TITULACIÓ:
GRAU EN ENGINYERIA EN VEHICLES AEROESPACIALES

ALUMNE:
XAVIER GUERRERO PICH

TÍTOL TFG:
NUMERICAL STUDY ON ACTIVE FLOW CONTROL USING SYNTHETIC JET ACTUATORS OVER A NACA 4421 AIRFOIL

DIRECTOR DEL TFG:
DAVID DEL CAMPO SUD

CONVOCATÒRIA DE LLIURAMENT DEL TFG:
JUNY 2015

CONTINGUT D'AQUEST VOLUM:

DOCUMENT 1. - MEMÒRIA
Numerical Study on Active Flow Control using Synthetic Jet Actuators over a NACA 4421 Airfoil

REPORT

Author
Xavier Guerrero Pich

Director
David Del Campo Sud

Co-Directors
Vanessa Del Campo Gatell
Josep Maria Bergadà Granyó

June 12, 2015
Abstract

This study is focused on evaluating the effects of using a Zero Net Mass Flux (ZNMF) actuator on a NACA 4421 airfoil for active flow control. First part of the study presents the fundamentals of boundary layer and a study of the available devices which are more used for flow control, focusing on the ZNMF. The steps for creating the mesh to perform numerical simulations of the airfoil are explained, and the results of the CFD simulations are compared with experimental data as a baseline validation. In the second part, the ZNMF is studied in order to set the parameters of the actuator and to simulate its effect on CFD, and moreover the numerical simulations of the airfoil with the ZNMF set up are performed and the results are evaluated. The evaluation will show the most optimum parameters for the actuator, as well as the effects that the ZNMF has on the airfoil’s behaviour.
Summary

INTRODUCTION...........................................................................................................................................1

1.1. Aim of the study......................................................................................................................................2

1.2. Scope of the study...............................................................................................................................3

1.3. Requirements......................................................................................................................................4

1.4. Justification..........................................................................................................................................5

1.5. State of the art......................................................................................................................................6

THEORETICAL INTRODUCTION..............................................................................................................8

2.1. Aerodynamics introduction................................................................................................................9

2.1.1. Boundary layer fundamentals......................................................................................................9

2.1.2. The stall phenomenon....................................................................................................................13

2.1.3. Types of airfoils............................................................................................................................15

2.2. High lift devices..................................................................................................................................18

2.2.1. Passive high lift devices...............................................................................................................19

2.2.1.1. Trailing edge high lift devices................................................................................................21

2.2.1.2. Leading edge high lift devices.................................................................................................23

2.2.1.3. Combining leading edge and trailing edge high lift devices................................................25

2.2.2. Boundary layer control..................................................................................................................26

2.2.2.1. Zero-net-mass-flux (ZNMF)....................................................................................................27

2.2.2.2. Vortex Generators....................................................................................................................32

2.2.2.3. Plasma actuators.......................................................................................................................33

2.2.2.4. Wing fences...............................................................................................................................34

AIRFOIL AND ZERO-NET-MASS-FLUX ANALYSIS......................................................................................36

3.1. Characteristics of the NACA 4421 airfoil.........................................................................................37

3.1.1. NACA 4421 analysis.......................................................................................................................37

3.1.1.1. Airfoil characteristics...............................................................................................................37

3.1.1.2. Airfoil applications...................................................................................................................38

3.1.1.3. Operating conditions...............................................................................................................39

3.1.2. NACA 4421 simulation..................................................................................................................40

3.1.2.1. Meshing the airfoil...................................................................................................................40

3.1.2.2. Choosing the solver..................................................................................................................46

3.1.2.3. Comparison with experimental data.........................................................................................49

3.2. The Zero-Net-Mass-Flux actuator........................................................................................................52
3.2.1. Boundary condition ......................................................... 52
3.2.2. Simulation of the ZNMF effect ........................................ 55

ANALYSIS OF THE EFFECT OF THE ZNMF ACTUATOR ON THE NACA 4421 BEHAVIOUR ................................................................. 56

4.1. Evaluation of the results ....................................................... 57
  4.1.1. Lift coefficient behaviour ............................................... 58
  4.1.2. Evaluation of the velocity profile ..................................... 60
  4.1.3. Velocity magnitude contour ........................................... 62
  4.1.4. Streamlines .............................................................. 64
  4.1.5. Boundary layer separation point ..................................... 65
  4.1.6. Vorticity ................................................................. 68

4.2. Comparison with experimental data ....................................... 69
  4.2.1. Lift coefficient .......................................................... 69
  4.2.2. Streamlines ............................................................. 70
  4.2.3. Vorticity ................................................................. 71

CONCLUSIONS ........................................................................... 72

5.1. Conclusions and future work .............................................. 73
  5.1.1. Conclusions ............................................................... 73
  5.1.2. Future lines of investigation .......................................... 74
    5.1.2.1. Planning of the future work ........................................ 75

5.2. Sustainability study .......................................................... 78

5.3. Bibliography ................................................................. 79
Figure Summary

Figure 1. Experiment setup to evaluate Vortex Generators effects (Lin 2002) ...........6
Figure 2. Airfoil with plasma actuators at the leading edge and mid-chord locations (Post 2004) ................................................7
Figure 3. NACA 0012 with a synthetic jet actuator array (Ugrina 2007) .................7
Figure 4. Profile of a boundary layer (Roland 1985) ...........................................10
Figure 5. Laminar boundary layer thickness δ and velocity profile (Schlichting and Gersten 1958) ...........................................11
Figure 6. Laminar and turbulent velocity profiles (Glauser) (Glauser 2014) .........12
Figure 7. Laminar to turbulent transition (www.avalanche-center.org) ...............13
Figure 8. Formation of a separated (reversed) flow (Loftsi anskii 1970) ..........14
Figure 9. Smooth airflow (left) and boundary layer separation (right) (Newman 2000) ..................................................15
Figure 10. Stall in thick airfoils (Meseguer et al. 2004) .....................................15
Figure 11. Stall in medium thickness airfoils (Meseguer et al. 2004) ...............16
Figure 12. Stall in thin airfoils (Meseguer et al. 2004) ....................................17
Figure 13. Example of a passive high lift device (Brooksby 2010) ..................18
Figure 14. Example of an active high lift device (www.russianpatents.com) ....18
Figure 15. Typical effect of flaps on lift (Mason 2007) ....................................19
Figure 16. Effect of flap extension on lift (Mason 2007) ..................................20
Figure 17. Trailing edge flaps (Swatten 2010) ..............................................21
Figure 18. Multiple slotted flaps - BOEING 747 (Prisacariu and Luchian 2014) ...22
Figure 19. Effect of leading edge devices in lift (Cashman et al. 2000) ..........23
Figure 20. Leading edge devices (www.skysaber.com) ..................................24
Figure 21. Effect of flaps and slats in cl − a curve (Prisacariu and Luchian 2014) .25
Figure 22. Effect of flaps and slats in cl − cd curve (Prisacariu and Luchian 2014) 25
Figure 23. Slat with slotted flap (www.eroftac.org) .....................................26
Figure 24. Formation of a zero-net-mass-flux jet (Zhang et al. 2008) ..........27
Figure 25. Hairpin vortices produced by the ZNMF. Orifice at the top, cross flow from right to left (Tang et al. 2007) ...........................................28
Figure 26. Interaction of the ZNMF with the cross flow. (a)Suction (b) Blowing. (Schaeffler 2003) .................................................28
Figure 27. Co-rotating vortex generators (Titchener and Babinsky 2015) ..........32
Figure 28. Plasma actuator scheme (Post 2004) .........................................33
Figure 29. Fence location (www.cap-ny153.org) .......................................34
Figure 30. Lift coefficient spanwise using or not a fence (www.b2streamlines.com):34
Figure 31. NACA 4421 airfoil (Abbott and Von Doenhoff 1959) .................37
Figure 32. Airfoil parameters (www.grc.nasa.gov) .....................................38
Figure 33. Vanguard Omniplane (Mcchesney 2012) ...................................38
Figure 34. Far-field extension .....................................................................41
Figure 35. Split edges ..................................................................................42
Figure 36. Mesh zoom around the airfoil .....................................................42
Figure 37. Stages of the wake .....................................................................43
Figure 38. Zoom out of the wake's stages .................................................................43
Figure 39. Mesh around the airfoil .........................................................................44
Figure 40. Mesh of the whole fluid field .................................................................44
Figure 41. Aspect ratio of the mesh (X axis: Aspect Ratio of the element; Y Axis: Number of elements with that aspect ratio) .................................................................45
Figure 42. Eriksson skewness evaluation of the mesh .............................................46
Figure 43. General quality of the mesh .................................................................46
Figure 44. Comparison of velocity profiles for NACA 4412 airfoil using different turbulence models (Menter 2011) .................................................................47
Figure 45. Contours of vorticity magnitude using different turbulence models (Sandberg 2015) ........................................................................................................47
Figure 46. Choosing the turbulence model in Fluent .............................................48
Figure 47. NACA 4421 $Cl - \alpha$ curve (Abbott and Von Doenhoff 1959) ..........49
Figure 48. Comparison of experimental data (Abbott and Von Doenhoff 1959) and results of the simulations ..............................................................................51
Figure 49. Lift convergence in transient simulation at $\alpha = 7^\circ$ .......................54
Figure 50. Boundary condition UDF code ...........................................................55
Figure 51. Lift coefficient, $\alpha = 7^\circ$ .................................................................58
Figure 52. Lift coefficient, $\alpha = 12^\circ$ ...............................................................59
Figure 53. Lift coefficient, $\alpha = 14^\circ$ ...............................................................59
Figure 54. Velocity profile on the airfoil (14$^\circ$, 45 Hz, 55% of the chord) ..........60
Figure 55. Velocity profile of the boundary layer separation point .....................61
Figure 56. Velocity profile of the region where the boundary layer is separated .....62
Figure 57. Velocity magnitude contour (m/s) at $\alpha = 12^\circ$ (top) and $\alpha = 14^\circ$ (bottom) .......................................................................................................................63
Figure 58. ZNMF velocity contour (m/s) ...........................................................64
Figure 59. Streamlines of the airfoil, $\alpha = 14^\circ$ and $f = 45$ Hz .........................65
Figure 60. Boundary layer separation point .........................................................66
Figure 61. Vorticity around the airfoil (1/s). 12$^\circ$, 45 Hz .....................................68
Figure 62. Streamlines of a NACA 0015 with ZNMF actuator (Zhang Panfeng 2007) ....................................................................................................................70
Figure 63. Vorticity magnitude (1/s), with the ZNMF set at 0.15c (Durrani and Haider 2011) ........................................................................................................71
Figure 64. Gantt Diagram of the future work .....................................................77
## Equation Summary

<table>
<thead>
<tr>
<th>Equation</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Equation 1.</strong></td>
<td>Reynolds number</td>
<td>10</td>
</tr>
<tr>
<td><strong>Equation 2.</strong></td>
<td>Simplified Navier-Stokes equation (Burr et al. 2013)</td>
<td>13</td>
</tr>
<tr>
<td><strong>Equation 3.</strong></td>
<td>Stroke length</td>
<td>29</td>
</tr>
<tr>
<td><strong>Equation 4.</strong></td>
<td>Blowing velocity</td>
<td>29</td>
</tr>
<tr>
<td><strong>Equation 5.</strong></td>
<td>Stroke length ratio</td>
<td>30</td>
</tr>
<tr>
<td><strong>Equation 6.</strong></td>
<td>Reynolds number based on the blowing stroke</td>
<td>30</td>
</tr>
<tr>
<td><strong>Equation 7.</strong></td>
<td>Velocities ratio for vortex formation criteria</td>
<td>30</td>
</tr>
<tr>
<td><strong>Equation 8.</strong></td>
<td>Strouhal number</td>
<td>30</td>
</tr>
<tr>
<td><strong>Equation 9.</strong></td>
<td>Stokes number</td>
<td>30</td>
</tr>
<tr>
<td><strong>Equation 10.</strong></td>
<td>Jet formation criterion</td>
<td>30</td>
</tr>
<tr>
<td><strong>Equation 11.</strong></td>
<td>Momentum coefficient</td>
<td>31</td>
</tr>
<tr>
<td><strong>Equation 12.</strong></td>
<td>Relative error</td>
<td>50</td>
</tr>
<tr>
<td><strong>Equation 13.</strong></td>
<td>Oscillation function</td>
<td>52</td>
</tr>
<tr>
<td><strong>Equation 14.</strong></td>
<td>Shedding frequency of the airfoil</td>
<td>54</td>
</tr>
<tr>
<td><strong>Equation 15.</strong></td>
<td>Function that will represent the ZNMF effect</td>
<td>54</td>
</tr>
</tbody>
</table>
Chart Summary

Chart 1. Operating conditions of the airfoil..............................................................39
Chart 2. Experimental data of $c_l - \alpha$ curve (Abbott and Von Doenhoff 1959)............50
Chart 3. Results of the numerical simulations using ANSYS Fluent..........................50
Chart 4. Relative error between the experimental data and ANSYS Fluent simulations
........................................................................................................................................50
Chart 5. Frequencies that will be evaluated ...............................................................54
Chart 6. Pulses that will define the boundary condition.............................................54
Chart 7. Evaluated scenarios......................................................................................57
Chart 8. Lift coefficient oscillation amplitude for each scenario...............................60
Chart 9. Boundary layer separation point position and correspondent wall shear stress,
$\alpha = 7^\circ$ ..................................................................................................................67
Chart 10. Boundary layer separation point position and correspondent wall shear stress,
$\alpha = 12^\circ$ .................................................................................................................67
Chart 11. Boundary layer separation point position and correspondent wall shear stress,
$\alpha = 14^\circ$ .................................................................................................................67
Chart 12. Lift coefficient obtained by Durrani (Durrani and Haider 2011)...............69
Chart 13. Lift coefficient obtained through ANSYS Fluent simulations...............69
Chart 14. Tasks interdependencies and level of effort.............................................76
PART I

INTRODUCTION
1.1. Aim of the study

Since the very first days, investigations about how to improve aircraft’s performance has been a subject of study all over the world. Creating devices in order to increase the efficiency of these vehicles is something which is still in development, and every year there are innovative methods and designs which able this sector to keep going on.

However, nowadays the designs have achieved such a high level of efficiency and accuracy that every single part of each component is constantly being revised in order to be improved. One of the aspects that is included in this revisions is the possibility of increasing the lift of an aircraft independently of the aircraft actuations, and without spending a lot of money in fuel or other resources, which are becoming more expensive every day.

The devices which increase the lift changing aircraft’s features (such as geometry, or the fluid behaviour in the surrounding area) are called high lift devices. These devices, in turn, are different from each other depending on the principle in which they are based. The first high-lift devices came into common use in the 1930s (Perkins and Hage 1949), and since then they are being improved in order to give more efficiency without affecting other characteristics like the reliability of the aircraft, the money expenditure or the fuel consumption.

