TFG TITLE: Simulation of an hypersonic gas turbine ramjet engine intake in the supersonic regime

DEGREE: Grau en Enginyeria d’Aeronavegació

AUTHORS: Ismael El Houas Ghouddana

ADVISOR: Fernando Pablo Mellibovsky Elstein

DATE: October 23, 2017
Resum

Des del començament dels temps, hem fixat els nostres ulls en l’espai, imaginant la possibilitat d’arribar allà. Després, d’uns centenars d’anys d’investigació, en el segle XX, vam assolir aquest objectiu. Ara, en el segle XXI, fer l’espai accessible és el nou repte que ens hem proposat. Hi ha diverses formes per tal d’aconseguir aquest objectiu: coets reutilitzables, pico-satèl·lits o vehicles hipersònics. La idea d’utilitzar vehicles que són capaços d’anar d’una pista d’un aeroport a l’espai és la idea que m’ha dut a realitzar aquest projecte. Per aquesta raó, he decidit estudiar una entrada d’aire hipersònica i un perfil alar en forma de diamant utilitzant dinàmica de fluids computacional (CFD).

Un dels objectius d’aquest projecte és l’estudi d’una entrada d’aire hipersònica d’un motor ramjet, aquesta és utilitzada per vehicles que volen en el règim supersònic. En aquest projecte, s’analitza la geometria d’una entrada d’aire hipersònica a un número de Mach de 2. Tanmateix, ha estat impossible aconseguir una solució totalment convergida a causa de la manca de temps i recursos. No obstant això, els resultats obtinguts són presentats en aquesta tercera secció. A més a més, el rendiment aerodinàmic d’un perfil alar en forma de diamant és estudiat. Per aquest projecte, he estudiat una geometria determinada amb angles d’atac, velocitats i gruix diferents. També, en aquesta secció vaig experimentar problemes per tal d’obtenir solucions convergides, però aquest fet no ha evitat que obtingui uns resultats precisos que m’han permès analitzar el comportament aerodinàmic del perfil.

Finalment, es fa una revisió de tot el projecte i es presenten les conclusions obtingudes. Al llarg d’aquest projecte s’han realitzat diverses simulacions, algunes d’elles no han convergit, però totes han ajudat a comprendre com funciona la dinàmica computacional de fluids (CFD) i el programa de Nektar++ i, finalment, obtenir la caracterització aerodinàmica del perfil aerodinàmic en forma de diamant. No obstant això, es necessita un estudi més profund del rendiment aerodinàmic en més situacions per validar-lo com una solució real. A més, es proposen futurs estudis per continuar l’estudi aerodinàmic dels vehicles supersònics.
Overview

Since the beginning of time, we have fixed our eyes in space, imagining the possibility to arrive there. After hundreds of years of investigation, in the 20th century, we have achieved this goal. Now, in the 21st century, making the space accessible is the new challenge which we have proposed. There are different ways in order to achieve this goal: reusable rockets, pico-satellites or hypersonic vehicles. The idea of using vehicles that are able of going from a runway to space is the base of this project. For this reason, I have decided to study a hypersonic intake geometry and a diamond shaped airfoil using commercial computational fluid dynamics (CFD).

One of the objectives of this project is to study a ramjet engine intake, this engine which is used by vehicles that fly at supersonic-hypersonic regimes. In this project, the geometry of a hypersonic intake in a supersonic regime at a number of Mach of 2 is analyzed. However, it has been impossible to achieve a full converged solution due to the lack of time and resources. Though, the obtained results are presented.

Moreover, the aerodynamic performance of a diamond shaped airfoil is studied. Concretely, a diamond geometry in different angles of attack, at different velocities and different thickness to chord ratios is analyzed. Also, in this section, I experienced problems in order to obtain converged solutions, but this fact has not avoided me to obtain good results and analyze its aerodynamic behaviour.

Finally, a review of all the project is made and the obtained conclusions are presented. Along with this project several simulations have been made, some of them have not converged, but all of them have helped to improve the knowledge about computational fluid dynamics (CFD) and how Nektar++ software works and, finally, obtain the aerodynamic characterization of the diamond airfoil. However, it is needed a deeper study of the aerodynamic performance in more situations in order to validate it as a real solution. Besides, future studies are proposed in order to continue the aerodynamic study of supersonic vehicles.
It has been a long journey. Now I am here, at the end of my university career, it has not been easy to arrive at this point. A lot of obstacles in my personal and academic life are been present all this time. For this reason, I have to thank all the people who have helped me to achieve this objective. I am very grateful to the professors who have really helped me to improve as an engineer. Overall, I want to thank with my girlfriend who has helped me to push harder in the worst time and makes me smile at any moment. Finally, my family, they are very special for me, they have really helped from the beginning of my academic career, at any time. My mother always has take care of me, always, in order to help at anything she can, she is incredible. My parents have always believed that I could be an excellent engineer and I want to make them proud of me. So, I finish this period, but a new stage will open and I will continue working to achieve all my purposes.
CONTENTS

Introduction ......................................................... 1

CHAPTER 1. Compressible Flow ................................. 3
  1.1. Overview .................................................... 3
    1.1.1. Density ................................................. 3
  1.2. Integral Form of the Conservation Equations .......... 4
  1.3. Differential Conservation Equations .................. 5
  1.4. Shock Waves ............................................... 6
    1.4.1. Shock Waves Generation ............................ 6
    1.4.2. Pressure Waves ...................................... 7
    1.4.3. Types of Shock Waves .............................. 8
  1.5. Summary ..................................................... 12

CHAPTER 2. Nektar++ ............................................. 13
  2.1. Overview .................................................... 13
  2.2. FEM methods ............................................... 13
  2.3. Spectral method ......................................... 14
  2.4. Spectral element method ............................... 14
    2.4.1. Lagrange polynomials .............................. 15
    2.4.2. Gauss-Lobatto-Legendre polynomials ............ 16
  2.5. Spectral hp/element method ............................ 16
  2.6. Summary ..................................................... 16

CHAPTER 3. Supersonic Profile Simulation .................. 17
  3.1. Overview .................................................... 17
  3.2. Geometry ................................................... 17
  3.3. Results ..................................................... 18
    3.3.1. Varying AoA .......................................... 19
LIST OF FIGURES

1 Some of the future vehicles which will make access to space affordable. 1

1.1 Pressure waves in a stationary, subsonic and supersonic motion. 7
1.2 Diagram of a normal shock. 9
1.3 Concave corner. 10
1.4 $\theta-\beta-M$ relation. 10
1.5 Detached bow shock. 11
1.6 Prandtl-Mayer expansion wave. 12

2.1 Rectangular grid. 13
2.2 Example of interpolation divergence for a set of Lagrange polynomials. 15

3.1 Diamond Airfoil scheme. 17
3.2 Scheme of the diamond airfoil with its dimensions. 18
3.3 Diamond Airfoil scheme. 18
3.4 Supersonic profile with the faces labeled. 19
3.5 Visualization of Mach field at Mach 2 and AoA=0°. 20
3.6 Pressure, Mach, Density and Specific linear momentum distribution at $M=2$ and AoA=0°. 21
3.7 Visualization of Mach field at Mach 2 and AoA=2.5°. 23
3.8 Pressure, Mach, Density and Specific linear momentum distribution at $M=2$ and AoA=2.5°. 25
3.9 Deflection angle explanation. 26
3.10 Visualization of Mach field at Mach 2 and AoA=5°. 26
3.11 Prandtl-Meyer expansion wave angle explanation. 28
3.12 Visualization of Mach field at Mach 2 and AoA=7.5°. 29
3.13 Visualization of Mach field at Mach 2 and AoA=10°. 30
3.14 Global $C_L$ as a function of AoA. 31
3.15 Global $C_D$ as a function of AoA. 32
3.16 Drag Polar graph where the two forms of drag are presented. 33
3.17 Global $C_L^2$ as a function of $C_D$. 35
3.18 Lift to Drag ratio as a function of angle of attack. 36
3.19 Pressure coefficient distribution along airfoil's chord. 37
3.20 Pressure coefficient distribution over airfoil's thickness. 39
3.21 Scheme of the pressures forces acting on the airfoil. 41
3.22 Global $C_{M_L}$ respect to the leading edge as a function of AoA. 42
3.23 Global $C_{Max}$ as a function of AoA. 44
3.24 Global $X_{C_F}$ as a function of AoA. 45

25 Riemann Problem. 56

26 Scheme of a ramjet engine sections and their functions [1]. 59
27 Scheme of the ramjet intake with its dimensions. 60
28 Supersonic intake geometry mesh. 60
29 Visualization airflow's Mach field in the ramjet intake. 62
30 Aerodynamic forces acting on a body. 63
31 Oblique Shock Waves and Prandtl-Meyer Expansion diagram. 65
32 Visualization of simulation results at Mach 2 and AoA=0 deg 67
33 Visualization of simulation results at Mach 2 and AoA=2.5 deg 67
34 Visualization of simulation results at M=2 and AoA=5 deg 68
35 Visualization of simulation results at M=2 and AoA=7.5 deg 68
36 Visualization of simulation results at M=2 and AoA=10 deg 69
37 Pressure, Mach, Density and Specific linear momentum distribution at M=2 and AoA=5° 71
38 Pressure, Mach, Density and Specific linear momentum distribution at M=2 and AoA=5° 72
39 Pressure, Mach, Density and Specific linear momentum distribution at M=2 and AoA=10° 73
40 Visualization of Mach field at M=2 and AoA=12.5° 75
41 Visualization of Mach field at M=2 and AoA=15 deg 76
42 Visualization of Mach field at M=2 and AoA=17.5° 77
43 Visualization of Mach field at M=2 and AoA=20° 78
44 Visualization of Mach field at M=2, AoA=2.5° and θ=2.5° 79
45 Visualization of Mach field at M=2, AoA=2.5° and θ=7.5° 80
46 Diamond airfoil at subsonic velocity. 83
47 Pressure, Mach, Density and Specific linear momentum distribution at M=0.9 and AoA=0 deg 84
48 Busemann's Biplane CFD simulations. 85
49 Busemann's Biplane CFD simulations. 85
50 Busemann's Biplane concept [2]. 86
LIST OF TABLES

3.1 Table M=2 and AoA=0° .................................................. 22
3.2 Aeroforces table M=2 AoA=0° ......................................... 23
3.3 Table M=2 AoA=2.5° ...................................................... 24
3.4 Aeroforces table M=2 AoA=2.5° ......................................... 24
3.5 Table M=2 AoA=5° ......................................................... 27
3.6 Aeroforces table M=2 AoA=5° ........................................... 27
3.7 Table M=2 AoA=7.5° ....................................................... 29
3.8 Aeroforces table M=2 AoA=7.5° ......................................... 29
3.9 Table M=2 AoA=10° ....................................................... 30
3.10 Aeroforces table M=2 AoA=10° ........................................ 31
3.11 Wave drag coefficient comparison between theoretical and experimental values 34
3.12 Pressure coefficients obtained experimentally over the airfoil at different AoA 40
3.13 Aerodynamic coefficients obtained by experimental $C_p$ values .............. 40
3.14 Moment coefficients obtained by experimental pressure coefficients .......... 43

15 Table M=2 AoA=12.5° ...................................................... 75
16 Aeroforces table M=2 AoA=12.5° ........................................ 75
17 Table M=2 AoA=15° ....................................................... 76
18 Aeroforces table M=2 AoA=15° ........................................... 76
19 Table M=2 AoA=17.5° ...................................................... 77
20 Aeroforces table M=2 AoA=17.5° ........................................ 77
21 Table M=2 AoA=20° ....................................................... 78
22 Aeroforces table M=2 AoA=20° .......................................... 78

23 Table M=2, AoA=2.5° and delta=2.5° ................................. 79
24 Aeroforces table M=2 AoA=2.5° and $\theta=2.5°$ ...................... 80
25 Table M=2 AoA=2.5 deg and $\theta=7.5°$ ................................. 80
26 Aeroforces table M=2 AoA=2.5 deg and $\theta=7.5°$ .................... 81
INTRODUCTION

Nowadays, it exists a big movement in the aerospace industry in order to make space access affordable. Since the beginning of the space race, access to space has been only possible for big governmental companies: NASA, ESA and RosCosmos. This fact has made that the space development has slowed down, once the space race between USA and USSR finished.

However, now, big private companies are working so as to space will become more accessible for small companies which want to put a small satellite in orbit or research groups which want to do experiments in a zero-gravity environment. These efforts which will make that more people work towards space investigation will help to develop new engineering concepts, reduce costs and enlarge the possibilities of space business. Some of these companies are: Reaction Engines Ltd, Space X, Bristol Spaceplanes, Starchaser Industries Ltd, Blue Origin, Bigelow Aerospace or Stratolaunch Systems.

(a) Spacecab vehicle by Bristol Spaceplanes[3].

(b) Space vehicle by Reaction Engines[4].

(c) Reusable Rocket by Space X[5].

(d) Reusable Rocket by Blue Origin[6].

Figure 1 – Some of the future vehicles which will make access to space affordable.

In Fig. 1, you can see some of the actual solutions in order to make the space accessible. All of these solutions need to overcome the sound barrier and flight in supersonic regime. Therefore, compressible flow theory has to be used so as to make these solutions a real product.
What is more, some of these companies want to establish the space tourism with space hotels and travels. Also, there are companies which want to use these new transport concepts in order to reduce the time of travelling large distances. For example, Reaction Engines propose 4 hours travel from London to Sydney or Space X wants to travel from New York to Shanghai in 39 minutes. These two incredible concepts, space tourism and fast travelling, will help the economic development of these innovative companies and give them a strong presence in the space industry which will help them to bring more new solutions in the space business.

The motivation for this project has been the airplane-rocket concept by Reaction Engines. For this reason, the project studies two critical points of a vehicle which moves in the supersonic regime: its air intake and airfoil.

Firstly, a review of the compressible flow is made in order to put the reader in situation. Before to start making simulations with CFD, the equations which are used and phenomenology which will appear have to be understood in order to know the difficulties which we will have to handle and make the right simulation setup.

Secondly, an explanation of the CFD software, Nektar++, used to make the aerodynamic simulations is made. Through this explanation, some of the numerical methods which are used are briefly explained so as to the reader could have a better comprehension of what is computational fluid dynamics. In this section, the type of used methods, their advantage and disadvantages and the difference with other software are some of the topics which are presented.

Thirdly, ramjet-scramjet engine intake is studied, this is a type of engines which are used for vehicles which fly at supersonic-hypersonic regimes, respectively. In a ramjet engine, the incoming air is slowed down to subsonic speeds before combustion starts. A hypersonic flow is defined at velocities above Mach 5. At this regime of velocities, the flow starts to present some phenomena: very high temperatures, the chemical bounds of the particles start to broke, chemical reactions, thin shock layers and very high drag. Due to those facts, the simulation of this regime with the available time, tools and knowledge is impossible. Nevertheless, I decided to apply the geometry of an hypersonic intake in a supersonic regime, at a number of Mach of 2. Note that a study of the hypersonic intake in this regime makes sense due to the fact that the vehicle will go through it while accelerates to the desired hypersonic velocity, see Appendix 3.4.

Fourthly, the aerodynamic behaviour of a diamond shaped airfoil is studied. The use of a diamond shaped airfoil in the compressible regime avoids the detachment of the shock at the leading edge due to the sharp wedges. Then, I will study this airfoil in several configurations in order to determine its aerodynamic performance: lift and drag coefficients, efficiency and pressure and momentum coefficients.

