UNIVERSITAT POLITÈCNICA DE CATALUNYA
BARCELONATECH
Escola d'Enginyeria de Telecomunicació
i Aeroespacial de Castelldefels

TREBALL DE FI DE GRAU

TFG TITLE: Aerodynamic performance optimisation of an airfoil with Synthetic Jets Actuation

DEGREE: BSc in Aerospace Systems Engineering

AUTHOR: Jordi Larruy Gilabert

ADVISOR: Fernando Pablo Mellibovsky Elstein

DATE: 21 de juliol de 2023

Títol: Optimització del rendiment aerodinàmic d'un perfil alar mitjançant l'Actuació de Jets Sintètics

Autor: Jordi Larruy Gilabert

Director: Fernando Pablo Mellibovsky Elstein

Data: 21 de juliol de 2023

Resum

Aquest TFG presenta un anàlisi de l'aerodinàmica del perfil d'ala Eppler149. L'Eppler149 és un perfil simètric dissenyat específicament per a nombres de Reynolds baixos, el que el fa adequat per a avions petits, drons i vehicles no tripulats. L'objectiu d'aquesta projecte és realitzar una anàlisi aerodinàmic del perfil Eppler149, en vol a nombre de Reynolds 60000, amb un angle d'atac de 16 graus, el qual està considerat post-stall, i podria ser especialment rellevant per a les operacions d'enlairament.
Per dur a terme l'anàlisi, s'ha creat un domini tipus C, amb malla hibrida, estructurada i desestructurada per optimitzar al màxim el rendiment computacional, utilitzant el programari gmsh. Posteriorment s'ha importat a OpenFOAM per fer les simulacions. S'han examinat tres models de turbulència, concretament els models k e, k wSST i Spalart-Allmaras, per determinar el model més precís i adequat per al nostre cas. A través d'una avaluació exhaustiva, el model de turbulència Spalart-Allmaras ha demostrat el millor rendiment i s'ha seleccionat per a una anàlisi més detallada.
Per millorar el comportament aerodinàmic del perfil alar, s'ha dissenyat un jet a succió, al extrados del perfil, a intensitats de 0%, 50%, 100% i 150% de la velocitat de Reeynolds de vol. Aquesta tècnica de control actiu del flux té com a objectiu millorar el rendiment i l'eficiència del perfil. A més, a més, es va dur a terme un estudi comparatiu variant la forma del leading edge del perfil per analitzar el seu impacte en el rendiment aerodinàmic. Els resultats obtinguts en aquest projecte, proporcionen una visió valuosa de les característiques aerodinàmiques del perfil Eppler149 a nombres de Reynolds baixos i angles d'atac elevats. A més a més, posen en manifest l'efectivitat del model de turbulència Spalart-Allmaras per predir amb precisió el comportament del flux al voltant del perfil d'ala. La implementació del control actiu del flux mitjançant aspiració ha demostrat millores en el rendiment aerodinàmic del perfil, mentre que l'estudi de les variacions del leading edge del perfil té una influència significativa en el rendiment aerodinàmic.

Title : Aerodynamic performance optimisation of an airfoil with Synthetic Jets Actuation
Author: Jordi Larruy Gilabert

Advisor: Fernando Pablo Mellibovsky Elstein

Date: July 21, 2023

Overview

This Bsc thesis presents an analysis of the aerodynamics of the Eppler149 airfoil profile. The Eppler149 is a symmetrical airfoil specifically designed for low Reynolds numbers, making it suitable for small aircraft, drones, and unmanned vehicles. The aim of this thesis is to perform an aerodynamic analysis of the Eppler149 airfoil, at flight conditions of Reynolds 60000, at an angle of attack of 16 degrees, which is considered as post-stall, and it could be particularly relevant for takeoff operations.
To conduct the analysis, a structured C-type domain has been designed, using an hybrid mesh structured and unstructured created on gmsh software, and subsequently imported into OpenFOAM, for simulations. Three turbulence models, $k \quad e$, $k \quad w SST$, and Spalart-Allmaras models, have been examined to determine the most accurate and suitable model for our case. Through a comprehensive evaluation, the Spalart-Allmaras turbulence model has demonstrated the best performance and was selected for further analysis.
To improve the aerodynamic behavior of the airfoil, a suction jet has been designed on the extrados surface of the airfoil, at intensities of 0%, 50%, 100% and 150% referenced to Reynolds in flight conditions. This active flow control technique aimed to enhance the performance and efficiency of the airfoil. Furthermore, a comparative study has been conducted by varying the shape of the leading edge to analyze its impact on aerodynamic performance.
The findings of this research provide valuable insights into the aerodynamic characteristics of the Eppler149 airfoil at low Reynolds numbers and high angles of attack. The results highlight the effectiveness of the Spalart-Allmaras turbulence model in accurately predicting the flow behavior around the airfoil. The implementation of active flow control through suction demonstrated improvements in the aerodynamic performance of the airfoil, while the investigation of leading-edge variations shed light on their influence on aerodynamic performance.

to my family

CONTENTS

Introduction 1

CHAPTER 1. Fundamentals of Aerodynamics 3

1.1. Principles and ow equations 3
1.1.1. Continuity equation 3
1.1.2. Momentum equation 3

1.2. Turbulent ows 4
1.2.1. Turbulent ows in computational uid dynamics 5
1.2.2. Reynolds Averaged Navier Stokes Equations 5

1.3. Turbulence models 6
1.3.1. k-e model 6
1.3.2. Spalart-Allmaras model 7
1.3.3. k-w model 8

CHAPTER 2. Boundary Layer 11

2.1. Boundary layer theory 11

2.2. Boundary layer detachment 13
2.2.1. Flow transition 13

2.3. BL control methods 14
2.3.1. Passive ow control 15
2.3.2. Active ow control 15

CHAPTER 3. Numerical methods CFD 19

3.1. Methodology 19

3.2. Domain setup 19

3.3. Meshing 21
3.3.1. Mesh considerations 21
3.3.2. Mesh development 26

CHAPTER 4. CFD tools 29

4.1. Flow Solvers 29

4.2. Boundary Conditions 30
4.2.1. k e and k wSST 31
4.2.2. Spalart Allmaras 32

4.3. Post Processing tools 32

CHAPTER 5. Results 33

5.1. Convergence study 33

5.2. Baseline case 35

5.3. Active ow control 38

5.4. In uence of a non-perfect leading edge 40
5.4.1. Active ow control for non-perfect LE 41

5.5. Aerodynamic coef cients comparison 43

Conclusions 45

Bibliography 47

LIST OF FIGURES

1.1 k-w SST model 9

2.1 Boundary layer sketch. Not to scale 12
2.2 Boundary layer development. Not to scale 13
2.3 Laminar and turbulent boundary layers velocity pro le. Not to scale. 14
2.4 Idealised operation of a synthetic jet actuator 17

3.1 Sketch of uid domain 20
3.2 Geometry of Eppler 169 airfoil 20
3.3 Structured vs unstructured mesh 22
3.4 a) Courant number, b) high AR cells, c) low AR cells 23
3.5 Examples of skewness 23
3.6 Boundary layer parameters in CFD for viscous sublayer case 24
3.7 Final airfoil geometry with in ation layers zone 25
3.8 First mesh approach; completely structured outside region 27
3.9 Second mesh approach; structured wake and unstructured outside region . . . 27
3.10 Final mesh approach: entire domain 28
3.11 Airfoil details in nal mesh approach 28