Some of them change mechanical features, and other ones are based on the study of the flow over the aircraft’s wings. These last devices are then used for what is called active flow control.

The aim of this project is to present the advantages of using high-lift devices through making different numerical simulations, in order to study active flow control methods. More specifically, the study will be focused on synthetic jet actuators, particularly those devices which do not provide mass addition, hence being called Zero-Net-Mass-Flux (ZNMF) actuators.

Moreover, numerical analysis with graphical evidences will be provided to analyse the efficiency that an aircraft wing can achieve using or not a ZNMF actuator, studying the range of the most important parameters which are optimum for the actuator configuration.
1.2. Scope of the study

In this study, an introduction to the fundamentals of aerodynamics are presented in order to facilitate the understanding of the project. More concretely, an introduction of the boundary layer and the different available airfoils currently in use are presented, giving reasons of the advantages and disadvantages of each one.

Furthermore, the most common high lift devices are explained and classified into different categories depending on the principles they use to work. The ones which modify the geometry of the airfoil are described, depending on if they modify the trailing edge, the leading edge or both of them.

In addition, the devices which interact with the surrounding fluid are outlined, focusing on ZNMF as it is the basis of the study. It is explained the way of working of the device, as well as the key parameters which must be dimensioned in order to set it on the airfoil, including parameters of the cross flow and how they affect to the ZNMF operating conditions.

Moreover, a brief description of other boundary layer control devices is presented, including some which are mechanically based (vortex generators and wing fences) and other ones which are based on the use of electrodes and electric fields (plasma actuators).

After describing the theoretical basis of the study, the airfoil which will be used in the simulations is presented, their characteristics, the operating conditions and some possible applications are mentioned.

Once the airfoil is presented, the next step is to mesh and simulate the airfoil in the already specified conditions. The objective is to compare the results of the simulations with the experimental ones in order to guarantee that the results which will be obtained in the following simulations are reliable.

When the meshing is verified, the high lift device (here the ZNMF) will be added to the airfoil, in order to simulate the same conditions while providing the effect of the device. This will be done to obtain enough data in order to compare the behaviour of the airfoil when using or not the ZNMF, and to underline the improvements that this device can offer.

These improvements will be evaluated as well as the airfoil parameters which are studied. These parameters will be the frequency of the oscillation and the position of the ZNMF, considering the amplitude as already settled.
1.3. Requirements

The basic requirement of the project is to get results when simulating the airfoil and the ZNMF and when simulating only the airfoil, in order to compare then and draw conclusions.

The first step of the simulations is to mesh the airfoil. The meshing will be done using ANSYS software, more specifically ICEM CFD. This software itself will evaluate the mesh in order to guarantee that the results will be reliable enough. When the mesh is done, the simulations will be performed using another ANSYS software, Fluent.

The operating conditions (Reynolds number, Angle of Attack, Mach number, atmospheric pressure) necessary to simulate will be obtained from previous experimental data obtained from Abbott, because this book will be the resource from where the experimental data will be extracted in order to be compared with the simulations.

It is also necessary to decide which turbulence model will be used. Through previous research, the best model will be chosen in order to get the more accurate data. The next step is then applying the ZNMF on the airfoil.

The ZNMF will be treated as a boundary condition in a small part of the extrados. The ideal function to simulate the actuator will be studied, including the evaluation of the parameters which will be used for the actuator configuration. Moreover, these parameters will be used to get the data to compare the airfoil performance when using or not the ZNMF.

Finally, a study of the environmental impact caused by this device will be presented calculating the fuel saving that the ZNMF provides using prices and ratios published by Industry Ministry and different electric companies.
1.4. Justification

The aerospace sector is one of the most expensive ones on the overall industry. Every component means a lot of money when it is manufactured, causing that a common commercial aircraft costs almost 100 million dollars (Airbus 2014). For that reason, all the possibilities which could involve a saving of money are studied in order to reduce the expenditure.

Most of this expenditure comes from the manufacture of the components. As in this sector the reliability is one of the most important aspects, the components must be designed and built sparing no expense. Therefore, trying to develop the cheapest components (without affecting the other features) has become an important part of the R&D task nowadays.

But there are other ways to save money. Another aspect which is being studied is the kind of fuel which is used by the aircrafts; the current one used today is the kerosene, but there are several studies which have been developed since the 1990s (Winter 1990) (Struminskii et al. 1996) which presents the hydrogen as the new fuel for its environmental impact and cost.

However, not only a change on the fuel is necessary, but also consumption decrease. For that reason, high lift devices are useful because they improve the aerodynamic efficiency of the aircraft, allowing to save money on both the design of the geometry and the diminishing of the consumption (Meseguer and Sanz 2011).

More particularly, and among all the available high lift devices, the best ones which are being studied are those which do not need any kind of external energy; they are called passive high lift devices (Mason 2007).

Moreover, the more compact and robust the device is, the cheaper will be in its overall cost. For this reason, many different high lift devices are being studied (Lin 2002; Mittal and Rampunggoon 2002) in order to improve their efficiency and usefulness. One of them is the ZNMF, an actuator which is included in the active high lift devices category; it only takes fluid of the external air and adds velocity (without any mass addition) to the fluid in order to delay the flow separation and hence improve the aerodynamic efficiency of the aircraft.
1.5. State of the art

As high lift devices are nowadays an issue which is being investigated, it is important to present the current studies which are currently being performed in order to have an overall view of the sector.

Due to the amount of types of devices which have been designed until today (Mason 2007), it has become more important not only to create new devices but also to improve the performance of the available ones.

Several numerical simulations have been performed in order to evaluate the performance of these devices (Rumsey 2004). The results have in common the importance of the parameters which configure the simulations, such as the turbulence model and the configuration of transient unsteady simulations, as concluded by Rumsey.

![Image](image-url)  

(c) 0.26-high vane-type counter-rotating VGs at 10 h upstream of baseline separation.  

**Figure 1.** Experiment setup to evaluate Vortex Generators effects (*Lin 2002*)

The diversity of the high lift devices which are studied is another issue to take into account. In 2002, Lin studied the effect of vortex generators over a wing and its capability of controlling the boundary layer separation (see **Figure 1**).

Moreover, Post studied in 2004 the possibility of adding a plasma actuator for the same purpose (as shown in **Figure 2**), something which worked according to its initial hypothesis (which were in turn based on numerical simulations).
Figure 2. Airfoil with plasma actuators at the leading edge and mid-chord locations (Post 2004)

Furthermore, it has increased the importance of doing experimental studies in order to get real, which is by far more reliable for development tasks. In 2007, Ugrina made an experimental study and analysis of the interaction between a synthetic jet actuator and the cross flow over a wing. As expected, the addition of the actuator ended up in an effectiveness along the lifting curve, mostly because it changed the velocity profile of the boundary layer.

Figure 3. NACA 0012 with a synthetic jet actuator array (Ugrina 2007)

In addition, it has been studied (Bhatia et al. 2014) the effects of passive high lift devices through experimental approach. According to Bhatia, changing the geometric profile lead to a reduction of boundary layer thickness, as well as an effect of the leading edge high lift device on the turbulence.

Although the several experiments described, the basic functions of the different devices and further information about them can be found in Section 2.2.
PART II

THEORETICAL INTRODUCTION
2.1. Aerodynamics introduction

Before entering to the study results and procedure, it is important to introduce the most basic theoretical concepts which are necessary to understand the whole meaning of the project.

In this chapter, the fundamentals of the physics phenomenon known as boundary layer are presented to give the previous concepts which involves this study. Particularly, the stall phenomenon will be explained as it is interesting for the study. Moreover, describing the different types of airfoil is an important previous tasks in order to classify further in the study the airfoil which is going to be used.

2.1.1. Boundary layer fundamentals

The main goal of this section is to facilitate the understanding of what is the boundary layer and how important it is in the aerodynamics environment.

It is usual to find most of the studies raised as if the solid would not have any speed, and there was an air speed which would represent the same situation. Therefore, the speed of the aircraft will be presented as the air speed or the fluid velocity, assuming then that the solid has no velocity.

It is necessary to start from the boundary condition, which describes the fact that velocity has to be tangent to the solid. This potential model cannot explain the appearing of drag in stationary motion around 2D obstacles (D’Alembert’s paradox)(Hoffman and Johnson 2008).

In addition, however high Reynolds number may be, the boundary condition over the airfoil is that fluid velocity is zero in its adjacent surface (Coelho 2013). This, consequently, implies boundary layer existence: a region in which velocity changes from zero to the value corresponding to free stream.

Besides, aerodynamic forces depend mostly on two fluid properties: the viscosity and the compressibility.

Regarding viscosity, as the fluid moves past the object (in this case, the airfoil), the molecules right next to the surface stick to it. Moreover, the molecules which are just above the surface are slowed down due to the collisions produced between them and the ones sticking to the surface.

These fluid particles in turn slow down the layer just above them, and this happening goes on and on; in conclusion, the farther one moves away from the airfoil’s surface, the fewer the collisions affected by the object surface. Therefore, there is a point where these collisions do not affect the behaviour of the fluid velocity.
According to GRC (Glenn Research Center 2014), the boundary layer is defined as the thin layer of fluid near the surface in which the velocity changes from zero to the free stream value, away of the surface (it is usually taken the point where the velocity is 99% of the free stream velocity value). The phenomenon has this name because it happens on the boundary of the fluid.

In Figure 4 it is presented the boundary layer in a graphical way in order to facilitate the comprehension of the phenomenon.

![Figure 4. Profile of a boundary layer (Roland 1985)](image)

Having explained the boundary layer, it is now necessary to describe its different behaviours. Although in the figure above we can see a 2D representation, the effects are three dimensional.

From the mass conservation in the three dimensions, a change in velocity in the streamwise direction produces a change in other directions’ velocity as well, as explained by GRC (Glenn Research Center 2014). Focusing on the velocity which is perpendicular to the surface, there is a small component which moves the flow above it, and this can be defined as the boundary layer thickness.

This thickness depends strongly on what is called the Reynolds number (Equation 1), which is represented as it follows.

$$Re = \frac{\rho \cdot v \cdot c}{\mu}$$

**Equation 1. Reynolds number**

Being the fluid density ($\rho$), its velocity ($v$), the airfoil chord ($c$) and the dynamic viscosity coefficient ($\mu$) the parameters needed to calculate this number. In physical terms, the Reynolds number represents the ratio of inertial forces in front of viscous forces.

Moreover, it is possible to find two different kinds of boundary layer, depending on how the fluid behaves inside this thin layer and the Reynolds number.
Laminar boundary layer

Its main feature is that the fluid is organised in parallel layers and there is no mixing between them. It is an ordered flow, where particles move creating streamlines, as shown in Figure 5. According to Meseguer and Sanz (2011), the only interaction between them is the viscous exchange of momentum, and at a molecular scale.

![Figure 5. Laminar boundary layer thickness $\delta$ and velocity profile (Schlichting and Gersten 1958)](image)

The laminar boundary layer appears at low Reynolds numbers (the Reynolds number at which laminar behaviour ends depends on several parameters, such as airfoil’s geometry (Bhatia et al. 2014). Therefore, it is found when the viscous forces are important against inertial forces (low velocities).

Another important feature is its low stability (Bhatia et al. 2014), together with the fact that it remains only in the front part of the airfoil. Moreover, the drag when the boundary layer has a laminar behaviour is much lower than when it is turbulent.

However, the boundary layer separation or stall phenomenon is more likely to happen, something which is not desirable and therefore expected to avoid.

Turbulent boundary layer

While the laminar boundary layer has a very ordered structure and the interaction is only produced in molecular scale, in the turbulent boundary layer this interaction is produced in a macroscopic way (Garcia 2012). The particles mix between them and the layers disappear, as presented in Figure 6, having a complex behaviour that will not be studied here.

It is particular from high velocities (high importance of inertial forces), where viscous forces lose importance in comparison to the laminar case. Because the turbulent mix is by far more effective than the laminar one(Garcia 2012), the turbulent boundary layer velocity profile is fuller. Consequently, there are two features someway antagonistic which must be remarked.
The first one is that, due to the high friction with the wall, the turbulent behaviour produces high values of drag. Moreover, the most rational way to avoid it is to keep the boundary layer in its laminar behaviour (Meseguer and Sanz 2011). On the other hand, its insensibility to pressure gradients allows the delay of the stall phenomenon, something which is desirable. Thus, this characteristic is a good reason to want the boundary layer to be turbulent.

![Laminar Boundary Layer](image1) ![Turbulent Boundary Layer](image2)

Figure 6. Laminar and turbulent velocity profiles (Glauser) (Glauser 2014)

In order to compare both velocity profiles, it can be noticed that the velocity gradient is by far softer in the laminar case; velocity increases almost uniformly as you move away from the surface. In contrast, turbulent behaviour has a more constant profile in the outset part of the boundary layer, offset by a very high velocity gradient near the surface (Fage and Preston 1941).

As it has been said, the boundary layer is turbulent at higher Reynolds numbers, while laminar behaviour appears at low values of this number. Because it depends on a characteristic longitude (in this case, the chord), it is easy to deduce that, measuring from the leading edge, the farer you go chordwise, the more turbulent the boundary layer will be (Glenn Research Center 2014).

It is therefore necessary to define the transition from laminar to turbulent. In Figure 7 it is graphically shown the behaviour from laminar to turbulent through the transition.
As the Reynolds number increases as you move away from the leading edge to the trailing edge, it is possible to define the region where the turbulent behaviour begins, relating it with a specific value of the Reynolds (Fage and Preston 1941).

Having explained the fundamentals of the boundary layer, the next step is to focus on the boundary layer separation, something also known as the stall phenomenon.

2.1.2. The stall phenomenon

As it has been said, the fluid layers near the surface slow down the immediate layer above them, and while this phenomenon influences fluid behaviour, the region is called boundary layer. However, in some cases these lowest layers can be so slowed down that the velocity even changes its direction. This happening is known as the boundary layer separation.

According to GRC (Glenn Research Center 2014), the most important consequences of the boundary layer separation, also known as stall, are the sudden loss of lift and a significant increase of the drag. Therefore, it is very important to take care of this situation in order to avoid it happening (although it is used for some acrobatic purposes).

The stall phenomenon occurs when the angle of attack (AoA) of the airfoil grows (in absolute value) more than a range of values (which depends on the airfoil), it produces a loss of lift which is not desirable. This is also known for being the critical boundary for defining the operation of the aircraft, because it can determine its maximum weight (although the take-off should also be taken into account).