Finally, once obtained the results from the simulation, an analysis of the aerodynamic coefficients and forces is presented.
CHAPTER 1. COMPRESSIBLE FLOW

1.1. Overview

First of all, the basic thing of this project is the fluid with I am going to work. For this reason, an explanation of the characteristics of a compressible flow and its implications in the numerical simulations is made in this chapter.

In the subsonic regime, the air flow is considered incompressible, therefore, the density remains constant along a streamline. However, the flow becomes compressible well below the supersonic regime. As the speed of sound is approached, shock waves start appearing in the transonic regime. Since then, all the characteristics of the fluid changes.

When we use the word compressible, we are instantly making reference to the change of density of something. Thus, density will not be constant so it will be a variable. In addition, a high-speed flow is a high-energy flow. Therefore, at the supersonic regime, energy transformations and temperature changes are important considerations.

1.1.1. Density

In real life, all the fluids are compressible. However, at small velocities, the assumption of incompressible flow holds. Therefore, it is possible to obtain results, using this assumption, with a high accuracy.

As velocity of an object through a fluid starts to increase, this assumption starts to fall. The Mach number, defined by Eq.1.1, allows us to know when we are at subsonic or supersonic regime.

\[ M = \frac{U}{a} \]  

assuming of isentropic pressure waves

\[ a = \sqrt{\gamma RT}. \]  

The different flow regimes are classified as

- \( M < 1 \) \( \rightarrow \) subsonic flow
- \( M = 1 \) \( \rightarrow \) sonic flow
- \( M < 1 \) \( \rightarrow \) supersonic flow

However, about a \( M > 0.3 \), the effects of compressibility start to show up. The compressibility of a fluid, \( \tau \), is defined as the relative variation of fluid’s volume when pressure varies. Depending on the process of compression, it could be defined as
\[ \tau = -\frac{1}{v} \left( \frac{dv}{dp} \right)_T \]  

(1.3)

Then, the previous equation depends on the temperature, it is the isothermal compressibility. Note that where we have a small element of volume \( v \) which is being compressed by the increase of pressure \( dp \) exerted by the fluid. Since the volume is reduced, \( \frac{dv}{dp} \) is negative.

In addition, if the compression is adiabatic and there are no dissipative forces, like viscosity. Then, compressibility is called the isentropic compression and it is defined as

\[ \tau = -\frac{1}{v} \left( \frac{dv}{dp} \right)_s \]  

(1.4)

As \( v \) is the specific volume (volume per unit mass), and the density \( \rho = \frac{1}{v} \). The final equation, in terms of density is

\[ \tau = \frac{1}{\rho} \frac{d\rho}{dp} \]  

(1.5)

hence

\[ d\rho = \rho \tau dp \]  

(1.6)

At this point, the fluid has been considered static, but the fluid is in motion. In particular, high-speed flows involve large pressure gradients, it causes that the fluid experiences great changes in density. Also, such pressure gradients create large velocity changes in the gas.

For most practical problems, it is considered that if the density changes by 5 percent or more, it will be a compressible flow.

**1.2. Integral Form of the Conservation Equations**

At this section, we are going to see the governing equations in the finite control volume that has been studied. These equations are used in their integral form, because when they are applied in certain important problems, algebraic equations can be extracted from them.

Due to this compressibility of air, thermodynamic variables of \( T \) and \( \rho \) are not constant. Now, we will need two more equations in order to solve these variables. Then, we must to solve: mass, linear momentum, energy and equation of state.
CHAPTER 1. COMPRESSIBLE FLOW

- **Continuity Equation**: Mass can be neither created nor destroyed. The net mass flow into the control volume must equal the rate of increase of mass inside the control volume. This statement is represented mathematically as:

\[
- \iiint_S \rho \mathbf{V} \cdot d\mathbf{S} = \frac{\partial}{\partial t} \iiint_V \rho dV
\]  
(1.7)

- **Momentum equation**: The time rate of change of momentum of the fluid that is flowing through the control volume at any instant is equal to the net force exerted on the fluid inside the volume. This statement is represented mathematically as:

\[
\iiint_S (\rho \mathbf{V} \cdot d\mathbf{S}) \mathbf{V} + \iiint_V \frac{\partial (\rho \mathbf{V})}{\partial t} d\mathbf{V} = \iiint_V \rho \mathbf{f} d\mathbf{V} - \iiint_S p d\mathbf{S} + F_{\text{viscous}}
\]  
(1.8)

- **Energy**: Energy can be neither created nor destroyed; it can only change in form. The rate of heat added to the fluid plus the rate of work done on the fluid is equal to the rate of change of energy of the fluid as it flows through the control volume. This statement is represented mathematically as:

\[
\iiint_V \dot{q} d\mathbf{V} - \iiint_S p \mathbf{V} \cdot d\mathbf{S} + \iiint_V \rho (\mathbf{f} \cdot \mathbf{V}) d\mathbf{V} = \\
\iiint_V \frac{\partial}{\partial t} \left[ \rho \left( e + \frac{V^2}{2} \right) \right] d\mathbf{V} + \iiint_S \rho \left( e + \frac{V^2}{2} \right) \mathbf{V} \cdot d\mathbf{S}
\]  
(1.9)

- **Equation of state**: This equation is applied to the fluids where the intermolecular forces are neglected.

\[
p = \rho RT
\]  
(1.10)

1.3. Differential Conservation Equations

Previously, we have used the integral form of the conservation equations in order to solve problems over a finite control volume. However, when in our flow is unsteady, it is necessary to apply the integral equations around a point of the flow. Then, these integral conservation equations become differential which perform the same physical principles as the integral ones.

This section is crucial, due to the software of computational fluid dynamics works over the discretization of these equations. For this reason, the differential form of the conservation equations have to be introduced:
• **Continuity Equation**: Equation (1.11) is the differential form of the continuity equation.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0 \tag{1.11}
\]

• **Momentum equation**: Equation (1.12) is the differential form of the momentum equation.

\[
\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u \mathbf{V}) = -\frac{\partial p}{\partial x} + \rho f_x \tag{1.12}
\]

where \( n \) indicates the direction of the momentum equation (x, y or z)

• **Energy**: Equation (1.13) is the differential form of the energy equation.

\[
\frac{\partial}{\partial t} \left[ \rho \left( e + \frac{V^2}{2} \right) \right] + \nabla \cdot \left[ \rho \left( e + \frac{V^2}{2} \right) \mathbf{V} \right] = -\nabla \cdot (p \mathbf{V}) + \rho \dot{q} + \rho (f \cdot \mathbf{V}) \tag{1.13}
\]

### 1.4. Shock Waves

At this section, we are going to deal with the physical phenomenon called shock wave. When our fluid overcomes the speed of sound, \( M > 1 \), this phenomenon becomes critical in the behaviour of the fluid around an object. Because, across this thin region, of about \( 10^{-5} \text{m} \) for air at standard conditions, the flow properties change drastically. Due to its relevance in the supersonic regime, I am going to explain along this section how this phenomenon is generated, what implications has on the fluid properties and the different types.

#### 1.4.1. Shock Waves Generation

When an airplane is flying at subsonic regime, all the fluid particles in front of it receives the “information” about the motion around the vehicle. This “information” is transmitted by pressure waves that travel at the speed of sound.

When we reach supersonic velocities, the fluid particles do not receive this “information” due to the velocity of the pressure waves is less than the velocity of the vehicle. Consequently, the fluid suffers a violent displacement that causes great changes in pressure, density and temperature. This phenomenon is called **shock wave**.

For this reason, when we talk about compressible flow, it is necessarily to talk carefully about shock waves. As it has been previously stated, a shock wave
is a violent disturbance of the fluid particles which causes a discontinuity in the physical quantities \((P,T,\rho,u\ldots)\).

We have previously mentioned the transport of “information”, we will go deeper with this concept. What is really going on is that molecules interact with the surface and they are reflected. Then, those molecules that are rebounded from the body travel forward, with a speed greater than the vehicle, and they interact with the other molecules from the fluid. This is how pressure waves or, in this case, sound waves are transmitted in a fluid. Consequently, the fluid in front of the body is being warned by the sound waves.

These pressure waves are similar to cars break lights in a highway. When a car slows down, the one behind it will respond and so on. In this situation, there is a transmission of information between the cars: when one slows down, the following car will do the same. Therefore, this passing of information is exactly what molecules do in order to know how to move along the way.

However, the situation explained before is for the subsonic regime. In our case, when the air reaches supersonic speed, the vehicle moves faster than the speed of sound; therefore, it has a velocity greater than the pressure waves. Consequently, pressure waves are not able to warn (pass the information) the other molecules of the fluid. For this reason, a big deceleration of the air comes about and a shock wave is produced.

### 1.4.2. Pressure Waves

In order to understand better how pressure waves propagate, some simple diagrams of Fig.1.1 will be used.

![Pressure waves in a stationary object](image1.png)

![Pressure waves in subsonic regime](image2.png)

![Pressure waves in supersonic regime](image3.png)

**Figure 1.1** – Pressure waves in a stationary, subsonic and supersonic motion.

Firstly, in a stationary object, pressure waves will propagate outwards, in all directions, as shown in Fig. 1.1(a).

Secondly, if the object starts to move forward, pressure waves will be compressed at the front and expanded in the back, as shown in Fig.1.1(b). The result is an increase in frequency of the sound ahead of the object and a decrease behind it. This phenomenon is known as a **Doppler Shift**.

Finally, if the object overcomes the speed of sound, it will out run the pressure
waves which it has generated. If we draw a tangent line between the pressure waves like in the Fig.1.1(c), it will form a cone, the Mach Cone, which represents the shock wave.

This line of disturbances is defined as Mach wave. The Mach angle, \( \mu \) defined at Eq.1.15, is computed from the geometry of Fig.1.1(c):

\[
\sin \mu = \frac{at}{Vt} = \frac{a}{V} = \frac{1}{M}
\]  

(1.14)

hence

\[
\mu = \sin^{-1} \frac{1}{M}
\]  

(1.15)

An observer situated in front of the object will not hear anything due to he will be outside the Mach Cone. This region is called the Zone of Silence. However, if the observer is situated inside the Mach Cone, he will hear the object. This is called the Zone of Action.

### 1.4.3. Types of Shock Waves

Furthermore, a shock wave is an instantly non-isentropic process which increases static pressure, density and temperature. In addition, these changes in the flow properties are irreversible and the entropy of the system increases. Moreover, the total enthalpy and the total temperature are constant due to a shock wave does no work.

However, the flow is non-isentropic. Then, the total pressure downstream of the shock is always less than the total pressure upstream of the shock. Because total pressure changes across the shock, we can not use the incompressible form of Bernoulli’s equation across the shock. Furthermore, the Mach number and speed of the flow also decrease across a shock wave.

Depending on the geometry of the object where the shock wave is generated, it could be different types:

- **Normal shock**: It is produced when the flow is not turned and then, it is perpendicular to the surface. Normal shocks waves could be attached or not. As we stated before and as you can see in Fig.1.2, the flow properties change: Mach and velocity are reduced; despite of density, pressure and temperature that increase.
The mathematical relations which we need to work with normal shocks, which came from the integral form of the conservation equations, are:

\[
M_2^2 = \frac{1 + \left[\frac{\gamma - 1}{2}\right] M_1^2}{\gamma M_1^2 - \left(\frac{\gamma - 1}{2}\right)} < M_1^2 \tag{1.16}
\]

\[
\frac{\rho_2}{\rho_1} = \frac{(\gamma + 1) M_1^2}{2 + (\gamma - 1) M_1^2} > 1 \tag{1.17}
\]

\[
\frac{p_2}{p_1} = 1 + \frac{2\gamma}{\gamma + 1} (M_1^2 - 1) > 1 \tag{1.18}
\]

\[
\frac{T_2}{T_1} = \left[1 + \frac{2\gamma}{\gamma + 1} (M_1^2 - 1)\right] \frac{2 + (\gamma - 1) M_1^2}{(\gamma + 1) M_1^2} > 1 \tag{1.19}
\]

- **Oblique shock**: It is produced when is inclined to the flow direction due to the geometry of the object. Oblique shocks waves could be attached shocks or not. In addition, the number of Mach decreases and the pressure, density and temperature increase, as shown in Fig.1.3. The changes that the flow suffers across an oblique shock are a function of two quantities, \( M_1 \) and \( \beta \).

In order to obtain \( M_2 \), deflection angle \( \theta \) has to be obtained. The deflection angle is obtained by using the following formula:

\[
\tan \theta = 2 \cot \beta \left\lfloor \frac{M_1^2 \sin^2 \beta - 1}{M_1^2 (\gamma + \cos 2\beta) + 2} \right\rfloor \tag{1.20}
\]

This equation is called the \( \theta-\beta-M \) relation. In Fig.1.4 you can see the results of it plotted, for \( \gamma = 1.4 \).
If we look at the results, we can see that there are two solutions for a given Mach number. These two possibilities are the weak shock or strong shock. The first one is more often in nature, but under some conditions the second one is possible. The fact which determine when appears a solution or another is the backpressure. If something increases the pressure downstream, the strong solution could be forced to occur. The big difference between these two solutions is the number of Mach downstream, $M_2$:

- **Weak shock**: $M_2$ is supersonic
- **Strong shock**: $M_2$ is subsonic.

To sum up, regarding at Fig.1.4, we can see the trends of the oblique shocks.

For the **weak solution**, as $\theta$ is increased with a fixed Mach number: $\beta$, $p_2$, $T_2$ and $\rho_2$ increase while $M_2$ decreases. However, if we fix $\theta$ while $M$ increases: $p_2$, $T_2$, $M_2$ and $\rho_2$ increase while $\beta$ decreases.

For the **strong solution**, on the one hand, as $\theta$ is increased with a fixed Mach number: $\beta$ decreases while $p_2$, $T_2$, $M_2$ and $\rho_2$ increase. On the other hand, if we fix $\theta$ while $M$ increases: $\beta$, $p_2$, $T_2$, and $\rho_2$ increase while $M_2$ decreases.

- **Bow Shock**: Also, called detached shock wave, it is produced in front of blunt
objects and is a normal shock wave followed by a continuum of oblique shocks. As you can see in Fig. 1.5, from point $a$ to $c'$ the flow behind the shock is subsonic and above point $c'$ it is supersonic. Hence, the flow field after the curved shock wave is a mixed subsonic-supersonic flow.

Additionally, they are difficult to study, because, they are very sensitive to the geometry of the body. The reason why this type of shock wave is produced in blunt objects is due to the fact that the needed rotation of the fluid exceeds the maximum achievable rotation angle for an attached shock. After it, the flow is subsonic.

In addition, this shock increases highly the drag. For this reason, supersonic aircrafts have a straights shapes. However, the return capsules have blunt shapes, in order to take benefit from this property and slow down during the atmospheric re-entry.

- **Expansions**: This phenomenon, also called Prandtl-Meyer expansion wave, is produced when there is an abrupt flow area increase, as shown in Fig. 1.6. As despite a shock wave which across it Mach number decreases, static pressure increases and there is loss of total pressure because the process is irreversible. Through an expansion wave: Mach number increases while pressure, density and temperature decreases, as shown in Fig. 1.6. However, total pressure remains constant because is an isentropic process.