5.1 Zoom at deattachment point of Cp comparison for the different studied meshes 34
5.2 Convergence of Cl respect to iterations number for k e, k wSST and SA for post stall AoA = 16° 35
5.3 Convergence of Cd respect to iterations number for k e, k wSST and SA for post stall AoA = 16° 36
5.4 Velocity pro le streamlines and turbulent viscosity contour at AoA 16° 36
5.5 Evolution of extrados velocity pro le at different x/C at AoA 16° 37
5.6 a) Pressure coef cient Cp(x/C) and b) Friction coef cient Cf(x/c) at AoA 16° . . 37
5.7 Cp a) and Cf b) at different suction cases at AoA 16° 38
5.8 a) Re=0, b) Re=30000, c) Re=60000, d) Re=90000 velocity streamlines at AoA 16° 39
5.9 Perfect a) and irregular b) LE comparison 40
5.10 a)Pressure coef cient Cp(x/C) and b) Friction coef cient Cf(x/C) for irregular LE at AoA 16° 41
5.11 Evolution of extrados velocity pro le at different x/C for irregular LE at AoA 16° . 41
5.12 Cp a) and Cf b) at different suction cases for irregular LE at AoA 16° 42
5.13 a) Re=0, b) Re=30000, c) Re=60000, d) Re=90000 velocity streamlines for irregular LE at AoA 16° 43
5.14 a) Re=0, b) Re=30000, c) Re=60000, d) Re=90000 Cf comparison for rounded and irregular LE 44

LIST OF TABLES

3.1 Boundary layer parameters 26

4.1 k e boundary conditions 31
4.2 k wsst boundary conditions 32
4.3 SA boundary conditions 32

5.1 Mesh convergence 33
5.2 Time Step convergence 35
5.3 Main coef cients for perfect LE at different suction cases 38
5.4 Main coef cients for non-perfect LE at different suction cases 42

INTRODUCTION

Aerodynamics plays a crucial role in the design of ef cient and sustainable transportation systems. Recently, the analysis of aerodynamics at low Reynolds numbers has gained relevance due to its numerous applications arising from the increased use of drones and smaller aircraft.

This work focuses on exploring how active ow control techniques can enhance aerodynamic performance, by applying different AFC methods, promoting a more optimal future without the need for moving surfaces in aerodynamic pro les.

The motivation for this research comes, in part, from the need to nd innovative solutions to reduce energy consumption and minimize the environmental impact of transportation systems. Currently, the transportation sector is responsible for a signi cant portion of the greenhouse gas emissions contributing to climate change. In this regard, improving aerodynamic performance at low Reynolds numbers emerges as a promising strategy to reduce drag, resulting in increased aerodynamic and energy ef ciency, thereby reducing CO2 emissions [1].

Traditionally, enhancing aerodynamic performance in off-design ight conditions has been achieved through the use of movable surfaces on wings. However, with the evolution of technology and the imperative to produce fewer CO2 emissions, the potential concept of active ow control has emerged [2].

State of the art

In the eld of aerodynamics, the study of the aerodynamic performance of Eppler149 airfoil pro les at Reynolds numbers of 60,000 for post-stall angles of attack has received extensive attention and analysis in the scienti c literature [2, 4, 46, 47]. The aerodynamic performance at this low regime of Reynolds numbers is particularly important for improving takeoff operations for drones, and smaller aircrafts among other applications.

Previous research has used various turbulence models to study and understand the ow behavior around the Eppler149 airfoil. The models employed include the $k \quad e$ turbulence model, the $k \quad w$SST model, and the Spalart-Allmaras model. These models were chosen for their ability to provide accurate and reliable results in the analysis of turbulent ow.

The baseline aerodynamic analysis allowed for the evaluation of the characteristics and performance of the Eppler149 airfoil at Reynolds numbers of 60,000 for post-stall angles of attack. The objective was to determine the ow behavior under these conditions, as well as to identify ow separation and associated phenomena.

In addition to conventional aerodynamic analysis, the implementation of active ow control techniques has been investigated to further enhance the performance and aerodynamic ef ciency of the Eppler149 airfoil. Speci cally, blowing and suction techniques have been employed to control the ow and optimize aerodynamic characteristics.

Among the active ow control techniques, particular attention has been focused on the suction technique. Experimental and numerical studies have been conducted to investigate the effects of suction on the ow around the Eppler149 airfoil. These studies covered

a range of suction velocities, from zero velocity to 90,000 Reynolds velocities, with the aim of maximizing the performance and aerodynamic ef ciency of the airfoil.

CHAPTER 1. FUNDAMENTALS OF AERODYNAMICS

This initial chapter is intended to introduce the basics of aerodynamics and review the most general and essential concepts.

1.1. Principles and ow equations

Governing equations are mathematical statements of the physical principles that we use to predict the evolution of the ow. Every phenomenon that involves a uid can be described through a mathematical model described by the Navier-Stokes equations, which are derived from fundamental physical laws.

- Mass conservation
- Momentum conservation
- Energy conservation

Any ow will be de ned depending on its characteristics; its velocity eld or Mach number, its friction and its Reynolds number.

1.1.1. Continuity equation

Starting from the continuity equation or mass conservation equation, where physical laws dictate that mass variation in a control volume follows from the uxes at its boundaries.

$$m_{in} \quad m_{out} = Dm_{cv} \tag{1.1}$$

- Mass equation in differential form:

$$\frac{\P r}{\P t} + \tilde{N}(r\,U) = 0 \tag{1.2}$$

1.1.2. Momentum equation

Due to approximations made depending on the scope of our problem, the Navier-Stokes equation will differ from each other. Nevertheless, our ow will be considered viscous. Real ows in movement will always experience effects due to frictional and viscous forces. If the ow was considered inviscid, the Navier-Stokes equations would be simpli ed as the shear stress tensor equals zero: $t_{i;j;k} = 0$.

- Momentum equation for incompressible, unsteady, viscous ow:

$$\rho \frac{\partial U}{\partial t} + \rho (U \cdot \nabla) U = -\nabla p + \nabla \tau^{T} \tag{1.3}$$

$$\rho \frac{\partial U}{\partial t} + \rho (U \cdot \nabla) U = -\nabla p + \mu \nabla^{2} U \tag{1.4}$$

$$\rho \left[\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \right] = -\frac{\partial p}{\partial x} + \mu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right] \tag{1.5}$$

$$\rho \left[\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} \right] = -\frac{\partial p}{\partial y} + \mu \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right] \tag{1.6}$$

$$\rho \left[\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} \right] = -\frac{\partial p}{\partial z} + \mu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right] \tag{1.7}$$

The momentum equation is a vector equation. Equation 1.4 can be divided into 3 different equations, one for each spatial component. For the incompressible case, where ρ is constant, the flow field variables are p and V. The continuity and momentum equations obtained earlier are two equations in terms of the two unknowns p and V. Hence, for a study of incompressible flow, the continuity and momentum equations are sufficient to solve the flow. Energy equation will not be necessary in this case. However for compressible flow, as ρ is introduced as a variable, we would need the energy equation to solve the system.

1.2. Turbulent flows

Turbulent flows, are those types of flows in which the fluid behaves in a chaotic, unpredictable way, Eddies, swirls, and flow instabilities occur. These flows, are characterized by sudden fluctuations in pressure and flow velocity and direction.

When fluids are regular and behave in an order and in clear manner, they are called laminar. Most flows of interest in aerodynamics, are turbulent except occasionally at the leading edge of moving solids and extremely close to solid surfaces [8, 9].

When convective forces outweight viscous forces, given by the increase of flow's velocity (U) or the characteristic length (L) of the solid model, or a decrease of fluid viscosity, laminar flow transitions into turbulent. The ratio between convective and viscous forces is known as Reynolds number. Reynolds number is used to categorize different types of flows. The greater the Reynolds number, the more turbulent the flow [8, 9].

$$Re = \frac{\rho U L}{\mu} \tag{1.8}$$

Turbulence can be seen in every day phenomena, from the smoke of a cigarette to the contrails of an airplane or the water flowing from a faucet. Furthermore, turbulence is caused by excess kinetic energy in some parts of the flow. For this reason, turbulence is mainly observed in low viscosity fluids as these kinetic energy is greater than the damping effect of the fluid's viscosity.

