figure it is possible to see the velocity V of airstream and the angle of attack of the airfoil α that is the angle between the direction of airstream and the x axis. In this thesis will be studied a two-dimensional case with a Reynolds number equal to 30000 with a constant angle of attack but with different meshes (with different values of y^+) in order to evaluate which mesh will be used for the case with AFC. However, since OpenFOAM source code uses dimensional quantities will be used a value of parameters that goes from 0 to 1 in order to insure consistency, only the kinematic viscosity, ν, will have a different value.

3.2.1 Pressure Distribution

The figure below shows the pressure distribution over the NACA 2412

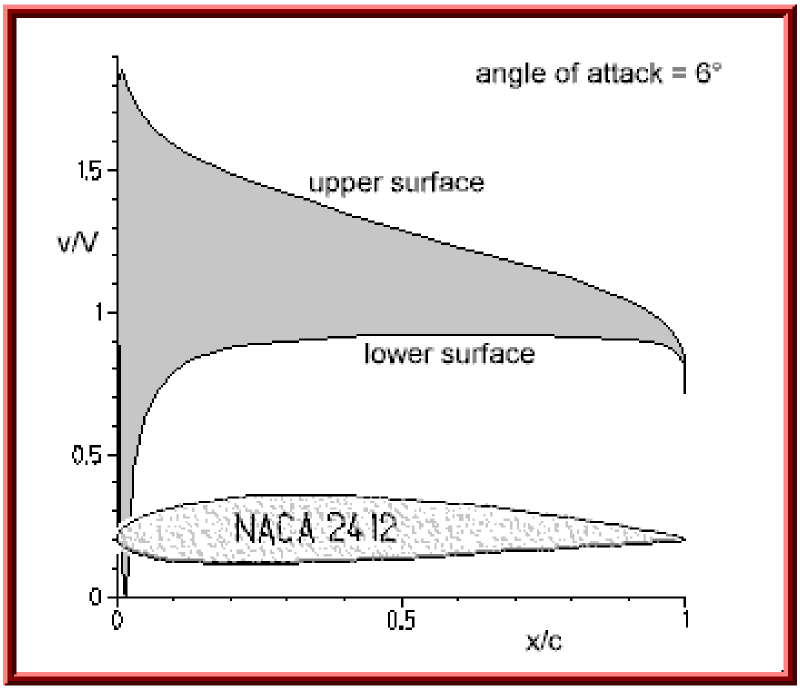

Figure 3.3: Pressure Distribution NACA 2412

3.3 Mesh

The generation of mesh is one of the most critical phase of a simulation because during this phase will be discretized the domain in some small cells in which the simulator will evaluate the Navier-Stokes equations. As told before the software chosen for generating the mesh was *Pointwise*. The domain is inspired to C-grid but instead of this in which it is possible to evaluate the case for every angle of attack (or direction of flow), using the grid in figure below the direction of flow cannot be larger than the angle of the oblique sides. The domain is discretized with a structured mesh around the airfoil to calculate the boundary layer. A small zone around the airfoil is made by unstructured cells, a zone thick of cells in which one wants to study how evolve the flow around the airfoil and finally there is a zone with large cells, a zone less important in which the accuracy is not fundamental for the purpose of this thesis. In the fig. 3.4 is shown the domain considered for this case, in the fig. 3.5 is shown the domain around the airfoil, in the fig. 3.6 it is possible to see the boundary layer.

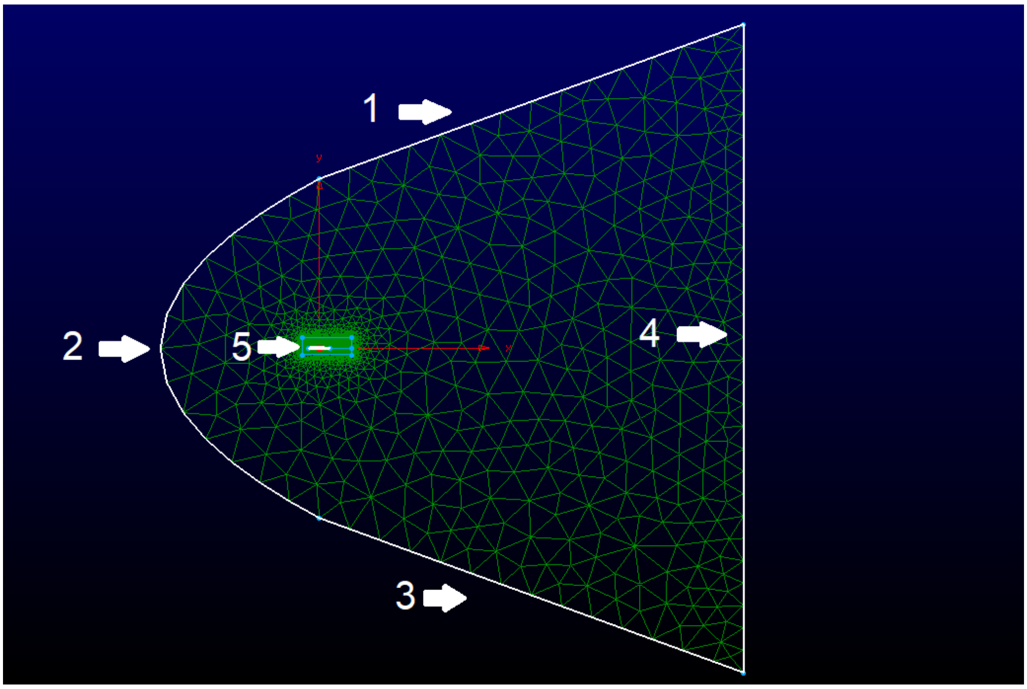

Figure 3.4: Domain

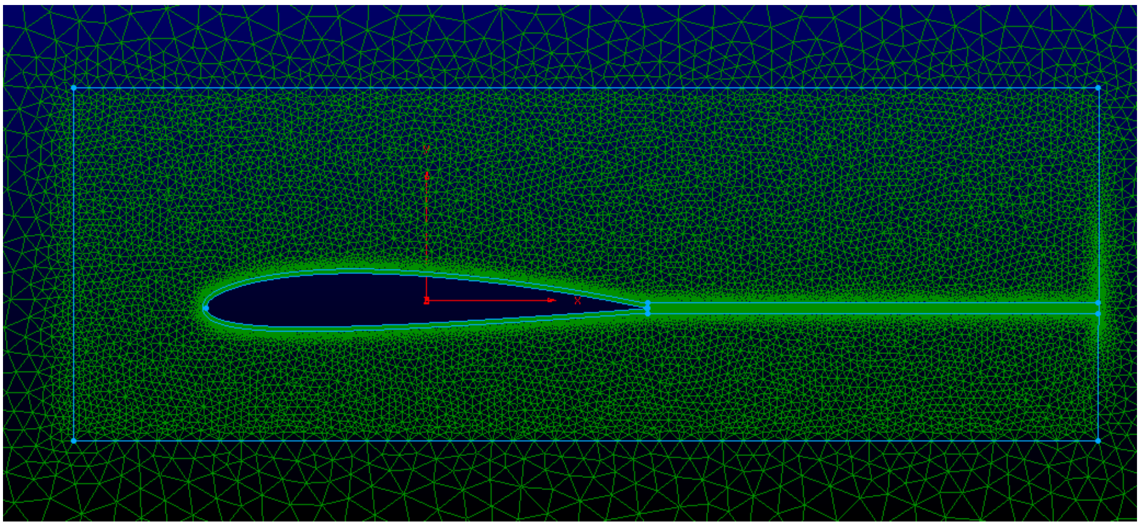

Figure 3.5: Domain around Airfoil

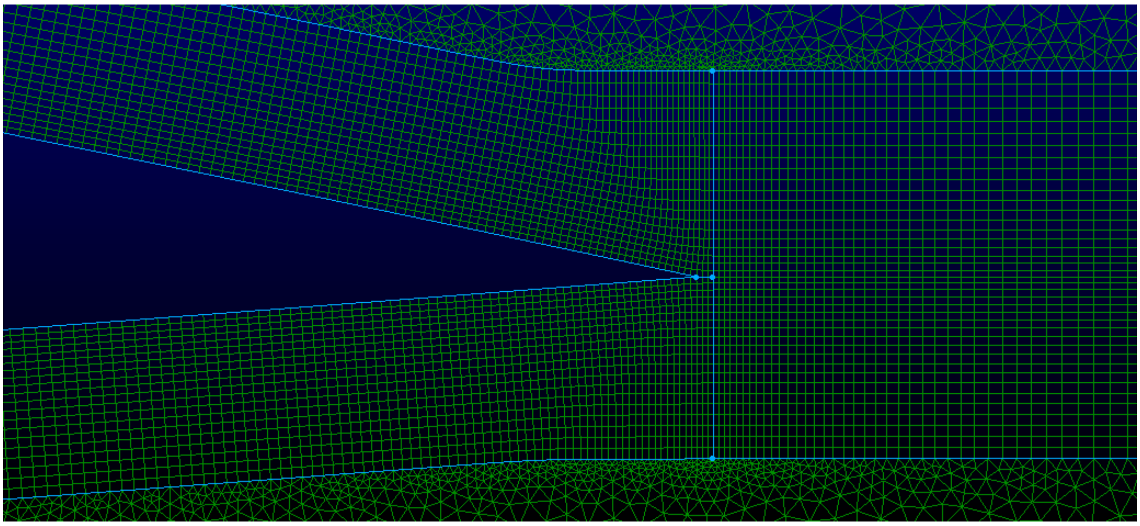

Figure 3.6: Trailing Edge, Boundary layer

3.4 Meshes analyzed

In this section will be presented various mesh considering firstly the case without AFC and later the case with AFC. A characteristic parameter of the creation of the mesh is the y^+, fixing the value of this parameter it is possible to vary the dimension of the first cell near the wall. How is it defined? y^+ is equal to $y^+ = \frac{u_* y}{\nu}$, where u_* is the friction velocity nearest to the wall, y is the distance to the wall of airfoil and ν is the kinematic viscosity of flow. The work of thesis consists in evaluate three different meshes with three different values of y^+. For this case are considered values of y^+ equal to:

- 1 (y equal to $6e - 4$);
- 0.7 (y equal to $3.8e - 4$);
- 0.5 (y equal to $3e - 4$).

What one could expect is that the results of the two last meshes will be more similar (and accurate) because the height of first cell differs less with respect to the first one that has got a larger cell height.
The mesh with AFC is similar to the one without it but it has got the groove in the position of the separation point, evaluated through the evaluation of friction coefficient C_f over the upper part of the surface.

It is possible to see the result in the figure below:

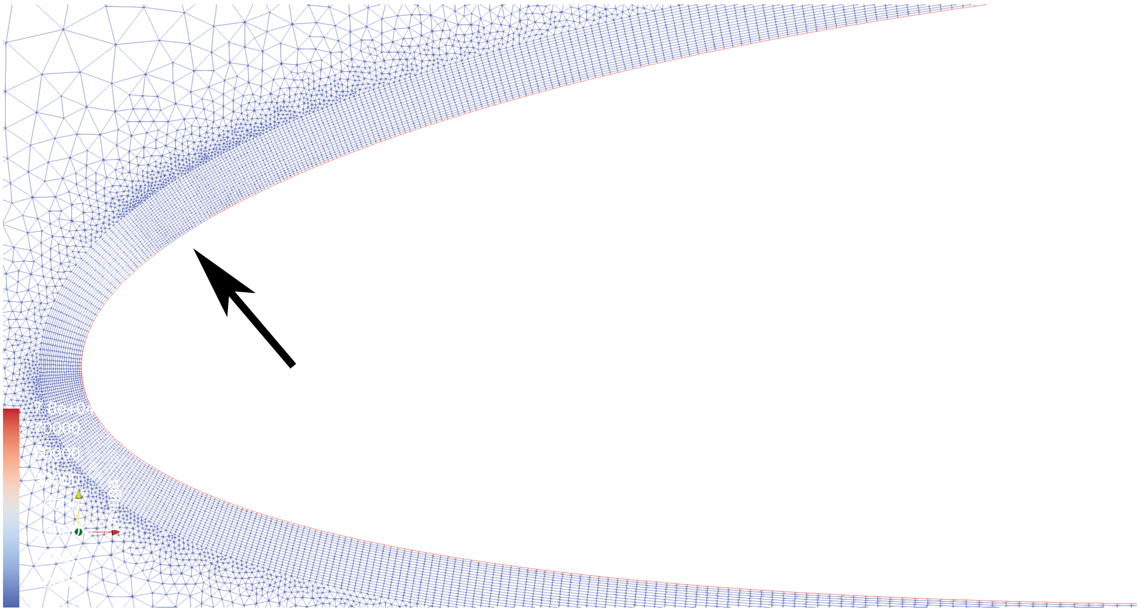

Figure 3.7: Groove near to the separation point

The zone colored in red is the surface of the airfoil whereas the zone indicated by the arrow is the AFC groove. From this point a flow directed in different conditions (depending from the case that one wants to study) in order to improve the performance over the airfoil. The case with a constant sucking is a little bit different, indeed there is not a better position where to put the groove of the AFC but usually it is better to put it upstream with respect to the separation point of flow over the upper surface.

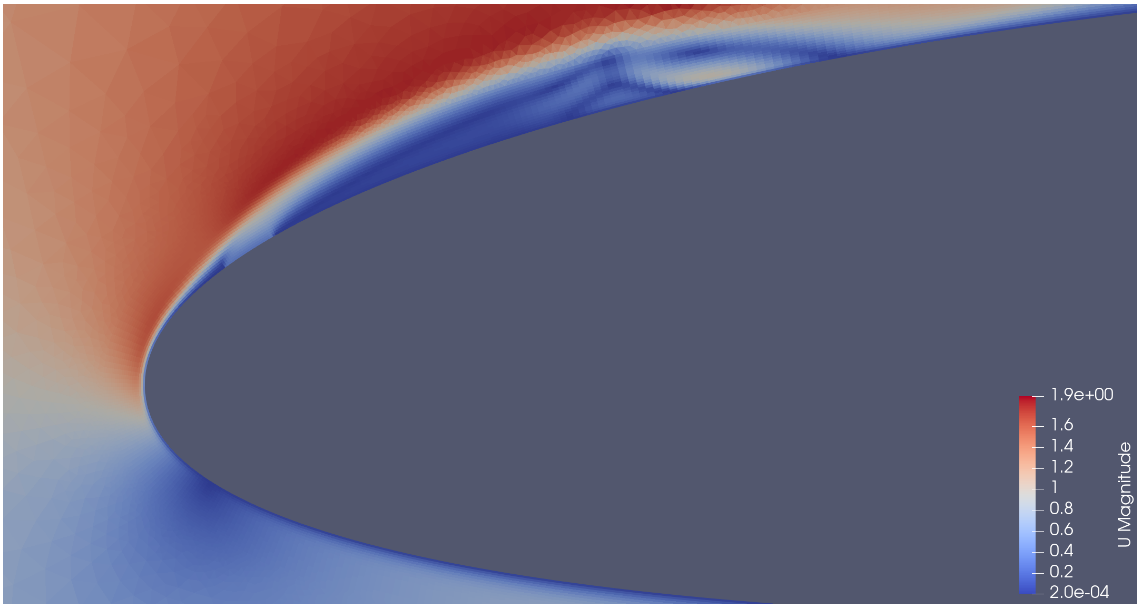

Figure 3.8: Groove near to the separation point

This last figure shows the contours of velocity around the airfoil when the AFC is blowing air (it has increased the thickness of boundary layer).