In order to make calculations about the expansions, it is necessarily the use of the Prandtl-Meyer function. This function is derived from conservation of mass, momentum, and energy for very small (differential) deflections. Then, the equation which describes the flow inside the expansion wave is:

\[
d\theta = \sqrt{M^2 - 1} \frac{dV}{V} \quad (1.21)
\]
12 Simulation of an hypersonic gas turbine ramjet engine intake in the supersonic regime

In order to analyze the entire Prandtl-Meyer expansion, it has to be integrated over the complete angle \( \theta \).

After the integration, we could find the Prandtl-Meyer function:

\[
\nu(M) = \sqrt{\frac{y+1}{y-1}} \tan^{-1} \left(\sqrt{\frac{y-1}{y+1}} \frac{M^2 - 1}{\tan^{-1} \sqrt{M^2 - 1}}\right)
\]  

(1.22)

1.5. Summary

Firstly, in this chapter, we have seen that the compressibility of a fluid, \( \tau \), is defined as the relative variation of fluid’s volume when pressure varies and starts to show up at \( M > 0.3 \). In addition, it enters at supersonic regime at \( M > 1 \). At this point, the fluid becomes a high-speed flow; therefore, it will be a high-energy flow. Then, the thermodynamics have to be taken into account in this regime, see Reference [8].

Secondly, the study of the control volume using the integral form of the conservation equations (mass, momentum and energy) has allowed us to analyze the flow and extract powerful algebraic relationships. Then, the discrete form of the conservation equations is presented, because they are the equations that the CFD software will solve in the numerical simulations.

Finally, after some manipulations of them, mathematical relations which describe the physical phenomenon of shock waves could be extracted. It is well known that the study of this phenomenon is a must done when we are studying the supersonic regime. Understanding the types of shock waves and their physical implications is crucial in the design of any geometry which works in this regime. In order to know more about compressible flow, see Reference [9].
CHAPTER 2. NEKTAR++

2.1. Overview

Nektar++ [10] is a tensor product based finite element package developed at the Imperial College of London, UK.

Before to continue, we will see where the method of this software come so as to understand better the advantages of using Nektar++ in front of other more extended solutions like: OpenFoam, ANSYS, Fluent...

Along the following subsections, the spectral/hp element method used by Nektar++ is going to be analyzed. So, with this overview, a brief and clear explanation of the method is made. Then, the reader could have an idea of the properties and the utilities of it.

2.2. FEM methods

First of all, finite element method (FEM) is a numerical method widely used in the field of solid and fluid mechanics, see References [11], [12] and [13].

FEM is a discretization technique used to solve partial differential equations (PDE’s) in its equivalent variational form. This means that the partial derivative terms are replaced with discrete algebraic difference quotients involving the flow field variables at discrete grid points, also called "nodes". FEM are used because these PDE’s cannot be solved analytically, so numerical methods are used in order to bring an approximate solution.

The classical approach of the finite element method is to divide the computational domain into a mesh of many small subdomains, called finite elements. Along the grid, the flow properties are predicted using Taylor’s series. As shown in Fig. 2.1, if we know, for example, the velocity at one node \( (u_{i,j}) \), we could calculate by Taylor expansion the velocity value at the following point, \( u_{i+1,j} \). The accuracy of the obtained value will depend on where we truncate in Taylor’s expansion.

Figure 2.1 – Rectangular grid.
Nevertheless, the accuracy of the obtained values not only depend on the order of Taylor’s polynomial. It also depends on the refinement of the mesh. In other words, as small the distances, $\Delta x$ and $\Delta y$ shown in Fig. 2.1, between the nodes are, a better accuracy we will obtain.

Nektar++ use high order FEMs, this means that it uses high-order polynomials to approximate the solution. The degree of the method essentially refers to the accuracy of the approximation. There are two types:

- **h-version FEM**: Thus type of high order FEM fixes the degree $P$ of the piecewise polynomial (Taylor expansion) basis functions and any change of discretization to enhance accuracy is done by means of mesh refinement.

- **p-version FEM**: Thus type of high order FEM fixes the size of the mesh and any change of discretization to enhance accuracy is done by changing the degree of $P$.

Taken this into account, Nektar++ works with the hp-version FEM. Thus, both ideas of mesh refinement and degree enhancement are combined.

### 2.3. Spectral method

Secondly, Nektar ++ is also based, in the spectral method which is a powerful tool in order to solve partial differential equations, see References [14]. This method, presented by Gottlieb and Orszag (1977), is convenient to use when the solution varies considerably in time or space, when a very high spatial resolution is required, and also when long time integration is needed.

Spectral methods appear with the necessity to speed up the computation time needed to solve certain problems and improve the accuracy.

In order to improve accuracy, spectral methods, as despite FEM methods, do not compute the derivatives of one point using the information of the "neighbours" points. As we see in Fig. 2.1, we compute the state at $u_{i+1,j}$ using the information of the previous point $u_{i,j}$. So, FEM methods are relying on local information, they are a *local approach*. However, spectral methods use all the available information, in the domain, in order to compute the derivatives. So, they are a *global approach*. This fact makes that spectral methods converged exponentially, which makes them more accurate than local methods.

### 2.4. Spectral element method

Thirdly, Nektar++ goes far away and uses a spectral element method. It was presented by Patera and combines the high accuracy of the spectral methods with the geometric flexibility of the finite element. Then, this combination makes the spectral element method conceptually similar to the above mentioned high-order finite element, but it adds the use of the nodal expansion basis, see Reference [15].
Spectral element method uses the philosophy of the global approach of spectral methods, but here the domain is the element of the grid (it is one of the grid squares which you can see at Fig. 2.1), because the global approach of the domain only can be done efficiently in very simple geometries. This method uses the Lagrange polynomials through the zeros of the Gauss-Lobatto-Legendre polynomials.

Before to continue, Lagrange polynomials and Gauss-Lobatto-Legendre polynomial in order to have a better concept of the method are briefly explained.

2.4.1. Lagrange polynomials

As we have discrete data, we want to know the value, or approximate it, of the intermediate values. In order to have a smooth function which represents the problem in the best way as possible; this is called data fitting, see Reference [16].

You can see this fact in Fig. 2.2, where the blue line represents the ideal function that fits all the data points and the orange line shows a great approximation to it using the Lagrange polynomials.

Lagrange polynomials allow us to interpolate in order to know these values. Keep in mind the grid of the Fig. 2.1, here we have our problem represented by discrete points. So, using the Lagrange polynomials we could obtain a smooth representation of the problem by the use of interpolation.

This polynomial is represented by the Eq. 2.1

\[ p_n = \sum_{k=0}^{n} y_k L_{n,k}(x) \]  

(2.1)
where $L$ is the Lagrange coefficient which has the property given by Eq. 2.2

\[ L_{n,k}(x_j) = \begin{cases} 
1 & \text{if } j = k \\
0 & \text{otherwise } j \neq k 
\end{cases} \tag{2.2} \]

### 2.4.2. Gauss-Lobatto-Legendre polynomials

These polynomials are a type of Jacobi polynomials. Probably, you have noted that the Lagrange polynomials do not fit well at the boundaries of the domain. In order to avoid these oscillations, Nektar++ uses the zeros of the Gauss-Lobatto-Legendre polynomials. This means that the integration includes the extreme points of the domain.

In addition, using these polynomials in each element we could have an exact equation until polynomials of order $2n$, where $n$ is the number of integration points. As despite other quadrature techniques which only arrive at polynomials of order $n$ and give less accuracy.

### 2.5. Spectral hp/element method

Finally, Nektar++ works with all methods mentioned above, the spectral/hp element method, as its name suggests, incorporates both the multidomain spectral methods as well as the more general high-order finite element methods.

### 2.6. Summary

Firstly, at this chapter, we have seen how Nektar++ use the combination of two high order methods, hp-version FEM, in order to make the discretization of the partial differential equations.

Secondly, the differential aspect of Nektar++ is the combination of the technique mentioned before with the spectral element method. This method implies a high accuracy without using fine meshes, so the possibility to speed up the simulations.

To sum up, the methods which use Nektar++ have allowed me to make numerical simulations in fine meshes and achieve good times of computation. In Appendix 3.4. you can see a brief explanation of the compressible solver used by Nektar++. However, if the reader wants more information about computational fluid dynamics, see Reference [17].
CHAPTER 3. SUPersonic profile simulation

3.1. Overview

In this section, Nektar++’s Compressible Flow Solver is used in order to make an aerodynamic study of a supersonic profile. The setup of the simulation is the same as the supersonic intake, see Appendix 3.4.

3.2. Geometry

First of all, the used geometry is presented. In this section, the dimensions of the geometry and the mesh are presented. You can see the used mesh in Fig.3.1. For this project, the mesh varies in function of the studied problem. For example, the increase of the angle of attack or the variation of the airfoil thickness needs that the zones of the mesh which capture the shock waves, have to be meshed fine in order to capture well the phenomenology and change their inclination in order to follow the generated shock wave.

Figure 3.1 – Diamond Airfoil scheme.

In Fig.3.2, you can see a scheme of the airfoil with its dimensions.
I am going to study a "Diamond Airfoil", Fig.3.3, this type of airfoil avoids the detachment of the shock at the leading edge due to the sharp wedges. Thanks to that, it eliminates the area of high pressure; then, there is much less drag, see Reference [18].

Through the variation of the angle of attack and the angle of the profile, I have studied how these changes affect the aerodynamic performance. A study of the influence of these changes, in the simulation results, is presented in the following sections.

**3.3. Results**

In this section, the obtained results from the simulations are presented. In order to extract the values, it is necessary to wait until the simulations achieved the convergence or a partial convergence. These simulations need from several days in order to have stabilized values.
Firstly, the supersonic profile is studied at different angles of attack (AoA) and Mach numbers. From the simulations, it has been extracted values of pressure, Mach, density and flow velocity. Then, the obtained values are shown in graphics and then compared with the predicted values in a table.

You can see how these values are obtained and, an explanation of the relative angles which sees the flow along the airfoil in Appendix 3.4. In addition, in Section 3.3.1. in each case, is only available the visualization of the Mach field (ParaView[19]) and the graphics (Octave) which show the distribution of the flow properties are only available in the cases of 0° and 2.5°, due to the other cases do not bring any different from them. However, if the reader wants to see all the cases, they are available in Appendix 3.4.

Secondly, aerodynamic changes due to the variation of the airfoil thickness are studied in Appendix 3.4.

Finally, the diamond airfoil is studied at a low Mach number is studied in Appendix 3.4.

3.3.1. Varying AoA

Maintaining the supersonic profile configuration stated in section 3.4., it has been studied with different angles of attack at Mach 2.

Before to continue, graphics are explained in order to help the reader with the presented data. Then, in the studied case, it will present different data on each face of the airfoil. In the graphics, you will see this data in blocks. Then, in Fig. 3.4 you can see how the faces are numbered.

![Figure 3.4](image)

**Figure 3.4** – Supersonic profile with the faces labeled.

For the following simulations, the pressure will be distributed as:

- The pressure in the zone 1 will be the upper forward pressure $P_{1UP}^P$
- The pressure in the zone 2 will be the upper back pressure $P_{3UP}^P$
• The pressure in the zone 3 will be the lower back pressure $P_{DOWN}^3$

• The pressure in the zone 4 will be the lower forward pressure $P_{DOWN}^2$

$\alpha=0^\circ$

With an angle of attack of $0^\circ$, we have the condition of symmetry. Therefore, the value of the studied variables will be the same at the upper and lower surfaces. It could be seen at Fig.3.5

![Visualization of Mach field at Mach 2 and AoA=0°.](image)

In addition, in Fig.3.6 you can see how block data is distributed, from left to right you will see the distribution of a variable over the four surfaces of the airfoil along the chord. All the following graphics will have the same data distribution: black circles for the variables in the upper surface and red crosses for the lower surface.

As we can see in Fig.3.6, all the presented variables (pressure, Mach, density and flow velocity) have the same value in the upper and lower surface. So, the zone 1 and 4 have equal conditions, the same with zone 2 and 3. Therefore, this symmetry condition of the variables produces "zero" lift.

As theory predicts, the data obtained from the simulation shows:

**After the shock wave:**
- Pressure increases $P_{\infty} = 101325 \rightarrow P_{UP}^2 = 133280$
- Density increases after a shock wave $\rho_{\infty} = 1.225 \rightarrow \rho_{UP}^2 = 1.49$
- Mach decreases after a shock wave $M_{\infty} = 2 \rightarrow M_{UP}^2 = 1.82$

**After the expansion wave:**
- Pressure decreases $P_{UP}^2 = 133280 \rightarrow P_{UP}^3 = 75767.75$
- Density decreases $\rho_{UP}^2 = 1.49 \rightarrow \rho_{UP}^3 = 0.99$
- Mach increases $M_{UP}^2 = 1.82 \rightarrow M_{UP}^3 = 2.17$
Figure 3.6 – Pressure, Mach, Density and Specific linear momentum distribution at $M=2$ and $AoA=0^\circ$.

In the following table, Table 3.1, theoretical and simulation result values are presented, in the columns “Predicted Data” and “Simulation Data”, respectively. Concretely, pressure (Pa), Mach number, density and specific linear momentum are computed in each surface of the airfoil. In order to compute the values obtained by the simulation, an average of the variables along each surface is made.

In this table, theoretical and simulation values in each surface of the airfoil are compared and the relative error ($\epsilon_r$) is computed in percentage.
Table 3.1 – Table M=2 and AoA=0°.

Once the different variables obtained from the simulation are presented in the previous table, the aerodynamic forces acting on the airfoil are computed using theoretical and simulation result values. Concretely, in the following table, Table 3.2, the resulting aerodynamic forces are presented, also you could notice three columns. The data of each column is obtained by:

- **Predicted Data**: In this column, the value of the forces are the theoretical values and are predicted by the theoretical pressures in each face. These pressures are the pressures in the horizontal ($P_x$) and vertical direction ($P_y$). Then, the aerodynamic coefficients are obtained from them. The used equations are:

$$
\begin{align*}
D &= F_x \cos \alpha + F_y \sin \alpha \\
L &= -F_x \sin \alpha + F_y \cos \alpha
\end{align*}
$$

$$
\begin{align*}
C_D &= \left( \Sigma P_x \cos \alpha \cos \theta + \Sigma P_y \sin \alpha \cos \theta \right) + \left( \Sigma P_y \cos \alpha \cos \theta \right) \\
C_L &= \left( -\Sigma P_x \sin \alpha \cos \theta + \Sigma P_y \sin \alpha \cos \theta \right) + \left( \Sigma P_y \cos \alpha \cos \theta \right)
\end{align*}
$$

- **Approximated Data**: In this column, the value of the forces are predicted by the average pressures along each face of the airfoil which is obtained from the simulation. The used equations are the same as mentioned before.

- **Real Data**: In this column, the value of the forces (Drag and Lift) are obtained directly from the simulation. In order to obtain them, "Aeroforces filter" provided by Nektar++ is used. Concretely, these forces are obtained
from the integration of the pressure above the surface. Finally, in order to obtain the aerodynamic coefficients the following equations were used:

\[
\begin{align*}
C_D &= \frac{2 \cdot Drag}{c \cdot P_\infty \cdot \gamma \cdot M_\infty^2} \\
C_L &= \frac{2 \cdot Lift}{c \cdot P_\infty \cdot \gamma \cdot M_\infty^2}
\end{align*}
\] (3.3)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
<th>Real Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag(N)</td>
<td>10025.57</td>
<td>9998.50</td>
<td>9949.53</td>
</tr>
<tr>
<td>Lift(N)</td>
<td>0</td>
<td>0</td>
<td>-153.11</td>
</tr>
<tr>
<td>CD</td>
<td>0.0177</td>
<td>0.0177</td>
<td>0.0176</td>
</tr>
<tr>
<td>CL</td>
<td>0</td>
<td>0</td>
<td>-0.0003</td>
</tr>
</tbody>
</table>

Table 3.2 – Aeroforces table M=2 AoA=0°.