Turbulent flows creates unsteady vortices that interact with each other. Due to these vortices and unlike laminar flows, turbulent flows past bluff bodies are characterized by high friction drag and low form drag [8, 9, 10].

1.2.1. Turbulent ows in computational uid dynamics

Turbulent aws involves tiny spatial scales, making the Navier-Stokes equations (NSE) inappropiate to characterize the ow.

On one hand, NSE are able to solve laminar ows. However, for turbulent ows, due to the enormous computational cost, some mathematical models have been developed to solve the aws numerically [11].

- Direct Navier Stokes (DNS)
  This method solves directly hte NSE, but requires a very high-resolution mesh. It allows observing the different stages and scales of turbulence. In that way, it can be assured that any turbulent scale can be directly computed without being modelled. Unfortunately, this strategy implies a huge computational cost and it is mainly used for small scale problems with low to moderate Reynolds number.

- Large Eddy Simulation (LES)
  This strategy consists in solving or modelling turbulence depending on the turbulent scales size. While the largest are resolved directly from the mesh, the smallest ones are modelled. This approach requires a coarser mesh resolution. As computational resources are not as high as in DNS, it can be employed in more applied uid ow problems.

- Reynolds Averaged Navier-Stokes (RANS)
  RANS equations are the least computationally demanding strategy as it uses a time-averaged solution of the ow elds. Due to this, RANS is the most used method in CFD. As it will be seen in following sections, RANS encompasses the k e, k w and Spalart-Allmaras turbulence models.

1.2.2. Reynolds Averaged Navier Stokes Equations

As it has been showed in previous sections (1.2), turbulent ows are characterized by their chaotic dynamics and their vortices along their trajectory. Due to the random nature of the uid particles, its physical variables can be decomposed into an average value plus a uctuating component. This procedure, which is known as Reynolds decomposition, could be akin as considering the ow entirely laminar with some little perturbations. By considering velocity and pressure variables as an average component plus the uctuating one:

$$U = \bar{U} + u^0 ! \begin{bmatrix} u \\ v \\ w \end{bmatrix} = \begin{bmatrix} \bar{u} + u^0 \\ \bar{v} + v^0 \\ \bar{w} + w^0 \end{bmatrix} \tag{1.9}$$

$$p = P + p^0 \tag{1.10}$$

Due to heavy and extensive equations, suf x notation or Einstein summation convention will be used in the following chapters.

The RANS equations, which result from averaging the NSE, are: [12, 13, 14].

- Continuity equation:

$$\frac{\partial \bar{U}_i}{\partial x_i} = 0 \tag{1.11}$$

- Momentum equation:

$$\frac{\partial \bar{U}_j}{\partial t} + \bar{U}_i \frac{\partial \bar{U}_j}{\partial x_i} = \frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_j} + \mu \frac{\partial^2 \bar{U}_j}{\partial \bar{x}_i \partial \bar{x}_i} \quad \frac{\partial \overline{u_i^0 u_j^0}}{\partial x_i} \tag{1.12}$$

With the result of the averaged in time of the momentum equation, emerge six additional stresses, called Reynolds stress ($\overline{u_i^0 u_j^0}$). Therefore, these Reynolds stress represent the mean momentum transfer caused by the uctuation velocity eld $u^0 v^0$.

1.3. Turbulence models

To resolve RANS equations for a turbulent ow, the closure with a model is needed. Generally, a 3D ow has four independent equations, the 3 components of momentum equation plus continuity equation. Usually, ows can be solved as only 4 variables are unknown. However, for turbulent ows, due to the emergence of the Reynolds stresses, more unknowns are added, what makes the system be undetermined. Turbulence models must provide enough assumptions to close RANS system.

One way to close RANS incompressible system is by Boussinesq's hypothesis. Boussinesq's noticed that the Reynolds stress could be related to the mean deformation rate. The Reynolds stress tensor can be divided into two different terms of turbulence; turbulent viscosity related term, and turbulent kinetic energy related term.

$$\rho \overline{u_i^0 u_j^0} = \mu_T \quad \frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \quad \frac{2}{3}\rho \quad k\delta_{ij} \quad = 2\mu_T S_{ij}^0 \quad \frac{2}{3}\rho k\delta_{ij} \tag{1.13}$$

where μ_T is the dynamic turbulent viscosity, k is turbulent kinetic energy, S_{ij} is the mean rate of stress tensor $S_{ij} = \frac{1}{2} \frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} + \frac{\partial \bar{U}_k}{\partial x_k}$ and δ_{ij} is the Kronecker delta.

Considering equation 1.21, the closure problem now depends on the turbulent viscosity resolution. Even though different turbulence models exist, to solve RANS equations, only linear ones will be considered in this report. Albeit some models are more precise and reliable than the studied ones, due to their extremely high computational cost and complexity they are left outside the scope of the present work.

1.3.1. k-ϵ model

k-ϵ model is the simplest model of turbulence. It is a semi-empirical model based on two transport equations, whose aim is to compute the turbulent viscosity μ_T, which one will be achieved via the turbulence kinetic energy k and its dissipation rate ϵ, which is the rate that k is converted into thermal energy by viscosity.

This model has been proven to be reliable for free and recirculating ows. Its popularity comes from its wide range of use and validation from industrial and aeronautical to environmental ows. However, it may not be the best model for problems involving complex ows with strong adverse pressure gradients and large separations.

The initial impetus for the k e model was to improve the mixing length model and nd alternatives to the algebraic notation of the turbulent ows of high complexity. Both turbulent kinetic energy (1.22) and its rate of dissipation (1.23) are obtained from the following transport equations:

$$\frac{\P k}{\P t} + U_i \frac{\P k}{\P x_i} = \frac{\P}{\P x_i} \quad \frac{n_T}{s_k} \frac{\P k}{\P x_i} \quad + P \quad e \tag{1.14}$$

Transport equations are divided into: advection terms, diffusion terms, production (P) and destruction (e) terms. Production terms P represents the rate at which turbulent ow motion is being induced due to the kinetic energy of the ow. s_k is the turbulent Prandtl number for the kinetic energy.

$$\frac{\P e}{\P t} + U_i \frac{\P k}{\P x_i} = \frac{\P}{\P x_i} \quad \frac{n_T}{s_e} \frac{\P e}{\P x_i} \quad + \frac{e}{k}(C_{1e}P \quad C_{1e}e) \tag{1.15}$$

Transport equation for dissipation is also formed by advection, diffusion, production and destruction terms. However, the turbulent Prandtl number is now for the dissipation rate instead that for kinetic energy. Production terms come from the generation of turbulence kinetic energy due to buoyancy and mean velocity gradients.

As it has been said at the beginning of section 1.3.1, turbulent viscosity $n_T(x_i;t)$ is the goal of k-epsilon model of turbulence. In accordance to k-epsilon main properties, turbulent viscosity can be de ned as:

$$\mu_T = r\, C_\mu \frac{k^2}{e} \tag{1.16}$$

These equations consist of some constant adjustable values. s_k s_e as we have seen before, in addition to C_{1e} and C_{2e}, which ones have been de ned after numerous iterations of data tting for a wide range of turbulent ows.

$$C_{1e} = 1{:}44; \quad C_{2e} = 1{:}92; \quad C_\mu = 0{:}09; \quad s_k = 1{:}00; \quad s_e = 1{:}30 \tag{1.17}$$

Some turbulence models such as k e are only applicable in the area of turbulence fully developed, and do not perform well in the area close to the wall. In order to deal with the near wall region, another two methods are usually suggested [18, 19, 20, 23, 49].

1.3.2. Spalart-Allmaras model

Spalart-Allmaras turbulence model is a RANS approach that solves a modelled transport equation for the kinematic eddy turbulent viscosity. This model was specially derived for aerodynamic applications involving wall-bounded ows. Furthermore, it has shown to give accurate results for boundary layers ows subjected to adverse pressure gradients.