3.5 Computational Fluid Dynamics - CFD

3.5.1 Navier-Stokes Equations

The *Navier Stokes* equations are a system of differential equation at partial derivatives that permits to model the motion of a Newtonian flow, that is, as told before, a flow in which the viscosity is constant varying the velocity with which it is measured. The *NSE* equation could be written in case of tridimensional and incompressible flow as:

$$\rho\frac{\partial\overrightarrow{V}}{\partial t}+\rho(\overrightarrow{V}\cdot\nabla)\overrightarrow{V}=-\nabla p+\rho\overrightarrow{g}+\mu\nabla^2\overrightarrow{V}$$

where

- ρ is the density of the fluid
- p is the pressure
- $\overrightarrow{g}$ is the vector of gravity acceleration

- $\overrightarrow{V}$ is the velocity vector
- μ is the dynamic viscosity

The solution of these equation is related to the solution of continuity equation. In case of a tridimensional and incompressible flow the continuity equation can be written as:

$$\nabla \cdot \overrightarrow{V} = 0$$

The first of these equation is obtained after the application of Newton's second law to fluid motion, assuming that the stress in the fluid is due to a diffusing viscous term (proportional to the ∇ gradient of velocity) and a pressure term. These equation are not conservative as the *Euler* equations but they are a dissipative system. There is not a solution to these equation, it presents four equation in four unknowns and usually it is solvable after some simplifications. In general case and in the case of this thesis no simplifications could be done and the solutions of these equation is made in iterative way, evaluating for each iteration the value of parameters.

3.5.2 Turbulence model

As told before, there are two conditions of flow, a laminar condition and a turbulence one. In the first case it is possible to predict the behavior of the flow, the second case is affected by causality because the fluid motion is characterized by chaotic changes in flow velocity and pressure. When one is dealing with CFD and wants to know the behavior of flow, has to implement a turbulence model in order to study the particular case with which he is dealing. In literature there are a lot of turbulence models that differs one to each other in how they predicts the effects of turbulence in a statistical manner.

The simulations of the airfoil are made for a $Re = 30000$ and usually one needs to implement a turbulence model for $lowRe$ in order to best

simulate the turbulence. In *OpenFOAM* one could select a vary large variety of turbulence models, the ones more important are the $k-\omega$, $k-\epsilon$ and $k-\omega SST$ and a one-equation Spalart-Allmaras model. In this specific case a *kOmegaSST* where chosen for the simulation. It is a two transport equations model, one for the turbulence kinetic energy, k, and one for the scale of turbulence ω. The *kOmegaSST* [2] model were chosen because it is able to capture the separation of flow that is the aim of this thesis. This particular type of turbulence combines the best characteristic of the $k-\omega$ and $k-\epsilon$ models, because $k-\omega$ is able to study the flow near the wall for studying the boundary layer. SST means shear stress transport because it switch to $k-\epsilon$ model when the flow is far from the wall because the $k-\epsilon$ model is better in free-stream condition. How are parameters defined?
The equation of the model are:

$$\frac{\partial\omega}{\partial t} + \Delta(u\omega) =$$
$$\Delta^2[(\nu+\nu_t\sigma_\omega)\omega] + \frac{\delta\omega}{k}\tau_{ij}\frac{\partial u_i}{\partial x_j} - \beta\omega^2 + 2(1-F_1)\sigma_{\omega^2}\frac{1}{\omega}\Delta k\cdot\Delta\omega + P_{sas}$$

that corresponds to the dissipation of turbulent kinetic energy, whereas the turbulent kinetic energy is modeled as:

$$\frac{\partial k}{\partial t} + \nabla(uk) = \nabla^2[(\nu+\nu_t\sigma_k)k] + \tau_{ij}\frac{\partial u_i}{\partial x_j} - \beta * \omega k$$
$$\delta(\frac{1}{a_p}\nabla p) = \nabla(\frac{H(U)}{a_p}) = \sum_f U(\frac{H(U)}{a_p})_f$$

The turbulence kinetic energy is defined as:

$$k = \frac{3}{2}(UI)^2$$

where U is the reference velocity (in this case the mean flow velocity) and I is the turbulence intensity which is equal to

$$I \equiv \frac{u'}{U}$$

whereu' is the root-mean-square of the turbulent velocity fluctuations and U is the reference velocity.
The turbulence scale is defined as:

$$\omega = \frac{k^{0.5}}{C_\mu L}$$

where C_μ is the turbulence model constant, it usually is equal to 0.09, k is the turbulence intensity defined before and L is the turbulence length scale [2]. The turbulence length scale describes the size of large energy-containing eddies in a turbulent flow. For a fully developed pipe flow this can be equal to

$$L = 0.07 d_h$$

where d_h is the hydraulic diameter.

3.5.3 Boundary conditions

After the discretization n of the domain there is an important step that pass through the definition of boundary conditions where one will define the most important characteristics of the flow. As told before all the coefficient are dimensionless, only the kinematic viscosity will be changed.

Boundary condition	**Value**	**Dimension**
Freestream velocity	1	$[m/s]$
Airfoil chord	1	$[m]$
Freestream pressure	0	$[N/m^2]$
Air Density	1	$[kg/m^3]$
Kinematic viscosity	$3.3e-5$	$[m^2/s]$

Table 3.1: Values of boundary conditions

Also the domain has to be discretize in such a manner the simulator knows which is the inlet and the outlet of the domain. The domain in *Pointwise* is a three dimensional domain because we want to create some domain in the transversal direction w.r.t. the domain and we will give to these domain the boundary patches. In the table below there is the discretization of the patches used for this case

Patches	
Empty	The two side of the three dimensional domain ($left$ and $right$)
Inlet	The $left$ side of the domain [1,2,3] (the two oblique sides and the curved one)
Outlet	The $right$ side of the domain [4]
Wall	The $surface$ of the airfoil [5]

Table 3.2: Type of boundary conditions

What does the words in table mean? The empty patch is referred to the left and right side of the domain, this means that the domain is extruded in the normal direction to the airfoil and the simulator *OpenFOAM* needs a three dimensional mesh but it will collapse the mesh creating a two dimensional mesh joining the left and right sides of the 3D mesh. Through the patch empty it is possible to told to *OpenFOAM* which are these two sides. The inlet (1-2-3 in Fig. 3.4) patch is the part of the domain that describes the inlet of velocity and pressure, in this particular case in which the flow is rotated w.r.t. the airfoil, this angle cannot be higher than the inclination of the oblique sides (1 and 3 in Fig. 3.4). Usually when we impose, as in this case, the velocity of flow, the pressure is set on $freestreamPressure$ condition. It is important to notice that the value of the parameters of turbulence are not affected by the value of velocity and pressure at inlet. The outlet (4 in Fig. 3.4) patch describes the outlet of the domain, as in the case of inlet the choice of velocity is set on $freestreamVelocity$ because its value is set at the inlet whereas the value of pressure is set to 0 because at the inlet a $freestreamPressure$ condition were chosen. The wall

(5 in Fig. 3.4) is the surface of airfoil, this particular patch needs particular attention to the *wallFunction* and *turbulence* because around the airfoil one wants to study the boundary layer and the turbulence. Usually for very low values of y^+ near the wall ($y^+ < 1$ and with 4-5 layers less than this value) the *wallFunction* are not needed. In the case of this thesis with $y^+ = 1$ some *wallFunction* were implemented in the boundary conditions.
First of all it is important to tell that *wallFunction* will be used only for two of these three parameters of turbulence (k, ν and ω) in order to start the simulation [3].

The *nutUWallFunction* were used for the ν near to the walls, this type of wall function permits to evaluate y^+ based on the velocity close to the wall. Whereas for ω were used a *omegaWallFunction*. In *OpenFOAM* the *omegaWallFunction* is a wallFunction that combines the viscous and log equation, and it is important to notice that it depends by the position of y^+ near to the wall, indeed it can switch from viscous to log- thanks to the value of y^+ near the wall. Their value were sets equal to $1e - 12$ because it is better to choose a very small value but different from 0. The third parameter of turbulence, k, were set as $fixedValue$ with this one equal to $uniform$ 0 both for the top and for the bottom surfaces of airfoil.

3.6 Setup

In this thesis work will be analyzed some different cases:

- Free stream condition;
- Active flow control - Periodic forcing;
- Active flow control - Constant sucking.

3.6.1 Case 1: Free stream condition

The free stream condition is the first one case in which the airfoil is subjected to a flow that strikes with an angle of attack $\alpha = 11$ degrees. The airfoil considered for this analysis, as told in the chapter 3, is the NACA 2412. The figures below shows the lift coefficient C_L and the drag coefficient C_D at various angle of attack (α)

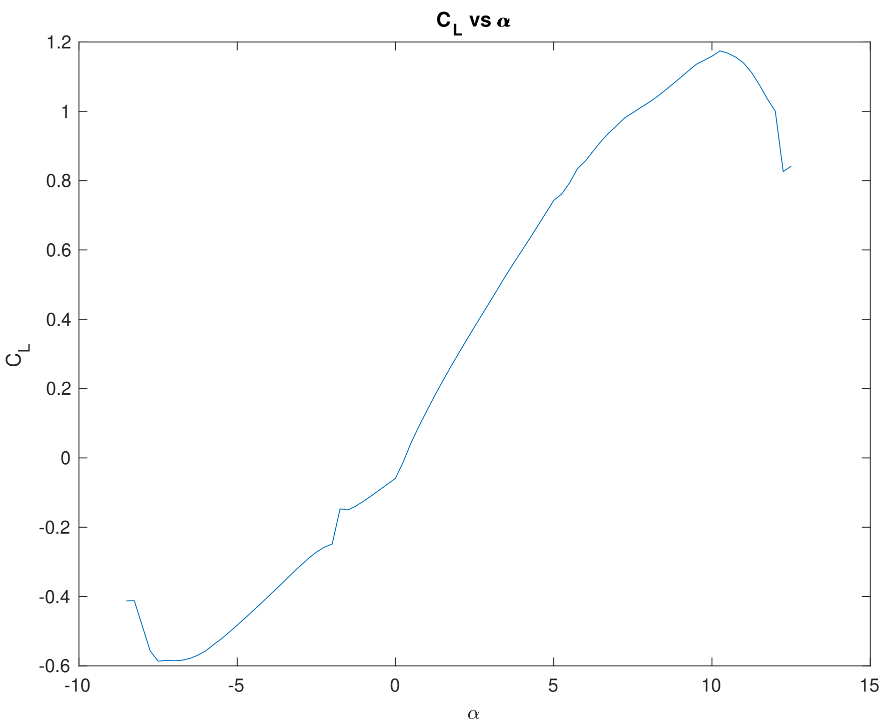

Figure 3.9: C_L vs α [6]

Considering the slope of curve starting from an angle of attack equal to 0 it is possible to notice that increasing the angle also the lift coefficient will be increased. Due to the gradient pressure, the deceleration of flow and other parameters, there is a condition around 10 degrees at which if the angle of attack will be increased the slope of curve change its behavior and it reduces with the increment of α. In this point of change of slope of curve we have the stall that corresponds to a rapidly decreasing of lift coefficient, i.e. a lost of aerodynamic properties.
For what regards the drag coefficient, the situation is counter-posed, indeed starting from 0 to 10 degrees the drag is almost the same but

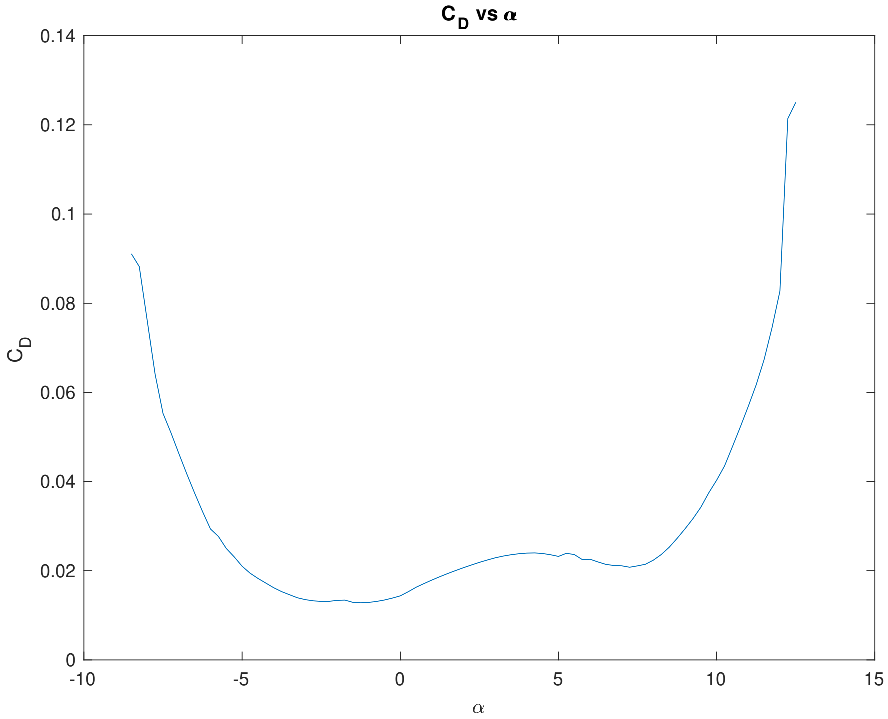

Figure 3.10: C_D vs α [6]

when the stall condition will be present on the wing the drag will increase rapidly and for what regards gasoline consumption, aerodynamic efficiency and noise, this cannot be accepted. For what said before the choice of an angle of attack higher than the angle of stall is a necessary condition to study the performance of wing in stall condition.