As we stated before and we can see in Table 3.2, this configuration produces no Lift, but there is Drag. This fact could be explained by the pressure differences. In order to produce lift in an airfoil, the pressures in the upper and lower surface have not to be equal. Concretely, the pressure on the lower surface has to be bigger. In Appendix 3.4, a mathematical explanation of how aerodynamic forces are created is presented.

- $\alpha=2.5^\circ$

When the angle of attack is increased, the condition of symmetry does not hold more. This increase of the angle is made by changing the boundary conditions, concretely, the velocity direction rather than inclining the geometry and fixing the flow direction.

In this case, if we see at Fig.3.3, it will appear a dull shock wave at the upper surface and an intense oblique shock wave at the lower surface. Therefore, the airflow at the top of the wing has less pressure than the lower surface, so the
flow is faster at the top than on the lower surface. This fact, cause a positive net force which is the lift force. Then, this phenomenon could be seen in the following results. In Fig.3.7, you can see the case at 2.5°.

In Table 3.3, the values of the flow properties are not more the same in the upper and lower surface. In addition, it shows how the flow is faster on the lower surface than in the upper, as we see also in Fig.3.8. In addition, the difference in velocities between the surfaces which causes the zone of high pressure in the lower surface and a zone of low pressure at the upper surface.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data UP</th>
<th>Predicted Data DOWN</th>
<th>Simulation Data UP</th>
<th>Simulation Data DOWN</th>
<th>Relative Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\beta)</td>
<td>34.57°</td>
<td>34.21°</td>
<td>34.85°</td>
<td>34.13°</td>
<td>0.810</td>
</tr>
<tr>
<td>(P_2(\text{Pa}))</td>
<td>116435.19</td>
<td>152038.71</td>
<td>116470</td>
<td>152050</td>
<td>0.0299</td>
</tr>
<tr>
<td>(M_2)</td>
<td>1.91</td>
<td>1.73</td>
<td>1.91</td>
<td>1.72</td>
<td>0.578</td>
</tr>
<tr>
<td>(\rho_2\ (\text{kg/m}^3))</td>
<td>1.35</td>
<td>1.63</td>
<td>1.35</td>
<td>1.63</td>
<td>0</td>
</tr>
<tr>
<td>(\rho u_2\ (\text{kg/m}^2s))</td>
<td>897.01</td>
<td>1021.26</td>
<td>893.07</td>
<td>1010.6</td>
<td>0.439</td>
</tr>
<tr>
<td>(P_3(\text{Pa}))</td>
<td>64996.72</td>
<td>87907.72</td>
<td>65031</td>
<td>87916</td>
<td>0.0527</td>
</tr>
<tr>
<td>(M_3)</td>
<td>2.28</td>
<td>2.09</td>
<td>2.27</td>
<td>2.07</td>
<td>0.439</td>
</tr>
<tr>
<td>(\rho_3\ (\text{kg/m}^3))</td>
<td>0.89</td>
<td>1.10</td>
<td>0.89</td>
<td>1.09</td>
<td>0</td>
</tr>
<tr>
<td>(\rho u_3\ (\text{kg/m}^2s))</td>
<td>650.71</td>
<td>769.42</td>
<td>641.04</td>
<td>754.85</td>
<td>1.486</td>
</tr>
</tbody>
</table>

**Table 3.3** – Table \(M=2\) AoA=2.5°.

In the following table, Table 3.4, you can see the aerodynamic forces acting on the body and its aerodynamic coefficients for this configuration. Note how pressure differences have created a net lift force, so the lift and \(C_L\) are not null.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
<th>Real Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\text{Drag}(N))</td>
<td>12605.61</td>
<td>12603.77</td>
<td>12500.231</td>
</tr>
<tr>
<td>(\text{Lift}(N))</td>
<td>57797.02</td>
<td>57747.72</td>
<td>57380.743</td>
</tr>
<tr>
<td>(C_D)</td>
<td>0.0223</td>
<td>0.0223</td>
<td>0.0221</td>
</tr>
<tr>
<td>(C_L)</td>
<td>0.1022</td>
<td>0.1022</td>
<td>0.1015</td>
</tr>
</tbody>
</table>

**Table 3.4** – Aeroforces table \(M=2\) AoA=2.5°.
In this case, as it could be seen in Fig. 3.9, the deflection angle which sees the airflow when it arrives at the upper surface is $\theta - \alpha$. So, the shock wave generated will depend on this resulting angle.

Therefore, with $5^\circ$ of AoA the Zone 1 in the upper surface will experience a flow deviation of $0^\circ$, so this will be a Flat Plate case. This could be visualized in Fig. 3.10.
Figure 3.9 – Deflection angle explanation.

Figure 3.10 – Visualization of Mach field at Mach 2 and AoA=5°.

In the following table, Table 3.5, we can see the results for each surface of the airfoil. Note how in front of the upper surface the values are the same as the far field conditions, this proves that the flow has any deviation and any shock wave has been generated. Also, at the lower back surface the values are close to the far field conditions.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Simulation Data</th>
<th>Relative Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>UP</td>
<td>DOWN</td>
<td>UP</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Flat Plate</td>
<td>34.31°</td>
<td>Flat Plate</td>
</tr>
<tr>
<td>$P_2$ ($\text{Pa}$)</td>
<td>101325</td>
<td>172914.77</td>
<td>101330</td>
</tr>
<tr>
<td>$M_2$</td>
<td>2</td>
<td>1.64</td>
<td>1.99</td>
</tr>
<tr>
<td>$\rho_2 \left( \frac{\text{kg}}{\text{m}^3} \right)$</td>
<td>1.225</td>
<td>1.79</td>
<td>1.225</td>
</tr>
<tr>
<td>$\rho u_2 \left( \frac{\text{kg}}{\text{m}^2 \cdot s} \right)$</td>
<td>833.72</td>
<td>1078.90</td>
<td>830.55</td>
</tr>
<tr>
<td>$P_3$ ($\text{Pa}$)</td>
<td>55522.68</td>
<td>101613.91</td>
<td>55536</td>
</tr>
<tr>
<td>$M_3$</td>
<td>2.38</td>
<td>1.99</td>
<td>2.37</td>
</tr>
<tr>
<td>$\rho_3 \left( \frac{\text{kg}}{\text{m}^3} \right)$</td>
<td>0.80</td>
<td>1.22</td>
<td>0.79</td>
</tr>
<tr>
<td>$\rho u_3 \left( \frac{\text{kg}}{\text{m}^2 \cdot s} \right)$</td>
<td>593.65</td>
<td>829.01</td>
<td>585.87</td>
</tr>
</tbody>
</table>

**Table 3.5** – Table M=2 AoA=5°.

In addition, in Table 3.6, we can see the aerodynamic forces acting on the airfoil for this configuration.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
<th>Real Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{Drag}(\text{N})$</td>
<td>20384.92</td>
<td>20378.31</td>
<td>20228.7</td>
</tr>
<tr>
<td>$\text{Lift}(\text{N})$</td>
<td>115897.55</td>
<td>115851.13</td>
<td>115273.38</td>
</tr>
<tr>
<td>$C_D$</td>
<td>0.0360</td>
<td>0.0361</td>
<td>0.0358</td>
</tr>
<tr>
<td>$C_L$</td>
<td>0.2050</td>
<td>0.2050</td>
<td>0.2039</td>
</tr>
</tbody>
</table>

**Table 3.6** – Aeroforces table M=2 AoA=5°.

• $\alpha=7.5^\circ$

At this point, when the flow arrives at the airfoil, it experiences an expansion wave at the upper surface, because it is turned away. It happens because the angle of attack is greater than the deflection angle of the airfoil. The flow will experience a Prandtl-Meyer expansion with an angle of $\epsilon = 2.5^\circ$: $\alpha - \theta = \epsilon$. You can see a scheme in Fig.3.11

The $\epsilon$ angle is obtained by:

\[
\begin{align*}
\eta &= 180^\circ - 90^\circ - \alpha \\
\eta &= 90^\circ - \alpha \\
\lambda + \delta + \eta &= 180^\circ \\
\lambda &= 180^\circ - \delta - \eta = 180^\circ - \delta - (90 - \alpha) \\
\lambda &= 90^\circ + \alpha - \delta \\
\lambda &= 90^\circ + \alpha - \delta
\end{align*}
\]
For the following angles of attack at the top surface, the flow will experience an expansion wave, at the upper surface rather than an oblique shock, when it arrives at the airfoil (Zone 1) followed with another expansion in the upper back surface (Zone 2). As you can see in Fig. 3.12.

\[
\begin{align*}
\phi &= 180^\circ - 90^\circ - \lambda \\
\phi &= 180^\circ - 90^\circ - (90^\circ + \alpha - \delta) \\
\phi &= -\alpha + \beta \\
90^\circ &= \phi + 2\theta + \epsilon \\
\epsilon &= 90^\circ - \phi - 2\theta \\
\epsilon &= \alpha - \delta - 2\theta \\
\epsilon &= 90^\circ + \alpha - \delta - 2\theta \\
\epsilon &= \alpha - \theta
\end{align*}
\]
In the following table, Table 3.7, you can see the obtained simulation values and their comparison with the theoretical ones for this configuration.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Simulation Data</th>
<th>Relative Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\beta$</td>
<td>$33.56^\circ$</td>
<td>$34.67^\circ$</td>
<td>$33.63^\circ$ $34.50^\circ$</td>
</tr>
<tr>
<td>$P_2$(Pa)</td>
<td>87806.98</td>
<td>196210.22</td>
<td>87807 $196170$</td>
</tr>
<tr>
<td>$M_2$</td>
<td>2.09</td>
<td>1.55</td>
<td>2.09 $1.54$</td>
</tr>
<tr>
<td>$\rho_2$ ($\frac{kg}{m^3}$)</td>
<td>1.11</td>
<td>1.95</td>
<td>1.11 $1.94$</td>
</tr>
<tr>
<td>$\rho u_2$ ($\frac{kg}{m^2*s}$)</td>
<td>771.29</td>
<td>1130.72</td>
<td>768.09 $1115.9$</td>
</tr>
<tr>
<td>$P_3$(Pa)</td>
<td>47147.87</td>
<td>116976.26</td>
<td>47262 $117000$</td>
</tr>
<tr>
<td>$M_3$</td>
<td>2.49</td>
<td>1.89</td>
<td>2.47 $1.87$</td>
</tr>
<tr>
<td>$\rho_3$ ($\frac{kg}{m^3}$)</td>
<td>0.71</td>
<td>1.35</td>
<td>0.71 $1.33$</td>
</tr>
<tr>
<td>$\rho u_3$ ($\frac{kg}{m^2*s}$)</td>
<td>538.71</td>
<td>886.69</td>
<td>532.65 $869.30$</td>
</tr>
</tbody>
</table>

| Table 3.7 – Table M=2 AoA=7.5°. |

In the following table, Table 3.8, you can see the aerodynamic forces acting on the airfoil and the aerodynamic coefficients in this configuration.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
<th>Real Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag(N)</td>
<td>33535.34</td>
<td>33502.97</td>
<td>33308.33</td>
</tr>
<tr>
<td>Lift(N)</td>
<td>174670.49</td>
<td>174543.50</td>
<td>173867.35</td>
</tr>
<tr>
<td>$C_D$</td>
<td>0.0593</td>
<td>0.0593</td>
<td>0.0589</td>
</tr>
<tr>
<td>$C_L$</td>
<td>0.3090</td>
<td>0.3088</td>
<td>0.3076</td>
</tr>
</tbody>
</table>

| Table 3.8 – Aeroforces table M=2 AoA=7.5°. |
• $\alpha = 10^\circ$

In the following figure, Fig. 3.13, the Mach field of the $10^\circ$ case is visualized.

![Mach field visualization](image)

**Figure 3.13** – Visualization of Mach field at Mach 2 and AoA=10°

It has to be noted that achieve this simulation has not been easy, several tries have been made in order to achieve a reasonable solution. You can see how the simulation has not totally converged, due to the lack of time, because other cases had to be studied. In addition, you can see dark zones in the figure, they are NaN values, it tells us that this simulation needs to be meshed better in this zone or a reduction of the step time is needed. Then, this state of the simulation has been used because the values were near the theoretical ones.

In Table 3.9, you can see the obtained values from the simulation which have not got a big divergence from the theory, the relative error is not big.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Simulation Data</th>
<th>Relative Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\beta$</td>
<td>UP 32.16°</td>
<td>DOWN 35.35°</td>
<td>UP 33.09°</td>
</tr>
<tr>
<td>$P_2 (Pa)$</td>
<td>75723.68</td>
<td>222373.09</td>
<td>75644</td>
</tr>
<tr>
<td>$M_2$</td>
<td>2.19</td>
<td>1.45</td>
<td>2.18</td>
</tr>
<tr>
<td>$\rho_2 \left(\frac{kg}{m^3}\right)$</td>
<td>0.99</td>
<td>2.12</td>
<td>0.99</td>
</tr>
<tr>
<td>$\rho u_2 \left(\frac{kg}{m^2 \cdot s}\right)$</td>
<td>710.12</td>
<td>1173.94</td>
<td>705.20</td>
</tr>
<tr>
<td>$P_3 (Pa)$</td>
<td>39813.65</td>
<td>134306.42</td>
<td>39881</td>
</tr>
<tr>
<td>$M_3$</td>
<td>2.60</td>
<td>1.79</td>
<td>2.58</td>
</tr>
<tr>
<td>$\rho_3 \left(\frac{kg}{m^3}\right)$</td>
<td>0.63</td>
<td>1.48</td>
<td>0.62</td>
</tr>
<tr>
<td>$\rho u_3 \left(\frac{kg}{m^2 \cdot s}\right)$</td>
<td>486.42</td>
<td>941.18</td>
<td>479.22</td>
</tr>
</tbody>
</table>

**Table 3.9** – Table $M=2$ AoA=10°.

However, the divergence appears when the aerodynamic coefficients are computed, you can see it in the following table, Table 3.10. Drag and Lift force diverge...
considerably from the simulation, therefore, \( C_D \) and \( C_L \) diverge \( \sim 0.02 \) points and \( \sim 0.01 \) points, respectively.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
<th>Real Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag (N)</td>
<td>52355.68</td>
<td>52303.47</td>
<td>43938.326</td>
</tr>
<tr>
<td>Lift (N)</td>
<td>234698.69</td>
<td>234561.04</td>
<td>196947.45</td>
</tr>
<tr>
<td>( C_D )</td>
<td>0.0926</td>
<td>0.0925</td>
<td>0.0777</td>
</tr>
<tr>
<td>( C_L )</td>
<td>0.4152</td>
<td>0.4150</td>
<td>0.3484</td>
</tr>
</tbody>
</table>

Table 3.10 – Aeroforces table \( M=2 \) AoA=10°.

3.4. Analysis

Once the numerical values from the simulations are obtained, these results are put in graphics in order to have a better perspective of what is happening in the aerodynamic behaviour of our airfoil.

First of all, in the following figures, Fig.3.14 and 3.15, it could be seen the aerodynamic coefficients showed in the previous section, Sec.3.3., visualized in Python graphs. In these figures, it is shown how the global \( C_L \) and \( C_D \) develop when AoA is increased.