Due to its high analytical complexity, not extensive and deeply explanation will be given about Spalart-Allmaras. As it has been said, due to dif culty of the (y^+) behaviour of n_T

near to the wall, n_T is considered as linear all way long to the wall, given name a new variable called $\tilde{n}$.

$$n_T = \tilde{n} f_v \tag{1.18}$$

where:

$$f_v = \frac{c^3}{c^3 + C_{v1}^3} \quad ; \quad c = \frac{\tilde{n}}{n}$$

Like any transport equation, it is composed of different terms, such us convection term and diffusion term. As Spalart-Allmaras is a turbulence model, it provide two additional terms regarding the production and destruction of the turbulence. Production term ($C_{b1}\bar{S}\tilde{n}$), generates turbulence in regions of high shear, which is proportional to shear rates in the mean velocity gradient. Regarding destruction terms ($C_{w1} f_w(\frac{\tilde{n}}{d})^2$), close to the wall we expect that the wall itself physically destroys turbulence by blocking inviscid pressure gradient uctuations and viscous damping.

$$\frac{\partial \tilde{n}}{\partial t} + \nabla \cdot (U\tilde{n}) = C_{b1}\bar{S}\tilde{n} + \frac{1}{s} \nabla \cdot (n + \tilde{n})\nabla\tilde{n} + C_{b2}(\nabla\tilde{n})^2 \quad C_{w1} f_w(\frac{\tilde{n}}{d})^2 \tag{1.19}$$

Even though Spalart-Allmaras is just a one-equation turbulence model, what at rst sight seems a liability in terms of accuracy, this model is able to provide turbulence results with an outstanding degree of accuracy. This fact, as it has been said at the beginning of section 1.3.1, is due to the speci c objective to modulate turbulence into aerodynamic eld, instead of trying to reach all generics ows. Some other advantages which made Spalart-Allmaras useful for studies, is its powerful use for boundary layer ows with arbitrary pressure gradients, and its dimensional reduction, which decreases the complexity of the problem and the overall simulation time [15, 16, 17, 18, 52].

1.3.3. k-w model

Like Spallart-Allmaras turbulence model, k w emerged with the goal to solve ows with adverse pressure gradients and boundary layers, specially due to the lack of accuracy of k e method spotting the separation point of the boundary layer, and not precise C_L and C_D.

k e is a two-linear-equation model, whose aim is to compute the eddy viscosity. Similarly to k e, turbulence problem is solved by the combination of two transport equations. First one, solved for the turbulent kinetic energy k, and the second one, for the speci c turbulent dissipation rate w.

$$w = \frac{e}{C_\mu k} \quad \text{where } C_\mu = 0{:}009 \tag{1.20}$$

$$\frac{\partial(rw)}{\partial t} + \nabla \cdot (rUw) = \nabla \cdot \left(\mu + \frac{\mu_T}{s_k}\right)\nabla w + \frac{g}{n_T}P_k - brw^2 \tag{1.21}$$

Where s_k, b, s are empirical coef cients, which are distinct from k e model.

What makes k e and k w so similar is that the previous transport equation can be solved either for w or e as they are directly related. However, k w is much better in comparison to k e model due to k e uses empirical damping functions to compute the right solution close to the wall. These damping functions are not accurate enough in the

presence of adverse pressure gradients. On the other hand, k w does not need these damping function,making it more ef cient in boundary layer regions.

Nevertheless, the main drawback of this model is its dependence on free stream turbulence. This means that it is a very sensitive model and any little change on initial conditions becomes in a considerably variation in ow results. In order to arrange this dependence a new method named k w SSTemerged.

Origins of SST model arrived to try to join in one single method the advantages of k e and k w, reaching in this way modulate free-stream ow with k e and boundary layers and adverse pressure gradients zones with k w. Notice that transition between both regions needs to be soft, what makes a blending region (basis of SST model) appear. [26, 27, 28].

Figure 1.1: k-w SST model
[26]

CHAPTER 2. BOUNDARY LAYER

2.1. Boundary layer theory

When a uid ows freely, without objects, walls or barriers obstructing its trajectory, its velocity eld is unaffected by viscosity. However, when freestream ow interacts with a surface, due to the shear stress of the wall, the incident ow velocity is altered originating the boundary layer phenomena.

The aerodynamic boundary layer was conceptualized by Ludwig Prandtl in 1904 in Germany. This theory simpli es the equations of uid ow by dividing the ow eld into two different regions. First, the boundary layer, which is located near the surface of the body and is dominated by the viscous effects of the uid. This boundary layer generates the majority of the drag experienced by the object. On the other hand, there is an outer region beyond the boundary layer where the effects of viscosity can be neglected without causing a signi cant impact on the solution.

Due to the interaction uid - object, aerodynamic forces are generated between them, which magnitude depends on the object geometry as long as several physical properties such as velocity, density, viscosity and compressibility of the uid. Viscosity plays an essential role as when the uid moves past an object,particles right next to the surface stick to it, originating the no-slip boundary condition. The particles just above the surface are slowed down in their friction with the particles sticking to the surface. These molecules in turn, slow down the ow just above them. The farther one moves away from the surface, the lesser the friction affected by the object surface. This creates a thin layer of uid near the surface in which the velocity changes from zero (at the surface) to the outer inviscid value (away from the surface) [30].

Even though the boundary layer is extremely thin in comparison to the rest of the ow domain, its understanding is crucial to predict the aerodynamic behaviour of a moving body.

In order to comprehend the boundary layer as a hole, the necessity to present some relevant BL properties arises [45].

- Velocity pro le
  The boundary layer velocity pro le follows an increasing trend away from the wall. This pro le generates a velocity gradient, which starts from the no-slip condition or zero velocity at the airfoil surface to reach the outer inviscid velocity at the edge of the boundary layer.

- Thickness (d_{99})
  The thickness or boundary layer width (d_{99}) is the distance between the geometry surface until where ow recovers 99% of its initial velocity. Notice that this property will vary along the chord. Depending on ow characteristics and the roughness of the surface, the boundary layer could be laminar or turbulent.

- Displacement Thickness (d_1)
  Displacement thickness quanti es the mass ow resolution due to the viscous block-

age.

$$d_1 = \int_0^d \left(1 - \frac{\rho u}{\rho_e u_e}\right) dy \tag{2.1}$$

- Momentum thickness
  Momentum thickness quantifies the amount of momentum missing due to the viscous blockage. In the same way as the displacement thickness, momentum thickness also varies along the chord.

$$q = \int_0^d \frac{\rho u}{\rho_e u_e}\left(1 - \frac{u}{u_e}\right) dy \tag{2.2}$$

- Skin friction
  Skin friction inefficient is caused by the viscosity of fluids and it is defined through the wall shear stress (τ_w) which quantifies the skin friction over the body's surface.

$$\tau_w = \mu \left.\frac{\partial u}{\partial y}\right|_w \tag{2.3}$$

  where μ is the dynamic viscosity of the fluid, ρ_e and u_e are the pressure and velocity just outside the boundary layer.

  The skin friction coefficient is a dimensionless skin shear stress which is nondimensionalized by the dynamic pressure of the outer inviscid flow velocity.

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho_e u_e^2} \tag{2.4}$$

  where μ is the dynamic viscosity of the fluid, ρ_e and u_e are the pressure and velocity just outside the boundary layer.

- Shape factor
  The shape factor, whose aim is helping to differentiate laminar and turbulent boundary layers is the ratio between the displacement thickness and the momentum thickness. Where shape factor equal to 2.59 (Blasius boundary layer) describe laminar flows, while shape factors around 1.3 describe turbulent flows and flows near separation respectively.

$$H = \frac{d_1}{q} \tag{2.5}$$

Figure 2.1 illustrates the previous properties, which ones have been substituted into equations to solve the boundary layer.