3.6.2 Case 2: Active flow control - Periodic forcing

The second case instead of the first one has got the active flow control system in periodic forcing case ON.
Which is the purpose of periodic forcing? An active flow control with periodic forcing let us to control the separation of flow. In general case the separation isn't controllable, a flow will separate due to several reason but with a system that sucks and blows air with a particular frequency we are able in controlling the behavior of vortices.
How was the approach to the problem? The purpose of this analysis is

to simulate some different cases, varying some characteristic parameters and evaluating which is the best combination that allows to obtain the best results.
But first of all there is an important step to be analyzed. Some chapters above the AFC coefficient (C_μ) that tells the strength of the AFC were introduced and the analysis of how much high it needs to be is the first analysis to do for this case.

Investigation about the value of C_μ

The purpose of this analysis is to obtain the best value of C_μ because if it is too much low it will not influence the flow over the airfoil and if it is too much high it changes too much the behavior of flow over the wing.
The analysis started from a value of $C_\mu = 1e - 5$, doing the first simulation it was clear that this value was very low and for this reason its value were increase and in the last simulations done 3 values where considered:

- $C_\mu = 5e - 3$;
- $C_\mu = 6e - 3$;
- $C_\mu = 7e - 3$;

The angle of outflow (α) is fixed to 30 degrees.

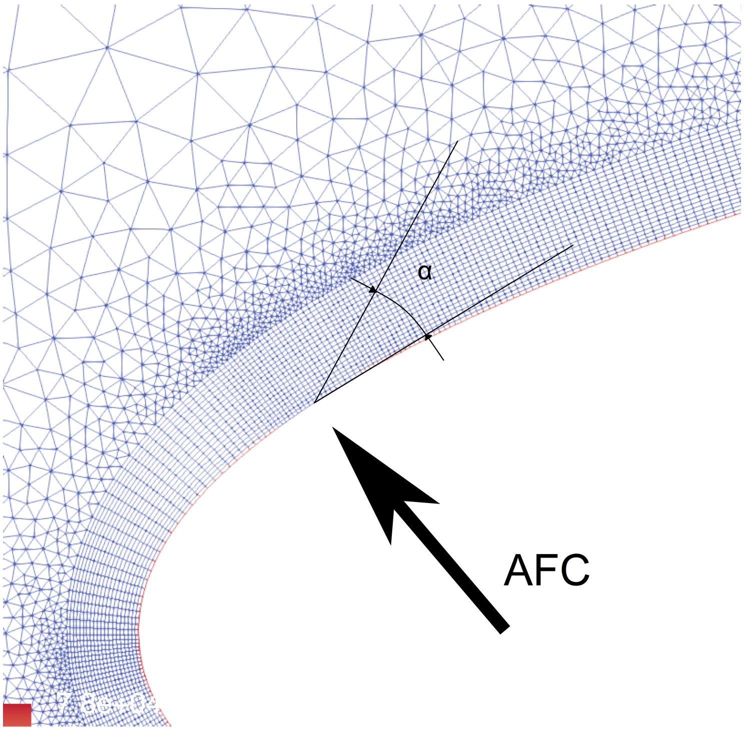

Figure 3.11: $C_\mu = 5e - 3$ vs *free stream flow*

Periodic forcing - Analysis

Which will be the setup of next simulation? A series of 3 different angle α of outflow will be considered and for every angle 5 different frequencies will be used in order to evaluate which configuration of angle and frequency is the best one.
The angle considered are:

- $\alpha = 25$ degrees;
- $\alpha = 30$ degrees;
- $\alpha = 35$ degrees;

The frequency used for every angle are:

- $f = 0.5 * \omega_n$;
- $f = 1 * \omega_n$;
- $f = 2 * \omega_n$;

- $f = 5 * \omega_n$;
- $f = 10 * \omega_n$;

where with ω_n is indicated the main characteristic frequency of generation of vortices. Combining angles and frequency 15 simulations will be made in this phase.

3.6.3 Case 3: Constant sucking

The case with constant sucking is important to understand how much different are the results obtained with a periodic forcing, that is the new frontier of AFC, with respect to the old and well known system with constant sucking.

Differences between periodic forcing and constant sucking

Which are the differences between a constant sucking system and a periodic forcing one? How told in the previous section, when a system blows air it usually wants a direction of outflow that is parallel to the surface, i.e. to the direction of flow near to the surface, in order to energize the particles of flow with low energy and delay the separation of flow (in this case the angle α is lower than 90 degrees). The other condition, the sucking condition, is different and the purpose is to eliminate the particles with low energy and the inlet direction is in the opposite direction with blowing (in this case the angle α is higher than 90 degrees). For this reason when a constant sucking is considered, the direction α of inlet chosen for this case was close to the normal direction to the surface but a little bit inclined on the left in order to facilitate the suction of low-energy particles. Usually a constant sucking system works better with respect to a constant blowing (because the boundary layer is thinner for the former) and prevents the stall condition.
Where is the AFC groove positioned? In the previous case the groove

where positioned near to the separation point founded in free stream condition, now the position of groove is totally casual and the choice made for this specific case study was $x_A = 30\%$ of chord, the figure below shows what just said:

The angle α chosen is equal to 100 degrees and the values of C_μ chosen

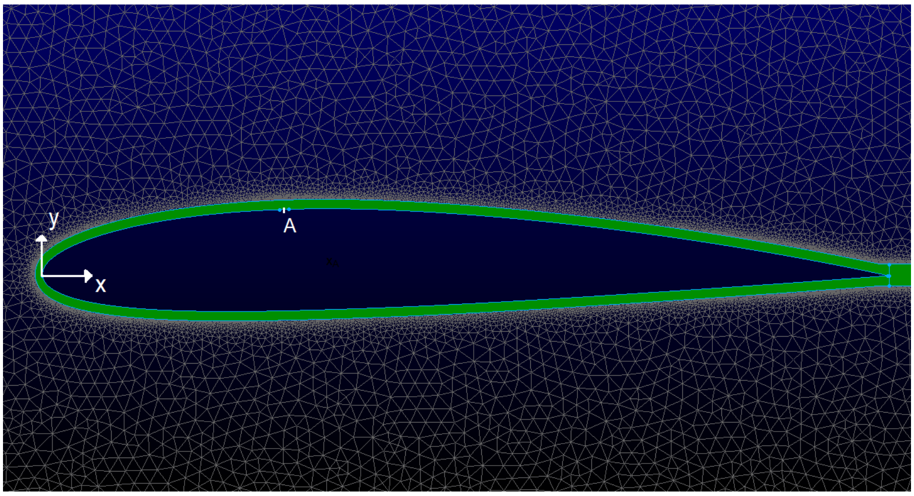

Figure 3.12: Position of groove

are:

- $C_\mu = 0.5$;
- $C_\mu = 1$;
- $C_\mu = 2$;
- $C_\mu = 3$.

Chapter 4

Results

4.1 Introduction

In this chapter the results of the simulation will be showed, starting from the case without AFC and showing the difference between some experimental data, passing through the simulation with AFC ON in $periodic\ forcing$ condition and the analysis of these results. As last case will be showed the standard case using a constant suction of boundary layer and the difference with the simulations in periodic forcing AFC and simulations without AFC.

4.2 Case 1: Free stream condition

In this first case it is important to show the results obtained with the three different meshes with the three different values of y^+. For this first step it is important to compare the results in term of lift coefficient and drag coefficient with respect to experimental values or results predicted through software like $xfoil$. The comparison between the three meshes used are shown in the figures below:

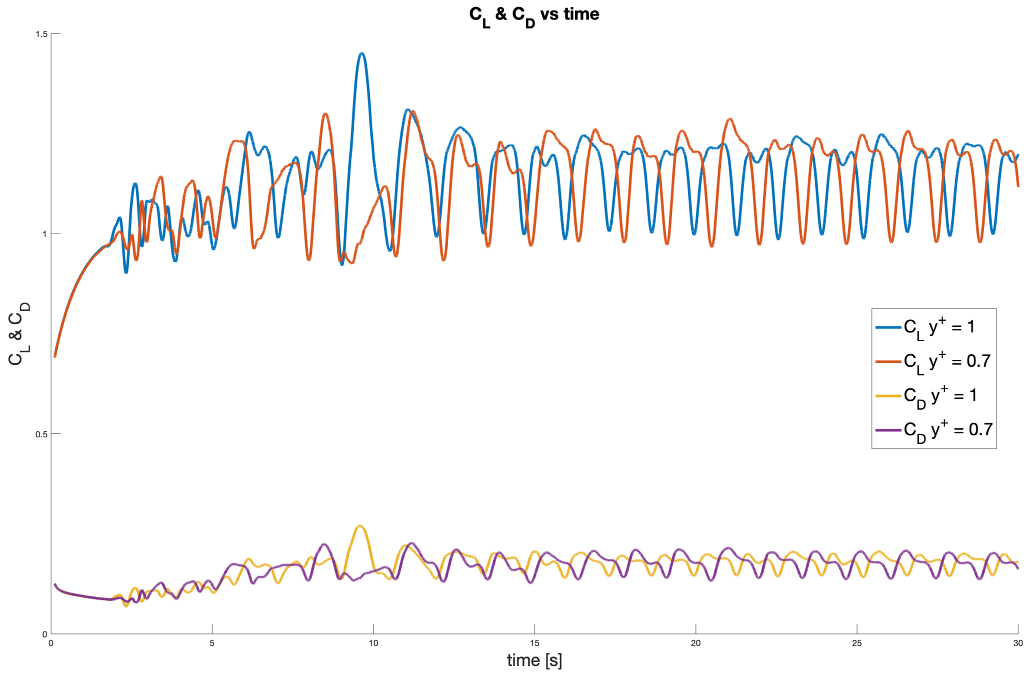

Figure 4.1: C_D vs α

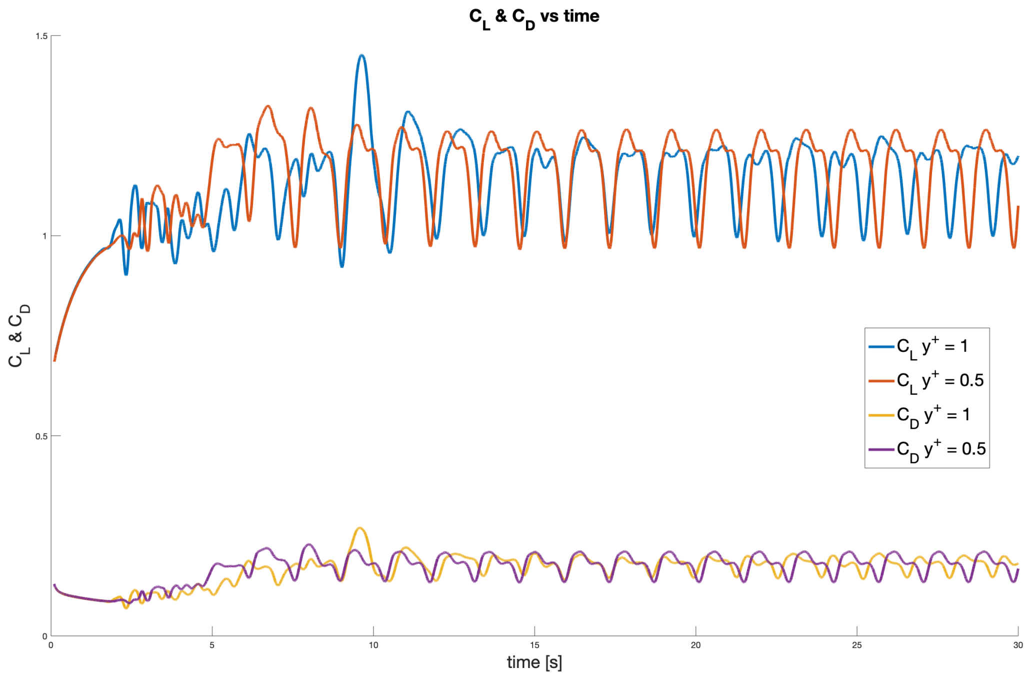

Figure 4.2: C_D vs α

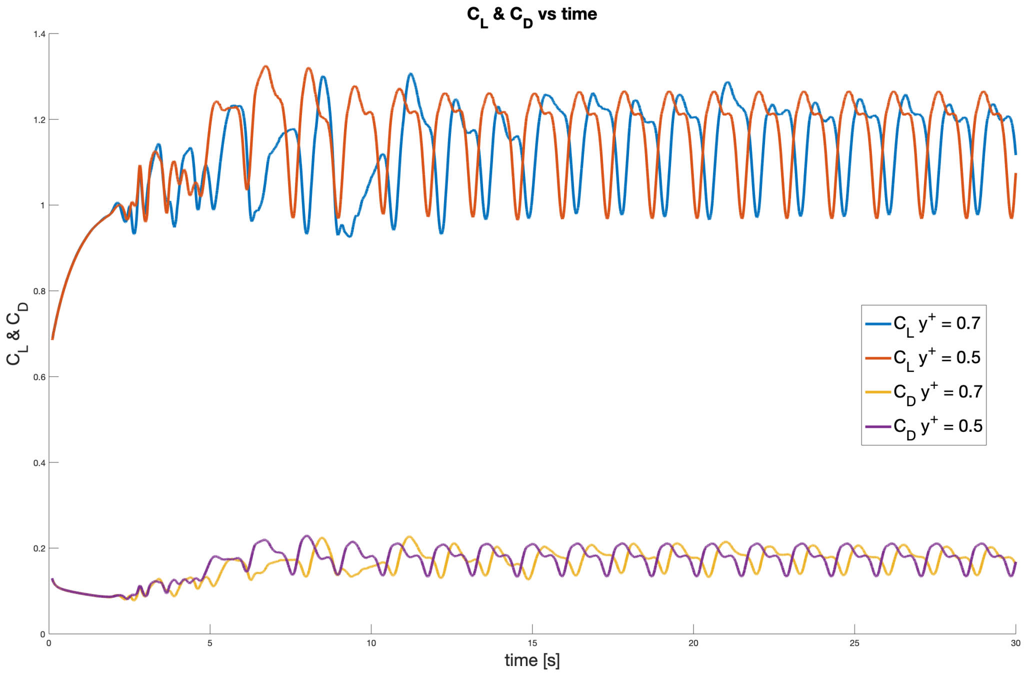

Figure 4.3: C_D vs α

What is it important to notice in these figures? First of all the analysis of the first three figures that regards the values of C_L and C_D of three meshes is necessary. The first picture shows the difference between the two first value of y^+. The key of this analysis resides into the reaching of a steady condition, in which the behavior of the C_L and C_D is almost the same during the time, this means that the detach of vortices has acquired an own shape. When this happens the simulator (*OpenFOAM*) has reached a "steady" condition and the simulation could be stopped. Comparing the two meshes they reach this "constant" behavior after about 15 seconds and it is clear that the behavior of the lift coefficient with $y^+ = 0.7$ is more constant than the one with $y^+ = 1$. Analyzing now the second picture (Fig. 4.4), it's very nice to see how the mesh with $y^+ = 0.5$ reaches a "steady" condition after less than 10 seconds and the shape of lift coefficient is equal after the detach of every vortex. These two first pictures are important because they show a behavior of $y^+ = 1$ that is different compared with the other two. The analysis culminates with the third picture, a comparison between the

two last meshes and apart from the fact that there is a shift due to the velocity with which the software has reached the solution, the behavior is the very similar.
How is it possible to prove this analysis? As told before when an airfoil is immersed in a flow there are vortices that detach from the upper surface of it, if this flow is steady also the frequency of detach of vortices will be steady and an analysis of this frequency is required. The figures below show the spectral analysis made on the lift coefficient after the reaching of "steady" condition.