Figure 3.14 – Global \( C_L \) as a function of AoA.
As you can see in Fig. 3.14, as theory predicts the lift coefficient increases as AoA increases due to the pressure difference between the upper and lower surface, it created by a greater velocity at the top than in the lower surface.

In the graphic, the red crosses represent the theoretical behaviour of $C_L$ as AoA increases, the blue scattering is $C_L$ based on the average pressures over the airfoil obtained from the simulation and the magenta crosses is $C_L$ based in the “Aeroforces” Nektar++’s filter, which is the integral of the pressure over the airfoil. In addition, the black line is a linear regression which was obtained by using a least-squares-fit procedure. This linear fit gives us a value of $R^2 = 0.99$, remind that R-squared is a statistical measure of how close the data are to the fitted regression line. Hence, in this case, the $R$ value means that the model fits well the data, it is an extremely good fit. This way of representation is applied in all the following figures.

It is seen that the simulation values are very close to the theoretical values, with a minimum error. However, the obtained value from the Aeroforces at 10° of AoA diverges from the theory, $C_{L_{\text{Theory}}} = 0.0926$ and $C_{L_{\text{Aeroforces}}} = 0.0777$. It could be explained by the way that in this case, the simulation setup would need a finer mesh and, especially, more time of computation so as to have a more converged simulation.

![Global $C_D$ as a function of AoA](image)

**Figure 3.15** – Global $C_D$ as a function of AoA.

In Fig. 3.15, the evolve of Global $C_D$ as a function of AoA is presented. It is seen how the $C_D$ is not zero at the symmetry condition, 0°, and how it does not follow a linear behaviour, it develops as parabolic. The ansatz for the dependence has been taken as $C_D = a \cdot \alpha^2 + b$ for the fit. Result values from the simulation are very
close to the theory except for the case at $10^\circ$ of AoA, of the Aeroforces filter; so the model fits very well.

Besides, it has to be noted how the $C_D$ increases rapidly when the angle of attack overcomes the deflection angle of the airfoil. In the previous section, it has been explained what occurs when the angle of attack increases the deflection angle, an expansion wave is produced at the upper surface instead of an oblique shock. Then, this is the $C_D$ behaviour before and after it occurs:

- $C_D$ at $2.5^\circ = 0.0223 \rightarrow C_D$ at $5^\circ = 0.0361$
- $C_D$ at $5^\circ = 0.0361 \rightarrow C_D$ at $7.5^\circ = 0.0593$
- $C_D$ at $7.5^\circ = 0.0593 \rightarrow C_D$ at $10^\circ = 0.0925$

As you can see, it could be seen how $C_D$ increases approximately 0.01 points before the angle of attack overcomes the deflection angle, but when it occurs $C_D$ increases $\sim0.02$ points from $5^\circ$ to $7.5^\circ$ and $\sim0.04$ points from $7.5^\circ$ to $10^\circ$. This fact has to be take into account when we are going to design our supersonic vehicle in order to find the optimum flight conditions.

![Global $C_D$ as a function of $C_L$](image)

**Figure 3.16** – Drag Polar graph where the two forms of drag are presented.

In Fig.3.16, drag polar graph is presented. Concretely, it is the relationship between the lift on a 2D wing and its drag, expressed in terms of the dependence of the lift coefficient on the drag coefficient. Then, in this graph could be seen how the experimental data fit the theoretical predictions, except for the case at $10^\circ$. 

$$C_D = kC_L^2 + C_{D_0}$$

- $k = 0.43438$
- $C_{D_0} = 0.01776$
Furthermore, at supersonic speeds in a 2D wing with no viscosity effects, the pre-dominant drag is the wave drag, \( C_{dW} \rightarrow F = \iint dF = x \text{ component of } \left[ -\iint p dS \right] \). 

Then, this type of drag follows, theoretically, the next mathematical relation:

\[
C_{dW} = \frac{4}{\sqrt{M_{\infty}^2 - 1}} \left[ \alpha^2 + \left( \frac{t}{c} \right)^2 \right] \quad (3.4)
\]

So, with a fixed Mach and thickness \( t \) to chord \( c \) ratio, if the AoA is increased, \( C_{dW} \) will increase. As it could be seen in the previous figure, this behaviour has been reproduced perfectly by CFD simulations. In the following table, Table. 3.11, the results obtained from Eq.3.4 and the simulation are analyzed. Note how close are between them:

<table>
<thead>
<tr>
<th>AoA</th>
<th>( C_{dW} ) from the equation</th>
<th>( C_D ) Approximated Data</th>
<th>( C_D ) Real Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>0.0177</td>
<td>0.0177</td>
<td>0.0176</td>
</tr>
<tr>
<td>2.5°</td>
<td>0.0221</td>
<td>0.0223</td>
<td>0.0221</td>
</tr>
<tr>
<td>5°</td>
<td>0.0353</td>
<td>0.0361</td>
<td>0.0358</td>
</tr>
<tr>
<td>7.5°</td>
<td>0.0572</td>
<td>0.0593</td>
<td>0.0589</td>
</tr>
<tr>
<td>10°</td>
<td>0.0880</td>
<td>0.0925</td>
<td>0.0777</td>
</tr>
</tbody>
</table>

**Table 3.11** – Wave drag coefficient comparison between theoretical and experimental values

Moreover, the reader can see how the curve is displaced from the origin, this is caused by the fact that wave drag is formed by two types of drag:

- **Zero-Lift** wave drag: \( D_0 \)
- **Lift induced** wave drag: \( D_i \)

It has to be noted that in this case, the study only cares about the drag produced by the shock waves, wave drag, see Reference [20]. In this study, other forms of drag are excluded:

- **Wake drag**.
- **Induced drag** produce by wing tips, it has no sense in a 2D wing.
- **Skin friction drag**.

In the figure, this Zero-Lift wave drag could be seen as the curve starts at the point 0.0177, see Table 3.2, so this is a kind of parasitic drag and gives us an idea of the aerodynamic design of the airfoil, in order to reduce this parameter.
it will be necessary to redesign the airfoil, so as to obtain a more streamlined solution. Besides, Lift-induced drag is responsible of the parabolic behaviour of the curve, so this form of drag makes that $C_D$ grows as the square of $C_L$. In the previous figure, Fig.3.16, a scheme of how these two types of drag, Zero-Lift and Lift induced wave drag, are represented in the $C_D$ vs $C_L$ graph.

In Fig. 3.17, it could be seen the relation between $C_L^2$ and $C_D$. This directly relation given by the equation:

$$C_D = kC_L^2 + C_D_0$$  \hspace{1cm} (3.5)

where

- $k=0.43438$
- $C_D_0=0.01776$

![Global $C_D$ as a function of $C_L^2$](image)

**Figure 3.17** – Global $C_L^2$ as a function of $C_D$.

As we see in Fig.3.17, $C_L^2$ and $C_D$ are directly proportional, as theory predicts, so this graph confirms the parabolic relation between $C_L$ and $C_D$.

In Fig.3.18 the Lift to Drag ratio is presented in a graph. In the design of an airfoil, one of the key parameters, in order to analyze wing performance is on the ability to obtain a high value of the lift to drag ratio, $\frac{L}{D}$. Essentially, the wing is designed to allow the airfoil to achieve its full performance. Then, in the graph could be seen how this parameter increases with the AoA, but when it surpasses
the 5° value this parameter starts to decrease. Therefore, we conclude that the point of maximum efficiency will be at a point near to the 5° of AoA. Concretely, it is the point of the flat plate case and when after it, at the upper surface an expansion wave is generated instead of a shock wave. Note that after this point \( C_D \) increases faster than \( C_L \), this fact explains why the \( \frac{L}{D} \) ratio decreases after 5° of AoA.

Besides, if we compare this analysis with the aerodynamic analysis made in References [21] and [22], where the aerodynamic performance of the supersonic airplane XB-70 is studied in real flight and wind tunnel. It could be seen, how their experimental results are very close to the theory and the behaviour of the aerodynamics is particularly the same as the graphs of this project have shown. In addition, they only studied the effects of the angle of attack between 0° and 6°, so they do not afford the problems of the shock wave detachment at the high angles of attack.

Secondly, in Sec.3.3.1., it is explained that the generation of the lift force is directly related to the pressure difference between the upper and lower surfaces. For this reason, until the airfoil does not acquire an angle of attack different of zero the airfoil does not generate lift. Especially, this fact could be visualized directly from the pressure graphic in Fig.3.6. In order to have a better comprehension of the phenomenology, the evolution of the \( C_p \) (pressure coefficient) while increasing the angle of attack will be analyzed, instead of pressure.

![Lift to Drag ratio as function of AoA](image)

**Figure 3.18** – Lift to Drag ratio as a function of angle of attack.
\[ C_p = \frac{P - P_\infty}{\frac{1}{2} \rho_\infty U_\infty^2} \]  

(3.6)

In Fig. 3.19, \( C_p \) values for each of the four surfaces in three different angles of attack (0°, 2.5°, and 7.5°) are presented. In this graph, it could be seen how the area between opposite \( C_p \) values increase when the angle of attack is increased. This increase of area is proportional to the increase of \( C_y \) coefficient, which at moderate AoA is the major contribution to \( C_L \). Mathematically this is explained by Eq. 3.7 and Eq. 3.8.
\[ \vec{F} = - \int_C p \star \vec{n} \, dl \]  
\[ F_y = \vec{F} \hat{j} = - \int_C p \star n_y \, dl = - \int_C p \star dx \]  
\[ C_y = \frac{F_y}{\frac{1}{2} \rho_\infty U_\infty^2 c} = \frac{\delta (-C_p dx)}{c} \]  
\[ C_y = \frac{F_y}{\frac{1}{2} \rho_\infty U_\infty^2 c} = \frac{\delta (-C_p dx)}{c} \] 

Furthermore, at this graph, we could note which parts of the airfoil have a more contribution to lift coefficient. It could be seen with the difference between upper and lower surfaces at the front and back of the airfoil. Concretely, this contribution is the area created between the two surfaces.

For example, at 0° this area has null value because the pressure at the upper and lower surfaces is the same. However, when the angle of attack is increased, at 2.5° for example, this area increases and we can note that this area is bigger at the front of the airfoil than at the back. So, at half of the chord, the contribution to lift will be higher than at the aft half. Especially, this fact could be seen better if we fix our attention to the case of 7.5°, where the airfoil experiences an expansion instead of an oblique shock wave, at the front is greater than at the back of the airfoil.

Moreover, the same principle could be applied to \( C_x, F_x \) coefficient, which is the major component to \( C_D \) at moderate AoA, see Eq.3.9 and 3.10.

\[ \vec{F} = - \int_C p \star \vec{n} \, dl \]  
\[ F_x = \vec{F} \hat{i} = - \int_C p \star n_x \, dl = - \int_C p \star dy \]  
\[ C_y = \frac{F_x}{\frac{1}{2} \rho_\infty U_\infty^2 c} = \frac{\delta (-C_p dy)}{c} \] 

Therefore, analyzing the difference between the \( C_p \) values over the airfoil thickness, we can obtain an idea of the evolution of the drag coefficient when the angle of attack varies. This could be seen in the following figure, Fig.3.20.

Also in this figure, the drag contribution from the different surfaces could be seen. As in the previous case, we are going to see the area created between surfaces, but in this case, we will see the area generated by the front and back parts of the airfoil in the upper and lower cases. Then, in this case, at 0° the value of this area is not zero, due to zero-lift wave drag, so it will exist a drag contribution from the surfaces, but the airfoil is in symmetric condition, so the contribution will be the same from the upper and lower surfaces. However, when the angle of attack is increased this contribution becomes unbalanced; at 2.5°, the contribution of the
lower half is greater than the upper one. Therefore, the area generated due to pressure differences between front and back surfaces in the lower case is bigger than the area generated in the upper case. Especially, this difference could be seen better at 7.5° where the area generated at the lower case is so big in comparison with the upper case.

Moreover, at the following table, Table 3.12, an average of pressure coefficients over each surface of the airfoil are presented. Because of pressure coefficients are quasi-constant along each surface. Also, at the table, it could be seen how $C_p$ is bigger for the lower surfaces.

From the values presented in the previous table, $C_y$ and $C_x$ values using Eq.3.11 and 3.12 are computed, respectively:
Parameter | 0°  | 2.5°  | 7.5°  \\
---|------|------|------\
$C_{pu f}$ | -0.113 | -0.053 | 0.048  \\
$C_{pub}$ | 0.090 | 0.123 | 0.191  \\
$C_{plb}$ | 0.090 | 0.047 | -0.055  \\
$C_{pl f}$ | -0.113 | -0.179 | -0.334  \\

Table 3.12 – Pressure coefficients obtained experimentally over the airfoil at different AoA

$$C_y = \frac{-(C_{pu f} - C_{pl f}) \frac{c}{2} - (C_{pub} - C_{plb}) \frac{c}{2}}{c} = \frac{\Delta C_{pf} + \Delta C_{pb}}{4} \quad (3.11)$$

$$C_x = \frac{-(C_{pu f} - C_{pub}) \frac{t}{2} - (C_{pl f} - C_{plb}) \frac{t}{2}}{c} = \frac{(\Delta C_{pu} + \Delta C_{pl}) \frac{t}{4 \cdot c}}{c} \quad (3.12)$$

Once the $C_y$ and $C_x$ are computed, aerodynamic coefficients, $C_L$ and $C_D$, are determined by projecting them in the right way, as you can see in Eq. 3.13 and 3.14, respectively:

$$C_L = -C_x \sin \alpha + C_y \cos \alpha \quad (3.13)$$

$$C_D = C_x \cos \alpha + C_y \sin \alpha \quad (3.14)$$

So the results are shown in the following table, Table 3.13. The values obtained for $C_L$ and $C_D$ are computed using the experimental pressure values presented in Sec.3.3.1. The results are so close to the ones from Sec.3.3.1., any type of small discrepancy is due to the number of decimals chosen when the computations were made.