In order to evaluate the causes which can influence on the separation, it is necessary to present the equation which involves the velocity profile, the Navier-Stokes equation(Burr, Akylas, and Mei 2013) (presented below as Equation 2). It is necessary to mention that the equation has been simplified in order to present only the terms which are interesting.

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + v \frac{\partial^2 u}{\partial y^2}
\]

Equation 2. Simplified Navier-Stokes equation(Burr et al. 2013)
It can be noticed that there are two terms on the right-hand side of the equation (Burr et al. 2013). The first one involves pressure gradient, and the other is related to viscosity. As it has been explained, the importance of each term depends on the situation, but it is necessary to evaluate both of them.

Regarding viscosity, the fluid particles inside the boundary layer are decelerated due to viscous forces effect, so there occurs a loss kinetic energy loss. In these conditions, it is possible to find a point on the surface where the fluid experience that change of velocity direction (Stern 2010).

Besides, it is easy to realise that a high adverse pressure gradient can make this reversal happen; when the pressure term is (in absolute value) higher than the viscous one, the velocity profile change its direction creating this flow separation.

Moreover, the point where this change is produced, when calculated chordwise, creates the streamline which defines the separation of the boundary layer, as it can be seen in Figure 8.

![Figure 8. Formation of a separated (reversed) flow (Loitsianskii 1970)](image)

Beyond that point where the line of zero velocity appear (also called the boundary layer separation point), it occurs a separation of fluid generating a wake. According to Stern (Stern 2010), this wake, which is characterized for being turbulent, creates a low pressure zone downstream. Therefore, the pressure distribution becomes asymmetric and it appears the pressure drag. This drag is different from the one created by viscous forces and friction, and it has to be taken into account as well.

In Figure 9 it is shown the boundary layer together with its later separation, where it can be noticed the wake which involves the process and the turbulent flow where the separation appears.
Now that the stall phenomenon it has been introduced, it is doubtlessly important to keep an eye on it. Because the stall phenomenon can occur in different ways, it is important to try to avoid the most dangerous ones, which appear suddenly, in order to give the pilot enough time to reattach the boundary layer and hence avoiding possible accidents.

There are several types of boundary layer separation, some of which are presented in the following section. As the separation depends on the type of airfoil, a classification of them is therefore explained as well.

### 2.1.3. Types of airfoils

Depending on the type of airfoil, the boundary layer separation occurs in different ways. This conclusion comes from the importance of the pressure gradient, which in turn depends on the design of the airfoil and its features.

Consequently, the geometric parameters of the airfoil have high relevance, and the classification given is created based on the thickness $\delta$ (dimensionless with the chord) of the airfoil. Moreover, it is possible to differentiate between three different types of airfoils. In this section, these types of airfoils are explained, following different criteria of several references (Meseguer and Sanz 2011)(Meseguer, Álvarez, and Pérez 2004).

#### Type A

The airfoils included here have a relative thickness $\frac{\delta}{c} > 0.15$. In these kind of airfoils, the stall is preceded by a transition from laminar to turbulent flow. The steps are graphically shown in Figure 10.

![Figure 10](image)

The stall phenomenon in thick airfoils can be described as follows.
1. When the AoA increases, the boundary layer on the extrados progressively thickens until the boundary layer separation appears on the trailing edge. The AoA at which this usually happens on these type of airfoils is known to be about 10°.

2. If the AoA continues growing, the separation point moves forward to the leading edge, while the slope of the lift curve decreases gradually. However, it doesn’t become negative because the suction on the leading edge, which contributes to lift, keeps growing when the AoA is increased.

It is necessary to underline that, when the separated zone of the boundary layer reaches half of the chord, the maximum lift coefficient is achieved.

In terms of moment coefficient, as the stall begins on the trailing edge the pressure coefficient on the extrados becomes more negative, so the consequence is a pitching moment in the nose-down direction, and it grows (in absolute value) with the AoA.

Regarding drag coefficient, it increases with the AoA, with sudden changes when the transition of the boundary layer is produced.

Finally, and to mention important properties of this kind of stall, it is important to know that the variations are soft, so the changes on the forces when the stall appears are not very sudden. This gives enough time to the pilot for correcting the problems, recovering the necessary lift (using nose-down moment) and overcoming the situation.

**Type B**

The next type of stall occurs on airfoils with a relative thickness $0.08 < \frac{\delta}{c} < 0.15$. The stall is produced near the leading edge, where a short bubble appears.

In these kind of airfoils, the boundary layer separation occurs when the flow is laminar before reaching the maximum AoA. Then, the shear layer becomes turbulent and it can be reattached on the airfoil, but a short bubble appears near the leading edge (as shown in Figure 11). As this bubble represents 1% of the chord, the forces are not affected.

![Figure 11. Stall in medium thickness airfoils (Meseguer et al. 2004)](image)

When the AoA grows, the bubble moves forward to the leading edge, complicating the reattachment. Moreover, the kinetic energy of the fluid inside the bubble grows until it is not possible to recirculate the flow.
Finally, these factors bring about the final burst of the bubble; consequently, the stall suddenly expands along the extrados and the forces suffer an abrupt change. The moment coefficient becomes negative (nose-down direction) while the drag coefficient increases significantly in the whole separated zone.

On the one hand, the maximum value of the lift coefficient for these airfoils is higher; however, as the stall appears suddenly, it can be dangerous because it appears with no advice, so the pilot cannot control the situation until the moment it begins.

Moreover, it is used in swept wings because the vortexes prevent bubble from bursting.

**Type C**

The thinnest airfoils are the ones included in this group, as well as the ones with a sharp leading edge. With a relative thickness $\frac{\delta}{c} < 0.08$, the bubble which appears in this kind of stall is by far longer than the one of medium thickness airfoils.

On these thin airfoils, a small value of AoA is enough to make the bubble of recirculation appear. The shear layer becomes turbulent and the boundary layer reattaches at a certain point along the extrados; however, that point moves backward to the trailing edge if the AoA is increased.

In the final step, when the reattachment point matches with the trailing edge, the maximum value of lift coefficient is achieved, but beyond that limit the stall occurs. The whole phenomenon is presented in Figure 12.

![Figure 12. Stall in thin airfoils (Meseguer et al. 2004)](image)

This airfoil is very characteristic due to the nose-up moment which appears before the stall. The bubble of recirculation creates an intense suction which in turn causes this moment. Moreover, the drag coefficient grows progressively until the separation point matches the trailing edge, when a significant drag appears due to the stall.
2.2. High lift devices

The efficiency of an aircraft is an important parameter, and it must be maximized as much as possible. In order to meet that goal, what are called high lift devices were created; more concretely, in 1918 the German scientist Gustav Lachmann developed the first prototype of this system. In this section the most common high lift devices are explained, together with the effects that they cause on the lift of the aircraft.

According to Prisacariu and Luchian (Prisacariu and Luchian 2014), high lift devices used for bearing surfaces are designed to expand the flight envelope by changing the local geometry (wing mechanization) according to phases of flight of the aircraft. The use of these devices came from the need for speed with low values for take-off and landing phases.

Moreover, there are two ways of increasing the maximum lift coefficient $c_{l_{\text{max}}}$: the first one relies on modifying the profile geometry, and these kind of devices are called passive systems. On the other hand, it is possible to control the boundary layer using these devices, which are in turn called active systems.

![Figure 13. Example of a passive high lift device (Brooksby 2010)](image13)

![Figure 14. Example of an active high lift device (www.russianpatents.com)](image14)

In the following paragraphs both types of devices are presented, beginning with the passive ones. These systems can be classified as well depending on the part of the geometry which changes; therefore, we can differentiate between the leading edge high lift devices and the trailing edge high lift devices, being possible (and usual) to combine both of them to achieve a better aerodynamic performance.
2.2.1. Passive high lift devices

The most common high lift devices are the ones which modify the trailing edge of the airfoil. As there are several ones, and it is not possible to cover all the existent devices, only the most important ones will be presented here.

According to Meseguer (Meseguer and Sanz 2011), there are two different ways of achieving high lift: the increasing of curvature of the airfoil and increasing the effective area of the wing. While the fundamental of the first one lies on devices which can be deflected, if it is added the ability of being extended, it is possible to increase the airfoil’s chord.

The ways of modifying the geometry and the consequences which can appear are still being studied, but the effects can be classified depending on which aspect it is changing (curvature or effective area).

In Figure 15 it is shown the effect of deflecting a passive high lift device (in this case a trailing edge device called flap). It can be noticed that, when the device is deflected (and therefore the curvature increases), the lift coefficient which is achieved at the same AoA grows as a consequence.

However, it is important to realise that the angle of attack at which the maximum lift coefficient is achieved decreases as well.

![TYPICAL VARIATION OF FLAPS](image)

\[ C_l \text{ vs. } \alpha \]

**NOTE:** NO L.E. DEVICE

Figure 15. Typical effect of flaps on lift (Mason 2007)
As it has been mentioned, changing curvature using high lift devices is not the only way of using them. The following Figure 16 shows the effect of not only deflecting the flap, but also extending it in order to increase the effective area of the airfoil.

**Figure 16. Effect of flap extension on lift (Mason 2007)**

As defined by Van Dam (Van Dam 2002), extending flaps increases the curvature of the wing, raising the maximum lift coefficient, this is the lift a wing can generate. This allows the aircraft to generate as much lift, but at a lower speed, reducing the stall velocity of the aircraft, or the minimum speed at which the aircraft will maintain flight. Moreover, extending flaps increases drag, which can be beneficial during approach and landing, because it slows down the aircraft.

On some aircraft, a useful side effect of flap deployment is a decrease in aircraft pitch angle which lowers the nose thereby improving the pilot's view of the runway over the nose of the aircraft during landing. However, the flaps may also cause pitch-up depending on the type of flap and the location of the wing.
Once explained the advantages of the high lift devices, it is necessary to classify them depending on the part of the airfoil you are modifying. As it has already been mentioned, it is possible to differentiate between the trailing edge devices and the leading edge devices.

2.2.1.1. **Trailing edge high lift devices**

The trailing edge high lift devices are the most common ones. As indicated by their name, the trailing edge of the airfoil is modified, adding different mobile systems which can be extended and/or deflected in order to get the effects explained in the previous paragraphs.

There are several types of trailing edge devices, and **Figure 17** shows some of the most used ones, the majority of which are explained further in this chapter. Every device definition will be presented together with an approximation of the increase of the lift coefficient due to the device usage, as well as the new maximum AoA. For further information, see Swatton (Swatton 2010) and Meseguer (Meseguer and Sanz 2011).

---

**Figure 17.** Trailing edge flaps (*Swatton 2010*)
• **BASIC AIRFOIL** \( \Delta c_l \approx 0\% / \alpha_{max} \approx 15^\circ \)

The basic airfoil, with no high lift devices. Its high lift features only depend on the airfoil’s shape.

• **CAMBER FLAP** \( \Delta c_l \approx 50\% / \alpha_{max} \approx 12^\circ \)

When it is deflected, the airfoil’s curvature increases. Its best performance is achieved when the cavity between the fixed part and the moving part is sealed. It is important to underline that, when the deflection exceeds the value of 10-15\(^\circ\), the flow separation is produced after the cavity, and while the lift progressively falls the drag increases suddenly.

Moreover, when the deflection is high enough, the effective area decreases (although this device cannot be extended) and it produces nose-down moment.

• **SPLIT FLAP** \( \Delta c_l \approx 60\% / \alpha_{max} \approx 14^\circ \)

It increases the curvature. The drag is high even if the deflection is low, so it is not proper for the take-off actuation. Moreover, and apart from the nose-down moment that it produces, its simplicity has induced this flap to be outdated.

• **(DOUBLE) SLOTTED FLAP** \( \Delta c_l \approx (70\%) 65\% / \alpha_{max} \approx (18^\circ) 16^\circ \)

This device, apart from increasing the airfoil’s curvature, it can be used to control the boundary layer. Because the air passes from the intrados to the extrados through the slot, it suction the boundary layer of the fixed part of the airfoil. Then, on the flap appears a new thin boundary layer which enables the flap to be deflected up to almost 40\(^\circ\).

Moreover, although the aerodynamic pros and cons depend highly on the slot’s shape, as there is no flow separation, the drag does not grow significantly. It also exists the double slotted flap (which is shown installed on a Boeing 747 in **Figure 18**), which features are very similar to the slotted flap, and only the lift coefficient increase and the maximum AoA grow.

![Figure 18. Multiple slotted flaps - BOEING 747 (Prisacariu and Luchian 2014)](image-url)
• (SLOTTED) FOWLER FLAP $\Delta c_l \approx (100\%) 90\% \, / \, \alpha_{\text{max}} \approx (20^\circ) 15^\circ$

This kind of flap is useful for all increasing curvature and effective area as well as for controlling the boundary layer. The high lift element passes through the entire cord using guides and then it deflects, what increases the drag but not more significantly that it does with the lift. Nose-down moment is created when this flap is used.

Moreover, nowadays one of the most trailing edge high lift devices used in the industry is the one which mixes the fowler flap and the slotted flap, due to the aerodynamic advantages which this combination can provide, although the structural complexity is higher.

Once the most common trailing edge high lift devices have been exposed, the next paragraphs present the leading edge devices in a similar way.

2.2.1.2. **Leading edge high lift devices**

Another way of modifying the geometry consists in changing the leading edge of the airfoil. The effect that produces is shown in Figure 19, and as explained by Cashman et al. (Cashman, Kelly, and Nield 2000), the use of a leading edge device increase the angle of attack at which the maximum lift coefficient is achieved, as well as the value of this coefficient.

![Figure 19. Effect of leading edge devices in lift (Cashman et al. 2000)](image)

The variety of leading edge of high lift devices is by far less than the trailing edge ones, but there are some systems which are worth mentioning, shown in Figure 20. Several writers have defined their features, like Swatton (Swatton 2010) and Meseguer (Meseguer and Sanz 2011).
• LEADING EDGE SLOT $\Delta c_l \approx 40\% / \alpha_{\text{max}} \approx 20^\circ$

It is used to control the boundary layer (delaying the flow separation), usually in Type $C$ airfoils. Although they are used in slow aircrafts, its highest disadvantage is the significant increase of the drag when at high velocities.

• SLOTTED LEADING EDGE FLAP (SLAT) $\Delta c_l \approx 50\% / \alpha_{\text{max}} \approx 20^\circ$

The definition of the slat is a little airfoil (with abrupt curvature) installed towards the leading edge of the main airfoil, with the goal of diminishing the suction in that part of the geometry.

The main consequence is the avoidance of the flow separation, and hence the AoA can be increased. However, it also increases the drag (although not significantly).

• KRUGER FLAP $\Delta c_l \approx 50\% / \alpha_{\text{max}} \approx 25^\circ$

The main advantage of this device is that it is possible to achieve an increase in the airfoil curvature, although it reduces the effective AoA. It is used to delay the flow separation, usually in Type $C$ airfoils (where it is easier to install the device).