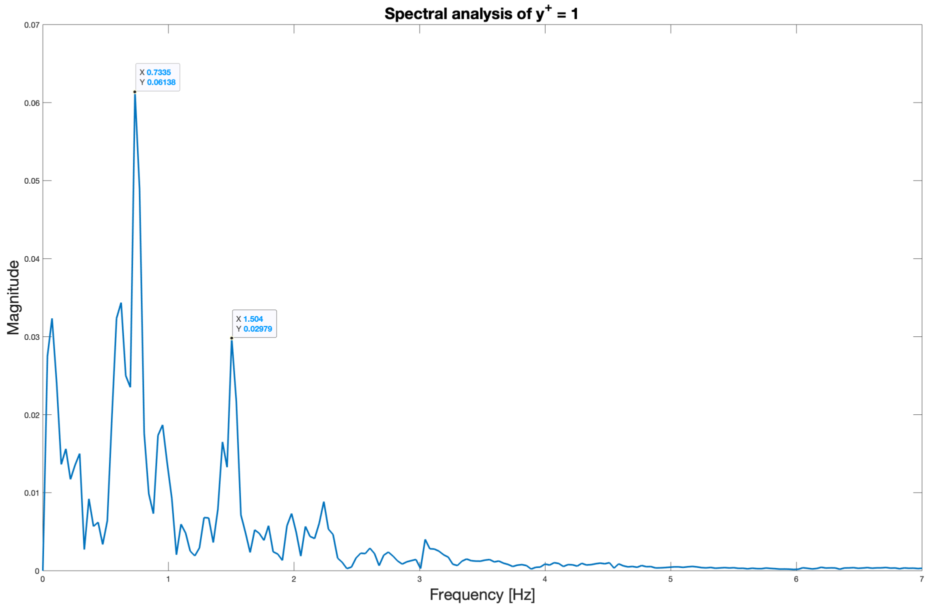

Figure 4.4: Spectral analysis of $y^+ = 1$

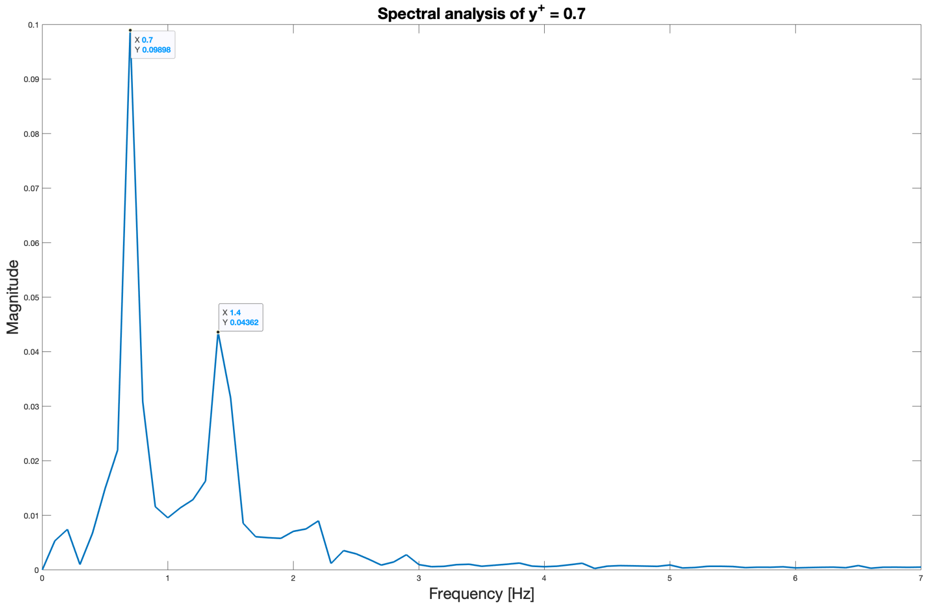

Figure 4.5: Spectral analysis of $y^+ = 0.7$

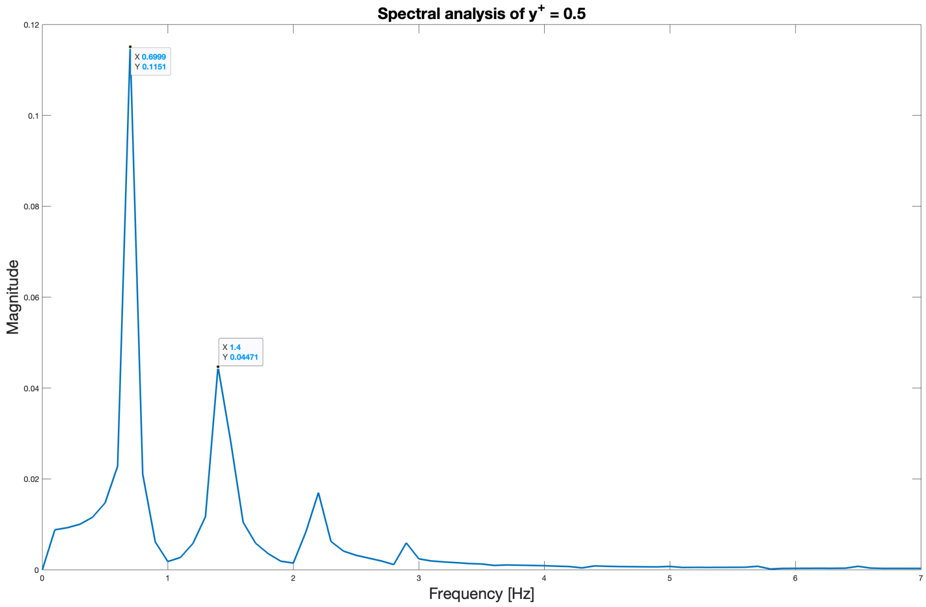

Figure 4.6: Spectral analysis of $y^+ = 0.5$

It is very clear that the two last meshes return an equal value of mean frequency (first harmonica) and of other important harmonicas. In the table are shown the average values of lift and drag and also the aerodynamic efficiency defined as:

$$E = \frac{L}{D} = \frac{C_L}{C_D}$$

	$y^+ = 1$	$y^+ = 0.7$	$y^+ = 0.5$
Lift	1.1445	1.1571	1.1608
Drag	0.1722	0.1777	0.1794
Efficiency	6.64	6.51	6.47

Table 4.1: Average values of aerodynamic coefficient over time
Free stream case

It is possible to notice from this last table the difference in terms of value of lift and drag coefficients from different meshes and it is clear that the meshes obtained using $y^+ = 0.7$ and $y^+ = 0.5$ return values of C_L and C_D more or less similarly compared with the mesh with $y^+ = 1$. The final analysis consists into comparing the results obtained from the simulation with experimental or other simulation in order to understand the trustworthiness of the simulation. The figures below show some results obtained with a three dimensional case study of flow over a NACA 2412 profile for different values of α and for different values of Re [5].

Table No. 4/ **Coefficient of lift, drag and moments at different angles of attack, at Reynolds number 60000.**

A	Cl	Cd	Cm 0.25
[°]	[-]	[-]	[-]
0	0.261	0.01532	-0.05
4	0.73	0.01841	-0.055
8	1.128	0.02794	-0.059
12	1.142	0.10236	-0.027

The two values of Re chosen for these analysis, published on the *International Journal of Engineering Research and General Science*, were

Table No. 6/ **Coefficient of lift, drag, pressure and moments at different angles of attack, at Reynolds number 100000.**

α	Cl	Cd	Cm 0.25
[°]	[-]	[-]	[-]
0	0.261	0.01197	-0.051
4	0.733	0.01483	-0.055
8	1.139	0.02418	-0.059
12	1.144	0.09473	-0.029

Figure 4.7: http://pnrsolution.org/Datacenter/Vol3/Issue2/27.pdf

60000 and 100000. Only the last row of these table will be considered as a point of reference in order to compare the values of C_L and C_D of this work with the one done in this thesis. The table that describes the results in this thesis work has got a value of C_L around 1.5 and it is important to consider the difference between the two dimensional case and the three dimensional case because in the latter case there exists phenomenon of circulation of flow that change the values of lift and drag coefficient compared with the two-dimensional case. For what told before it is possible to imagine that lift in two dimensional case is larger compared with the one in three dimensional case. The second problem is due to the number of Re and one can think that the Re is very different but comparing the results in the two tables, at 12 degrees the difference of lift coefficient is of 0.2% and for this reason it is allowable to compare these results with the simulation done in this thesis work. Also the drag coefficient needs to be analyzed. From the comparison of the same cases considered for lift coefficient is clear that drag coefficient evaluated in this thesis is larger with respect to the second one case. This difference could be due to the Re number because in this case the difference from the case with Re 60000 and Re 100000 is around 10% and reducing Re will increase the drag because the flow has low energy.

After all this analysis started from the consideration of the meshes used in free stream condition, the comparison with each other and with some results founded in literature in order to validate the quality of simulation, the mesh with $y^+ = 0.7$ is chosen because the results obtained are

very similar to the one with $y^+ = 0.5$ but the times required for the simulation in the former case in less than in the latter.
The last thing to do regards the evaluation of separation point over the surface, it is necessary because in this position will be put the groove of AFC. The figure below shows the friction coefficient over the cord , the point in which the friction coefficient will be equal to 0 is the separation point.

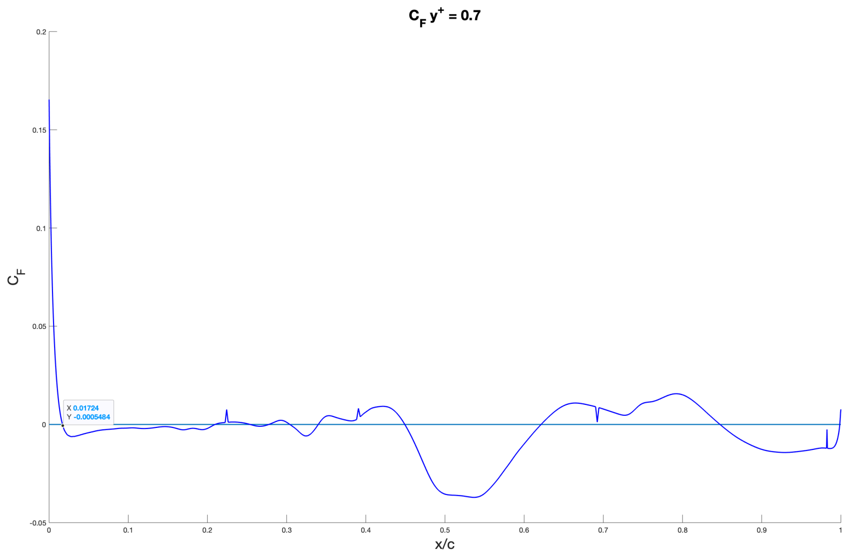

Figure 4.8: C_f vs x/c

The point at x/c = 0.01724 is separation point and in this point an AFC groove in forcing condition will be inserted in order to control the separation of the flow.

The Figures presented in this last part of this subsection shows some contours of velocity, pressure and flow streamline of this particular case. A very short discussion is necessary for the contour of pressure, the zones in blue are all the vortices that are generated over the wing, the farther is the one that has already detached the wing, there is another vortex that is detaching the wing with the lowest value of pressure (there is a circle, that correspond to the vortex, with the lowest value of pressure in the scale) and then over the surface there are other vortices that are increasing their dimension. The vortex could be seen also in the contour

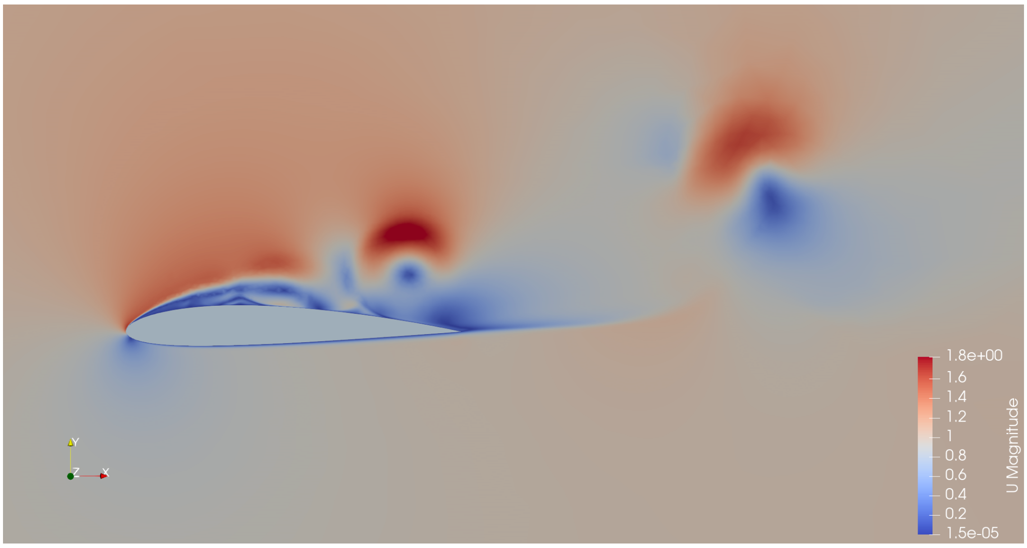

Figure 4.9: Contours velocity

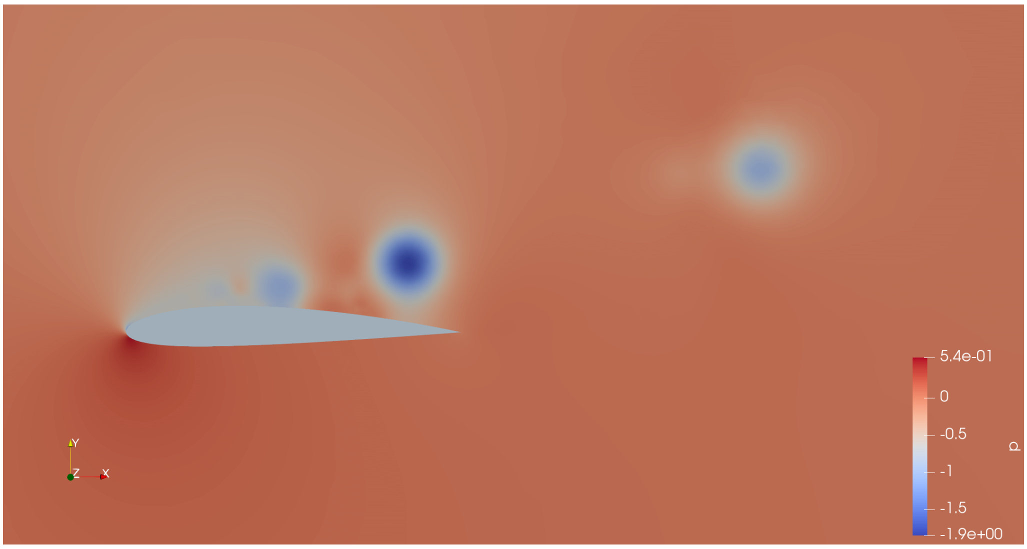

Figure 4.10: Contour pressure

of velocity (the circle blue and red) and in the streamlines.