Parameter | 0°  | 2.5°  | 7.5°  \\
---|------|------|------\
$C_y$ | 0 | 0.1030 | 0.3139  \\
$C_x$ | 0.0177 | 0.0178 | 0.0185  \\
$C_L$ | 0 | 0.1022 | 0.3088  \\
$C_D$ | 0.0177 | 0.0223 | 0.0593  \\

Table 3.13 – Aerodynamic coefficients obtained by experimental $C_p$ values

Thirdly, carrying on with the analysis, we are going to look at the coefficient moment variation through the increase of the angle of attack. The study of this coefficient gives us information about the stability of the airfoil. In order to obtain this information, $C_p$ values obtained from the simulation shown in the previous figures, Fig.3.19 and Fig. 3.20 have been used. Assuming $C_p$ is almost constant along each surface, in the following figure, Fig.3.21, you can see where the forces are applied in each surface.
From the scheme, you can see that the forces are assumed to be applied at the center of the surface, so it will be at $a_2$ and $b_2$. Then, the moment, $M \ [N \cdot m]$, of these forces respect to the leading edge, $x_{le} = -0.99$, take in account the right projections, is defined as:

\[
\begin{align*}
M_{uf} &= -F_{uf} \cos \delta \cdot \frac{a}{2} - F_{uf} \sin \delta \cdot \frac{b}{2} \\
M_{ub} &= -F_{ub} \cos \delta \cdot \frac{3a}{2} + F_{ub} \sin \delta \cdot \frac{b}{2} \\
M_{lf} &= F_{lb} \cos \delta \cdot \frac{a}{2} + F_{lb} \sin \delta \cdot \frac{b}{2} \\
M_{lb} &= F_{lb} \cos \delta \cdot \frac{3a}{2} - F_{lb} \sin \delta \cdot \frac{b}{2} \\
M_T &= M_{uf} + M_{ub} + M_{lf} + M_{lb}
\end{align*}
\] (3.15)

where

- $M_{uf}$ is the moment of the force applied at the upper forward surface.
- $M_{ub}$ is the moment of the force applied at the upper back surface.
- $M_{lf}$ is the moment of the force applied at the lower forward surface.
- $M_{lb}$ is the moment of the force applied at the lower back surface.
Therefore, the total moment respect to the leading edge, $M_T$, is the sum of the applied moments in each of the four faces.

$$M_T = M_{uf} + M_{ub} + M_{lf} + M_{lb} \quad (3.16)$$

Hence, the global coefficient moment respect to the leading edge, $C_{MT}$, is defined as

$$C_{MT} = \frac{M_T}{\frac{1}{2} \rho_\infty \gamma c^2 M^2} = \frac{1}{c^2} \left( C_{M uf} + C_{M ub} + C_{M lf} + C_{M lb} \right) \quad (3.17)$$

![Graph](image)

**Figure 3.22** — Global $C_{MT}$ respect to the leading edge as a function of AoA.

In the previous figure, Fig.3.22, a graph of the variation of total coefficient moment respect to the leading edge is presented ($C_{MT}$). The results show a negative $C_{MT}$, so the pitching moment has clock-wise movement. In Table 3.14, you can see the value of the coefficient moment in each surface and the total for each angle of attack.

Besides, there is a direct relation between the generated pitch moment and the for-aft and the up-down half asymmetry of the forces. Hence, if the front part is generating more lift than the back this tends to generate a nose-up pitching moment with respect to the airfoil center. If the drag comes predominantly from the lower part, this results in a nose-down pitching moment. Then, we can see in the results than the nose-up moment of the front part overcomes the other, so it is the clock-wise moment which we have obtained before.
Table 3.14 – Moment coefficients obtained by experimental pressure coefficients.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>0°</th>
<th>2.5°</th>
<th>5°</th>
<th>7.5°</th>
<th>10°</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_{Muf}$</td>
<td>0.0565</td>
<td>0.0267</td>
<td>0</td>
<td>-0.0238</td>
<td>-0.0453</td>
</tr>
<tr>
<td>$C_{Mub}$</td>
<td>-0.1331</td>
<td>-0.1899</td>
<td>-0.2396</td>
<td>-0.2829</td>
<td>-0.3216</td>
</tr>
<tr>
<td>$C_{Mlf}$</td>
<td>-0.0565</td>
<td>-0.0889</td>
<td>-0.1252</td>
<td>-0.1657</td>
<td>-0.2111</td>
</tr>
<tr>
<td>$C_{Mlb}$</td>
<td>0.1331</td>
<td>0.0702</td>
<td>-0.0015</td>
<td>-0.0820</td>
<td>-0.1726</td>
</tr>
<tr>
<td>$C_{Mr}$</td>
<td>0</td>
<td>-0.0458</td>
<td>-0.0922</td>
<td>-0.1397</td>
<td>-0.1891</td>
</tr>
</tbody>
</table>

After analyzing the pitch’s moment coefficient while AoA increases, we are going to find the center of pressure and aerodynamic center, respect the leading edge and see their variation with AoA. Remind that:

- $X_{Cp}$ The point along the chord of an airfoil where the lift and drag forces act, and there is no moment produced.
- $X_{Ac}$ is the point of action of lift and drag forces such that the value of the moment coefficient does not change with the angle of attack. The aerodynamic center does not move with the angle of attack.

The mathematical respective expressions are Eq.3.18 and 3.19:

\[
C_{Mxcp} = C_{Mxle} + \frac{X_{Cp} - x_{le}}{c} * C_L = 0
\]
\[
X_{Cp} = x_{le} - \frac{C_{Mxle}}{C_L} * c
\]

\[
\frac{\partial C_{mxic}}{\partial \alpha} = \frac{\partial C_{mxle}}{\partial \alpha} + \frac{X_{Ac} - x_{le}}{c} * \frac{\partial C_L}{\partial \alpha} = 0
\]
\[
X_{Ac} = x_{le} - \frac{\partial \alpha}{\partial C_L} * c
\]

where

- $\frac{\partial C_{Mxle}}{\partial \alpha} = -0.01891$
- $\frac{\partial C_L}{\partial \alpha} = 0.04150$

Hence, we have determined that the $X_{Ac} = -0.089$, it is situated at 45.56%, theory tells us that in the supersonic regime the aerodynamic center is at $\sim 50\%$, so it is a reasonable value. Once the aerodynamic center is determined, the value of the
moment coefficient respect to it, $C_{M_{Ac}}$, through the variation of the angle of attack is computed. In the following figure, Fig.3.23, you can see a graph that shows the evolution of the $C_{M_{Ac}}$ while AoA increases.

In the figure, the reader could see how the $C_{M_{Ac}}$ remains constant along the increase of AoA. The smallest value of $C_{M_{Ac}}$ is $-4.6977 \times 10^{-5}$ and the largest one is 0.0011, so we can determine that $C_{M_{Ac}}$ is constant and zero in all the cases. Linear potential theory tells us that it has to remain constant while AoA increases, $X_{Ac}$ does not vary as the $X_{Cp}$ when AoA is increased. Besides, for symmetric airfoils, if $C_{M_{Ac}} = 0$ when $C_L = 0$, then $C_{M_x}$ will be zero for any $x$ when $C_L = 0$. Hence, $C_{M_{Ac}}$ will be zero for any AoA or $C_L$.

Moreover, the value of the center of pressure, $X_{Cp}$, is computed, through the variation of the angle of attack. The center of pressure depends on the pressure distribution over the airfoil, so it changes when AoA varies, it causes that the center pressure does not remain constant as the aerodynamic center. In the following figure, Fig.3.24, you can see a graph that shows the evolution of the $X_{Cp}$ while AoA increases.

In the previous figure, we can see how the center of pressure varies when the AoA of the airfoil is increased. Note that when the angle of attack increases to 2.5° the $X_{Cp}$ is shifted to the left of the aerodynamic center,$X_{Cp} = -0.1033$, and when the AoA increases it is moved forward to the right and when it is at 10°, the $X_{Cp}$ is situated at the aerodynamic center, $X_{Cp} = -0.0887$. 

**Figure 3.23** – Global $C_{M_{Ac}}$ as a function of AoA.
CHAPTER 3. SUPERSONIC PROFILE SIMULATION

To sum up, the lift generation is linked directly to the pressure differences between the lower and upper surfaces. While the pressure at the lower surface remains greater than in the upper surface, lift will be generated. However, drag force has to be taken into account due to it increases also, but in these type of vehicles in order to overcome wave drag more fuel is burned so as to obtain more thrust, because fuel efficiency is not a key parameter as in the commercial airlines. Furthermore, it has to be noted that the angle of attack could be not increased infinitely. There is a point when the shock wave is detached and the lift coefficient drops. So, in each flight regime, the aerodynamic limitations have to be well-known in order to flight in the best conditions and avoid critical points.

Finally, if a vehicle, which flies in the supersonic regime, uses a symmetric diamond shaped airfoil, it will have to increase always its angle of attack in order to generate lift. Nevertheless, it has to take into account its aerodynamic limitations, due to shock wave detachment, in order to avoid any possible accident. The studied airfoil seems to be a suitable option for the development of a supersonic wing, but a more deeply study is needed in order to have a complete analysis of the aerodynamic performance.

Nektar++ demonstrates a strong and reliable behaviour in the compressible regime providing very close results that could be used to characterization and design of aerodynamic solution in the compressible regime [23].
Simulation of an hypersonic gas turbine ramjet engine intake in the supersonic regime
CONCLUSIONS

In this project, the aerodynamic performance analysis of a ramjet intake and diamond airfoil in the supersonic regime is made.

In order to understand better the aerodynamic conditions of the problem, an explanation of compressible flow is made at the beginning. In this flight regime, the condition of incompressible flow does not hold more; also, it is a high-energetic flow, so the energy changes are substantial enough to be taken into account due to the interaction with other properties of the flow, so thermodynamics has not been skipped.

Before to start to simulate, it is necessary to study deeply possibilities phenomenologies which will appear when the simulation begins. When a CFD simulation is going to be made in compressible flow, shock waves must be considered. Especially, take in account all the problems that appeared in order to capture them and achieve stability, it is necessary to explain how they are generated and change fluid properties. This phenomenology is produced when the flow finds an obstacle which changes its flow direction. As despite of incompressible flow, here the flow particles are not warned by the sound waves and shock with the obstacle, as a Newtonian fluid; this shock causes, instantly, greater changes in flow properties in velocity, density, temperature, energy and pressure.

For this reason, the efforts of this project have been in capture, in the best possible conditions, the appearance of oblique shock waves. However, the zone of the instant change of fluid properties is a unstable place where if the mesh is not fine enough or the step time is so big, it will cause NaN values. It has been proved that for studying this phenomenon the recommended step time is 1e-8, in some cases where you will not have difficult transient it could be increased to 1e-6. But, in the best critical situations, as for angles of attack bigger than 10°, this step time has not been enough. So, it could be that another type of setup simulation or more small time and finer mesh has to be used, but this could delayed the project significantly.

Firstly, a ramjet intake geometry was studied in order to analyze its aerodynamic performance under a determined conditions. This type of intake reduce the airflow velocity using oblique shock waves in order to reduce airflow, but the problem is that huge instabilities are generated in the throat through this process, NaN values are generated in the throat. For this reason, the simulation has been stopped many times during this project, I have meshed fine and reduce significantly the step time, 1e-11, but it has not been enough to achieve a final convergence. Even, the problem has been shown to the Nektar++ team and after their recommendations, the simulation has not been able to converge. Therefore, it is necessary a careful study of the possible instabilities which happen inside the throat, so as to find the right mesh dimension and simulation setup and obtain smooth and precise results.

Secondly, the aerodynamic performance of a diamond shaped airfoil has been studied. The advantage of this airfoil in the supersonic regime is the fact that the sharp wedges avoid the detachment of the shock wave at the leading edge, so
it reduces the drag. Concretely, when this type of airfoil is studied in the inviscid compressible regime, the only drag which appears is the wave drag. The studied airfoil is symmetric, thus, it can not generate lift at $0^\circ$ of AoA, this behaviour has been shown in the simulations and how due to the progressive increase of the AoA, the airfoil is able to generate lift force. It has been proven that the lift generation is directly related to the difference in pressure between the upper and lower surfaces. Besides, the maximum point of $\frac{L}{D}$ has been find at $5^\circ$ of AoA, this point gives the maximum efficiency. Note that the maximum efficiency is located before the appearance of the expansion wave at the upper surface, instead of the oblique shock wave. Therefore, in terms of efficiency, the AoA cannot be increased infinitely without losing efficiency. In addition, above $15^\circ$, as the airfoil sees $20^\circ$, the oblique shock wave is detached so it causes a drop of the lift force and the appearance of instabilities.

Unfortunately, the detachment of the shock wave could not be possible to capture, several tries have been made, even the detachment at a low Mach velocity has been tried, but the simulation setup could not capture it. If you compare this project with some of the articles and projects which I have referenced in the bibliography, the reader will see that they do not study a wide range of angles, most of them study until $6^\circ$ of AoA, due to the difficulty and instabilities that the detachment of the oblique shock wave represents.

I want to note that this project has carried a huge workload because for any simulation the minimum time has been a week and a half, some of them a month. Unfortunately, in a lot of occasions, I have waited more than a week to find that the simulation has not been able to converge and tried one more time. One of the problems has been the lack of examples using Nektar++ in the supersonic regime, so it has been a clearly unique project using this software. This is due to the fact that Nektar++ has been recently developed by the Imperial College, it is clear that almost all the researchers work in the incompressible regime, so the compressible flow solver has been less tried.

Besides, try and error has been the way to find lacks in the solver, like the symmetry condition and the aeroforces filter. The first one allows to use symmetry conditions on the wall of a domain that has symmetric conditions, so it allows to speed up simulations which are symmetric. The second one brings the possibility to obtain directly from the simulation the aerodynamic forces acting on a body. Then, thanks to this project and the collaboration of the high-skilled team of developers, these two features have been added to the solver for future studies. Furthermore, thanks to the project I have learned to implement certain tools to improve my simulation like the deliasing technique using the Gauss-Lobatto polynomials or using the NonSmooth Shock Capturing in order to obtain the aerodynamic forces over the airfoil.

Finally, during this project, I have learned how to use a CFD software, in my case Nektar++, so as to simulate any geometry and make an aerodynamic study of it. Besides, I have worked with a new software in a field that has not been well studied and I have been able to overcome all the difficulties and obtain the characterization of a supersonic airfoil in a good range of angles. Therefore, I have put myself in the place of a researcher who makes her/his research in
an unbounded field and without clear guidelines. In addition, in the realization of this thesis, I have to use the cluster provided by the university and the lack of nodes has made that I could not study several configurations because many students were using the cluster. Owing to that I have taste how difficult is the research world and how many scientists around the world have to think outside the box, so as to use their available resources and still being able to do excellent investigations.

Future Studies

In this aerodynamic study, there are some configurations which could be not studied due to the lack of time or the impossibility to make them converge. For this reason, for me, the first thing to has to be studied in the future is the ramjet intake, so it has to be found a way to achieve the convergence. Once it is achieved, different configurations of height throat and cowl lip have to be studied in order to observe their influence in the performance of the engine. Then, I will study other geometries like a circular ramp and cowl lip.

Secondly, also, in the study of the diamond airfoil there are configurations which could be not studied, so for future studies, I will find the way to make them converge and make a complete aerodynamic study take into account more possible configurations. In addition, the diamond airfoil has been studied in 2D without viscosity, so the next step is to make a 3D simulation using the Navier Stokes equations. This type of simulation requires powerful computational resources and several months to converge. The compressible regime has high Reynolds number, so the vortices and then the possible instabilities have to be carefully studied in order to make the right simulation setup. Besides, I will make this 3D simulation in an airfoil with a swept angle, because this improves its aerodynamic performance on low Mach velocities.

Finally, I have proposed in Appendix 3.4. a solution for a future supersonic airplane, the Biplane. It is based on the Busemann's problem and allows to cancel the drag generated by the shock waves, but it occurs in symmetric configurations, so find an optimum way to use this system in the future will be a great path of investigation.
Simulation of an hypersonic gas turbine ramjet engine intake in the supersonic regime

[2] Tohoku University Obayashi laboratory. Conceptual drawing of a supersonic biplane. xii, 86


[4] Reaction Engines Ltd. Reaction Engines Vehicle, Skylon. 1

[5] Space X. Space X, Falcon 9. 1


[26] CFD-Online. Discontinuous Galerkin. 55


[34] Auld & Srinivas. Shock-Expansion Techniques for Aerofoils. 85

b) els apendixs s’enquadernen amb el mateix projecte (sense portada).
APPENDICES
COMPRESSIBLE FLOW SOLVER

As it has mentioned before in Section, Nektar++ has multiple packages which allow you to make numerical simulations for different problems.

In this project, we are working in the supersonic regime, so the compressible flow solver is the used solver and it has allowed me to solve the unsteady compressible Euler representation of the variables.