Figure 2.1: Boundary layer sketch. Not to scale
[56]

2.2. Boundary layer detachment

Boundary layer separation is one of the most signi cant phenomena in airfoil aerodynamics. Separation occurs due to the presence of an adverse pressure gradient along the surface of the airfoil pro le, which slows down the ow within the boundary layer due to viscous and pressure forces. Conversely, inertia and its initial velocity are the only favorable forces for maintaining the boundary layer attached to the surface.

Figure 2.2: Boundary layer development. Not to scale
[55]

Figure 2.2 show the development of a boundary layer. When ow touches the leading edge, a stagnation point is encountered what results on a velocity increase on the streamwise direction. At some point of the chord, and analysing the upper surface, velocity starts to decelerate resulting on a pressure increase or a positive pressure gradient is originated to ful ll Bernoulli's theorem. This pressure gradient is dependent on several things, such as the angle of attack or the airfoil geometry.

Due to opposing forces against the ow, the boundary layer is prone to reverse and form recirculation zones. At this point, the boundary layer separates. The risk of separation increases with a more pronounced adverse pressure gradient, which is usually caused by a higher angle of attack.

Having the boundary layer separated results in an aerodynamic disadvantage. Firstly, there is a signi cant loss of lift, also known as stall, due to the signi cant increase in pressure. Additionally, there is a pressure drag that manifests as a net pressure force in the longitudinal direction, creating regions of lower pressure at the trailing edge than at the leading edge.

These separation points can be easily observed by plotting the friction coef cient along the airfoil's upper surface. The rst time the friction coef cient reaches zero, the boundary layer separates, and it it reattaches to the airfoil, the friction coef cient becomes zero again, typically at the trailing edge.

To con rm these separation points, the velocity vector can be plotted as a function of the y-coordinate for different chord locations (x/C). In this way, when the boundary layer is separated, a small reverse ow can be observed [30, 45].

2.2.1. Flow transition

The transition phenomenon is a process in which the ow starts as laminar at the leading edge of the airfoil and transitions to turbulent as the boundary layer develops. This transi-

tion can be caused by various factors such as surface roughness, ow velocity uctuations, or external disturbances. Transition can also be induced by using ow control techniques.

In theory, transition is considered to occur at a point along the chord, but in practice, it occurs over a short region known as the transition zone.

Laminar ow is characterized by being smooth and stable, with a relatively thin boundary layer compared to turbulent ow, resulting in low drag. Fluid particles move in parallel and ordered layers, and compared to turbulent ow, velocity uctuations are minimal, producing no turbulence and making laminar ows highly predictable.

On the other hand, turbulent ow is characterized by vortices, quick velocity uctuations, and chaotic uid motion, with a thicker boundary layer and higher drag.

The increased thickness of the turbulent boundary layer also affects the velocity pro le. While in a laminar boundary layer. The velocity pro le is more gradual throughout it. In a turbulent boundary layer, the velocity increases fast very close to the wall, as it can be seen in Figure 2.3.

Figure 2.3: Laminar and turbulent boundary layers velocity pro le. Not to scale. [54]

Not only laminar and turbulent ows are different in velocity pro le behavior, but they also experience differences in other properties such as wall shear stress and friction coef cient, which are higher for turbulent ows. Additionally, uid particles in turbulent ow possess higher kinetic energy, offering greater resistance to adverse pressure gradients, what helps to maintain the ow attached to the airfoil for a longer time, thereby minimizing pressure drag compared to laminar ow [30, 45].

2.3. BL control methods

As previously discussed in Chapter 2, the separation of the boundary layer greatly affects the behavior of uids in the presence of a solid. The current Chapter 2 aims to provide the reader with a comprehensive understanding of the boundary layer to proceed with the design of control methods to improve its aerodynamic performance.

The idea of boundary layer control is not new, and there is an extensive literature on the subject [4, 5, 46, 47]. However, the challenge lies in the complexity of implementing such control methods.

The concept of control methods aims to modify the velocity pro le within the boundary layer, reducing viscous friction and consequently aerodynamic drag. Depending on the solid or vehicle under study, the improvement objectives and any potential constraints will determine the choice of a particular boundary layer control method.

2.3.1. Passive ow control

It is called passive ow control because it does not require additional energy intervention for its operation. It mainly consists of altering the surface geometry or adding elements. This type of control method has not been addressed in the present work, but it offers a wide range of applications.

In comparison to active ow control, passive control is generally less expensive as it does not require additional actuators or an external power source. Although it is less exible due to its independence from time or potential limitations for different ight situations, the passive control method can be extremely useful in operations where continuous adjustments over time are not required.

2.3.2. Active ow control

Active Flow Control (AFC) is a rapidly evolving multidisciplinary scienti c technology that aims for change a state of natural ow to a more desirable state. This technology have been used during last decades with the purpose to modify the ow eld by an external interaction.

A wide variety of methods can be employed in order to achieve different goals, mainly of which, transition delay, drag reduction, lift enhancement, separation postponement, etc. The potential bene ts of ow control may include improved performance, affordability, fuel consumption economy, and environmental compliance [2].

Probably, the art of ow control dates back to 1900 with Prandtl's development of boundary layer theory, which one explained the separation phenomena and performed several experiments to control the boundary layer. Nevertheless, the gold period of ow control arrives with World War II, as well as during the Cold war. The unforeseen necessity to develop faster, lighter, highly maneuverable, more ef cient aircrafts makes that ow control methods were extensively studied. During that period was proved that suction or projection of ow along the surface of an airfoil pro le could alter the pressure distribution by momentum transfer to the boundary layer.

Lately, thanks to computation 2nd generation, was possible perform numerical simulations situations of complex ow [2, 3].

The emerging technology of active ow control (AFC) could provide a breakthrough in aeronautical science that would enable the engineering design of next-generation.

Active ow control methodology can be divided into predetermined and interactive. On one hand, predetermined ow control involves the introduction of actuators / steady or unsteady energy inputs without considering the state of the ow eld. On the other hand, interactive methods, the power input to the actuator is continuously adjusted based on some sensors measurements [2, 3].

From all the different active ow control methods that could be applied, blowing and suctions jets have been applied to the current work:

- Blowing
  Blowing technique consists on injecting air jets at high velocity in order to manipulate and control the ow over a surface. This AFC method implies the necessity to implement strategically jets or ori ces along the airfoil surface. When AFC control is turned on, these jets generates ow perturbations which alter and modify its behaviour. In this way the boundary layer is re-energized, what can be resulting on a later boundary layer detachment and a decrease in aerodynamic drag.

- Suction
  Suction method could be more intuitive than blowing technique. Its physical interpretation consists on suctioning the most decelerated part of ow within the boundary layer. After the jet implementation, which is required to be strategically implemented, the boundary layer becomes thinner and with higher momentum.

  The application of suction method is very robust. With this implementation we are no longer dependent on the geometry design as depending on jet suction velocity and jet width, it is possible to delay boundary layer separation until approximately AoA of 40 degrees. Not only the advantage of using high values of AoA are available to suction technique, but also its considerably reduction of aerodynamic drag by maintaining the ow attached along the airfoil surface.

  The fact of being an active ow control offers the possibility to modify the jet intensity at real time by adjusting the airfoil necessities at any time during the ight.

- Synthetic Jets
  Synthetic jet actuators are one of the many types of actuators that can be used in ow control, which ones recently have been extensively studied. They offer a wide variety of applications, highlighting separation, turbulence and propulsion. Due to its robustness, its low power consumption, fast response, and its reliability have become a recent breakthrough in actuators concepts [3, 4, 5].