4.3 Case 2: Active flow control - Periodic forcing

The results of the different sub-cases considered for periodic forcing condition with all the three cases described in chapter 3 will be presented in this section.

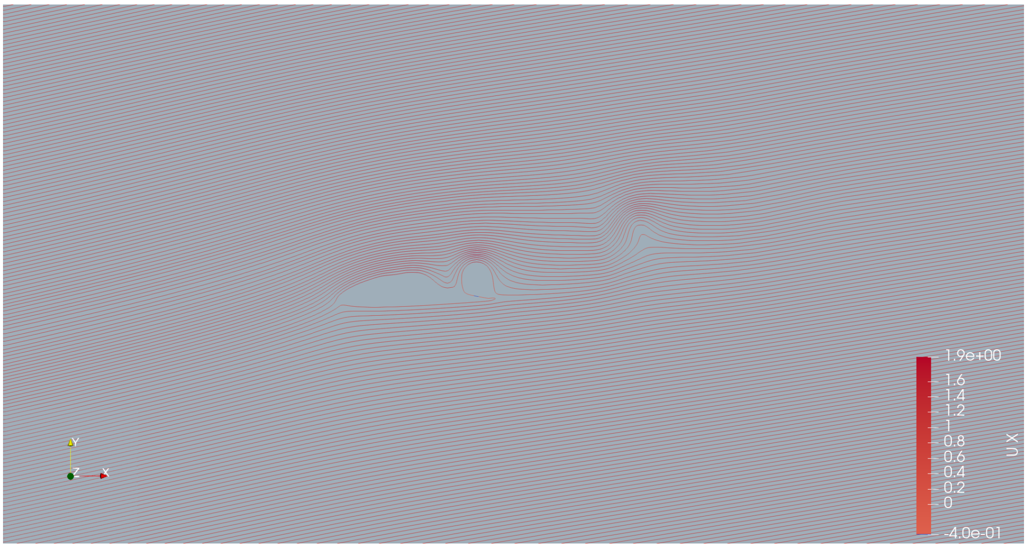

Figure 4.11: Streamlines without AFC

4.3.1 Investigation about the value of C_μ

In the figure below are shown the coefficients of lift and drag vs time in case of periodic forcing with a proper characteristic frequency equal to the one found with the spectral analysis and the flow in free stream condition using $y^+ = 0.7$.

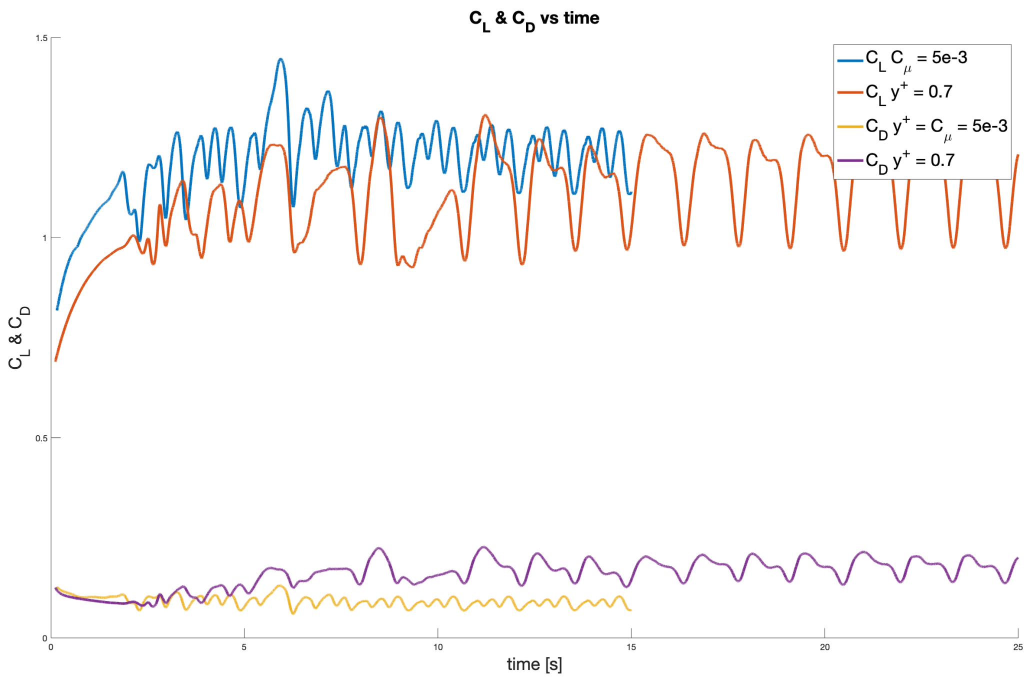

Figure 4.12: $C_\mu = 5e - 3$ vs *free stream flow*

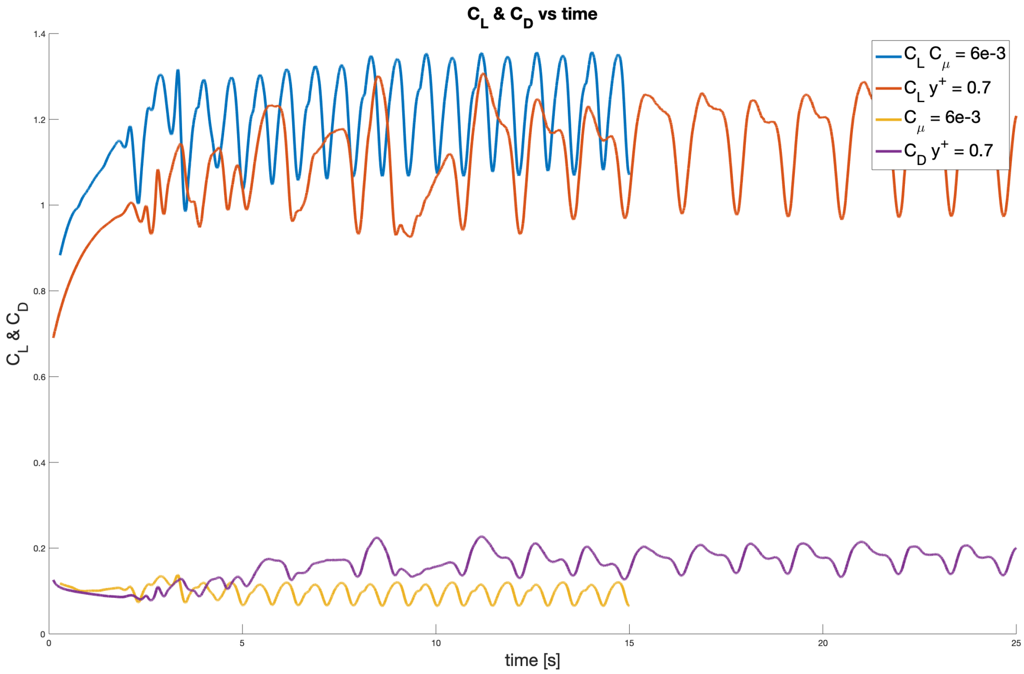

Figure 4.13: $C_\mu = 6e-3$ vs *free stream flow*

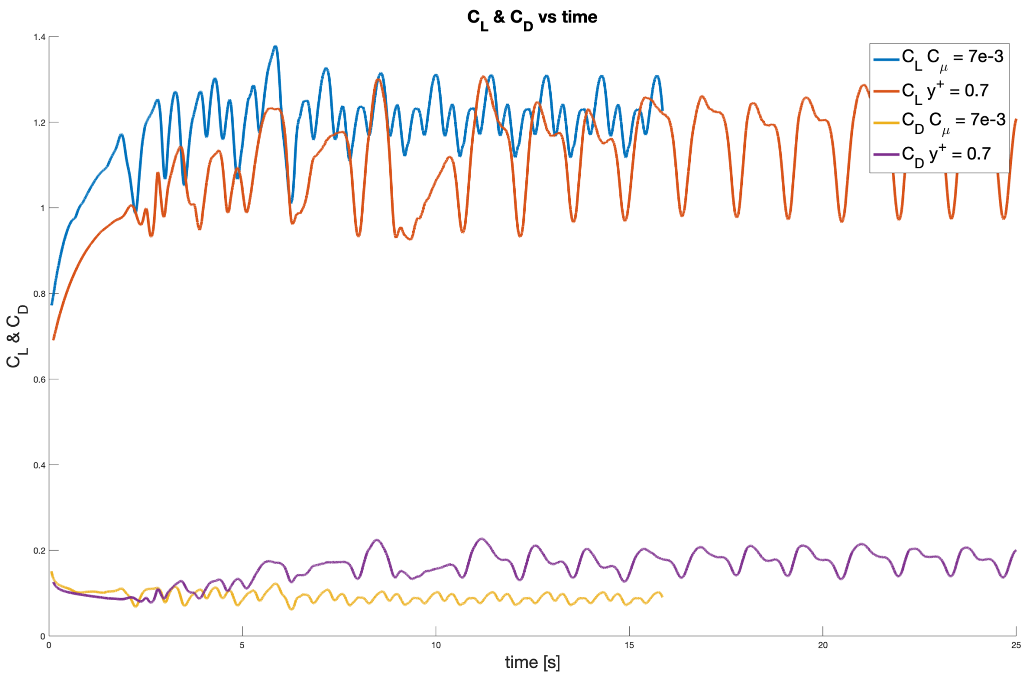

Figure 4.14: $C_\mu = 7e-3$ vs *free stream flow*

From these last figures what is it important to notice? As told a lot of times in the argumentation of this thesis, when an active flow control is introduced in a case like that, what one expects is the reduction of drag and an increasing of lift coefficients. These three cases are important because starting from the first case, the choice of the coefficient needs to be done on the basis that it is the best one coefficient that allows us to improve the expected characteristic. Going from the first case and analyzing the second one, the blue line (that describe the C_L coefficient when AFC is active) is better in second case, in terms of lift increase and stability. This improvement is evident also when the C_D coefficient is analyzed, both when we compare it with the free stream case and when we compare it with the $C_\mu = 5e - 3$. But what one expects is that once this coefficient increase, also the performance will be increased, but analyzing now the last case both the values of lift coefficient and drag are increased and at this point what we want to evaluate is the efficiency to understand the real difference. The table below shows these differences in terms of average values of lift and drag coefficients and efficiency.

	$C_\mu = 5e - 3$	$C_\mu = 6e - 3$	$C_\mu = 7e - 3$
Lift	1.2292	1.2132	1.2327
Drag	0.0952	0.0877	0.0955
Efficiency	12.9089	13.8345	12.9108

Table 4.2: Average values of aerodynamic coefficient over time AFC in periodic forcing mode

The lift and drag in the three cases are shown in the table, what does these values mean? First of all comparing these values with the free stream case we notice an improving both in terms of lift and drag (in free stream condition the lift coefficient was equal to 1.1571 and drag coefficient was 0.1777) and this reflects the last row of the table, where the efficiency is evaluated. With respect to AFC OFF the improvement of efficiency is around 73%, a very nice results if one thinks that this improvement is due only to some air that is blowed o sucked over the upper side of wing.

At this time the threshold between too low and too high values of C_μ is founded and for this reason the final value chosen for the next simulation is $6e-3$.

4.3.2 Periodic forcing - Analysis

In this subsection will be analyzed the main important study made in this thesis work indeed it is an analysis of performance of active flow control once some characteristic parameters will be changed. From the previous analysis the best value of C_μ at a prescribed frequency (in the previous cases it was equal to the characteristic frequency of detach of vortices) were founded. Now the analysis will be moved into varying the frequency of active flow control and the angle of outflow (Fig 3.11).