Discontinuous Galerkin method

In Nektar, the spatial discretization of the Euler and of the Navier-Stokes equations is projected in the polynomial space via a discontinuous projection. For this purpose, discontinuous Galerkin (DG) method is used. It is a hybrid method, firstly introduced by Reed and Hill in 1973, since it combines features of both finite element (FEM) and finite volume methods (FVM). The solution is represented by each element as a polynomial approximation (as in FEM), while the interelement convection terms are resolved with upwinded numerical flux formulas (as in FVM), so as to know more about DG method see References [24] [25] [26]

Firstly, in this approach, the physical domain is divided into a mesh of N non-overlapping elements (Ω), this is called broken space, and the solution is allowed to be discontinuous at the boundary between two adjacent elements. This fact is due to the discontinuous functions, with it works, within each element are set to zero everywhere outside of their associated element. In addition, this method allows the combination with adaptive refinement procedures, like the hp method.

Secondly, Euler, as well as the Navier-Stokes equations, are defined on each element of the computational domain. For this reason, it is necessary to define a term to couple the elements of the spatial discretization in order to allow information to propagate across the domain. This term, called numerical interface flux, naturally arises from the discontinuous Galerkin formulation.

To sum up, the two features which discontinuous Galerkin method have been used are:

• **High order accuracy**: The possibility to obtain a high order accuracy approximation to the exact solution in smooth regions.

• **High resolution**: DG produces sharp and non-oscillatory discontinuity transitions near discontinuous solutions including shocks and contact discontinuities.

Advection term

For the advection term, a Riemann solver is used. It is a numerical method used to solve a Riemann problem. It consists of a piecewise function with a single discontinuity, as shown in Fig.25.

The Riemann problem is very useful for understanding equations like Euler co-
In Nektar++, there are different Riemann solvers, one exact and nine approximated. The exact Riemann solver applies an iterative procedure to satisfy conservation of mass, momentum and energy and the equation of state. The left and right states are connected either with the unknown variables through the Rankine-Hugoniot relations, in the case of shock, or the isentropic characteristic equations, in the case of rarefaction waves.

Across the contact surface, conditions of continuity of pressure and velocity are employed. Using these equations, the system can be reduced to a non-linear algebraic equation in one unknown that is solved iteratively using a Newton method. The use of the Riemann solver implies high computational cost due to its accuracy. For this reason, I have also applied the approximated Riemann solvers that are simplifications of the exact solver. In order to know more about Riemann solvers see [27] and [28]

**Diffusion term**

Concerning the diffusion term, the coupling between the elements is achieved by using a local discontinuous Galerkin (LDG) approach.

The local discontinuous Galerkin (LDG) is a suitable rewrite of the convection diffusion system, into a larger, first order system and then discretize it with the discontinuous Galerkin (DG) method.

**Boundary Conditions**

The boundary conditions are also implemented by exploiting the numerical interface fluxes just mentioned, see Reference [29] to know more about the implementation of boundary conditions in compressible aerodynamics.
Governing Equations

Compressible Euler Equations

The Euler equations can be expressed as a hyperbolic conservation law in the form

$$\frac{\delta q}{\delta t} + \frac{\delta f_i}{\delta x} + \frac{\delta g_i}{\delta y} + \frac{\delta h_i}{\delta z} = 0$$

(20)

where $q$ is the vector of the conserved variables, $f_i = f_i(q)$, $g_i = g_i(q)$ and $h_i = h_i(q)$ are the vectors of the inviscid fluxes:

$$q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ E \end{bmatrix}, f_i = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uw \\ u(E + p) \end{bmatrix}, g_i = \begin{bmatrix} \rho v \\ \rho uv \\ p + \rho v^2 \\ \rho vw \\ v(E + p) \end{bmatrix}, h_i = \begin{bmatrix} \rho w \\ \rho uw \\ \rho vw \\ p + \rho w^2 \\ w(E + p) \end{bmatrix}$$

(21)

Where $\rho$ is the density, $u$, $v$ and $w$ are the velocity components in $x$, $y$ and $z$ directions, $p$ is the pressure and $E$ is the total energy. In this project, I have considered a perfect gas law for which the pressure is related to the total energy by the following expression:

$$E = \frac{p}{\gamma - 1} + \frac{1}{2} \rho \left( u^2 + v^2 + w^2 \right)$$

(22)

where $\gamma$ is the ratio of specific heats.
RAMJET INTAKE SIMULATION

Overview

In this chapter, after the explanation of the characteristics of the flow and some tips about the solver, it will be explained how ramjet engines work and the importance of the supersonic inlet, the used geometry, the conditions that we impose in the setup of the simulation and an analysis of the obtained results.

How Ramjet works

While our engine is going to the hypersonic regime, it will have to pass the supersonic regime. Along with this phase, it will behave like a ramjet engine. In order to be operable in this regime, the use of shock waves by the supersonic inlet is crucial, see [30],[31] and [23].

First of all, this type of engine does not use turbomachinery in order to compress the air which enters into the engine, because at the intake the airflow is supersonic, so the flow becomes compressible. In this regime of speed, no turbojet engine compressor is able to handle with the supersonic flow. Then, its supersonic inlet uses only its geometry in order to slow down the velocity of the flow to subsonic by using oblique shock waves. So, it converts the kinetical energy of the air in pressure using only its geometry, the pressure acquired is called “ram” pressure. Therefore, this type of engine will be lighter than a turbojet.

Secondly, after three or four oblique shocks, the fluid stops in a normal shock before entering the combustion chamber. The position of the normal shock is crucial in order to obtain a right performance of the engine. It is clear that the inlet’s job is crucial, because the core engine of gas turbine and ramjet engines operates at subsonic speed for optimal efficiency, regardless of flight speed.

Finally, flame holders in the burner localize the combustion process. In the ramjet engine, burning process occurs, always, subsonically. Leaving the burner, the hot exhaust passes through a nozzle, which has a divergent shape in order to accelerate the flow.
Geometry

In Fig. 27, you can see the dimensions of the intake. The change of the height of the throat, the angle of the cowl lip and the position of it will change dramatically the obtained results.

![Figure 27](image_url) – Scheme of the ramjet intake with its dimensions.

In Fig. 28, the used mesh is presented, it has been created by Gmsh [32]. You can observe that the mesh is very fine the throat and the cowl lip. Thanks to that, the solver will be able to capture well the phenomenology produced in these regions.

![Figure 28](image_url) – Supersonic intake geometry mesh.
Simulation setup

Before to start the simulation, it is necessary to give some inputs to the solver: TimeStep, pressure, velocity, boundary conditions... The setup of the simulation is established in a file which we are going to call session file. We are going to see some aspects of this session file:

Firstly, the Expansions of the simulation, it has been imposed the use of: Gauss-Lobatto-Legendre and Lagrange polynomials as a basis with a number of modes of 4 and Gauss-Lobatto-Legendre and Lagrange points of a number of points of 8.

Note that the use of a number of points which doubles the number of modes is a deliasing technique in order to avoid the noise of the high frequencies interferes in the low frequencies. Then, the accuracy is improved.

Secondly, the far field conditions which are imposed on the domain are:

- $\gamma = 1.4$
- $P_\infty = 101325$ Pa
- $\rho_\infty = 1.225 \left( \frac{kg}{m^3} \right)$
- $M_\infty = 2$
- $u_\infty = M_\infty \frac{\gamma P_\infty}{\rho_\infty}$

Thirdly, flow is inviscid, therefore, viscosity effects are neglected, so it will only exist the wave drag. Nektar++ solves the Euler equations for the compressible regime.

Finally, when a simulation in the compressible regime is made, always the Shock Capturing technique has to be activated, see Reference [33].

Results and Analysis

In this section, the obtained results from the simulation of the ramjet intake are presented. As it has been stated before, it has been impossible to obtain a full converged solution due to the lack of time and resources. In the following figure, Fig.29, you can see the Mach field of the airflow in the intake.

In the figure, you can see the evolution of the shock waves through the throat and the cowl lip. The shock waves from the cowl lip and the beginning of the ramp still need more time to arrive at the final state. However, it could be appreciated how the flow slows down after it arrives at the cowl lip, Mach number goes from 2 to $\sim 1.7$. The multiple shock waves interactions through the cowl lip generate instabilities which cause the appearance of a NaN value inside the throat. This produces that the simulation stops and it is not able to continue. In addition, it is
Figure 29 – Visualization airflow’s Mach field in the ramjet intake.

an acceleration of the flow inside the throat, which is not the desired behaviour, but due to the simulation has not converged, it can not be concluded if this will be the final state of the flow inside the throat.

In order to achieve a final convergence, it has been meshed very fine and reduced the step time to 1e-11, which is very small and makes that a simulation needs several months to finish. However, NaN values have been generated inside the throat, they are the dark zones that you can see in the figure. Unfortunately, after several tries and months of work, it has not been possible to achieve the convergence of the simulation.
Aerodynamic forces acting on a body:

\[
F = \iiint dF = -\iiint pdS + \iiint \tau_m \ dS
\]

(23)

In inviscid flow \( \iiint \tau_m dS = 0 \). Then,

- \( L = y \) component of \(-\iiint pdS\)

- \( D = x \) component of \(-\iiint pdS\)

- \( dS = n \ dS \)

We need to decompose the force which contributes to Lift and Drag in the \( x \) and \( y \) components:
1. \( F = P \times S \)
   In our case \( F = P \times c \rightarrow F = P \left( \frac{l}{2} \right) \times \frac{1}{\cos\beta} \)
   Note that,
   \( a = \frac{l}{2} \)
   \( b = a \times \tan\beta = \frac{l}{2} \times \tan\beta \)
   \( l = 2c \times \cos\beta \rightarrow c = \frac{l}{2} \times \frac{1}{\cos\beta} \)

2. \[
\begin{align*}
F_x &= P_x \times \frac{l}{2} \times \frac{1}{\cos\beta} \times \sin\beta \\
F_y &= P_y \times \frac{l}{2} \times \frac{1}{\cos\beta} \times \cos\beta 
\end{align*}
\]

3. \[
\begin{align*}
F_x &= P_x \times \frac{l}{2} \times \tan\beta \\
F_y &= P_y \times \frac{l}{2}
\end{align*}
\]

4. \[
\begin{align*}
F_x &= P_x \times b \\
F_y &= P_y \times a
\end{align*}
\]

5. \[
\begin{align*}
D &= F_x \times \cos\alpha + F_y \times \sin\alpha \\
L &= -F_x \times \sin\alpha + F_y \times \cos\alpha
\end{align*}
\]

Oblique Shock Waves and Prandtl-Meyer Expansion
Handmade Computations

Through "Oblique shock Properties" \( \beta \), the shock wave angle is obtained:

- \( M_{n1} = M_1 \times \sin\beta \)

- \( \frac{P_2}{P_1} = 1 + \frac{2 \times \gamma}{\gamma + 1} \left( M_{n1}^2 - 1 \right) \rightarrow P_2 \)

- \( M_{n2}^2 = \frac{M_{n1}^2}{\left( \frac{2\gamma}{(\gamma - 1)} \right) \left( M_{n1}^2 - 1 \right)} \rightarrow M_{n2} \)
Figure 31 – Oblique Shock Waves and Prandtl-Meyer Expansion diagram.

- \( M_2 = \frac{M_{\infty}^2}{\sin(\beta - \theta)} \rightarrow M_2 \)

Through Prandtl-Meyer function and Mach angle obtained by the Table A5 in Anderson.

- \( M_2 \rightarrow \nu(M_2) \) interpolate table values if it is necessary.
- \( \psi = \nu(M_3) - \nu(M_2) \rightarrow \nu(M_3) = \psi + \nu(M_2) \rightarrow \nu(M_3) \)

with \( \nu(M_3) \rightarrow \) Go Table A5 Anderson \( \rightarrow M_3 \)

- \[
\frac{P_2}{P_3} = \left[ \frac{1 + \frac{\gamma - 1}{2} \cdot M_2^2}{1 + \frac{\gamma - 1}{2} \cdot M_1^2} \right]^{\frac{\gamma}{\gamma - 1}} \rightarrow P_3
\]
VARIABLE FIELDS

In this section the reader can visualize each of the four variables studied in Section 3: pressure, Mach, density and specific linear momentum.

\( \alpha = 0^\circ \)

Figure 32 – Visualization of simulation results at Mach 2 and AoA=0 deg

\( \alpha = 2.5^\circ \)

Figure 33 – Visualization of simulation results at Mach 2 and AoA=2.5 deg
\( \alpha = 5^\circ \)

(a) Pressure field  
(b) Mach field

(c) Density field  
(d) Flow Velocity field

**Figure 34** – Visualization of simulation results at \( M=2 \) and AoA=5 deg 

\( \alpha = 7.5^\circ \)

(a) Pressure field  
(b) Mach field

(c) Density field  
(d) Flow Velocity field

**Figure 35** – Visualization of simulation results at \( M=2 \) and AoA=7.5 deg
$\alpha = 10^\circ$

(a) Pressure field  (b) Mach field

(c) Density field  (d) Flow Velocity field

**Figure 36** — Visualization of simulation results at $M=2$ and $AoA=10$ deg
In this section, you can see in the following figures, Fig.37,38 and 39, the distribution of the variables, for the airfoil at 5°, 7.5° and 10° of angle of attack, which have not been showed in Section 3, along the chord of the airfoil.