  These jets are characterized for being entirely synthesized from the surrounding uid and without the need of mass injection to the external ow. For this reason synthetic jets are also known as Zero Net Mass Flux (ZNMF) jet since it is created by oscillating the uid around the body periodically. Nevertheless, actuators devices transfer momentum, energy and vorticity affecting the external ow in a local and global way. These actuators produce suction and ejection of the external uid at a determined frequency. While the exit of the jet, ow is separated and creates a train of rotating vortices that convect far away of the external ow eld. In case the intensity of the jet (velocity amplitude and frequency) are not high enough, the vortices are sucked back into the cavity.

  A synthetic jet actuator consists of an ori ce or slot, an oscillating membrane, and a cavity (see Figure 2.4). The cavity plays an important role in the performance of the overall device since it is where the uid is periodically compressed and expanded [4, 5].

  In general, the cavity shape could be of any type, rectangular, cylindrical... However, the most important aspect of the cavity is its volume. The cavity plays an essential

Figure 2.4: Idealised operation of a synthetic jet actuator
[6]

role in determining ow characteristics; compressible or incompressible. Regarding the ori ce, its design and its size will de ne the actuators response and the nature of ow inside the cavity [4, 5].

Synthetic jets can be produced in different ways; with the use of piezoelectric, electrodynamic or even mechanical actuators. The main purpose of these actuators is to move large volumes of uid at desired frequencies. To reach it, these actuators cause a deformation/bending moment in the diaphragm, which produces the displacement of volumetric ow.

CHAPTER 3. NUMERICAL METHODS CFD

This chapter provides the reader the enough acknowledgment to perform a CFD simulation taking into consideration the essential parameters. CFD is a branch of uid mechanics which by numerical analysis and data structure predicts uid ow by solving the governing equations using computational power.

As it has been explained in Chapter 1, fundamentals of aerodynamics, beginning with Navier Stokes equations, led to RANS equations and its different turbulence models, are the basis of any CFD software.

Due to the extreme dif culty of solving analytically RANS equations, the necessity to solve uid ows using numerical methods came up. Historically, methods were initially developed to solve linear potential equations, but they were quite limited. However, these recent years with the improvement of technology and the emergence of super-high-speed computers better and accurate solutions can be offered for any type of equation and turbulence model.

However, the process to perform a CFD simulation has also its own dif culties. Even though RANS equations are computationally solved, software requires the users a wide variety of parameters and considerations which need to be thoroughly studied [30, 31, 32].

3.1. Methodology

In order to perform a successful CFD simulation, as it has been said, some parameters, steps and approaches are required:

1. Pre-processing
   Firstly, a de nition and construction of the studied geometry and uid domain. Secondly the discretization of the domain is essential to mesh enough accurate in accordance with our requirements. Followed by de nition of boundary and turbulence conditions as long as control parameters.

2. Simulation
   Appliance of numerical methods to solve iteratively our equations.

3. Post-processing
   Analysis and visualizations of ow resulting behaviour and coef cients.

3.2. Domain setup

First step in any project is decide what is wanted to reach and how shall be reached it. After some research about CFD in airfoils, [33, 34, 35] and considering different meshes, C-mesh was the most appropriated one among other types.

C-mesh is the name given due to its C shape on the inlet surface, which it provides ow capturation variation without making dense meshing on complete domain. Furthermore, the fact of meshing with c-mesh gives better ow convergence over the airfoil and the curvature of the mesh matches the leading edge curvature for good ow separation.

Figure 3.1: Sketch of uid domain

Figure 3.1 shows the domain of study, from its boundary surfaces (Inlet, Outlet, Upper and Lower), airfoil position, which its leading edge is located at (x,y)=(0,0), and dimensions referred to airfoil's chord length.

Domain dimensions has been considered as previous, after carefully developed some simulations based on OpenFoam tutorials. Considering ows over geometries around Re=60000, the eld downstream needs to be large enough to do not lose detailed results during simulations. Field upstream is considered half of downstream's eld to let the incoming ow interacts airfoil's walls as completely free stream.

The airfoil selected is an Eppler 169 [36] (Figure3.2), a symmetric airfoil used for low Reynolds numbers. As it has been said on Abstract chapter, Reynolds number is set at 60000, just above the edge between laminar and turbulent regimes. Problem variables have been de ned as following to make easier the evaluation of results: Reynolds number 60000, chord's length has been taken as $c = 1$, $U_{¥} = 1$, density is considered constant and $r = 1$ and viscosity $\mu = \frac{1}{Re} = 1{:}667\ 10^{5}$.

Figure 3.2: Geometry of Eppler 169 airfoil

3.3. Meshing

Mesh creation is the discretization of the domain of interest, where RANS equations will be solved, and it is an essential preliminary step for the CFD simulation.

Intuitively , it is thought that as ner and smaller the mesh, as closer the nodes to each other, as more continuum condition imposed for solving governing equations, results as higher its accuracy. Being these restrictions applied to our mesh, results would be extremely closer to reality ows, however not always is allowed to design a so high-precision mesh. Limited computational resources and huge time demanding simulations make unfeasible this types of meshes.

On the other hand, the studied mesh could not employ too wide or large cells. This fact would produce a fatal error in results computation as the ow behaviour would not be captured properly through the cells. Nevertheless, a not so re ned mesh is available to the vast majority of CFD users due to its low computational resources needed, and for being low time demanding simulations [37].

As it has been said in previous paragraphs, none of both presented meshes are worth to be set due to its clear drawbacks. Achieving a proper mesh for our requirements is one of the main hard and demanding tasks when performing a CFD simulation, so a trade off between the two mesh options presented above will be needed. There is not only one target mesh feasible for our simulation, but experience and intuition on uid mechanics are a key aspect to reach the most optimal one.

Initially, mesh development would had to be achieved through the mesh options incorporated in OpenFoam software, which will be used later on to compute the different simulations. However, OpenFoam requires 3D geometries, and create external C-mesh with BlockMesh command was quite a hard task. Even though re ne the domain next to airfoil's wall was not as complicated as it should be, thanks to snappyHexMesh command that allows the user interact with layer cells width and cell growth ratio. However, transition between layers close to the airfoil and the ones far from it was not clear and a huge re nement was needed to mesh the entire 20x30 domain.

Finally, mesh generation have been made with Gmsh software, together with a Matlab script to compute the required boundary layers cells width and numbers depending on Reynold's number, $U_{¥}$, r, μ and the characteristic length L. Gmsh is an open source 3D nite element mesh generator with a built-in CAD engine and post-processor. Its design goal is to provide a fast, light and user-friendly meshing tool with parametric input and exible visualization capabilities [38].

3.3.1. Mesh considerations

When discretizing a domain, several considerations should be taken into account:

1. Structured or unstructured mesh
   Structured or unstructured meshes are totally dependent on its connectivity. Structured connectivity use to be Quadrilateral in 2D and hexahedra in 3D, reaching a highly space ef cient model, since the neighborhood relationships are de ned by storage arrangement.

On the other hand, an unstructured mesh is characterized by its irregular connectivity, which typically employs triangles in 2D and tetrahedra in 3D and requires store additional connectivity arrays to keep information about their adjacent nodes and positioning, what implies more computational capacity [39], as it can be seen on Figure 3.3.

Figure 3.3: Structured vs unstructured mesh
[40]

To avoid drawbacks of previous mesh structures, an hybrid structured and unstructured mesh was designed in the most ef cient manner.

Whichever grid type is nally chosen, its quality is crucial to the analysis as a whole. Better mesh quality, for instance, boosts convergence rate. Numerical error is introduced by improperly constructed meshes, increasing the amount of computation necessary for convergence. The numerical order may be locally decreased for extremely suboptimal elements to aid in convergence and stability, or, even worse, the solution may completely fail to converge.

Additionally, a more accurate solution is produced by meshes of higher quality. Not only better convergence is achieved, but also better alignment due to the alignment between structured cells and ow behaviour. An excellent illustration of this is the capability to enhance the delity of solutions in the region by meshing re nement at speci c regions of the geometry where gradients are large. The precision of the solution might be constrained if the mesh is not suf ciently re ned [37, 39].