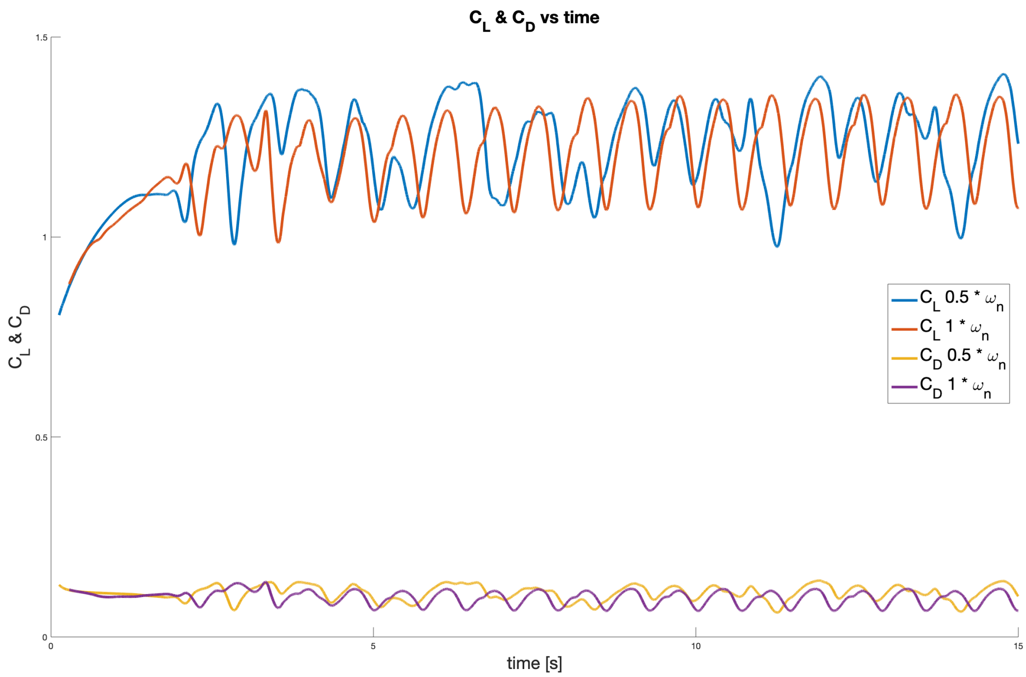

Figure 4.15: Comparison between $f = 0.5 * \omega_n$ and $f = 1 * \omega_n$

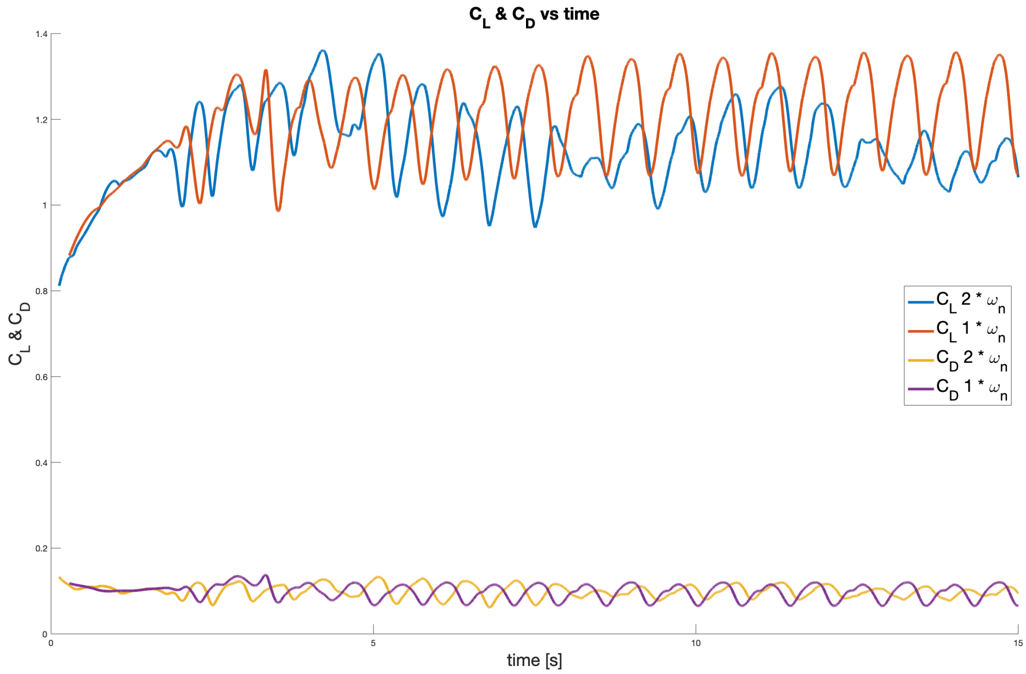

Figure 4.16: Comparison between $f = 2 * \omega_n$ and $f = 1 * \omega_n$

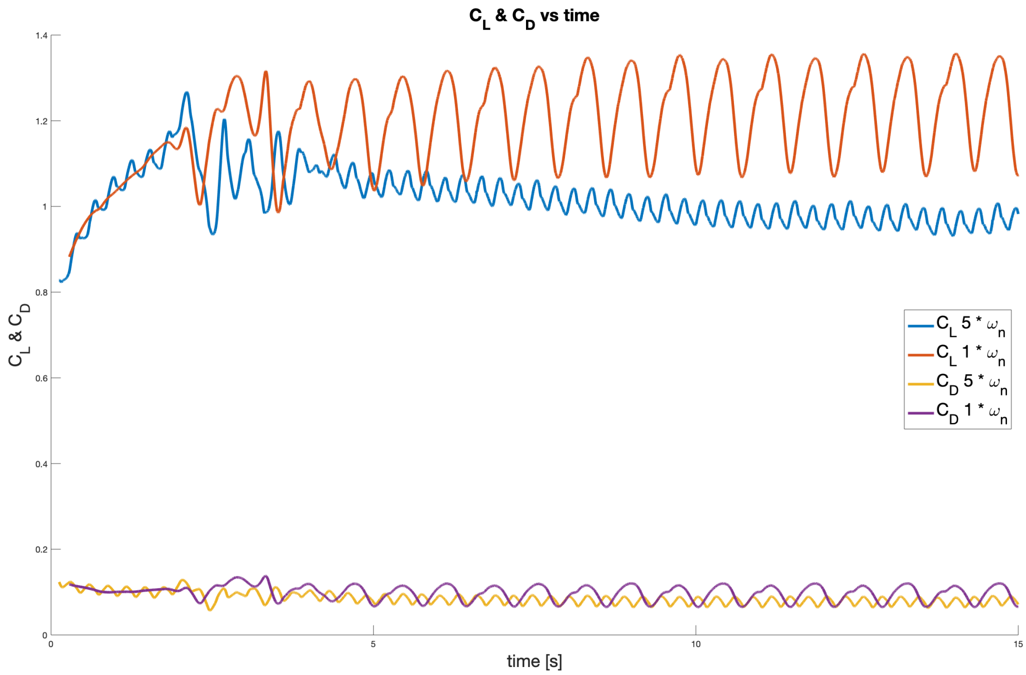

Figure 4.17: Comparison between $f = 5 * \omega_n$ and $f = 1 * \omega_n$

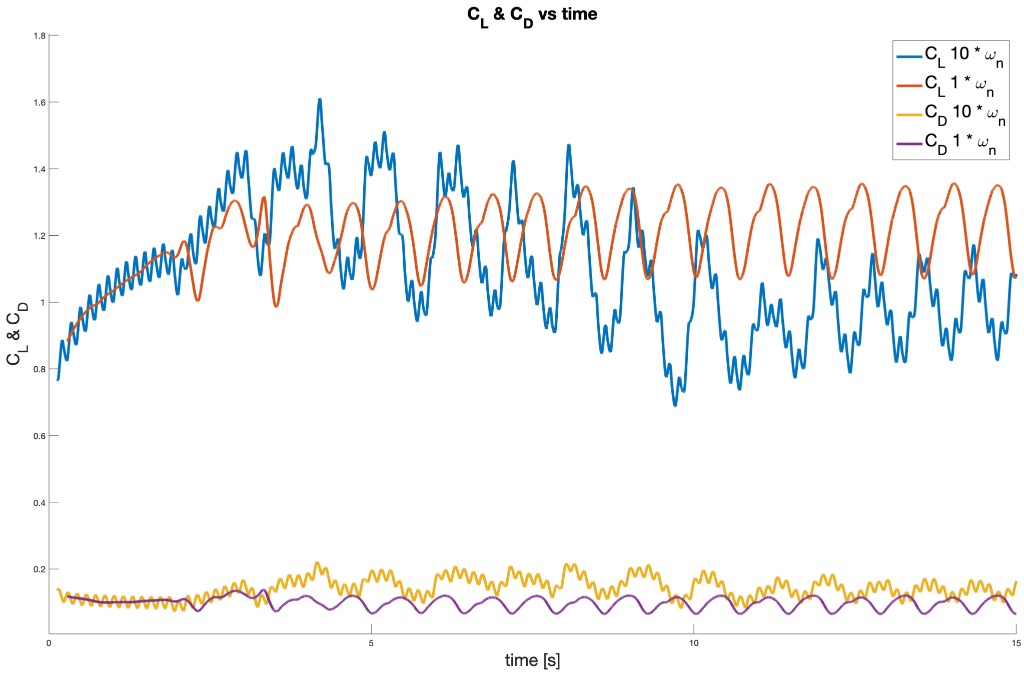

Figure 4.18: Comparison between $f = 10 * \omega_n$ and $f = 1 * \omega_n$

Analyzing the first four graphs it is possible to see the difference between the usage of 5 different frequencies. The comparison is between the main natural frequency and some multiple and submultiple of this frequency. It is clear that the main frequency is used for AFC, the behavior is the more linear possible compared with the other 4 frequencies considered. What seems to be clear is the effect on the performance due to high and low frequencies, indeed at low frequencies (less than ω_n) the lift and drag remains almost similar to the main case, increasing the frequency over the main one, the lift will be reduced and also the drag is reduced, this might through at an increasing of efficiency, but this last parameter will be evaluated later. An important thing to observe is the Fig. 4.18, in this case the multiple of 10 times the main frequency is considered and it is clear that it destroy completely the boundary layer and the aerodynamics over the wing, the lift is decreased and the drag is increased, this is counter-posed with the objective of this thesis.