• $\alpha=5^\circ$

**Figure 37** – Pressure, Mach, Density and Specific linear momentum distribution at $M=2$ and $\text{AoA}=5^\circ$
\( \alpha = 7.5^\circ \)

Figure 38 – Pressure, Mach, Density and Specific linear momentum distribution at \( M=2 \) and \( A_oA=5^\circ \)
\[ \alpha = 10^\circ \]

**Figure 39** – Pressure, Mach, Density and Specific linear momentum distribution at M=2 and AoA=10°
NO CONVERGED SIMULATIONS

\[ \alpha = 12.5^\circ \]

**Figure 40** – Visualization of Mach field at M=2 and AoA=12.5°

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Simulation Data</th>
<th>Relative Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>UP</td>
<td>DOWN</td>
<td>UP</td>
</tr>
<tr>
<td>( \theta )</td>
<td>25.96°</td>
<td>48.98°</td>
<td></td>
</tr>
<tr>
<td>( P_2(Pa) )</td>
<td>64997.76</td>
<td>252272.32</td>
<td></td>
</tr>
<tr>
<td>( M_2 )</td>
<td>2.28</td>
<td>1.34</td>
<td></td>
</tr>
<tr>
<td>( \rho_2 \left( \frac{kg}{m^3} \right) )</td>
<td>0.89</td>
<td>2.30</td>
<td></td>
</tr>
<tr>
<td>( \rho u_2 \left( \frac{kg}{m^2 \cdot s} \right) )</td>
<td>650.79</td>
<td>1204.58</td>
<td></td>
</tr>
<tr>
<td>( P_3(Pa) )</td>
<td>33421.26</td>
<td>153721.29</td>
<td></td>
</tr>
<tr>
<td>( M_3 )</td>
<td>2.71</td>
<td>1.68</td>
<td></td>
</tr>
<tr>
<td>( \rho_3 \left( \frac{kg}{m^3} \right) )</td>
<td>0.55</td>
<td>1.61</td>
<td></td>
</tr>
<tr>
<td>( \rho u_3 \left( \frac{kg}{m^2 \cdot s} \right) )</td>
<td>436.94</td>
<td>989.53</td>
<td></td>
</tr>
</tbody>
</table>

**Table 15** – Table M=2 AoA=12.5°

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
<th>Real Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>( Drag(N) )</td>
<td>77390.53</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( Lift(N) )</td>
<td>296686.45</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_D )</td>
<td>0.0498</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_L )</td>
<td>0.5312</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Table 16** – Aeroforces table M=2 AoA=12.5°
\( \alpha = 15^\circ \)

**Figure 41** – Visualization of Mach field at \( M=2 \) and \( \text{AoA}=15 \, \text{deg} \)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Simulation Data</th>
<th>Relative Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>UP</td>
<td>DOWN</td>
<td>UP</td>
</tr>
<tr>
<td>( \theta )</td>
<td>24.85(^\circ)</td>
<td>53.42(^\circ)</td>
<td></td>
</tr>
<tr>
<td>( P_2 (Pa) )</td>
<td>55527.36</td>
<td>288052.74</td>
<td></td>
</tr>
<tr>
<td>( M_2 )</td>
<td>2.38</td>
<td>1.21</td>
<td></td>
</tr>
<tr>
<td>( \rho_2 \left( \frac{\text{kg}}{\text{m}^3} \right) )</td>
<td>0.80</td>
<td>2.50</td>
<td></td>
</tr>
<tr>
<td>( \rho u_2 \left( \frac{\text{kg}}{\text{m}^2 \cdot \text{s}} \right) )</td>
<td>593.68</td>
<td>1215.52</td>
<td></td>
</tr>
<tr>
<td>( P_3 (Pa) )</td>
<td>27882.91</td>
<td>175509.56</td>
<td></td>
</tr>
<tr>
<td>( M_3 )</td>
<td>2.83</td>
<td>1.56</td>
<td></td>
</tr>
<tr>
<td>( \rho_3 \left( \frac{\text{kg}}{\text{m}^3} \right) )</td>
<td>0.49</td>
<td>1.76</td>
<td></td>
</tr>
<tr>
<td>( \rho u_3 \left( \frac{\text{kg}}{\text{m}^2 \cdot \text{s}} \right) )</td>
<td>390.43</td>
<td>1027.64</td>
<td></td>
</tr>
</tbody>
</table>

**Table 17** – Table \( M=2 \) \( \text{AoA}=15^\circ \)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
<th>Real Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \text{Drag(N)} )</td>
<td>109818.01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( \text{Lift(N)} )</td>
<td>362639.07</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_D )</td>
<td>0.0642</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_L )</td>
<td>0.6496</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Table 18** – Aeroforces table \( M=2 \) \( \text{AoA}=15^\circ \)
\[ \alpha = 17.5^\circ \]

**Figure 42** – Visualization of Mach field at \( M = 2 \) and \( \text{AoA} = 17.5^\circ \)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Simulation Data</th>
<th>Relative Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \theta )</td>
<td>UP 23.19 DOWN 60.40</td>
<td>UP ( \text{AoA} )</td>
<td></td>
</tr>
<tr>
<td>( P_2 ) (Pa)</td>
<td>43622.11</td>
<td>340582.35</td>
<td></td>
</tr>
<tr>
<td>( M_2 )</td>
<td>2.54</td>
<td>1.03</td>
<td></td>
</tr>
<tr>
<td>( \rho_2 \left( \frac{\text{kg}}{m^3} \right) )</td>
<td>0.67</td>
<td>2.77</td>
<td></td>
</tr>
<tr>
<td>( \rho u_2 \left( \frac{\text{kg}}{m^2 \cdot \text{s}} \right) )</td>
<td>514.11</td>
<td>1180.13</td>
<td></td>
</tr>
<tr>
<td>( P_3 ) (Pa)</td>
<td>23090.19</td>
<td>191510.72</td>
<td></td>
</tr>
<tr>
<td>( M_3 )</td>
<td>2.96</td>
<td>1.46</td>
<td></td>
</tr>
<tr>
<td>( \rho_3 \left( \frac{\text{kg}}{m^3} \right) )</td>
<td>0.43</td>
<td>1.84</td>
<td></td>
</tr>
<tr>
<td>( \rho u_3 \left( \frac{\text{kg}}{m^2 \cdot \text{s}} \right) )</td>
<td>346.76</td>
<td>1025.65</td>
<td></td>
</tr>
</tbody>
</table>

**Table 19** – Table \( M = 2 \) \( \text{AoA} = 17.5^\circ \)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
<th>Real Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag(N)</td>
<td>153507.94</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Lift(N)</td>
<td>437707.57</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_D )</td>
<td>0.0902</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_L )</td>
<td>0.7852</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Table 20** – Aeroforces table \( M = 2 \) \( \text{AoA} = 17.5^\circ \)
\( \alpha = 20^\circ \)

![Visualization of Mach field at M=2 and AoA=20°](image)

**Figure 43** – Visualization of Mach field at M=2 and AoA=20°

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Simulation Data</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \theta )</td>
<td>22.63° detached</td>
<td>UP DOWN</td>
<td></td>
</tr>
<tr>
<td>( P_2(Pa) )</td>
<td>39813.65</td>
<td>NaN</td>
<td></td>
</tr>
<tr>
<td>( M_2 )</td>
<td>2.60</td>
<td>NaN</td>
<td></td>
</tr>
<tr>
<td>( \rho_2 \left( \frac{kg}{m^3} \right) )</td>
<td>0.63</td>
<td>NaN</td>
<td></td>
</tr>
<tr>
<td>( \rho u_2 \left( \frac{kg}{m^2 * s} \right) )</td>
<td>486.43</td>
<td>NaN</td>
<td></td>
</tr>
<tr>
<td>( P_3(Pa) )</td>
<td>18991.55</td>
<td>NaN</td>
<td></td>
</tr>
<tr>
<td>( M_3 )</td>
<td>3.09</td>
<td>NaN</td>
<td></td>
</tr>
<tr>
<td>( \rho_3 \left( \frac{kg}{m^3} \right) )</td>
<td>0.37</td>
<td>NaN</td>
<td></td>
</tr>
<tr>
<td>( \rho u_3 \left( \frac{kg}{m^2 * s} \right) )</td>
<td>306.24</td>
<td>NaN</td>
<td></td>
</tr>
</tbody>
</table>

**Table 21** – Table M=2 AoA=20°

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
<th>Real Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag(N)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Lift(N)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_D )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_L )</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Table 22** – Aeroforces table M=2 AoA=20°
VARYING THICKNESS

In this section, AoA at 2.5° is fixed. Then, the deflection angle of the airfoil (θ) is changed, in order to change the thickness (t) of the airfoil.

• θ=2.5° and t=0.087

Figure 44 – Visualization of Mach field at M=2, AoA=2.5° and θ =2.5°

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Simulation Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>β</td>
<td>Flat Plate</td>
<td>31.8°</td>
</tr>
<tr>
<td>P₂(Pa)</td>
<td>101325</td>
<td>133283.03</td>
</tr>
<tr>
<td>M₂</td>
<td>2</td>
<td>1.82</td>
</tr>
<tr>
<td>ρ₂((\text{kg/m}^3))</td>
<td>1.225</td>
<td>1.49</td>
</tr>
<tr>
<td>ρ₂u₂((\text{kg/m}^2\cdot\text{s}))</td>
<td>833.72</td>
<td>960.02</td>
</tr>
<tr>
<td>P₃(Pa)</td>
<td>75723.68</td>
<td>101361.93</td>
</tr>
<tr>
<td>M₃</td>
<td>2.19</td>
<td>1.99</td>
</tr>
<tr>
<td>ρ₃((\text{kg/m}^3))</td>
<td>0.99</td>
<td>1.22</td>
</tr>
<tr>
<td>ρ₃u₃((\text{kg/m}^2\cdot\text{s}))</td>
<td>710.12</td>
<td>833.07</td>
</tr>
</tbody>
</table>

Table 23 – Table M=2, AoA=2.5° and delta=2.5°

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag(N)</td>
<td>5016.63</td>
<td>5014.16</td>
</tr>
<tr>
<td>Lift(N)</td>
<td>57377.25</td>
<td>54391.08</td>
</tr>
<tr>
<td>C_D</td>
<td>0.0088</td>
<td>0.0088</td>
</tr>
<tr>
<td>C_L</td>
<td>0.1012</td>
<td>0.0959</td>
</tr>
</tbody>
</table>

Table 24 – Aeroforces table M=2 AoA=2.5° and θ=2.5°
\( \theta = 7.5^\circ \) and \( t = 0.2611 \)

**Figure 45** – Visualization of Mach field at \( M = 2 \), \( \text{AoA} = 2.5^\circ \) and \( \theta = 7.5^\circ \)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Simulation Data</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>UP</td>
<td>DOWN</td>
</tr>
<tr>
<td>( \beta )</td>
<td>34.30(^\circ)</td>
<td>36.81(^\circ)</td>
</tr>
<tr>
<td>( P_2(Pa) )</td>
<td>133283.03</td>
<td>172914.77</td>
</tr>
<tr>
<td>( M_2 )</td>
<td>1.82</td>
<td>1.64</td>
</tr>
<tr>
<td>( \rho_2 \left( \frac{kg}{m^3} \right) )</td>
<td>1.49</td>
<td>1.79</td>
</tr>
<tr>
<td>( pu_2 \left( \frac{kg}{m^2 \cdot s} \right) )</td>
<td>960.02</td>
<td>1078.90</td>
</tr>
<tr>
<td>( P_3(Pa) )</td>
<td>55565.58</td>
<td>76028.91</td>
</tr>
<tr>
<td>( M_3 )</td>
<td>2.38</td>
<td>2.17</td>
</tr>
<tr>
<td>( \rho_3 \left( \frac{kg}{m^3} \right) )</td>
<td>0.80</td>
<td>0.99</td>
</tr>
<tr>
<td>( pu_3 \left( \frac{kg}{m^2 \cdot s} \right) )</td>
<td>593.41</td>
<td>706.96</td>
</tr>
</tbody>
</table>

**Table 25** – Table \( M = 2 \) \( \text{AoA} = 2.5 \) deg and \( \theta = 7.5^\circ \)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Predicted Data</th>
<th>Approximated Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \text{Drag}(N) )</td>
<td>25367.50</td>
<td>25987.81</td>
</tr>
<tr>
<td>( \text{Lift}(N) )</td>
<td>58530.14</td>
<td>49179.49</td>
</tr>
<tr>
<td>( C_D )</td>
<td>0.0451</td>
<td>0.0462</td>
</tr>
<tr>
<td>( C_L )</td>
<td>0.1040</td>
<td>0.0874</td>
</tr>
</tbody>
</table>

**Table 26** – Aeroforces table \( M = 2 \) \( \text{AoA} = 2.5 \) deg and \( \theta = 7.5^\circ \)

As the reader can see in Fig.44 and 45, the simulations still need time to converge. For this reason, in Tables 23 and 25, there are values that are still far from the theoretical ones. Nevertheless, it could be seen how drag increases with the thickness of the airfoil as theory predicts. For this reason, we will want sharp airfoils, but we will have taken into account the forces that actuate on the structure in order to avoid structural problems.
In this section, the diamond airfoil of the original configuration is studied at subsonic velocity, $M = 0.9$, as you can see in Fig. 46. The diamond airfoil is studied at this Mach number, because the wing has to pass the subsonic regime in order to achieve the supersonic velocity, so study our airfoil in this configuration is necessary. In addition, it is also a transonic regime, this regime produces a lot of instabilities which makes suffer the structure of the airframe. For this reason, always we want to surpass the regime as fast as possible.

![Figure 46](image.png) 

**Figure 46** – Diamond airfoil at subsonic velocity.

In the previous figure you can see that the flow is subsonic in all the domain, but a bubble of supersonic velocity is generated at the middle of the upper surface of the airfoil. This supersonic velocity is generated because a favorable pressure gradient accelerates the flow in the corner, but after the shock wave return to be subsonic.

In the following figure, Fig. 47, you can see the distribution of the variables (pressure, Mach, density and specific linear momentum) along the chord of the airfoil. Note that at the second half of the chord the the distribution of the variables changes drastically two times: before the shock wave and after it. So, this transonic flow produces an increase of the pressure, density and specific linear momentum and decreases the Mach drastically in this region of the airfoil. This shock at the second half of the airfoil will cause instabilities and reduce the aerodynamic performance of the airfoil.

A diamond airfoil at subsonic velocity has a poor performance, in order to improve it a swept angle has to be introduced to the wing. The swept angle increases $C_L$ and decreases $C_D$, so the lift to drag ratio is increased. Besides, it allows a better aerodynamic performance at take-off and landing, maneuvers that are make at subsonic regime. However, if the swept angle is going to be introduced, we will
Figure 47 – Pressure, Mach, Density and Specific linear momentum distribution at M=0.9 and AoA=0 deg

have to make a 3D simulation. Then, the possible physic phenomenology which will appear has to be studied in order to make the correct simulation setup. In addition, a 3D simulation needs a lot of time, we are talking of months, so we have to look if the available resources allows this type of simulation.
BUSEMANN’S BIPLANE

Busemann’s Biplane is a conceptual airplane design invented by Adolf Busemann which avoids the formation of shock waves and thus does not create a sonic boom, so the wave drag is null. In this chapter, this solution for a supersonic airframe is presented because is an excellent idea for the future of the supersonic jets and it has to be investigated.

Concretely, it consists of two triangular cross-section plates a certain distance apart, with the flat sides parallel to the fluid flow. Thanks to that the spacing between the plates is sufficiently large that the flow does not choke and supersonic flow is maintained between them. In addition, the flat upper and lower surfaces generate no shock waves because the flow is parallel.

(a) Busemann’s Biplane under off-design conditions [34].

(b) Busemann’s Biplane under design conditions [34].

Figure 48 – Busemann’s Biplane CFD simulations.

In the previous figure, Fig.48, you can see the theoretical behaviour of the geometry. In the first figure, Fig.49(a), you can see the biplane under-off design conditions, this means that the oblique shocks are not perfectly symmetric, this causes that the wave drag is not cancelled. However, if the design conditions are created, it will create a perfect symmetric system of shock waves which will cause the exit is wave-free, no wave drag.

(a) CFD simulation of Busemann’s Biplane under off-design conditions.

(b) CFD simulation of Busemann’s Biplane under design conditions.

Figure 49 – Busemann’s Biplane CFD simulations.

It is like if we cut our previous diamond airfoil in two symmetric parts, so it has
been done and simulated using Nektar++, as you can see in Fig. 49. In Fig.49(b) it has been simulated under off-design conditions and it has reproduced the expected behaviour, the oblique shock waves were not cancelled so the at the exit of the system, wave drag is created. Then, when the design conditions are simulated, Fig.49(b), there is a small slip of the shock wave which produces that the oblique shocks waves were not cancelled totally, so a quantity of wave drag is still generated. It could be produced due to a small error by the software, it could be fixed using a finer mesh or reducing the step time.

![Figure 50 - Busemann's Biplane concept](image)

**Figure 50** – Busemann’s Biplane concept [2].

Hence, it shows that the Busseman’s biplane is a good concept, but it is very sensitive to a small error that could cause that the system of shock waves were not cancelled. Therefore, it needs to be modified in order to improve its aerodynamic performance and become a real solution. Besides, the flat external surfaces and internal symmetry also mean that the airfoil does not produce any lift at the design point zero angle of attack.

One of the possible solutions to avoid break the design conditions is to use movable parts which could be adapted at cruise and avoid the creation of the sonic boom.