As important characteristics, if the deflection of the flap is low enough, the lift can be reduced as the consequence; moreover, its design produces a nose-up moment.
2.2.1.3. Combining leading edge and trailing edge high lift devices

Despite the available devices available for achieving higher lift values, the best airfoil performance is achieved when a leading edge device and a trailing edge device work together in the same airfoil. The most suitable combinations of these high lift devices mix the aerodynamic improvements as well, providing the advantages of both kind of systems, explained below from the information from Prisacariu et al. (Prisacariu and Luchian 2014). The effects are graphically exposed in Figure 21 and Figure 22.

Figure 21. Effect of flaps and slats in $c_l - \alpha$ curve (Prisacariu and Luchian 2014)

The effects can be noticed in Figure 21; flaps help to achieve a higher lift coefficient at the same angle of attack, while adding the slats increase the maximum lift coefficient as well as grow up the maximum angle of attack in which the flow separation appears.

Figure 22. Effect of flaps and slats in $c_l - c_d$ curve (Prisacariu and Luchian 2014)

Despite being a different graphic, Figure 22 shows similar information; using only the trailing edge high lift provides a higher lift coefficient associated to a specific drag coefficient. Likewise, adding the slats enables the airfoil to have a higher maximum lift coefficient, although the drag coefficient associated to a specific lift coefficient increases.
Despite all the available combinations of trailing edge and leading edge high lift devices, the most common ones are explained based on Meseguer and Sanz (Meseguer and Sanz 2011).

- **SLAT WITH SLOTTED FLAP** $\Delta c_L \approx 75\% / \alpha_{max} \approx 25^\circ$

In this combination, the slat avoids the flow separation on the leading edge (which can appear when the flap is used). Furthermore, the moments from the aerodynamic centre can be compensated when combining two opposite effects: the one created by the slat and the one created by the flap.

![Figure 23. Slat with slotted flap (www.eurofacs.org)](image)

- **SLAT WITH DOUBLE SLOTTED FOWLER FLAP**

The most used in the current airplanes. They increase both curvature and effective area. The flap slot suctions the boundary layer while the slat stabilize it in the nearby of the leading edge, and the moments can be compensated.

This combination is said to be the most optimum one (Mason 2007). For that reason, we can find it in several commercial aircrafts.

### 2.2.2. Boundary layer control

Although passive high lift devices are commonly used to modify the geometry or the curvature, it is also possible to control not the geometry of the object but the fluid behaviour. According to Hazen (Hazen 1948), boundary layer control is defined as those methods created to control the fluid behaviour of fluid boundary layers.

There are several ways of modifying that behaviour, such as controlling the velocity profile or vary the fluid properties. The first one can manage the flow using blown devices or air suction. On the other hand, fluid properties can be modified using magnetic or electric gases, according to Lachmann (Lachmann 2014).

In the following paragraphs some of the boundary layer control devices which can be implemented on an airfoil are presented, together with a brief description of their function.
2.2.2.1. Zero-net-mass-flux (ZNMF)

Since 1950, when the Zero-net-mass-flux (ZNMF) was found, there have been more institutions and articles every year who have started researching about these devices. According to Zhang et al. (Zhang, Wang, and Feng 2008), a ZNMF jet can be defined as a jet produced by a sinusoidal oscillating membrane or piston to alternatively force fluid through an orifice into the external flow field and entrain fluid back.

A usual ZNMF, which is shown in Figure 24, consists of two strokes: the blowing stroke and the suction stroke. In the first process, the ejected fluid separates at the sharp edges of the orifice (which is a slot orifice in 2D and a circular orifice in 3D) creating a vortex pair in 2D or a ring in 3D.

Then, when the suction stroke begins, the fluid is far enough from the orifice to keep on propagating away due to its self-induced velocity. Therefore, the suction does not entrain back the fluid into the cavity, but it makes the fluid coalesce to synthesize a jet which transfers momentum to the embedding flow (Zhang et al. 2008).

![Figure 24. Formation of a zero-net-mass-flux jet (Zhang et al. 2008)](image)

The use of zero-net-mass-flux devices has some advantages compared with other traditional methods of blowing and suction control. As presented by Lachmann (Lachmann 2014), the ZNMF does not require an external fluid supply nor a complex piping system. This enables the device to be a more reduced size and weight than other boundary control devices. Other features which make these systems interesting to be investigated are the low cost or the high reliability.

To understand why the ZNMF are interesting, it is important to present the way they interact with the cross flow. As described by Tang et al. (Tang et al. 2007), the vorticity of the vortex ring (if described in 3D) in the upstream branch is significantly suppressed because the resident vorticity in the boundary layer has the opposite sign. This, in turn, produces an intensification of the vorticity in the downstream part (because the sign is the same).
Therefore, and as concluded by Tang et al., the vortex ring undergoes both stretching and tilting, while it propagates downstream. Because of that, what is called a hairpin vortex is formed, the legs of which are connected to the orifice. The hairpin vortices produced by the ZNMF are shown in Figure 25.

![Hairpin vortices](image)

**Figure 25.** Hairpin vortices produced by the ZNMF. Orifice at the top, cross flow from right to left (Tang et al. 2007)

As concluded by Rumsey (Rumsey 2004), the induced hairpin vortices cause the high speed fluid out of the boundary layer to be injected in the low speed region in the boundary layer. Consequently, the relative low speed fluids are lifted up into the external flow, which enhances the momentum of the boundary layer and helps the flow to overcome the adverse pressure gradient. This, in turn, delays the boundary separation.

The results of the performance of a ZNMF can be seen in Figure 26, where both strokes interact with the cross flow. The orifice is located at x=48-56 mm.

![Interaction of the ZNMF with the cross flow](image)

**Figure 26.** Interaction of the ZNMF with the cross flow. (a)Suction (b) Blowing. (Schaeffler 2003)

The way in which the jet adds energy to the boundary layer depends on the stroke (Schaeffler 2003). During the suction stroke, most of the fluid which enter to the cavity come from the upstream boundary layer. This occurs due to the velocity component of the flow around the actuator, which is positive (in the streamwise direction). As a result, the boundary layer becomes thinner and the momentum of the upstream boundary layer increases.
On the other hand, in the blowing stroke, the jet is tilted due to the effect of the cross flow, creating the hairpin vortices. Parallel to the suction stroke, the fluid with higher momentum moves downstream, and this complements the momentum deficit in the boundary layer.

According to these interactions, and as concluded by Schaeffler, the actuator both produces momentum and allows the boundary layer overcoming pressure gradients, due to the mixing of the low momentum fluid (located near the surface) and the high momentum external flow (which is induced by hairpin vortices).

**Important parameters of the Zero-Net Mass Flux actuator**

In order to define the ZNMF actuator, it is important to define its parameters. We can classify the parameters on the following categories:

- Geometric parameters
  - Cavity diameter $D$
  - Cavity depth $H$
  - Slot orifice width $D_0$
- Parameters of the working fluid, explained in section 3.1.1.3.
- Actuation parameters
  - Frequency $f$
  - Diaphragm oscillation amplitude $A$

However, and according to the research done (Amitay and Glezer 2006), it is possible to combine them to create two independent dimensionless parameters. The parameters are the *stroke length ratio* and the *Reynolds number based on the blowing stroke*.

The stroke length is, as defined by Amitay and Glezer, the length of a fluid column containing the fluid which is pushed out of the orifice during the blowing stroke:

$$L_0 = U_0 T$$

Equation 3. Stroke length

Being $T$ the period of actuator excitation and $U_0$ the blowing velocity over the period, time-averaged, expressed as follows:

$$U_0 = \int_0^{T/2} \bar{u}_0(t) dt$$

Equation 4. Blowing velocity

And being $\bar{u}_0$ the instant stream-wise averaged velocity over the orifice section.
Moreover, the stroke length ratio is obtained as the stroke length dimensionless with the slot orifice width:

\[ L = \frac{L_0}{D_0} \]

**Equation 5. Stroke length ratio**

It is important to underline that the stroke length ratio can represent the geometry characteristics of the actuator (Tang et al. 2007).

On the other hand, the Reynolds number of the ZNMF, is presented as follows:

\[ Re_{U_0} = \frac{U_0 D_0}{\nu} \]

**Equation 6. Reynolds number based on the blowing stroke**

Where the cinematic viscosity of the fluid \( \nu \) is needed. As presented by Amitay and Glezer, the criteria for defining if the vortex pair entrain back into the cavity during the suction stroke depends mostly on two different parameters: the self-induced velocity of the vortex formed during blowing stroke (\( V_I \)) and the velocity during the suction stroke (\( V_S \)).

The ratio of these two velocities can be considered, as found out by Utturkar (Utturkar et al. 2003), as the parameter which is compared with a constant K. This constant, following Utturkar et al. investigations, is approximately 2.0 for two dimensional ZNMF jets.

The velocities ratio can be approximated as follows (Utturkar et al. 2003):

\[ \frac{V_I}{V_S} = \frac{1}{St} = \frac{Re_{U_0}}{S^2} \]

**Equation 7. Velocities ratio for vortex formation criteria**

Where the dimensionless parameters used to define the ratio are the Strouhal number (Equation 8) and the Stokes number (Equation 9)

\[ St = \frac{2\pi f D_0}{U_0} = \frac{2\pi}{L} \]

**Equation 8. Strouhal number**

\[ S^2 = \frac{2\pi Re_{U_0}}{L} \]

**Equation 9. Stokes number**

The most common criterion according to these researches is the expression with the Strouhal number. So, as expressed in **Equation 10**, if the inverse of this number is higher than the constant K, the vortex pair will not entrain back to the cavity during the suction stroke.

\[ \frac{1}{St} = \frac{L}{2\pi} > K \]

**Equation 10. Jet formation criterion**
During similar investigations (Holman et al. 2005), it was found that the jet formation was also restricted by the stroke length ratio \( L > 0.5 \) for the 2D slot by theoretical analysis. It was also concluded that, although the circulation of the vortex pair increases with \( L \), when this parameter exceeds 4 the circulation remains constant and the excess vorticity forms a second vortex behind the primary vortex.

Moreover, there are also important parameters associated with the cross flow (Ugrina 2007). The most important ones are the incoming flow velocity \( U_\infty \), and the boundary layer thickness \( \delta \) at the location where the actuator is set. Similarly to the other parameters, these ones are also dimensionless with the blowing velocity, called the velocity ratio \( R = U_0/U_\infty \), and with the scale of the orifice \( \delta/D_0 \).

However, the most important parameter is the momentum coefficient \( C_\mu \), which is defined as the ratio of the momentum of the actuator over the momentum of the cross flow. According to Durrani (Durrani and Haider 2011), the momentum coefficient is presented as \textbf{Equation 11}.

\[
C_\mu = \frac{h \cdot V_{\text{max}}^2 \cdot \rho}{c \cdot U_\infty^2 \cdot \rho}
\]

\textbf{Equation 11.} Momentum coefficient

Where \( h \) is the orifice width and \( V_{\text{max}} \) is the maximum value of the blowing/suction velocity. There are several studies which calculate the momentum coefficient, and the most accepted value ranges from 0.0005 to 0.003 (Zhang et al. 2008)(Holman et al. 2005).

Moreover, and according to Ugrina, it was necessary to study different velocity ratios to know which ones would allow the ZNMF to penetrate the boundary layer, and the result was a minimum ratio of 1.0. Furthermore, smaller ratios would cause the jet to be deflected and buried into the boundary layer, not perturbing the farthest flow.

Moreover, in this study it was also concluded that as the velocity ratio increases, the strength of the jet for pushing the boundary layer away from the wall increases as well. There are more studies which supports these conclusions (Tang et al. 2007), (Mittal and Rampunggoon 2002).
2.2.2.2. Vortex Generators

Since it is important to control the boundary layer and the flow separation, many studies have been done in order to investigate the use of vortex generators (VGs), for these purposes.

According to (Titchener and Babinsky 2015), the main goal of these devices is to introduce vorticity into the flow which is close to the surface or wall. This brings about a transfer of fluid and momentum in the plane which is normal to the vortex (for this reason the vortex generators are normal to the surface where the flow is being controlled).

This momentum transfer increases the shear stress in the wall, making the boundary layer more resistant to adverse pressure gradients. The configuration of the locations of the VGs may differ depending on the purpose, and in Figure 27 it is shown one of the possibilities together with the contours which are obtained.

![Figure 27. Co-rotating vortex generators (Titchener and Babinsky 2015)](image)

Moreover, the vortex generators are used for different purposes related to fluid mechanics, such as boundary layer separation (Schubauer and Spangenberg 2006), aircraft wing lift enhancement (Bragg and Gregorek 1987), or afterbody drag reduction (Calarese, Crisler, and Gustafson 1985).
2.2.2.3. Plasma actuators

Another technique for flow separation is based on plasma actuators. These actuators consist of two electrodes (Post 2004): one of them insulated using some dielectric insulation (i.e. Kapton), and the other one exposed to the surrounding air. About their location, they are asymmetrically located, with a little overlap (order of mm). The scheme of the actuator is shown in Figure 28.

![Figure 28. Plasma actuator scheme (Post 2004)](image)

The working of these actuators is based on the formation of low-temperature plasma. According to Post, when the AC voltage is high enough between the electrodes, the air is ionized forming the plasma. Finally, and when the air is ionized, its molecules (which are in the air surrounding the electrodes) are also ionized, and consequently accelerated through an electric field.

Moreover, plasma actuators have some advantages over other flow control devices (Md Daud et al. 2015). They are fully electronic, with no mechanical parts, so the high loadings are not expected to be a problem. Regarding response, their high frequency actuation guarantees a fast response.

Furthermore, they are low mass and scalable (they can range from less than 1 mm to 20 cm), and in addition their energy requirements are relatively low, due to the high energy density of the plasma.
2.2.2.4. Wing fences

Wing fences are defined (Stremel 1996) as passive aerodynamic devices for flow control which are attached to aircraft wings. More specifically, they are flat plates located in the extrados (see Figure 29) which main function is to prevent the entire wing of suffering stalling at the same time. If these devices are located in the intrados, they are called vortilons.

![Wing fences](www.cap-ny153.org)

According to Stremel, when the wing slows toward the stall speed, part of the airfoil moves sideways towards the wing tip. Therefore, at the tip the airflow can be almost all spanwise instead of chordwise, something which in turn causes the airspeed to drops below the stall. As the stall phenomenon is something which is desirable to be avoided (see section 2.1.2), fences are used in some aircrafts in order to control the stall.

The main goal of fences is to delay, or eliminate, the stall by preventing the spanwise airflow to arrive at the wingtip. This is because if the airflow moves too far along the wing it can gain speed, which would increase the effect of the spanwise airflow in the situation of stall.