Considering simple geometries, such an airfoil, structured mesh is preferred due to the brief knowledge of how the ow will behave. In that way, high degree of quality and control must be applied to those potentially con icted areas. However, for complex geometries, ow prediction could be less intuitive and with more variable patterns implying that engineers have to sacri ce high control levels in certain areas.

While a structured mesh is normally preferred, CPU time must also be considered. When the number of cells per unit area is at its highest for a highly re ned mesh, a signi cant amount of CPU time will be needed. Nevertheless, highly re ned structured mesh will consume less computational capacity than unstructured one due to the necessity to save storing mechanism as it has been previously said.

2. Aspect ratio, skewness warnings and orthogonality
   Previous parameters must be considered when designing a mesh because they are

of vital importance to avoid potential quality mesh problems.

- Aspect ratio
  Aspect ratio is the measure between the longest side and the shortest side of a cell. When a CFD mesh contains cells with the adequate aspect ratio, usually close to 1, cells tend to be more squared or cubic. Depending on the ow solver, steady state or transient, aspect ratio will be slightly different. Transient simulations are limited by the Courant number (Co), which is the fraction of the cell moved across in a single time step Mt (see Figure 3.4.a).

(a) (b) (c)

Figure 3.4: a) Courant number, b) high AR cells, c) low AR cells

If Co $<$ 1 means that Mt is running cell per cell without missing any information, what implies higher stability and accuracy.

However, aspect ratio are warnings, not errors, because depending on the location of the mesh Co $<$ 1 can not be guaranteed. Boundary layer regions, which are highly limited by y+ will not be a problem as at the boundary layer cell width increases at same time than velocity .

- Skewness
  Skewness is considered as the distorsion measure of cell elements respect to its ideal shape, see Figure 3.5. It is mainly used to evaluate mesh quality regarding regularity and uniformity. Skewness usually is computed for every cell and can be positively distorsioned or negatively (distorsioned to the opposite direction compared to the reference one).

Figure 3.5: Examples of skewness
[57]

- Orthogonality
  Orthogonality in CFD is considered as the goal to maximize the perpendicularity between cells boundaries to each other. This property is of vital importance due to the necessity to ensure the accuracy of the gradients and derivatives of ow variables. By reducing non-orthogonal cells, numerical errors are being minimized as soon as enhancing numerical methods convergence.

3. Boundary layer re nement
   One of the most demanding tasks of meshing procedure is to ensure that the mesh cells are ne enough to capture boundary layer behaviour.

   To make clear the comprehension of boundary layer computation some parameters have been declared:

Figure 3.6: Boundary layer parameters in CFD for viscous sublayer case [41]

- y_p: dimensionless distance from the centeroid of the cell to the nearest wall.
- $y+$: dimensionless distance equal to the cell width. $y+ = 2y_p$.
- y_t: total distance from the wall to outer in ation layer.
- d_{99}: boundary layer width.
- N: number of layers.
- G: growth ratio within in ation layer.
- u_t : friction velocity.
- t_w= wall shear stress.

As it has been mentioned in chapter 1.4, boundary layer is composed by different sublayers. Visocus sublayer is the layer closest to the wall and where shear stresses are exceedingly important.

In order to perform with enough accuracy at the wall vicinities (viscous sublayer), $y+$ needs to be $y+ < 5$ [42]. To achieve the maximum accuracy $y+$ is set to 1, as by its own de nition, as the smaller $y+$ value, the smaller cell height and more accuracy close to the wall. However, depending on the sublayer studied of the boundary layer, $y+$ value will be different.

To compute the previous parameters, two approaches are required:

(a) Boundary layer thickness (d_{99})
As it has been seen in chapter 1.4, boundary layer is computed different depending on the Reynolds Number. For turbulent boundary layers:

$$d_{99} = \frac{0:38L}{Re_L^{\frac{1}{5}}} \tag{3.1}$$

(b) Evaluation of y_p
In order to obtain a realistic y_p value, a CFD simulation is required to check several times that y_p chosen is accurate enough, known as initial guess. Nevertheless, often the wall is modelled as a at plate boundary layer because empirical data is available for this simple ow scenario.

By assuming that the boundary layer is fully turbulent and arranging t_w and u_t equations:

$$y_p = \frac{(y+)\,\mu}{r\,u_t} \tag{3.2}$$

Once y_p and $y+$ are computed, a simulation have been run, and $y+$ values from a post-processor have been extracted to check if its value is low enough. Moreover, should be taken into consideration that $y+$ value changes along surface, what provokes the necessity to emphasise with the lowest $y+$ values that zones that we care about.

Considering the number of layers N and growth ratio G, the following formulas are obtained:

$$y_T = \sum_{k=0}^{N\ 1} y_H G^k = y_H \frac{1\ \ G^N}{1\ \ G} \tag{3.3}$$

The purpose of boundary layer calculations is to mesh by using the least computational resources, what is reached by:

$$y_T = d_{99} \tag{3.4}$$

For the studied airfoil, not perfect equidistance could be reached due to the lack of equidistant tool in gmsh, making impossible ful ll equation 3.4. However, to perform boundary layer with enough re nement and avoid the loss of boundary layer data outside the in ation layers, the following condition must be ful lled: $d_{99} < y_T$.

Figure 3.7: Final airfoil geometry with in ation layers zone

Figure 2.4 shows airfoil Eppler169 with its ”equidistant” airfoil, which is the outer zone of the structured mesh. Previously, an on online calculator have been used [43] to calculate the rst cell height for the desired simulation values. After that, a Matlab script have been implemented to compute all CFD parameters involved in the boundary layer:

PARAMETER	VALUE
$U_{¥}$	1
r	1
μ	1.667e-05
L	1
y+	1
Re	60000
G	1.1
N	39
y_H	2.941e-04
d_{99}	0.0437
y_T	0.011

Table 3.1: Boundary layer parameters

3.3.2. Mesh development

In computational uid dynamics, mesh development is considered one of the most critical tasks. As it has been previously mentioned, a mesh serves as a discretized representation of the uid domain and plays an essential role in capturing ow behaviour accurately. Depending on mesh quality and resolution, the accuracy, convergence and computational ef ciency could be signi cantly impacted.

This subsection focuses on the important aspects of mesh development for CFD studies. Various considerations and techniques have been explored to ensure the capture of complex geometries and ow features with suf cient resolution at the same time that computational costs are minimized.

Three different meshes have been designed in order to achieve the nal optimal desired mesh.

First approach, seen in Figure 3.5, consists on a C-mesh completely structured. This approach ensures that no ow behaviour escapes from our grid due to its surplus of extremely re ned cells. The idea of splitting the domain into different sections is to produce softer and smoother transitions from closer areas to the airfoil to the domain boundaries. However when mesh check was performed, some failures appeared such as cells with high aspect ratios, non homogeneous regions, or maximum skewness values.

Figure 3.8: First mesh approach; completely structured outside region

Unfortunately solve a uid problem with this grid was not feasible due to the extremely huge time required to iterate steps far from the geometry, where freestream ow is not perturbed.

With the challenge to acquire maximum precision while solving previous mesh issues, mesh in Figure 3.6 was designed.

Figure 3.9: Second mesh approach; structured wake and unstructured outside region

Re ned structured regions were restricted to those zones were ow is altered such as regions around the airfoil wall and the entire wake. When reducing the number of structured cells the possibility to reduce non orthogonal and large aspect ratio cells is also reduced, but it stills appear when running this mesh. Moreover, a less smoother transition between

both regions appeared. Even though it would be the most accurate mesh due to the accurate re nement along the entire downstream wake and computation time is not has high as in Figure 3.8, the mesh did not reach convergence fast enough.