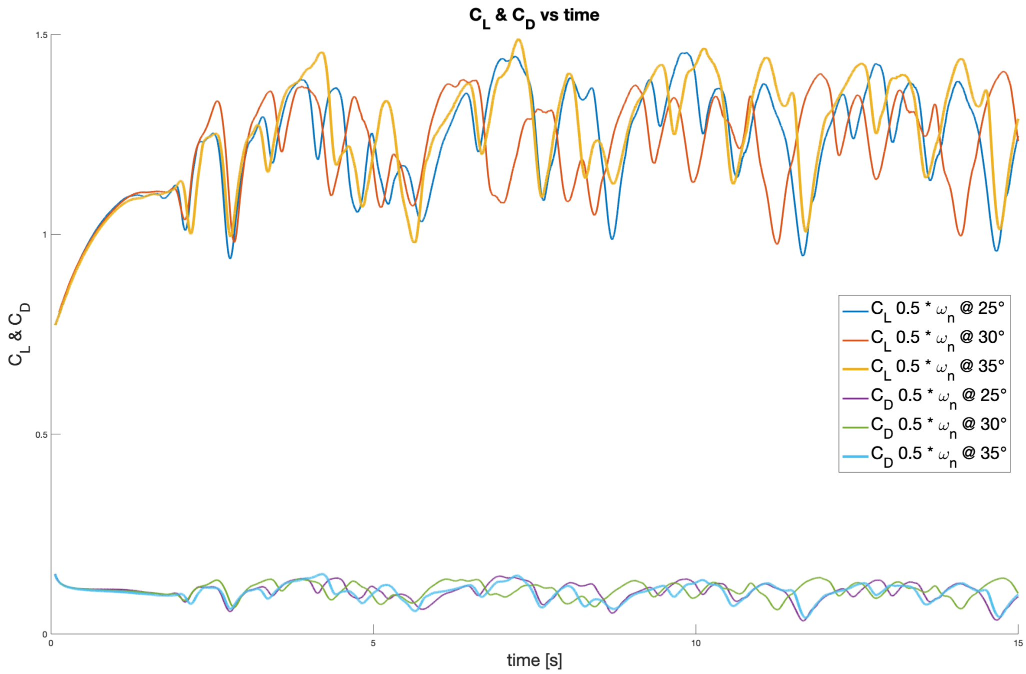

Figure 4.19: Comparison between three angles at $f = 0.5 * \omega_n$

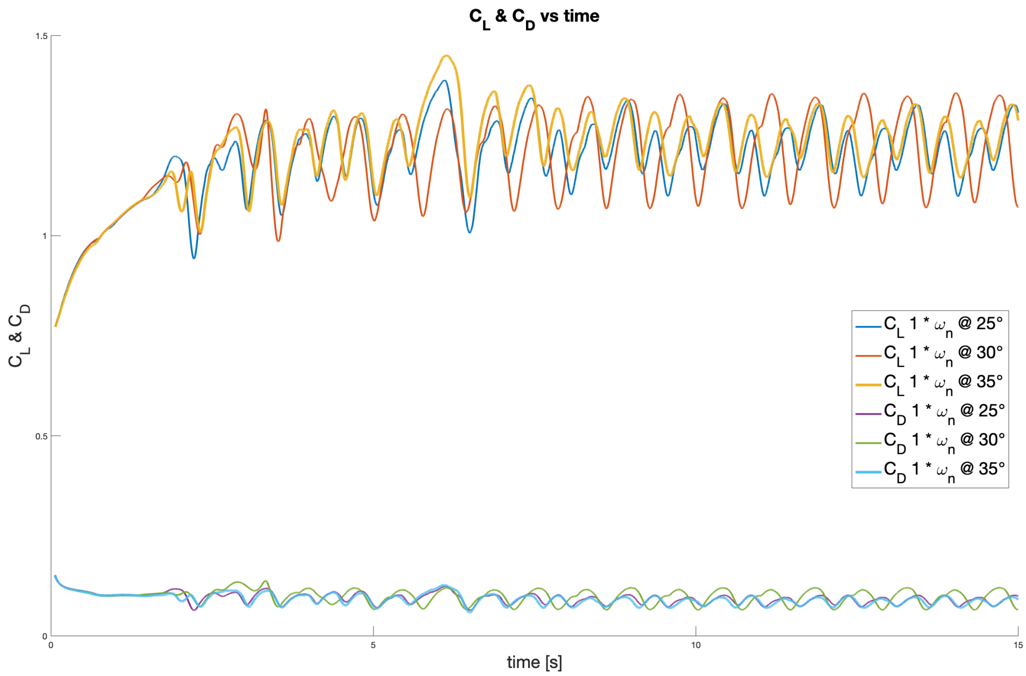

Figure 4.20: Comparison between three angles at $f = 1 * \omega_n$

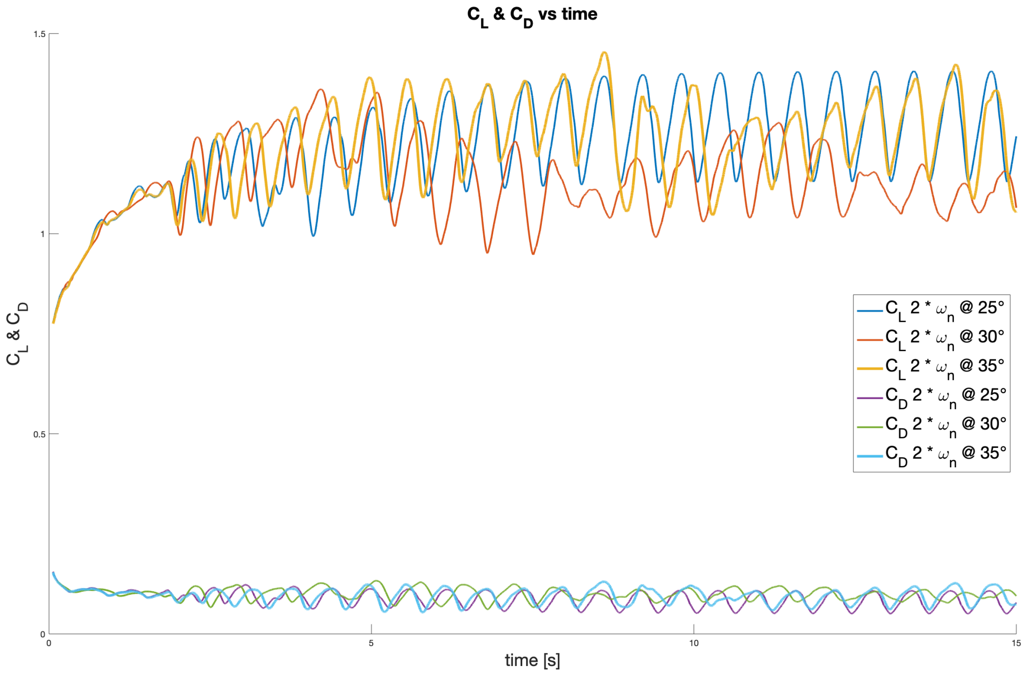

Figure 4.21: Comparison between three angles at $f = 2 * \omega_n$

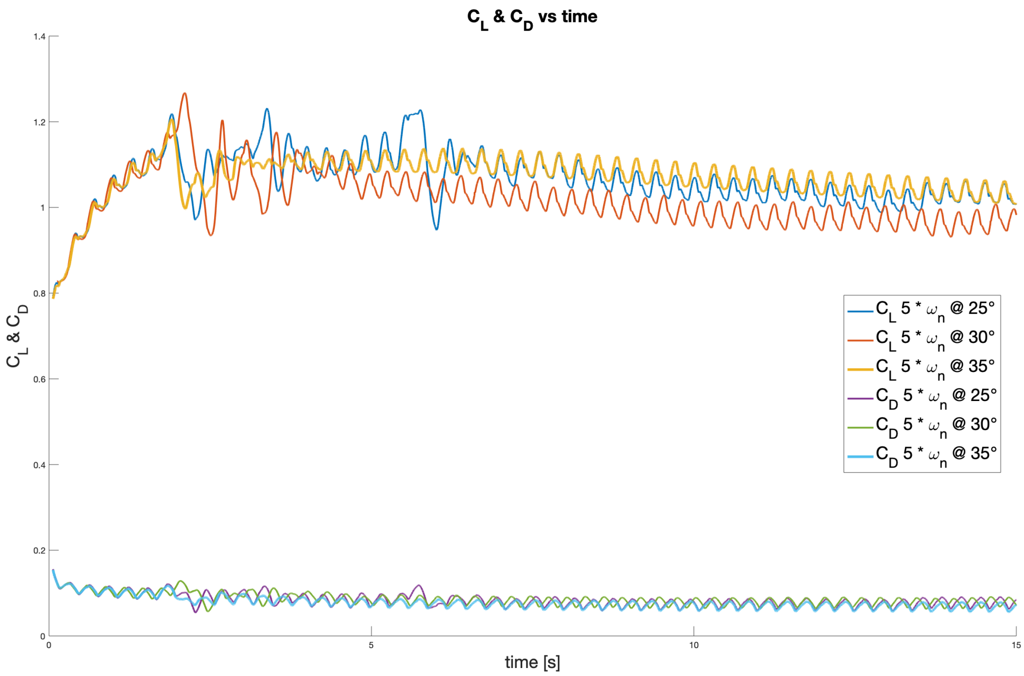

Figure 4.22: Comparison between three angles at $f = 5 * \omega_n$

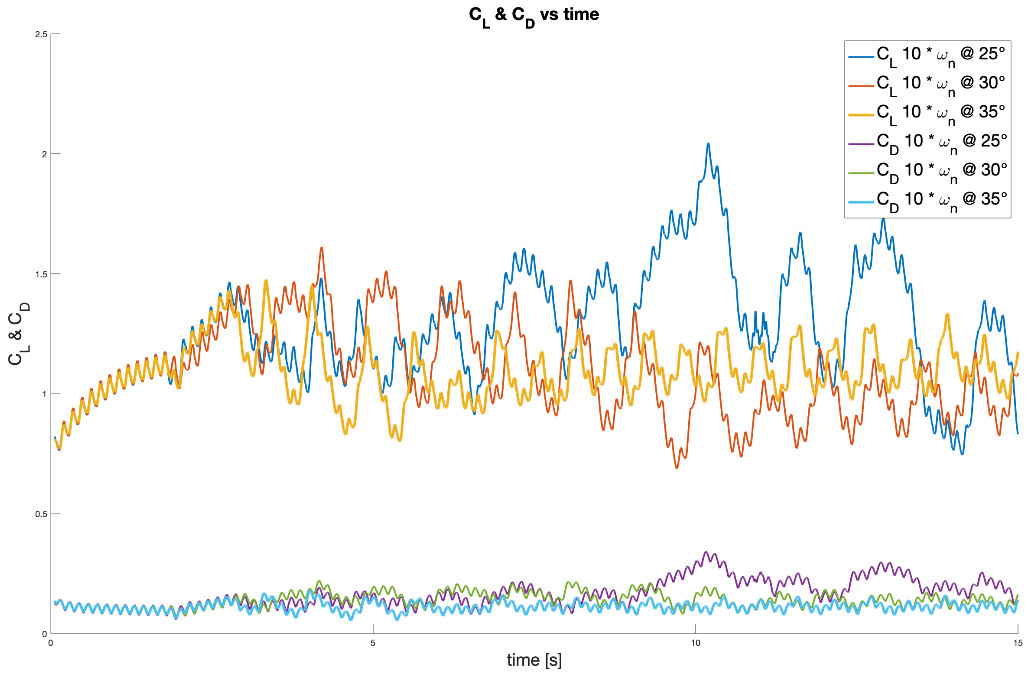

Figure 4.23: Comparison between three angles at $f = 10 * \omega_n$

After the first analysis of difference between the frequencies considered with a fixed angle of outflow from AFC, these five last figures show the differences between the three angles considered, maintaining the same frequency for all the cases. This type of analysis is interesting because it is possible to understand how much the variation of α could change the final solution and if it is better to impose an outflow directed more versus the surface of wing or more versus the normal direction to the wing.
The Fig. 4.19 shows the comparison using a frequency $f = 0.5 * \omega_n$, it is clear that at low frequencies the difference between the three angle is very small and one may use which angle he wants because the results probably will be similar. The Fig. 4.20 seems to be more curious, on the one hand the behavior of lift and drag with an angle of 30 degrees is perfect, but what is very interesting considering for example the yellow line that corresponds to an angle of 35 degrees is that the linearity is similar but the amplitude of these oscillations is reduced and this means

that the vortices that will be created one time the flow is inclined vs the surface of wing are smaller, this translates in less instability of flow. The best results obtained for the angle equal to 25 degrees is showed in the next figure, a very constant behavior of flow that it is not valid for the other two angles. Increasing the frequency it is possible to notice that only the configuration with outflow more directed versus the wing surface has the best results with small oscillations but instead of this it could not be the best choice for AFC.
The evaluation of numerical values of data presented in the figures above is done in the tables below:

LIFT	$f = 0.5 * \omega_n$	$f = 1 * \omega_n$	$f = 2 * \omega_n$	$f = 5 * \omega_n$	$f = 10 * \omega_n$
25 Degrees	1.2702	1.2196	1.2730	1.0641	1.2502
30 Degrees	1.2552	1.2327	1.1248	0.9782	0.9643
35 Degrees	1.2974	1.2558	1.2639	1.0764	1.0850

Table 4.3: Average values of lift over time
AFC in periodic forcing mode

DRAG	$f = 0.5 * \omega_n$	$f = 1 * \omega_n$	$f = 2 * \omega_n$	$f = 5 * \omega_n$	$f = 10 * \omega_n$
25 Degrees	0.1064	0.0905	0.0865	0.0787	0.1871
30 Degrees	0.1113	0.0955	0.0970	0.0787	0.1414
35 Degrees	0.1042	0.0882	0.0947	0.0726	0.1131

Table 4.4: Average values of drag over time
AFC in periodic forcing mode

EFFICIENCY	$f = 0.5 * \omega_n$	$f = 1 * \omega_n$	$f = 2 * \omega_n$	$f = 5 * \omega_n$	$f = 10 * \omega_n$
25 Degrees	12.1662	13.4655	14.8782	13.1408	6.6817
30 Degrees	11.2736	12.9107	11.5908	12.4297	6.8215
35 Degrees	12.4523	14.2406	13.3441	14.8204	9.5956

Table 4.5: Average values of efficiency over time
AFC in periodic forcing mode

The analysis of results also in this case requires also a numerical analysis that is the most important thing because it is possible to appreciate how much the performance are increased. The three table above shows obviously the three important parameters analyzed in this work thesis,

lift coefficient, drag coefficient and efficiency that relates the first and second coefficient. A total number of 15 simulations were performed, 5 for every angle of inlet of flow from the AFC groove. The first case analyzed is the 30 degrees case, what happens to the flow one time the frequency is changed? In this case reducing the frequency by the half of natural frequency of vortices increases the lift coefficient and it reaches its maximum value. Then increasing the frequency the value of lift is decreased (as we have seen in the figures from 4.15 to 4.18). The analysis must be done also on the drag coefficient, what one wants to obtain is a decrease of drag, indeed when the case with maximum lift coefficient is considered it is possible to notice that the drag is increased more than 20% with respect to the second frequency analyzed before. The lowest value of drag is obtained for a frequency five time higher than ω_n (but in this case the lift is reduced w.r.t. the free stream case and so it is not very good). The best frequency for 30 degree case is the second one.
At this moment the angle of inlet is changed and the situation is obviously different. Reducing the α angle of inlet, and so considering the 25 degree case, the best value of lift coefficient is obtained when a frequency that is two time higher than the natural frequency is consider and also in this case the lowest value of drag coefficient is obtained in the fourth case. It is nice to see the value of lift for a frequency 10 times ω_n, it is higher than the precedent but it has a value of drag higher more than 200% and for this reason it could not be a good combination. Now increasing the angle α it is possible to notice a behavior very similar to the 30 degree case but with higher values of lift and lower value of drag and so it is better of the second case.
What is it possible to say? With the analysis made above when the flow is inclined more versus the normal to the surface the situation results to be better, a flow too much inclined versus the surface not produce the results that one expected and why do we get this result? This is due to the fact that we deal in this case with a periodic forcing case,

where both a sucking and a blowing condition are spaced out. The sucking condition wants that the direction of sucks is counter-posed to the direction of flow and so with an angle α higher that 90 degrees, a blowing condition wants, ideally, a direction of inlet of flow tangential to the surface in order to energize the flow. When the periodic forcing is considered a condition that is good for sucking and blowing must be considered and for this reason the case with 35 degrees is better than the case with 25 degrees, because 25 degrees is better for blowing but it is more dangerous when the sucking condition works. The compromise is 35 degrees.

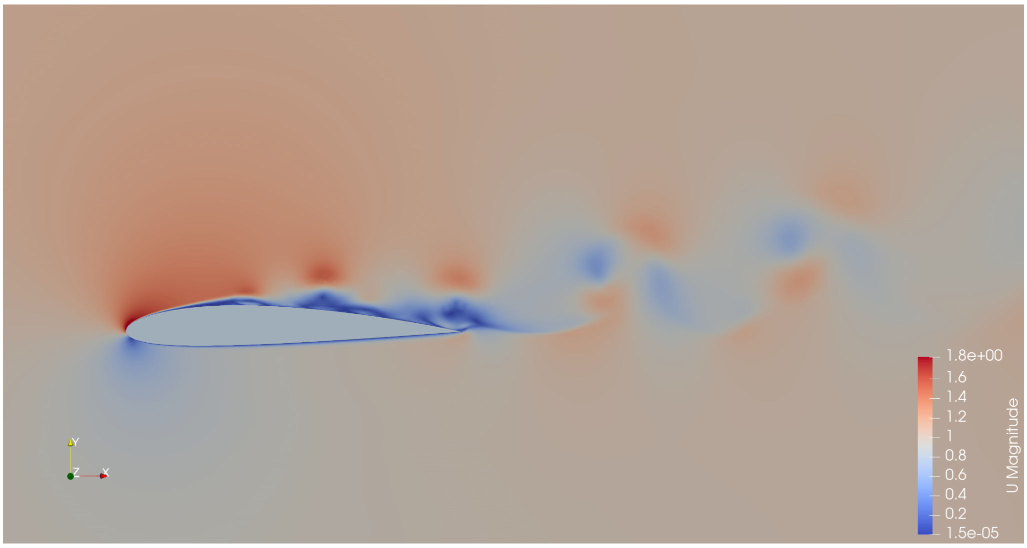

Figure 4.24: Contour velocity periodic forcing

These last figures shows also in this case some contour of pressure and velocity over the wing. The first figure compared with the contour of velocity without AFC shows the possibilities of AFC, the boundary layer is thiner and the dimension of vortices, that is possible to see in contour of pressure, are smaller and are also regulated by the frequency of AFC. The streamlines appear less casual and more clean.