![Lift coefficient spanwise using or not a fence](www.b2streamlines.com)

Figure 29. Fence location (www.cap-ny153.org)

Figure 30. Lift coefficient spanwise using or not a fence (www.b2streamlines.com)
As shown in Figure 30, and according to several researches (Kuhlman 1997), the effect of using or not a fence changes the lift behaviour over the wing. Due to the airflow which moves sideways, if a fence is not used the maximum lift coefficient is reached at the wingtip, something not desirable because the moments would be high at the root of the wing.

On the other hand, if a fence is used at the middle of the wing, the maximum lift coefficient is achieved in the fence, so the moment would not be so great that if the fence is not used. Moreover, the moment created from the root to the fence has contrary sign that the one created from the fence to the wingtip, which helps to reduce the moments at the root noticeably.
PART III

AIRFOIL AND ZERO-NET-MASS-FLUX ANALYSIS
3.1. Characteristics of the NACA 4421 airfoil

In every study which is going to be made, it is important to define all the parts in depth, to have knowledge enough to study the interactions between them. For this reason, it is necessary to start presenting the airfoil around which the study will revolve about.

The basis of this project revolves around the NACA 4421 airfoil, and in these sections they will be analysed both the airfoil within its real applications or the surrounding conditions and its simulation using ANSYS software (including ANSYS ICEM CFD for the mesh and ANSYS Fluent for the simulations).

3.1.1. NACA 4421 analysis

This section will present the basic parameters which define the airfoil, as well as some applications in the reality and the operating condition in which it will be studied during this project. It is important to define well the airfoil, because it is the basis of the whole study.

3.1.1.1. Airfoil characteristics

The NACA 4421 airfoil is classified as a thick airfoil, because as defined by Abbott (Abbott and Von Doenhoff 1959), the maximum thickness of the airfoil (dimensionless with its chord) is \( \frac{\delta}{c} = 0.21 = 21\% \). As defined in Section 2.1.3, the airfoils whose thickness exceeds 15% are considered as thick airfoils, with its respective way of entering stall.

\[
\delta/c = 0.21 = 21\%
\]

![Image of NACA 4421 airfoil](image31)

**Figure 31.** NACA 4421 airfoil (Abbott and Von Doenhoff 1959)

The established chord of the airfoil is in this study of \( c = 1 \, m \). According to the UIUC Data Base (Selig 2010), the maximum thickness is reached at 30% of the chord, as it can be noticed in **Figure 31**.

Moreover, the maximum (upper) camber (see **Figure 32**) is the line formed as the average of the extrados and the intrados. In this airfoil, the maximum camber is located at 40% of the chord, being 4% dimensionless with the chord.
3.1.1.2. Airfoil applications

Although this airfoil is not one of the most common among the available for using in typical aircrafts (such as Airbus A320 or similar), it is possible to find specific uses of this geometry in different systems.

Nowadays, it is not very common to find VTOL (Vertical Take Off and Landing) aircrafts commercially sold, but in the past some of these systems were implemented as a typical aircraft, mostly for particular uses (aerodromes), and one of the applications of the airfoil is one of these VTOLs (Billingsley 2002).

The name of the VTOL is Vanguard Omniplane (shown in Figure 33), and its interest comes from being a VSTOL (Vertical and Short Take Off and Landing), which has the advantage of needing less space to manoeuvre. It was developed in 1959 (Mcchesney 2012), and it has the singularity of having two ducted fans which use the NACA 4421 airfoil.

Figure 32. Airfoil parameters (www.grc.nasa.gov)

Figure 33. Vanguard Omniplane (Mcchesney 2012)
3.1.1.3. Operating conditions

The operating conditions at which the airfoil will be studied depend on the studies which will be used for the comparison. Parameters such as the velocity, pressure and temperature have to be defined, using dimensionless parameters such as the Reynolds number or the Mach number.

Firstly, it is important to determine the Reynolds number at which the study will be performed. As presented in Section 2.1.1, the Reynolds number depends on the velocity \( U_\infty \), the chord \( c \), and both the density \( \rho \) and the dynamic viscosity \( \mu \).

As the fluid is air, we will study the dynamic viscosity as the one at the temperature \( T = 15^\circ C \). The density will be treated as an ideal gas, and while the chord is already settled, the Reynolds will depend on the velocity of the free stream.

Because the comparison will be done using Abbott experimental data (Abbott and Von Doenhoff 1959), and because the turbulences are an important part of the study, the chosen Reynolds is \( Re = 3 \cdot 10^6 \). Is then possible to find the upstream velocity and present it as the Mach number, being then \( M_\infty = 0.13 \).

As the Mach number is low enough, we could consider the fluid as incompressible. Moreover, the density will be then even constant.

The pressure used for the study is considered as the reference at sea level, being \( P = 101325 \ Pa \). However, this is the operating pressure at which we set the study, and the software (Fluent) will be set to perform a pressure-based simulation.

In order to facilitate the identification of the data to the lecturer, Chart 1 shows the variables at which ANSYS Fluent has been set up.

<table>
<thead>
<tr>
<th>Reynolds number ( Re )</th>
<th>( 3 \cdot 10^6 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density ( \rho ) [kg/m³]</td>
<td>1.225 (Ideal gas law)</td>
</tr>
<tr>
<td>Viscosity ( \mu ) [kg/(m·s)]</td>
<td>( 1.79 \cdot 10^{-5} )</td>
</tr>
<tr>
<td>Chord ( c ) [m]</td>
<td>1</td>
</tr>
<tr>
<td>Mach number ( M_\infty )</td>
<td>0.13</td>
</tr>
<tr>
<td>Gauge pressure ( P_g ) [Pa]</td>
<td>101325</td>
</tr>
</tbody>
</table>

Chart 1. Operating conditions of the airfoil
3.1.2. NACA 4421 simulation

Now that the basic parameters of the study have been settled, it is time to present the whole configuration of the ANSYS Fluent simulations. The airfoil mesh, the selection of the solver and the results obtained, together with the comparison with the experimental data are topics which are developed in this section.

3.1.2.1. Meshing the airfoil

Meshing fundamentals

The basis of a simulation, however the type of simulation studied, is the mesh of the object or field which is being simulated. The mesh consists on a spatial discretization with a specific number of nodes (or elements), which are in turn inside cells; these nodes are the points at which the equations will be solved (ANSYS 2013).

Due to the interactions of these nodes, it is important to guarantee a good quality mesh, because it is one of the keys for the success of the study. For that reason, some possibilities have been studied before beginning the mesh; how to do it and how to know if the mesh is good enough are important aspects which must be taken into account.

There are two types of meshes: structured and unstructured. The difference lies in the nodes organisation, which in turns affects to the interaction between the elements (Owen 1998).

As the mesh is a discretization of the geometry involved in the simulation, the unstructured mesh is capable of recreating the geometry with more accuracy, but the interaction of the partial differential equations is not so controlled as in the structured mesh.

In this study, the structured mesh has been used in order to let the software control the interaction, but guaranteeing the quality of the mesh in order to get the highest accuracy, which has been also limited to the computational power. However, the mesh parameters have been configured following several criteria.

NACA 4421 mesh characteristics

The far-field extension is presented in Figure 34. It can be seen the extension of the far-field, which it has been studied and has to be ranged from 11 to 20 times the airfoil’s chord. In this study, the highest number has been chosen, in order to guarantee the most reliable far-field from which the reference values will be taken.
Moreover, and to present the mesh, it is necessary the number of nodes. Here, there are 220586 nodes, and in each node the equations will be solved using approximations and models that are explained in Section 3.1.2.2.

Furthermore, the mesh has been created in what are called stages. It is important to mention that the mesh has different refinement options, where refinement is the amount of nodes in a specific zone. In other words, the more refined the mesh is, the more nodes in the same area will be.

As the region which is more interesting is the boundary layer hence the region near the airfoil, the mesh has to be very refined in this zone. On the other hand, the furthest part of the mesh is not very important, so we can reduce the refinement (it is then said that the mesh is coarse).

Therefore, the stages are the different zones with different refinements, which in turn creates different blocks. The edges which separates stages and blocks are called split edges, and they are shown in Figure 35.
It is important to take a look at the stages. The most coarsened stage is the furthest one (identified as 1), which includes the whole far-field; then, another one quite closer to the airfoil (number 2) can be also identified. To present the other stages, it is necessary to zoom the picture, something which is presented in Figure 36.

In this zoom it is possible to identify another stage very close to the airfoil (number 3); and the last one, which has a length of mm, is the most refined stage because it is the one which will study the boundary layer behaviour (number 4).

Once the stages are settled, it is time to define the airfoil’s wake. After the boundary layer, the wake is the most important part because is where vortices and turbulences will be located.
For that reason it is important to separate the wake into stages as well. In Figure 37 the first four stages are presented, being the first one the most refined stage (as it is the closest to the airfoil).

Moreover, the 4th and 5th stages are presented in Figure 38. These stages are more coarsened, because they are further from the airfoil and closer to the far-field.

![Figure 37. Stages of the wake](image)

Once the blocks are created, it is time to define the mesh with its cells. For that purpose, it has to be decided the number of separations per edge, as well as the function that these separations will follow. For example, in the stages which surround the airfoil, the separations will be further from each other as you move away from the geometry; the same occurs with the wake’s stages.

In Figure 39, it can be noticed these differences. The region of the boundary layer is almost full, while the space between separations further from the airfoil increases.

Finally, in Figure 40 the entire mesh is presented, in order to let the reader have an overall view of the mesh.
Figure 39. Mesh around the airfoil

Figure 40. Mesh of the whole fluid field
Mesh quality

When the mesh is finished, the next step consists of evaluating it. ICEM CFD has several tools in order to let the user have an approximate idea of the overall quality of the mesh.

Before using these tools, it is important to present a restriction that the mesh has to accomplish in order to be reliable. More specifically, it has been studied that the height of the first cell (being the one adjacent to the airfoil) is very important for guaranteeing the reliability of the mesh. According to Salim (Salim and Cheah 2009), the dimensionless parameter related to this height is called $y^+$.

This parameter depends on the Reynolds number, the chord length and the first cell height itself. Then, knowing your desired $y^+$, and introducing the other parameters, it is possible to find the needed height for accurate results. Here, our $y^+$ had to be under 5 to be good enough, so with the parameters presented in Section 3.1.1.3 it was found that the minimum first cell height had to be $4.27 \cdot 10^{-5} \text{ m}$. As the height of our mesh is $3.78 \cdot 10^{-5} \text{ m}$, the mesh is good enough in this aspect.

Back to ICEM CFD tools, there are three important parameters which must be taken into account to evaluate the mesh, according to ANSYS. The first one is the Aspect Ratio ($AR$), this ratio evaluates the area of a cell in relation to its adjacent cells. The perfect mesh would have a maximum aspect ratio of 1; however, values below 250 are considered as correct enough to have reliable results (ANSYS 2013).

In Figure 41 the AR of the mesh is evaluated. As shown, the maximum aspect ratio is 259, but the majority of the elements have an aspect ratio below 151. This provides evidences that the mesh is, in this aspect, good enough.

![Figure 41. Aspect ratio of the mesh (X axis: Aspect Ratio of the element; Y Axis: Number of elements with that aspect ratio)](image)

The second parameter is the Eriksson skewness, this is a parameter which evaluates the symmetry of the cells. In other words, to evaluate the distortion, because a high distortion (so a low skewness) would be prejudicial for the solver.

In Figure 42 the skewness of the mesh is presented. As shown, the minimum skewness is 0.841, and due to skewness greater than 0.8 are considered good enough (ANSYS 2013), the mesh is reliable in this characteristic.
Finally, the general quality of the mesh can be evaluated. Again, values over 0.8 are good enough (ANSYS 2013), and in Figure 43 it can be noticed that almost all the elements are over 0.85. Then the quality of the mesh is good enough.

![Eriksson Skewness](image)

Figure 42. Eriksson skewness evaluation of the mesh

![Quality](image)

Figure 43. General quality of the mesh

In conclusion, it is possible to confirm that according to ICEM CFD, the mesh is good enough to do the studies. The following step will be the simulations in order to compare the results with the experimental data.

### 3.1.2.2. Choosing the solver

Before launching the simulations, it is necessary to define the solver that the software (here, ANSYS Fluent will be used) will use in order to solve the equations. Before choosing the solver, it is necessary to present a brief introduction of the available Turbulence Models

**Turbulence models**

As the computational power is not available to everyone, in general it is not possible to solve the Navier-Stokes equations that simply. For this reason, several methods to approximate the solution have been created.

According to Capote (Capote, Alvear, and Abreu 2008), the most basic computational models for calculating the turbulence are called *Reynolds Averaged Navier Stokes* (*RANS*). As its name claims, they are based on the average of the fluid equations, so the variables are calculated as an averaged number and a fluctuation term.

But the problem comes with the *Reynolds turbulence tensor*. As investigated by Boussinesq and summarised by Bona (Bona, Chen, and Saut 2002), these tensor consists, among other parameters, on the dynamic viscosity coefficient $\mu_t$. And in order to calculate this coefficient, it is necessary to add more equations to the original equation system.
It is possible to choose how many equations will be added. From zero to two equations is the most common range, even though there are possibilities which involve more equations (although they are beyond the scope of the study).

In order to give evidence of the accuracy of the \( k - \omega \) SST compared to other models, ANSYS provide results of the same simulation using different solvers, and comparing the obtained data with the experimental values. This comparison is presented in Figure 44.

![Figure 44](image.png)

**Figure 44.** Comparison of velocity profiles for NACA 4412 airfoil using different turbulence models (Menter 2011)

It can be easily noticed that the SST model works with higher accuracy than the other ones, while it does not require a lot of extra computational capacity.

Moreover, recent studies have compared the accuracy of the results depending on the type of solver which is chosen. In Figure 45 the vorticity of the same simulation test is presented using many different turbulence models, in order to show the differences between them.

![Figure 45](image.png)

**Figure 45.** Contours of vorticity magnitude using different turbulence models (Sandberg 2015)

The figure evidences that RANS solvers are not so accurate calculating and presenting the results. While RANS models only solve an averaged form of the Navier-Stokes equations, the DNS models solve the equations itself. For this reason, the results presented in this study will not be full realistic, and they will be an approximate idea of the ranges in which the values will oscillate.
Besides, in the other solvers the vorticity is well defined with its own shape (although SRS is less precise), while RANS models only show blur regions with no shape. Then, RANS models are useful for nothing but identifying these regions.

The most used models are those ones which include two equations (Gao et al. 2008). The first one, which has been used since nowadays and it is still used) is the $k-\epsilon$ model. The equations which this model add to the fluid equation system include these two parameters. The first one ($k$) is the kinetic energy of the turbulent fluid, while $\epsilon$ represents the dissipation velocity of the fluid.