Finally mesh from Figure 3.9 was replaced by Figure 3.10 mesh. This mesh was designed as explained in Figure 3.4, creating a boundary layer equidistant to airfoil wall at a distance of 0.1 meter. Within this region, cells will be structured and re ned, while outdoors mesh follows an unstructured pattern. The fact of splitting the airfoil in different sub-regions is also bene cial. Non orthogonality is reduced in comparison to previous meshes, and more emphasis to relevant regions around the airfoil can be applied. Airfoil has been split three times over jet region, one for the jet re nement, and the rest in order to deal with 4-corners when creating trans nite regions (Figure 3.8.b). Upper surface have been split also three times to reach higher detailed cells over the region where boundary layer will be detached, while two splits have been done done along lower surface (Figure 3.8.a).

Figure 3.10: Final mesh approach: entire domain

(a) (b)

Figure 3.11: Airfoil details in nal mesh approach

Furthermore, as it is seen in Figure 3.8.a, two wake regions of 0,1 x 0,1 dimensions were created in order to capture ow behaviour close to the trailing edge.

This nal approach also offers a smooth transition along overall domain, producing low-density cells on the vicinities of external boundaries at once high density cells are reached close to the airfoil.

CHAPTER 4. CFD TOOLS

OpenFOAM is a CFD simulation software, which uses the nite volumes technique. It provides a exible and powerful platform for solving complex problems related to uid ow and heat transfer. Among other computational uid dynamics softwares such as anysis, 3d experience, etc. OpenFOAM is widely used for being an open-source code. It was developed by OpenCFD Ltd in 2004 due to the necessity to offer available and cheaper alternatives to everybody than other CFD softwares.

As it has been mentioned, the main advantage of OpenFOAM is its open-source nature, which implies that the source code is available, free to use, modify and distribute. That fact allows users use the software as its own speci c needs and maximize its performance. Moreover, exists a large community of developers who contribute to its continuous improvement.

OpenFOAM offers a wide range of functionalities and it is capable to solve various mechanical problems related to uid ow. It can simulate uid and turbulence, heat transfer, uid interactions, chemical reactions, sound, among others. It is designed to be exible and scalable, meaning it can be adapted to both simple and complex problems, from academic research to industrial applications.

4.1. Flow Solvers

OpenFOAM solvers are modules that implement speci c algorithms and numerical models to solve different CFD problems. Selecting the proper solver could be a hard task, as it must be the adequate one depending on the needs of each uid ow problem [48].

Compressible ows are the rst issue to deal with. This separation between compressible and incompresible solvers will allow to discard half of available solvers. For the current work, as active ow control (AFC) is usually employed at low speeds to maximize hyperlift, incompressible ow will be assumed.

In addition, two more approaches need to be done to select the adequate solver, as the nature of the ow could be laminar or turbulent. Most common OpenFOAM solvers for aerodynamics applications are listed as follows:

- simpleFoam
  Steady-state solver for incompressible, turbulent ows or ows where turbulence is not very signi cant.
- pimpleFoam
  Transient solver for incompressible, turbulent ow of Newtonian uids on a moving mesh.
- pisoFoam
  Transient solver for incompressible, turbulent ow.

PisoFoam is more suitable for problems with complex pressure-velocity and transient turbulent ows, while PimpleFoam provides an ef cient solution for transient ows with not

high complexity pressure-velocity zones. The choice between them depends on the speci c characteristics of the problem and the requirements for accuracy and computational ef ciency.

PisoFoam was selected as the preferred solver for the current work. This choice has been made because PisoFoam is capable of handle effectively transient phenomena, which is essential for analyzing the ow behavior during different stages of AFC membrane oscillation. Since the setup involves actuation through synthetic jets, the ability of PisoFoam to accurately capture transient behaviors is particularly valuable.

4.2. Boundary Conditions

Boundary conditions play a crucial role in computational uid dynamics (CFD) studies. These conditions specify how the uid ow interacts with the boundaries of the simulation domain and are essential for obtaining accurate and realistic results. Boundary conditions are important because they de ne the system's inputs and outputs and represent the environment. These conditions in uence the ow behavior and, therefore, the obtained results. By setting the correct boundary conditions, it is possible to simulate real-world situations with a high degree of accuracy and analyze complex phenomena.

It is important to mention the two-different types of boundary conditions employed in the followings OpenFOAM simulations: Dirichlet and Neumann boundary conditions.

Dirichlet boundary conditions are used to specify xed property values at the boundaries of the simulation domain, which provide a precise and detailed description of the ow behavior at speci c locations. On the other hand, Neumann boundary conditions are used to specify the ow of a property across the simulation domain boundary. Instead of xing a speci c value, a gradient or rate of change of the property in the direction perpendicular to the boundary is de ned.

To set boundary conditions, is essential a good understanding of which physical variables will vary depending on the turbulence model used. Regarding pressure and velocity, values will remain constant for the three turbulence models. Moreover, exists some literature regarding boundary conditions to p and $U_{¥}$.

- Velocity
  Velocity eld is set at the inlet physical boundary (Figure 3.1) as $\forall = (U_{¥x}; U_{¥y}; 0)$, as a Dirichlet boundary condition. $U_{¥x}$ and $U_{¥y}$ will be changed in order to perform the desired angle of attack. However, ”zero gradient” Neumann condition is set at the outlet. Lastly, as walls are treated as impermeable, no-slip condition $\forall = (0; 0; 0)$ is applied to the airfoil and to the jet in the baseline case.

- Pressure
  Pressure eld is set as 0 Pa in the entire domain. However, as OpenFOAM does not allow to apply zero-gradient condition on the inlet and outlet at same time, inlet pressure is set as a xed value 0, while zero-gradient function has been set on the outlet boundary. Walls are treated as impermeable, what oblige to impose zero gradient also at the airfoil boundary and the jet during baseline case.

On the other hand, when performing AFC while employing synthetic jets, jet boundary will

be treated as a patch, such an inlet in order to set the opportune conditions for every actuated case.

4.2.1. k e and k wSST

For these two turbulence models, shared physical variables will have same values in both methods, while e and w will be independent for each one.

- nut
  In order to take a value for the turbulent viscosity (n), the lecture from P. Catalano and R. Tognaccini has been considered [46]. n is computed via the free-stream ratio between air kinematic viscosity and turbulent viscosity:

$$\frac{n_{air}}{n} = 10^{\ 09} \tag{4.1}$$

  leading in that way to a turbulence viscosity of $1{:}667^{\ 14}\ m^2/s$.

- k
  Considering speci c documentation for k e turbulence model [52], k or turbulent kinetic energy is computed as follows:

$$k = \frac{3}{2}(I|U_\infty|)^2 \tag{4.2}$$

  where I is the turbulence intensity measured in %. For low turbulence cases [51], I has been set as 0,1.

- e
  Regarding the turbulence dissipation rate, which can be estimated as: [51]

$$e = \frac{C_\mu^{0:75} k^{1:5}}{L} \tag{4.3}$$

- w
  The turbulence speci c dissipation rate can be estimated as: [50]

$$w = \frac{k^{0:5}}{C_\mu^{0:25} L} \tag{4.4}$$

  where C_μ is a model constant set as 0.09, and L is our reference length scale.

Tables 4.1 and 4.2 show the most important parameters considerations of k e and k w turbulence models.

VARIABLE	INLET	OUTLET	WALL	FRONT AND BACK
U_∞ [m/s]	$(U_x; U_y; 0)$	zeroGradient	no-slip	empty
p [Pa]	zeroGradient	0	zeroGradient	empty
n [m^2/s]	1.667E-14	0	nutLowReWallFunction	empty
k [J/kg]	0.0150	zeroGradient	kLowReWallFunction	empty
e [J/kg s]	3.0187E-04	zeroGradient	epsilonWallFunction	empty

Table 4.1: k e boundary conditions