The contour of velocity when the frequency is 10 time the natural frequency of detach of vortices is added to this final analysis because it is important to notice how the increasing of frequency has completely

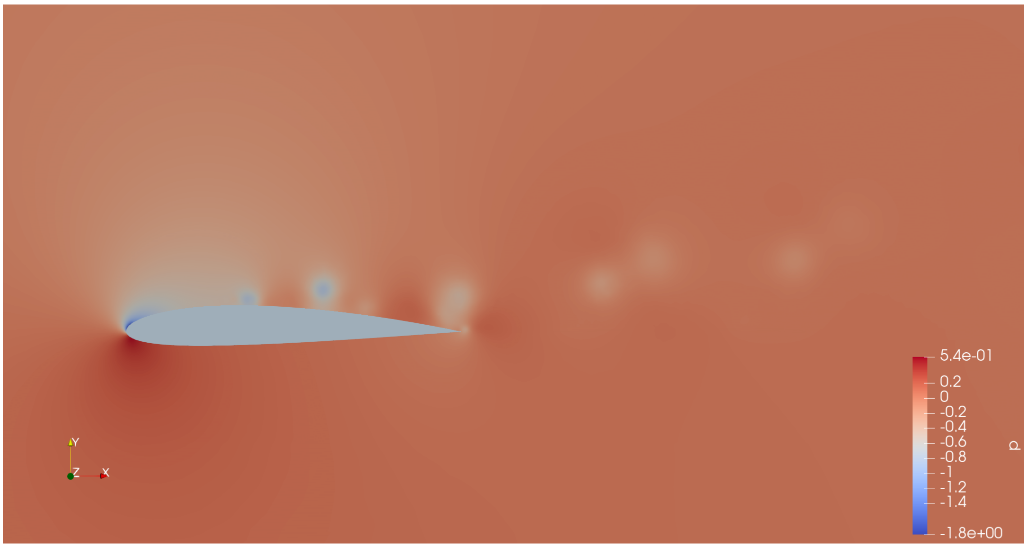

Figure 4.25: Contour pressure periodic forcing

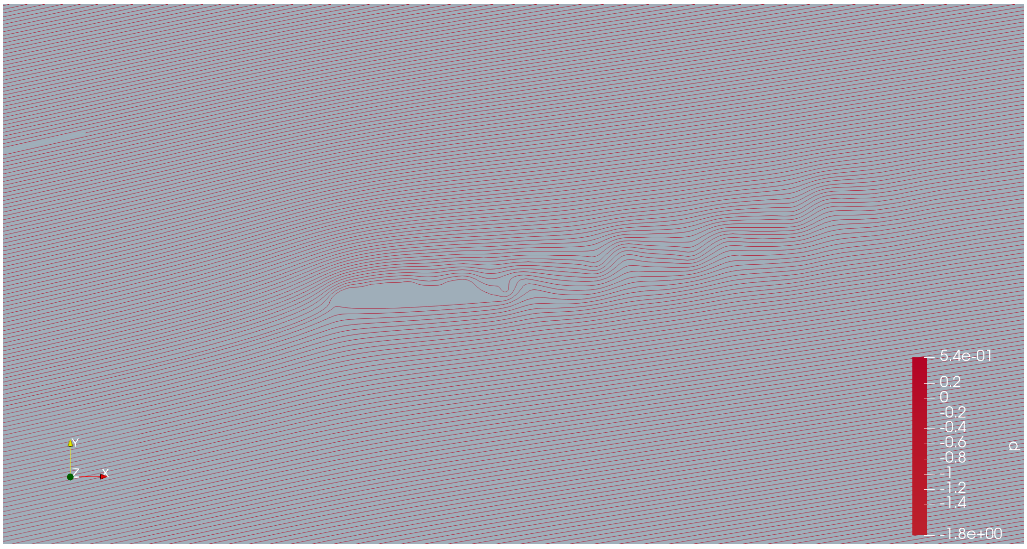

Figure 4.26: Streamlines with AFC - Periodic forcing

destroyed the boundary layer, it is larger compared with the case without AFC and it is obviously counterproductive. For this reason this case is discarded.

4.4 Case 3: Constant sucking

The results obtained in this third case will be presented.

This last figures shows the lift and drag coefficients versus time, as did in the previous sections, and what it is very nice to see are the improvement with respect to the case with periodic forcing, i.e. w.r.t.

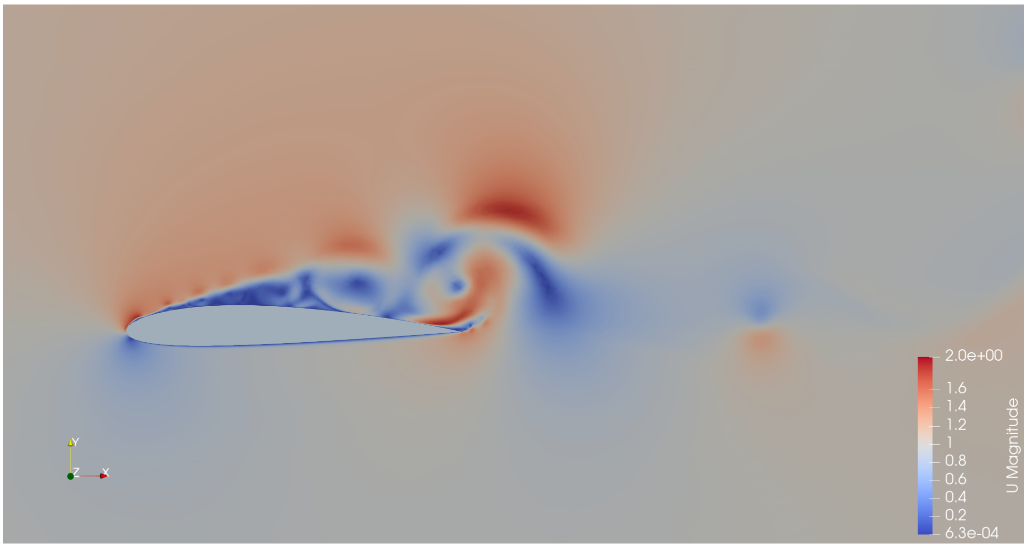

Figure 4.27: Contour velocity when $f = 10 * \omega_n$

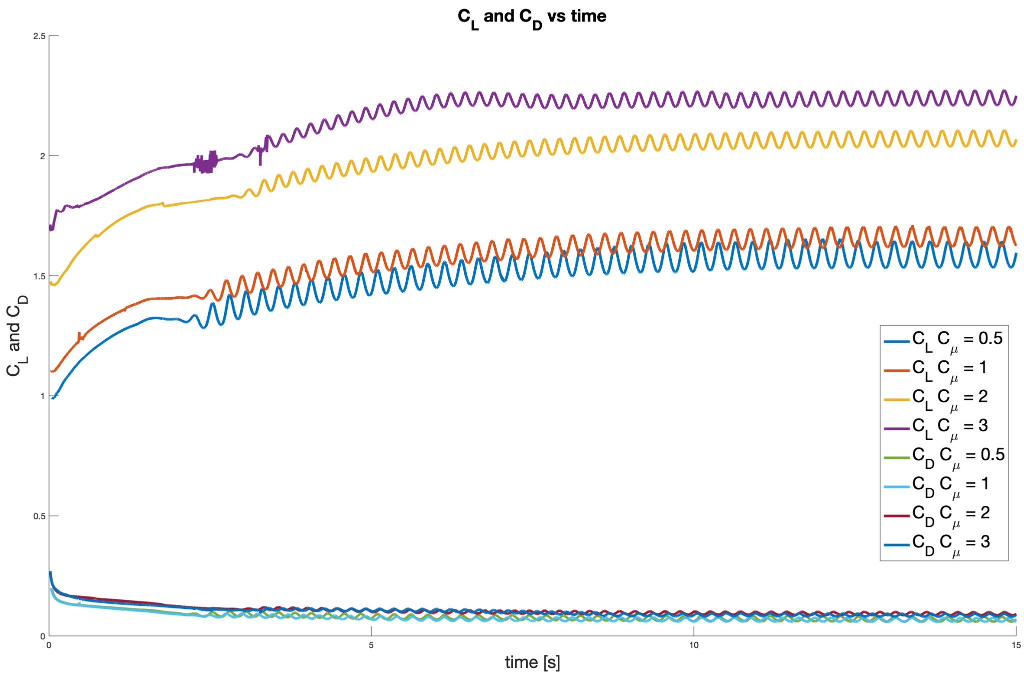

Figure 4.28: Position of groove

the free stream condition. Increasing the value of C_μ the lift is increased more and more, from an average value around 1.5 for $C_\mu = 0.5$ to a value around 2.3 for the $C_\mu = 3$ case, the improvements are significantly. We just need to remember that the average value of lift coefficent in free stream case was around 1.15 and so the increase of this coefficient is around 200%.

	$C_\mu = 0.5$	$C_\mu = 1$	$C_\mu = 2$	$C_\mu = 3$
Lift	1.5748	1.6572	2.0565	2.2318
Drag	0.0708	0.0668	0.0949	0.877
Efficiency	22.2411	24.8237	21.6796	25.4382

Table 4.6: Average values of aerodynamic coefficients over time - AFC in constant sucking mode

This table shows the average values of lift and drag coefficients versus the four different values considered. The best value of efficiency and lift coefficient is obtained for a $C_\mu = 3$, the efficiency is 200 % higher than the periodic forcing AFC and 400 % higher than the free stream case.

Figure 4.29: Contour velocity when $f = 10 * \omega_n$

As usually some contours of velocity and pressure respectively are shown. The improvements w.r.t. the other cases is very clear, where we have the zone with lower pressure is inserted the AFC groove. Analyzing the figure of velocity contours we see a flow that is attached to the surface of airfoil with very small vortices that detach from the trailing edge. This small vortices reflects the behavior of graph where the lift and drag coefficients are plotted, the oscillations around the average values are very small. The streamline of velocity show a very linear flow of air

Figure 4.30: Contour velocity when $f = 10 * \omega_n$

Figure 4.31: Streamlines with AFC - Constant Sucking

that once leaves the airfoil, follows the undisturbed velocity of air, apart from a little crinkling on the trailing edge due to vortices that leave the airfoil.

This last graph below represents shows the significantly difference between the simulations in free stream case, periodic forcing case (the choice is gone on the simulation that gives the best efficiency) and the best simulation with constant sucking AFC

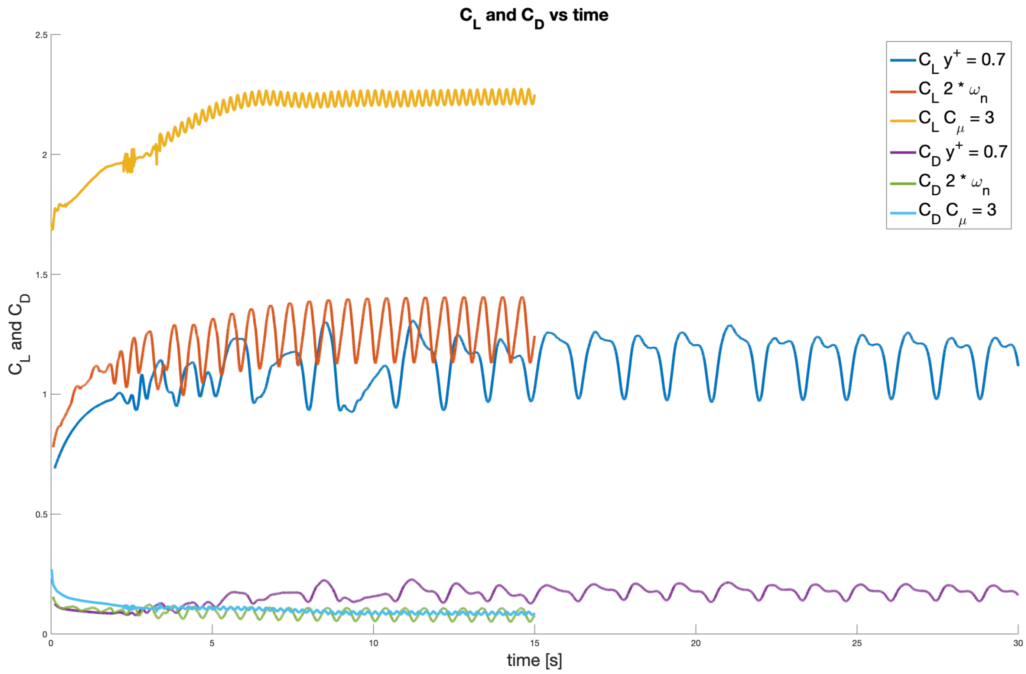

Figure 4.32: Final graph of C_L and C_D vs time

Chapter 5

Conclusions

In the pages above the problem of implementation of an active system for flow control on a wing is illustrated. The reason of designing of an AFC system reside into helping the flow through the introduction or the elimination of particles of flow that help the wing not to reach the stall conditions.

First of all the study of an airfoil in free stream condition where made with the purpose of finding with type of mesh will be used for next simulations. Three different meshes with three different values of y^+ where used, some simulations where made in order to find which mesh is the best one. Through the analysis of results the mesh with $y^+ = 0.7$ where chosen because gives us the best results in terms of lift and drag coefficients compared with the time required by simulation.

Secondly, the AFC system in forcing condition where implemented on the wing once all the data founded in free stream case were analyzed. The goal of this phase is to understand how much some characteristic parameters of this control system influence the overall solution. For this reason a series of 5 different frequencies and 3 different angles of inlet of flow from AFC groove were considered, 15 simulations were made in order to find the best combination for this specific case.

Thirdly, a case study now well known were simulated because the system in constant suction conditions is the best case known so far and the goal is to understand if the system in periodic forcing could be better compared to the older system. Four different simulations where performed with four different values of C_μ that increases the power of system.

What is it possible to tell about these results? Analyzing the case study in periodic forcing the results give us an increase of performance related to our system in terms of lift and drag coefficient once these results are compared with the free stream case, but the last case, the one in constant sucking condition, is obviously better, returns larger possibilities of design because the range of values of C_μ is larger and a limit where not found in this thesis instead of the periodic forcing case in which we have seen that increasing too much the value of C_μ also the performance could be compromised.
But the reason of the choice of an AFC system in periodic forcing where made because it is a system less energivorous and with this system were founded that it is possible to control the frequency of detach of vortices.

In conclusion, it is possible to state that the results obtained with periodic forcing are very good, the goal of obtaining best results compared with free stream case were reached. An improvement of efficiency around 100 % were founded.
The benefits of this system mounted on a wing are clear, more simulations and experimental must be performed to understand the real possibilities that this system could give back. Surely, it isn't better than older system considered here but an evaluation of ratio between energy consumption and performance increasing rate must be performed in order to understand which system is better.

Bibliography

[1] http://www.hotelsrate.org/kinematic-viscosity-of-water-temperature-equation/, Kinematic Viscosity Of Water Temperature Equation

[2] https://www.simscale.com/docs/content/simulation/model/turbulenceModel/kOmegaSST.html, K-Omega SST

[3] http://www.tfd.chalmers.se/ hani/kurser/OS_CFD_2016/FangqingLiu/openfoamFinal.pdf, *Fangqing Liu*, A Thorough Description Of How Wall Functions Are Implemented In OpenFOAM, Jenuary 2017

[4] https://en.wikibooks.org/wiki/Structural_Biochemistry/Chemical_Bonding/Van_der_Waals_interaction, *Wikibooks*, Structural Biochemistry/Chemical Bonding/Van der Waals interaction

[5] http://pnrsolution.org/Datacenter/Vol3/Issue2/27.pdf *Er. Shivam Saxena , Mr. Rahul Kumar*, Design of NACA 2412 and its Analysis at Different Angle of Attacks, Reynolds Numbers, and a wind tunnel test, Pag. 197-198

[6] http://airfoiltools.com/polar/details?polar=xf-naca2412-il-50000 *AirFoilTools*, NACA 2412 (naca2412-il) Xfoil prediction polar at RE=50,000