The other turbulence model is the $k-\omega$ model. The parameters involved are the kinetic energy $k$, and now $\omega$ is the specific dissipation. The difference of both solvers are beyond the scope of the project, but further information can be found in Gao (Gao et al. 2008), Capote (Capote et al. 2008) and Bona (Bona et al. 2002).

According to the mentioned articles, the $k-\omega$ turbulence models approximates better the behaviour of the fluid near the walls and surfaces. As both models are good enough in the rest of the fluid field, applying $k-\omega$ near the walls and $k-\epsilon$ in the rest of the field (as it requires less computational power), mixing both of them would be the best option.

The solver which mixes both approximations is the $k-\omega$ SST model, being the Shear Stress Transport (SST) the formulation which combines the best of both worlds, and therefore it is the most suitable option. In Figure 46 the choice of the turbulence model in Fluent can be seen.

![Figure 46. Choosing the turbulence model in Fluent](image)

Figure 46. Choosing the turbulence model in Fluent
3.1.2.3. Comparison with experimental data

Now that Fluent has been set up, it is now time to launch the simulations. The experimental data which will be used for the comparison is extracted from Abbott (Abbott and Von Doenhoff 1959), from where the $C_l - \alpha$ graphic will be extracted and compared with the numerical simulations.

The experimental graphic is presented in Figure 47, taking into account that the used Reynolds number is $Re = 3 \cdot 10^6$.

![Figure 47. NACA 4421 $C_l - \alpha$ curve (Abbott and Von Doenhoff 1959)](image)

However, in order to get results more quantitative instead of only qualitative, points from the curve have been exported using G3DATA software. The angles of attack which have been exported are the same which will be evaluated through the simulations, in order to get the same level of accuracy and to facilitate the comparison.
In Chart 2 the values taken from the experimental data are presented, while in Chart 3 the results correspond to the different simulations performed using Fluent.

<table>
<thead>
<tr>
<th>Angle of Attack ((\alpha) [(^{\circ})])</th>
<th>Lift coefficient (c_l)</th>
</tr>
</thead>
<tbody>
<tr>
<td>-7</td>
<td>-0.3379</td>
</tr>
<tr>
<td>-3</td>
<td>0.1103</td>
</tr>
<tr>
<td>0</td>
<td>0.3621</td>
</tr>
<tr>
<td>7</td>
<td>1.0221</td>
</tr>
<tr>
<td>12</td>
<td>1.3255</td>
</tr>
<tr>
<td>14</td>
<td>1.3287</td>
</tr>
</tbody>
</table>

Chart 2. Experimental data of \(c_l - \alpha\) curve (Abbott and Von Doenhoff 1959)

<table>
<thead>
<tr>
<th>Angle of Attack (AoA) [(^{\circ})]</th>
<th>Lift coefficient (c_l)</th>
</tr>
</thead>
<tbody>
<tr>
<td>-7</td>
<td>-0.3144</td>
</tr>
<tr>
<td>-3</td>
<td>0.1305</td>
</tr>
<tr>
<td>0</td>
<td>0.3640</td>
</tr>
<tr>
<td>7</td>
<td>1.0036</td>
</tr>
<tr>
<td>12</td>
<td>1.3202</td>
</tr>
<tr>
<td>14</td>
<td>1.3429</td>
</tr>
</tbody>
</table>

Chart 3. Results of the numerical simulations using ANSYS Fluent

It is now possible to evaluate the relative error between experimental data and simulations results, using the expression presented in Equation 12. The results are shown in Chart 4.

\[
RE \, (\%) = \frac{c_{l_t} - c_{l_s}}{c_{l_t}}
\]

Equation 12. Relative error

<table>
<thead>
<tr>
<th>Angle of Attack (AoA) [(^{\circ})]</th>
<th>Relative error (RE)</th>
</tr>
</thead>
<tbody>
<tr>
<td>-7</td>
<td>7.47 %</td>
</tr>
<tr>
<td>-3</td>
<td>15.48 %</td>
</tr>
<tr>
<td>0</td>
<td>0.52 %</td>
</tr>
<tr>
<td>7</td>
<td>1.84 %</td>
</tr>
<tr>
<td>12</td>
<td>0.40 %</td>
</tr>
<tr>
<td>14</td>
<td>1.06 %</td>
</tr>
</tbody>
</table>

Chart 4. Relative error between the experimental data and ANSYS Fluent simulations
It can be noticed that, apart from the first two values, the other AoA present almost the same value, and taking into account that the angles which are more important for the study are the ones closer to the stall (around 13°), we can conclude that the results are reliable enough.

In order to give more evidences, both $c_l - \alpha$ graphics have been plotted at the same time (see Figure 48), in order to have visual information about the accuracy of the results.

The graphic highlights the similarity of both results. The two curves are almost overlapped, showing the data closeness, something which is consistent with the previous argument involving relative errors.

![Figure 48. Comparison of experimental data (Abbott and Von Doenhoff 1959) and results of the simulations](image)

Therefore, it has been proved that the mesh is good enough because the results of the simulations will be very similar to the experimental results that would be obtained if the experiments were performed. The next step is, therefore, to study the way the ZNMF will be introduced to the simulation.
3.2. The Zero-Net-Mass-Flux actuator

Once the mesh and the airfoil have been successfully simulated to guarantee the reliability of the results, it is now time to insert the Zero Net Mass Flux into the simulation. The theoretical basics of the actuator are described in Section 2.2.2.1, and in this section all the characteristics of the studied ZNMF will be settled, as well as the way in which the ZNMF will be simulated in Fluent.

3.2.1. Boundary condition

The Zero Net Mass Flux actuator can be described as an orifice on the airfoil’s surface which connects the extrados with a cavity. In this cavity, a membrane (which can be activated in many different ways, such as piezo-electrically) oscillates in order to give velocity to the cross flow, and add momentum to the boundary layer.

As the aim of this study is to simulate the actuator effect more than its geometry itself, a way of simplify this effect needs to be found. The most practical way of simulating the effect is to study the interaction of the actuator with the cross flow, and simulate this interaction as a boundary condition.

This boundary condition will be defined on the airfoil’s extrados, and the length of the part which will be affected by the condition will be the same as the theoretical length of the actuator’s orifice.

It must be underlined that the parameters, in its majority, have been settled before simulating using the available bibliography. To begin with, the position of the orifice has been established at a 17% of the chord, according to several experiments performed by Glezer (Amitay and Glezer 2006).

The first step is to define the type of function which will define the boundary condition. As the membrane oscillates, the two strokes (blowing stroke and suction stroke) will cause a change on velocity direction and magnitude. The blowing stroke will make the fluid move away from the extrados (perpendicular to the surface), while the suction stroke will create a velocity with the same direction but different sense, causing this suction effect.

Therefore, it can be assumed that the velocity will have the same behaviour than the membrane movement, being a sinusoidal function. This sinusoidal function of the oscillation, with the parameters defined, will recreate the effect of the ZNMF over the airfoil’s surface.

Now that it has been decided the kind of function, the next step is to identify which parameters have to be defined. An oscillation function is, in general, as presented as in

\[ f(t) = C_1 + A \cdot \sin(\omega t) \]

Equation 13. Oscillation function
Where $C_1$ is the offset of the function with respect to the X axis, $A$ is the amplitude of the oscillation, $t$ is the time (here the variable, as the oscillation will depend on the time) and $\omega$ is the pulse, and can be defined as $\omega = 2\pi f$. Here, $f$ is the oscillation frequency, while its inverse is $T = 1/f$ the oscillation period.

As it is not possible to evaluate more than one parameter at a time with the simulations, and due to the available time to do the study, some parameters will be already settled. Firstly, the constant will be eliminated as there won’t be any offset ($C_1 = 0$). This is because the constant in the actuator would represent a mass addition, and as it will be simulated a ZNMF and no mass will be added, this constant has no sense.

The orifice width ranges from 1mm to 5mm, depending on the case. Here, the orifice width will be settled to $h = 2mm$, because it is the value used in most of the experiments (Holman et al. 2005)(Zhang et al. 2008)

Moreover, the amplitude can be established following dimensionless parameters. As presented in Section 2.2.2.1, and according to Durrani (Durrani and Haider 2011), the momentum coefficient $C_\mu$ is the relation between the upstream velocity $U_\infty$ and the blowing velocity, which is the amplitude of the function $A$. Both are weighted by its characteristic length (the chord and the orifice width respectively).

As used by Durrani, and according to his conclusions, a proper value for $C_\mu$ would be about 0.008. Therefore, the amplitude of the velocity can be easily found following the momentum coefficient expression (see Section 2.2.2.1):

$$C_\mu = \frac{h \cdot A^2 \cdot \rho}{c \cdot U_\infty^2 \cdot \rho} \rightarrow A \approx 31 \text{ m/s}$$

The last step is to define the frequency or the period of the velocity oscillation. This is the only parameter which will vary with the simulation, in order to find the most suitable frequency to delay as much as possible the boundary layer separation.

However, it is important to establish an optimum range of frequencies. As described by Mittal (Mittal and Rampunggoon 2002), the actuator frequency should be approximate to the shedding frequency of the airfoil. On the other hand, Amitay (Amitay and Glezer 2006) concluded that the oscillation frequency should be an order of magnitude higher than the shedding frequency.

For that reason, both conclusions will be performed in order to find the most suitable one. First of all, it is necessary to obtain the shedding frequency of the airfoil. This frequency can be measured as the frequency of the oscillations of the lift coefficient during the transient simulation.

Therefore, the next step is to plot the lift coefficient versus the flow time at a specific AoA, for example $\alpha = 7^\circ$. The result is presented in Figure 49.
These oscillations are the result of the transient simulation, and they are caused by the change on the vorticity throughout the flow time. From this graphic we can find the shedding frequency, which is the inverse of a period. The calculations are presented in Equation 14.

$$f = \frac{1}{T} = \frac{1}{3.42 \cdot 10^{-1} - 3.2 \cdot 10^{-1}} \approx 45 \text{ Hz}$$

Equation 14. Shedding frequency of the airfoil.

The result is that the studied frequencies will oscillate between this frequency and an order of magnitude higher. From these conclusions, together with the several data available (Gul, Uzol, and Akmandor 2014) (Tang et al. 2007) the frequencies at which the ZNMF will be studied are presented in Chart 5.

<table>
<thead>
<tr>
<th>Evaluated frequencies (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_1 \approx 45 \text{ Hz}$</td>
</tr>
<tr>
<td>$f_2 \approx 250 \text{ Hz}$</td>
</tr>
<tr>
<td>$f_3 = 500 \text{ Hz}$</td>
</tr>
</tbody>
</table>

Chart 5. Frequencies that will be evaluated.

Having settled the frequencies, $\omega$ can be easily found as $\omega = 2\pi f$. Therefore, the boundary condition will be simulated as presented in Equation 15.

$$f(t) = 31 \cdot \sin(\omega t) \text{ [m/s]}$$

Equation 15. Function that will represent the ZNMF effect.

Where the evaluated $\omega$ are presented in Chart 6.

<table>
<thead>
<tr>
<th>Evaluated pulses $[\text{rad/s}]$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\omega_1 \approx 285 \text{ rad/s}$</td>
</tr>
<tr>
<td>$\omega_2 \approx 1500 \text{ rad/s}$</td>
</tr>
<tr>
<td>$\omega_3 \approx 3150 \text{ rad/s}$</td>
</tr>
</tbody>
</table>

Chart 6. Pulses that will define the boundary condition.
3.2.2. Simulation of the ZNMF effect

Now that the boundary condition has been defined, it is necessary to insert it to Fluent. As it is a custom function, Fluent does not integrate all the possible functions, so it is necessary to create what is called a User Defined Function (UDF).

For that purpose, it is necessary to write the function (in Fluent these functions are called macros) in the same language that Fluent uses. The basic language used by this software is C#, which is in turn based in the worldwide known C language.

The UDF is presented in Figure 50, followed by the explanation of the code.

```c
#include "udf.h"

DEFINE_PROFILE(unsteady_velocity, thread, position)
{
    face_t f;
    real t = CURRENT_TIME;

    begin_f_loop(f, thread)
    {
        F_PROFILE(f, thread, position) = 31*sin(1500.*t);
    }
    end_f_loop(f, thread)
}
```

Figure 50. Boundary condition UDF code

First of all, the UDF library must be included in order to create the macro. Moreover, as Fluent is identified as the main program, the macro has to be prepared for being called by the main program.

Here, the macro is called DEFINE_PROFILE, and the first input ("unsteady_velocity") is the name of the UDF. The second input ("thread") is the boundary zone in which the UDF will be applied. The last input is the position of this zone.

Then, the macro defines a variable whose type is face variable. This variable will save the faces where the UDF is defined. As the function depends on the real flow time, it is necessary to write it in the macro, which will use the current time of the simulation for the equation.

Moreover, the "begin_f_loop" function defines the function which is applied to the zone, and this zone is the input of such function. To end the region where the UDF is applied, the "end_f_loop" function must be called as well. The type of function is "F_PROFILE", which involves instructions which are beyond the scope of the study.
PART IV

ANALYSIS OF THE EFFECT OF THE ZNMF ACTUATOR ON THE NACA 4421 BEHAVIOUR
4.1. Evaluation of the results

The airfoil mesh has been evaluated, comparing numerical results with Abbott experimental data. Then, the UDF function has been settled, defining the parameters of the ZNMF as a boundary condition.

The next step is then running Fluent simulations that include both airfoil and actuator. As the mesh is the same than the presented in Section 3.1.2.1, it is not explained again, and therefore this section is focused on the results of the simulations.

There are several monitors and magnitudes which could characterize the numerical obtained data, but only some of them (which are considered the most important ones) are presented.

The simulations have been done for different $\alpha$ and ZNMF frequencies. As this study is focused on the stall behaviour of the airfoil, AoA close to the stall are studied, while the frequencies have been introduced in Section 3.2.2. However, the scenarios are summarised in Chart 7.

<table>
<thead>
<tr>
<th>SCENARIOS (Frequencies and Angles of Attack)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Angle of Attack</strong></td>
</tr>
<tr>
<td><strong>Frequency (Hz)</strong></td>
</tr>
</tbody>
</table>

Chart 7: Evaluated scenarios

Due to the similarity of the qualitative results, only one frequency will be presented in depth, and in on AoA. Moreover, different graphics comparing the effects in all the situations will be shown and analysed, giving the most important information of all scenarios.

First of all, the lift coefficient behaviours during a period will be presented, for each AoA at each ZNMF frequency. Then, the velocity profile is presented, explaining how it will be evaluated which frequency is the most optimum one.

Besides, qualitative information is presented through the velocities field, in order to give a global view of the velocity profile behaviour on the whole airfoil. This monitor will be accompanied by the streamlines configuration, in order to study in depth all the singularities which appear.

Once the singularities and behaviours are introduced, the boundary layer separation point is evaluated. This will give an overall idea of the performance of the ZNMF actuator at each frequency, while evaluated at different AoA. This results will lead to the choice of the best parameters for the ZNMF, which is one of the main goals of the project.

Finally, information about the vorticity is presented, something interesting when the separation occurs and when the recirculation bubble appears.
4.1.1. Lift coefficient behaviour

The first parameter which must be evaluated of the numerical simulations is the new lift coefficient behaviour. One graphic is presented for each combination of angle of attack and frequency, plotting the lift coefficient value versus the time (dimensionless with the period).

The amount of points used for plotting the oscillation varies depending on the frequency; having one point per millisecond, with the highest frequencies only few points are obtained.

It can be noticed that the value of the maximum lift coefficient is under the value of the lift coefficient achieved without the actuator; this is because the blowing velocity produced by the actuator affects the lift, reducing it. This is a general pattern which is followed on almost all the numerical simulations.

However, the most important result is the amplitude of the oscillation. One of the main goals of the ZNMF is to eliminate the oscillations as far as possible, so the amplitude is the factor which must be evaluated.

In Figure 51, the lift coefficient of the airfoil with the ZNMF is presented at the angle of attack $\alpha = 7^\circ$. The difference of the amplitude depending on the frequency can be easily noticed, being the lowest amplitude the frequency $f = 45 \text{ Hz}$. Moreover, the oscillation is quite similar in all cases, although at 250 Hz the shape suffers a distortion.

![Figure 51. Lift coefficient, $\alpha = 7^\circ$](image)
Parallel to the previous graphic, the lift coefficient behaviour at $\alpha = 12^\circ$ is shown in Figure 52. Again, the lowest amplitude is achieved at 45 Hz, although the lowest value is achieved before in the period than in the other two frequencies. The maximum value in all cases is $C_l \approx 1.32$, and both 250 Hz and 500 Hz curves are symmetric.

Figure 52. Lift coefficient, $\alpha = 12^\circ$

Figure 53. Lift coefficient, $\alpha = 14^\circ$
The last graphic (presented in Figure 53) plots the lift coefficient versus the flow time (dimensionless with the period) at an angle of attack $\alpha = 14^\circ$. As it can be observed, the oscillation of 250 Hz is far from the other two curves’ behaviour. Moreover, and despite having less maximum values, the oscillation associated to 45 Hz has less amplitude than the one which represents 500 Hz.

Finally, oscillation amplitude of all the frequencies is presented in Chart 8.

<table>
<thead>
<tr>
<th></th>
<th>7 °</th>
<th></th>
<th></th>
<th>12 °</th>
<th></th>
<th>14 °</th>
</tr>
</thead>
<tbody>
<tr>
<td>45 Hz</td>
<td>0.046</td>
<td>0.071</td>
<td>0.078</td>
<td>0.104</td>
<td>0.130</td>
<td>0.053</td>
</tr>
<tr>
<td>250 Hz</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>500 Hz</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Chart 8. Lift coefficient oscillation amplitude for each scenario

It can be noticed that the oscillation amplitude is lower when the frequency of the ZNMF is similar to the frequency of the oscillation of the same coefficient but without the airfoil (further explained in Section 3.2.1). This can be interpreted as an evidence that the frequency of 45 Hz is the most optimum one.

4.1.2. Evaluation of the velocity profile

Once the lift coefficient has been evaluated, it is time to present the velocity profile obtained through the numerical simulations. As presented in Section 2.1.1, the velocity profile depend on if the boundary layer is laminar or turbulent.

Moreover, the zone where the boundary layer is detached has been defined to be the region where the velocity direction is contrary to the upstream velocity. In order to give graphic data, the velocity profile in the region where the boundary layer is still not separated is shown in Figure 54.

Figure 54. Velocity profile on the airfoil (14°, 45 Hz, 55% of the chord)
It can be observed that the velocity change is very abrupt near the wall, while it becomes more progressive further from the airfoil surface. This fits perfectly with the description of the turbulent velocity profile given in the theoretical part.

Furthermore, it is possible to observe the velocity profile when the boundary layer separates (see Figure 55). It can be noticed that near the wall the velocity vector is almost a point, which indicates that the velocity is zero and therefore it is going to change its direction, producing the boundary layer separation.

![Figure 55. Velocity profile of the boundary layer separation point](image)

After this point, the velocity direction near the wall is opposite to the upstream velocity direction, due to the adverse pressure gradient and other factors explained in Section 2.1.2.

Figure 56 presents a velocity profile from the region where the boundary layer is already separated, being possible to observe the velocity behaviour and how it inverses its direction in the region near the wall.
4.1.3. Velocity magnitude contour

Although the velocity profile defines the region where the boundary layer is separated, the velocity magnitude contour consists of a solid image with colours defining the velocity magnitude at each node, and the value is extrapolated in order to fill the entire field. As it is a contour, the measures are more qualitative, and in order to give an overall view to the lector how the boundary layer has been separated.

A contour for each simulation can be obtained, and in Figure 57 two different velocity contours are presented.
Figure 57. Velocity magnitude contour (m/s) at $\alpha = 12^\circ$ (top) and $\alpha = 14^\circ$ (bottom)

It can be noticed that there are no significant changes apart from the boundary layer separation, but as this is qualitative information there has not any sense in presenting a velocity magnitude contour of each scenario. For that reason, only one scenario will be presented for the qualitative results (the one which is more characteristic).

Moreover, in the contours, the region where the boundary layer is separated can be clearly observed, printed as the blue zone. This fits with the expected result, as in the region where the boundary layer is separated the velocity direction is inverse to the upstream velocity direction, and therefore the velocity inside this region should be less than in the rest of the fluid field.

Moreover, the maximum velocity is achieved near the leading edge, where the maximum pressure is achieved as well. This also fits with the theory, being the zone where the pressure gradient is less adverse.
Besides, it must be mentioned a zone near the leading edge where the tangential velocity is zero (see Figure 58). One could think that his result is not reliable, as the zone is surrounded by an area where the velocity magnitude achieves its maximum value.

![Figure 58. ZNMF velocity contour (m/s)](image)

However, this result is coherent with the simulation, as that is the region where the actuator has been set. Due to the blowing velocity that the ZNMF adds to the cross flow, the vertical velocity blocks the cross flow velocity, creating this small region where the velocity tangential to the airfoil is zero.

### 4.1.4. Streamlines

As the velocity magnitude contour gives information about the parameter in a filled field (something which is introduced as qualitative), the streamlines are also presented in order to give more accurate data to be interpreted.

The first particle path line is presented in Figure 59. The stagnation point in the intrados can be noticed, and the region where the boundary layer is separated is represented as well.
However, the most important part is located at the trailing edge. As the image shows, there are some streamlines which are recirculated on the extrados before going downstream. This phenomenon occurs most likely because the airfoil does not have a sharp trailing edge.

The absence of sharpness in the trailing edge produces a local recirculation of some streamlines, which pass through the turbulent flow. A little bubble can be observed near the trailing edge (a region where there are no streamlines), a bubble produced probably for the same reason.

However, this phenomenon has no significant meaning in the simulations, because as it is local and does not affect the global circulation around the airfoil, in reality this effect would not have prejudicial effect more than creating a fluctuation on the drag.

In addition, only few of the streamlines are affected by this recirculation, and because the amount of streamlines which follows the expected behaviour is by far higher, this effect is not taken into consideration for evaluating the airfoil performance.

4.1.5. Boundary layer separation point

Although the previous results have helped to present the data and the simulation conditions, as the aim of this study is to delay the boundary layer separation, the parameter in which is necessary to be focused on is the point where the boundary layer separates.

As presented at the beginning of the section, nine different simulations have been performed (three frequencies at three angles of attack), apart from the ones used for evaluating the mesh reliability. The goal of this comparison is to give a definitive evidence that the Zero Net Mass flux delays the separation and therefore improves the airfoil performance.

For this reason, the graphics presented below are very important to evaluate if the performance is better using the actuator, and which frequency is the most optimum one.
The key for finding the point where the boundary layer separates is to study the shear stress of the extrados. The point with the minimum shear stress (theoretically, it should be zero) is the point where the velocity profile is going to invert its direction, being then the point where the boundary layer separates.

Once the shear stresses have been evaluated, the next step is to plot the positions (dimensionless with the chord) of the boundary layer separation point.

![Boundary layer separation point](image)

**Figure 60.** Boundary layer separation point

In **Figure 60** the point where the boundary layer separates is represented. In order to facilitate the comparison, the reference line of the separation point without the ZNMF has been included in the plot.

It is important to mention that the AoA of 7° is not very significant for the results, as the boundary layer is still adhered to the airfoil. As the AoA at which the stall begins is near 13°, the other angles are more important and are worth interpreting.

The results of the numerical simulations shows that, at $\alpha = 12°$, the separation point is near 75% of the chord. On the one hand, when the actuator is settled with the frequency of 45°, the most significant delaying is achieved, followed by the delaying produced by the actuator settled at 250 Hz. On the other hand, the frequency of 500 Hz is not optimum for delaying the boundary layer separation.

Furthermore, contrary to the results obtained for $\alpha = 12°$, the behaviour totally changes at $\alpha = 14°$. The separation point moves forward independently of the frequency at which the actuator is settled. Therefore, none of the frequencies is optimum for delaying the separation at 14°.
Through these results it is possible to conclude that the actuator has to be precisely set up, with the proper parameters conveniently studied and, if possible, having the chance of doing some experimental tests for investigating these parameters.

For this reason, the results achieved evidence that an improvement on the performance of the airfoil has been achieved at 12°, while at 14° none of the simulations have been successful for delaying the boundary layer.

In order to provide more accurate data, the position of the boundary layer separation point (dimensionless with the chord) is presented in Chart 9, Chart 10 and Chart 11.

<table>
<thead>
<tr>
<th>7°</th>
<th>No ZNMF</th>
<th>45 Hz</th>
<th>250 Hz</th>
<th>500 Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Position (x/c)</td>
<td>0.816</td>
<td>0.817</td>
<td>0.810</td>
<td>0.797</td>
</tr>
<tr>
<td>Shear stress (Pa)</td>
<td>0.0248</td>
<td>0.0149</td>
<td>0.0129</td>
<td>0.0188</td>
</tr>
</tbody>
</table>

Chart 9. Boundary layer separation point position and correspondent wall shear stress, α = 7°

<table>
<thead>
<tr>
<th>12°</th>
<th>No ZNMF</th>
<th>45 Hz</th>
<th>250 Hz</th>
<th>500 Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Position (x/c)</td>
<td>0.763</td>
<td>0.786</td>
<td>0.771</td>
<td>0.693</td>
</tr>
<tr>
<td>Shear stress (Pa)</td>
<td>0.0181</td>
<td>0.0251</td>
<td>0.0216</td>
<td>0.0259</td>
</tr>
</tbody>
</table>

Chart 10. Boundary layer separation point position and correspondent wall shear stress, α = 12°

<table>
<thead>
<tr>
<th>14°</th>
<th>No ZNMF</th>
<th>45 Hz</th>
<th>250 Hz</th>
<th>500 Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Position (x/c)</td>
<td>0.669</td>
<td>0.485</td>
<td>0.486</td>
<td>0.501</td>
</tr>
<tr>
<td>Shear stress (Pa)</td>
<td>0.0206</td>
<td>0.0255</td>
<td>0.0288</td>
<td>0.0250</td>
</tr>
</tbody>
</table>

Chart 11. Boundary layer separation point position and correspondent wall shear stress, α = 14°

This last results give the definitive evidence that the frequency of 45 Hz is the most optimum one, because at α = 12° that is the frequency which delays the most the boundary layer separation point. In addition, the shear stress does not achieve a null value, but the shear stress is irrelevant enough to neglect it.
4.1.6. Vorticity

Now that the boundary layer separation has been evaluated, it is possible to present the vorticity which appears on the region where the velocity profile is adverse to the upstream velocity.

According to Sicot (Sicot et al. 2006), the vorticity can be defined as a measure of the rotation of a fluid element as it moves in the flow field. The vorticity used to be located in the region where the boundary layer is separated, because the rotation which brings about vorticity is created by eddies that are inside the boundary layer.

However, as in the numerical simulations a ZNMF actuator has been settled, there is a vorticity interaction between the actuator and the flow field. As the boundary condition was a blowing velocity (which follows a sinusoidal function), the velocity perpendicular to the surface mixes with the cross flow creating hairpin vortices, as presented in Section 2.2.2.1.

It can be noticed that very close to the actuator position the vorticity is very high. This is caused by the rotation created by the vortices which are generated in the orifice, more specifically in the orifice limits. The vortices are created due to the interaction between the blowing stroke and the suction stroke (see Section 2.2.2.1).

The vorticity diminishes while you move away from the airfoil’s surface, but it is dragged by the cross flow, creating this vorticity wake. Figure 61 shows a qualitative idea of the length of this wake.

Finally, the general vorticity can be observed in Figure 61. All the vorticity is concentrated in the boundary layer and its wake, being created by eddies. The most important part of the wake is presented in the picture. The scale is settled accurate enough to allow the identification of the regions with more vorticity.

The highest vorticity values are achieved both near the actuator and in the beginning of the wake. However, in order to understand the importance of the vorticity and its effect on the airfoil’s wake, Figure 61 represents the length of the vorticity wake downstream, although the scale has been modified until the length can be properly evaluated.

![Figure 61. Vorticity around the airfoil (1/s). 12º, 45 Hz](image)

68
4.2. Comparison with experimental data

Once all the results have been evaluated, it would be useful to compare them with experimental results of similar analysis. As it is difficult to find an experiment performed in the same conditions as the simulations exposed here, the results will be compared with other studies which use a ZNMF over different airfoils.

Although the airfoil of the experiments is different, its changes in performance will change in the same way as the NACA 4421 presented in this study. For that reason, the comparison will show if the behaviour of the simulations is the same as expected.

4.2.1. Lift coefficient

The values of lift coefficient are different depending on if the ZNMF is set or not. The aim of this section is to provide evidences that the behaviour of the parameter is the same as in other performed studies.

The chosen data comes from the simulations made by Durrani (Durrani and Haider 2011), who developed a study of a ZNMF over a NASA GA(W)-2 airfoil using ANSYS Fluent, with a Spalart-Allmaras (1 equations RANS solver).

Although the article evaluates the ZNMF performance at different chordwise locations, they are presented only the results of the baseline and when the ZNMF is set at 15% of the chord, in order to do the comparison. They are all presented in Chart 12.

<table>
<thead>
<tr>
<th>$\alpha$ [°]</th>
<th>Baseline</th>
<th>ZNMF at 0.15c</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>1.8246</td>
<td>1.8039</td>
</tr>
<tr>
<td>18</td>
<td>1.7994</td>
<td>1.7749</td>
</tr>
<tr>
<td>20</td>
<td>1.5338</td>
<td>1.5543</td>
</tr>
<tr>
<td>22</td>
<td>1.1097</td>
<td>1.1421</td>
</tr>
</tbody>
</table>

Chart 12. Lift coefficient obtained by Durrani (Durrani and Haider 2011)

It is firstly necessary to mention that the angle of the maximum lift coefficient for this airfoil is approximately 18°, according to the article. It can be noticed that when the ZNMF is set, the lift coefficient increases once the airfoil stalls, while before that the value of the parameter is lower than the one of the baseline. In order to compare, the lift coefficients of the NACA 4421 obtained in the study are presented in Chart 13.

<table>
<thead>
<tr>
<th>$\alpha$ [°]</th>
<th>Baseline</th>
<th>ZNMF at 0.17c</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>1.0036</td>
<td>1.0065</td>
</tr>
<tr>
<td>12</td>
<td>1.3202</td>
<td>1.2789</td>
</tr>
<tr>
<td>14</td>
<td>1.3429</td>
<td>1.2896</td>
</tr>
</tbody>
</table>

Chart 13. Lift coefficient obtained through ANSYS fluent simulations
Although the values cannot be compared because the airfoils of both studies are different, the behaviour of the parameter should be similar, as the ZNMF is implemented in both cases analogously.

The comparison shows that, similarly to Durrani’s results, the value of the lift coefficient before the stall decreases (at $\alpha = 12^\circ$). However, at $\alpha = 7^\circ$ the lift coefficient is a bit higher, something which proves that the simulations need a higher level of accuracy. But this is fully evidenced when the lift coefficient at $\alpha = 14^\circ$ is compared; the parameter should increase due to the ZNMF, while the CFD simulations show that the value decreases.

Therefore, it can be concluded that the level of accuracy when performing the simulations (and calibrating the ZNMF) must be increased, in order to get reliable data which would be similar to other studies’ results.

### 4.2.2. Streamlines

Another result which can be compared with similar studies is the streamlines plot. As presented in Section 4.1.4, the plot shows a particular behaviour near the trailing edge, where some streamlines that come from the intrados are partially redirected to the extrados.

According to other studies, this behaviour is expected. The experiments presented by Zhang (Zhang Panfeng 2007) set a ZNMF actuator over a NACA 0015 and studied this recirculated region. The results are shown in Figure 62.

![Figure 62. Streamlines of a NACA 0015 with ZNMF actuator (Zhang Panfeng 2007)](image)

Comparing Zhang streamlines results with the ones obtained in the present study (introduced in Section 4.1.4), it can be affirmed that the recirculation region is reliable and was expected to appear when performing the simulations.
4.2.3. Vorticity

The vorticity is another result which can be compared with other studies. However, as the airfoil of the present study is different from the other ones, only the vorticity created by the ZNMF can be used for the comparison.

The study from where the data has been extracted is, again, the one performed by Durrani (Durrani and Haider 2011). As the ZNMF implemented in the article is similar to the used in the present study, the vorticity created by the actuator will be similar, although the overall vorticity will be different.

The contours of the vorticity magnitude obtained by Durrani are presented in Figure 63.

![Figure 63. Vorticity magnitude (1/s), with the ZNMF set at 0.15c (Durrani and Haider 2011)](image)

Comparing these results with the ones obtained through CFD simulations (which are presented in Section 4.1.6), it can be noticed that the contribution of the ZNMF to the vorticity is significant. Moreover, the vorticity near the orifice is so high that it is out of scale in both figures.

Therefore, and as evidenced by these results, the contours of vorticity magnitude obtained using ANSYS Fluent are the expected ones, being quite similar to the results obtained by Durrani.

Finally, it is important to underline that the vorticity wake in both studies is different, basically because the performance of both airfoils (NASA GA (W)-2 in the article and NACA 4421 in this study) is significantly different.
PART V

CONCLUSIONS
5.1. Conclusions and future work

5.1.1. Conclusions

According to the aim of the study, the performance of the airfoil using a Zero Net Mass Flux has been evaluated. Using ANSYS ICEM CFD the mesh has been developed and evaluated, while the numerical simulations have been performed using ANSYS Fluent.

As presented in the scope, the mesh quality has been guaranteed through comparing simulations results with experimental data. The results have been successful, giving accurate results and therefore reassuring the reliability of the mesh.

Parallel to this, the ZNMF theoretical fundamentals have been presented and the parameters which define the actuator behaviours have been studied, based on bibliography and considering the scenario which is being studied. The chosen frequencies have been selected through interpreting the lift coefficient of one’s own simulations, and papers from several researchers support the obtained values.

However, once the simulations have been performed it has been demonstrated that the parameters of the ZNMF must be carefully chosen, otherwise the airfoil performance when using the actuator does not improve as expected. This can be concluded because only one of the frequencies and at one specific angle of attack has shown better performance, while other combinations and parameters have reduced significantly the airfoil baseline performance.

This brings about the necessity of a refined mesh, with an accurate value of the y+. However, the value achieved in this study is about 5, and is good enough in order to get approximate data which can be used for studying the performance.

Moreover, there are lots of results which can be evaluated, but the most significant ones are presented in the report. The lift coefficient has an expected behaviour; the actuator eliminates the amplitude of the oscillations. Moreover, the boundary layer evolution agrees to the theory; it has the behaviour of the thick airfoils (as NACA 4421 is a thick airfoil), so the simulations are successful in this aspect

The boundary layer separation point is the most important part of the results. This point moves backward in the extrados only in certain cases, something which demonstrates the level of accuracy needed to settle the ZNMF.

Both the lift coefficient behaviour and the boundary layer separation point demonstrate therefore that the most optimum frequency is 45 Hz. This fits with the majority of the bibliography, which conclude that the same order of magnitude that the shedding frequency of the airfoil is the most suitable one. Moreover, the best improvements have been achieved at an AoA of 12°, just before the stall happens.
Finally, the vorticity has been presented. Agreeing to the other results, the expected vorticity behaviours only appears in the scenario that presents the most delayed boundary layer separation point. As the behaviour of the other scenarios differs significantly from the expected results, the most reliable results will be the ones taken when the variables are settled at $\alpha = 12^\circ$ and $f = 45 \, Hz$.

All the results demonstrated, in conclusion, that the Zero Net Mass Flux improves the airfoil performance, so the initial hypothesis of the study, which was that the airfoil had better performance in presence of the actuator, is finally confirmed. The scope has been accomplished, and all the applications explained during the study could take profit of a device that, although still in development, it is called to be the future of the high lift devices in the aerospace sector.

5.1.2. Future lines of investigation

Although high lift devices have been studied since long time ago, there is still a lot of research to do. The aim of this section is to set and plan the basis of the possible future work which could be done around the numerical study which has been performed.

First of all, the possibility of improving the mesh is an option which must be taken into consideration. The mesh quality has been evaluated and numerical results have been obtained to guarantee the reliability of the mesh.

However, and according to Salim (Salim and Cheah 2009), the $y+$ should be under the value of 1. As the mesh presented in our study reaches 4, it is necessary to improve the mesh quality (particularly on the boundary layer region) to get more accurate data in the numerical simulations.

Moreover, the solver is an aspect which should be reviewed. The presented solver ($k - \omega$ SST) is reliable enough to evaluate the parameters and the airfoil general performance, however if the results are desired to be more specific to study the airfoil behaviour in detail, a more exact solver must be used.

The available solvers which could be used are basically the ones based in LES (Large Eddy Simulation) (Volkov 2006). Moreover, it also exists the DNS (Direct Numerical Simulation) (Sandberg 2015) which directly solves the Navier-Stokes equations, but the computational power needed to run those simulations is not available for everyone nowadays.

To continue with, the actuator must be studied in depth. As the results of the study claim, the choosing of the parameters of the actuator affect significantly on its performance on the airfoil (Zhang et al. 2008). Likewise, in this study only frequencies have been evaluated, while the position of the actuator and its velocity function amplitude were settled. A lot of work can be done about this; every single parameter of the actuator must be studied in depth in order to find the most suitable values for each one.

Moreover, experimental analysis could be done in order to have own real data to compare the actuator behaviour in the reality with the numerical simulations.
In addition, it must be underlined that 2D solutions are not so accurate, because in the reality the vortices are generated and then they continue tightening themselves. This effect (which is produced spanwise) cannot be simulated in 2D, so the terms related to the dissipation are different and consequently the final results change.

Apart from all the improvements which can be done regarding this study, it could be possible to study other applications for these kind of actuators. According to Holman (Holman et al. 2005), applications such as heat transfer or mixing enhancement have to be investigated in the field of the ZNMF actuators.

5.1.2.1. Planning of the future work

Once all the improvements have been settled, it is possible to make an approximate calendar of the tasks which should be done, giving a brief description of each one. Moreover, interdependencies and an approximate level of effort is presented, as well as the Gantt Diagram.

Brief task description

1. **Mesh improvement**: Refinement of the mesh until a good value of the $y+$ is achieved.

2. **Solver choice**: Selecting the most suitable solver according to the level of accuracy needed for the results and depending on the available computational power.

3. **Study of the ZNMF parameters**: A study of the amplitude of the velocity function of the actuator (blowing velocity) or the position of the actuator, among other possible parameters.

4. **Definition of the operating conditions**: Definition of the simulation background such as the Mach number (depending on if it will be studied in subsonic, transonic or supersonic regime) or the Reynolds number (depending on the fluid behaviour it is going to be evaluated).

5. **Numerical simulations**: Simulations of the studied case, defining different scenarios to evaluate actuator’s parameter(s) as well as to study in depth if the actuator fit for the application for which it has been designed.

6. **Performance of experimental analysis**: Experimental simulations which are going to be performed in order to have own experimental data to compare the simulations.

7. **Comparison of the results with experimental data**: Search for experimental data of similar studies in order to compare the numerical simulations with real experiences, or use own experimental results.
**Interdependencies among tasks and level of effort**

The main tasks have been defined. The next step is to settle the interdependencies among the described tasks, as well as the level of effort of each task; they are all presented in **Chart 14**.

<table>
<thead>
<tr>
<th>Task code</th>
<th>Preceding task(s)</th>
<th>Level of effort (h)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>---</td>
<td>40</td>
</tr>
<tr>
<td>2.</td>
<td>1.</td>
<td>30</td>
</tr>
<tr>
<td>3.</td>
<td>---</td>
<td>80</td>
</tr>
<tr>
<td>4.</td>
<td>3.</td>
<td>30</td>
</tr>
<tr>
<td>5.</td>
<td>4.</td>
<td>150</td>
</tr>
<tr>
<td>6.</td>
<td>3.</td>
<td>80</td>
</tr>
<tr>
<td>7.</td>
<td>6.</td>
<td>50</td>
</tr>
</tbody>
</table>

*Chart 14. Tasks interdependencies and level of effort*

TOTAL: Approximately 460 hours will be spent in doing the suggested study
Gantt Diagram

<table>
<thead>
<tr>
<th>Nombre de tarea</th>
<th>Duración</th>
<th>Comienzo</th>
<th>Fin</th>
<th>Precio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Future Work</td>
<td>50 días</td>
<td>Lun 01/06/15</td>
<td>Vie 07/08/15</td>
<td></td>
</tr>
<tr>
<td>Mesh improvement</td>
<td>15 días</td>
<td>Lun 01/06/15</td>
<td>Vie 19/06/15</td>
<td></td>
</tr>
<tr>
<td>Solver choice</td>
<td>5 días</td>
<td>Lun 22/06/15</td>
<td>Vie 26/06/15</td>
<td>2</td>
</tr>
<tr>
<td>Study of the ZNMF parameters</td>
<td>20 días</td>
<td>Lun 01/06/15</td>
<td>Vie 26/06/15</td>
<td></td>
</tr>
<tr>
<td>Definition of the operating conditions</td>
<td>5 días</td>
<td>Lun 29/06/15</td>
<td>Vie 03/07/15</td>
<td>4</td>
</tr>
<tr>
<td>Numerical simulations</td>
<td>25 días</td>
<td>Lun 06/07/15</td>
<td>Vie 07/08/15</td>
<td>5</td>
</tr>
<tr>
<td>Performance of experimental analysis</td>
<td>15 días</td>
<td>Lun 29/06/15</td>
<td>Vie 17/07/15</td>
<td>4</td>
</tr>
<tr>
<td>Comparison of the results with experimental data</td>
<td>5 días</td>
<td>Lun 20/07/15</td>
<td>Vie 24/07/15</td>
<td>7</td>
</tr>
</tbody>
</table>

Figure 64. Gantt Diagram of the future work
5.2. Sustainability study

Nowadays, sustainability is becoming every time an aspect which is taken into account when evaluating if performing or not a project or study. The aim of this section is to study the environmental impact of the improvements which have been investigated.

Therefore, it is important to present the ways the object of the study (here the ZNMF actuator) can help reducing the environmental impact of the applications where it will be used.

As explained during the study, the actuator adds momentum to the boundary layer in order to delay its separation. When the stall occurs, the drag of the aircraft increases significantly, and in order to solve the problem the pilot has to increase the thrust of the aircraft, among other measures.

This thrust increase brings about the necessity of fuel, so the fuel consumption increases in order to compensate the drag. As the ZNMF actuator delays the stall phenomenon, and is even capable of let the aircraft avoid it, this thrust increment perhaps is not necessary, so it could help to save fuel, and therefore to reduce the CO₂ emissions.

Moreover, and according to studies of the actuator for drag reduction, Agarwal (Agarwal 2012), investigated the amount of fuel that these actuator could save in a vehicle consumption. Firstly, he studied ground vehicles and concluded that the drag could be reduced in a 15%, with a consequent saving of the 7% of the fuel.

Assuming the ground vehicle uses diesel, and with a fuel tank of 50 L., the 7% is approximately 3.5 L. Therefore, and knowing that a diesel car emits 2.5 kg CO₂/L, the device could avoid the emission of 8.75 kg CO₂.

Moreover, and assuming (according to Agarwal) that in an Airbus A320 the fuel saving is the 3% of the total consumption (which is about 2500 kg/h), in a hourly flight 75 kerosene litres would signify avoiding the emission of 240 kg CO₂ to the atmosphere, knowing that the aircraft emits 3.23 kgCO₂/kgkerosene.

If one figures out the amount of flights and hours that the airplanes spent flying, the amount of CO₂ emissions which could be avoided are very significant, proving therefore that the use of the ZNMF would have a positive impact on the environment and would contribute to the sustainability of the industry.
5.3. Bibliography


Hazen, David C. 1948. “Boundary Layer Control.” *Film.*


80


