Escuela Politécnica Superior de Jaén

Grado en Ingeniería Mecánica

UNIVERSIDAD DE JAÉN

Escuela Politécnica Superior de Jaén

Trabajo Fin de Grado

ESTUDIO AERODINÁMICO DE LAS ALAS DE UNA AERONAVE

Alumno: Bali Rania

Tutor: Prof. D. Miguel Angel Rubio Paramio
Prof. D. Jorge M. Mercado Colmenero

Dpto: Ingeniería Gráfica, Diseño y Proyectos

Septiembre, 2018

Universidad de Jaén
Escuela Politécnica Superior de Jaén
Departamento de Ingeniería Gráfica, Diseño y Proyectos

Don Miguel Angel Rubio Paramio , tutor del Proyecto Fin de Carrera titulado: Estudio aerodinámico de las alas de una aeronave, que presenta Bali Rania, autoriza su presentación para defensa y evaluación en la Escuela Politécnica Superior de Jaén.

Jaén, Septiembre de 2018

El alumno: Los tutores:

Bali Rania

Miguel Angel Rubio Paramio

Jorge Manuel Mercado Colmenero

Dedication

To my dear father Houcine

I owe it all, during my studies, he never stopped supporting me and encouraging me to persevere.
He is at the base of my success by his sacrifice and his love.

I hope I can realize today one of his dreams and always live up to his expectations. I dedicate it, in testimony of my deepest love, this modest work.

To my dear mother Sabeh

For her immeasurable sacrifices and her infinite love. May God keep it her to me so that her prayers protect me and her eyes follow my destiny.

No word would be loquacious enough to show the feelings of gratitude I feel for her.

To my best friends dadou, misou and leila

I find no words to thank you for your support throughout my studies, for supporting me and for believing in me. I hope that you will find in this work all my gratitude and all my love.

To my brothers Walid, Tarek and Yacine

I wish them good life with my sincere gratitude, and a lot of success and happiness.

Acknowledgments

Our awards are for all those who have contributed directly or indirectly to the accomplishment of this project thanks to their support, their friendliness, and their technical knowledge.

I would first like to thank the teachers of the University college of Aviation and Technology of Tunisia (ESAT) and University of Jaén (UJA) for their support throughout our courses.

I would like to thank my supervisors Mr Miguel Angel Rubio Paramioand and Mr Talel Ben Mbarek for their quality of the supervision that he assured me, for their availability, their support and their valuable advice that they has given me throughout this internship.

And it is with gratitude that I thank all those who contributed to the accomplishment of this work and the accomplishment of my mission.

Thank you to the members of the jury, who kindly attended the defense of this internship report.

Table of Contents

List of figures

List of tables

General introduction

Flows around objects are a phenomenon that occurs frequently in practice. They are indispensable in the mechanical and thermal design of many engineering systems such as: aircraft, automobiles, buildings, including electronics. The identification and study of hydro and aerodynamic phenomena that arise in the wake of an obstacle remains a subject of current interest in various fields. Indeed the knowledge of the turbulent structures generated last these obstacle and their different regimes is of paramount utility in the design of the structures exposed to the flows of fluids. The choice of the study of the obstacles cylindrical and square results from their geometric simplicities allowing to have experimental and numerical facilities. Much research has been done to model the flow around the obstacles and precisely around the wing for this purpose several experiments in the field have been carried out and confronted with numerical methods. For the latter a wide range of mathematical methods have been developed in order to approach the reality of the flow and to provide the maximum of information that can occur. For our case, the fluent code, which represents a digital simulation tool to simulate a flow around three different wings? This code of calculation finds more and more place in different fields of industry especially in the option of flows of fluid and transfer of heat.

In this context, the work consists of numerically analyzing the flow of a fluid around the wing of the aircraft. The finite element method is used for the spatial discretization of the partial differential equations written for the fluid. The systems of equations obtained are solved using the 'ansys' Software.

Chapter 1 :

Bibliographic studies

Introduction

In this chapter we will study a bibliographical synthesis that includes several works done in the area of aerodynamics.

1- Historic

It has become clear that the research is numerous and rich, and in addition, they are dominant. Here are some examples of some searchers:

Hamdullah Ozogul: studied the flow of stress-threshold fluids around a cylindrical obstacle in a confined environment. A viscoplastic numerical comparison was performed throughout the study using the finite volume method. The numerical model for describing the behavior of the threshold fluid is based on the Herschel-Bulkley law. The study is based on an elastohydrodynamic model of slippage established in the literature. The effect of sliding has been demonstrated on essential characteristics of the flow. Hamdullah Ozogul has shown that the effect of slippage has been demonstrated on essential characteristics of flow. He found that the flow generally becomes more confined by sliding effect by decreasing the shear zone around the cylinder. This same flow regime was then analyzed experimentally in a confined environment with a rough obstacle.

BEDJEGHIT EL-HOUAS: studied the laminar and turbulent flows around the obstacles. The different geometric shapes of the obstacles retained are the obstacle with square section and cylindrical section. He did the study in 2D in the case of the square configuration and in 2D and 3D in the case of the cylinder. According to the study of the flow in the case of the cylindrical obstacle, he compared the numerical simulation carried out on the Fluent Software and the experimental one made on a test bench. This work enabled him to draw observations for the different Reynolds numbers, especially in laminar flow.

A satisfactory result obtained showing the evolution of the friction coefficient profiles along the generatrices of the cylinder and square.

Marcin Huptas , Witold Elsner : Worked on a stationary and unsteady flow around two square obstacles.They presented with the use of FLUENT Numerical results concerning the flow around a square obstacle in the first case, and around two square obstacles on the same line in the second case. For the first resolution, they showed that the thickness of the boundary layer can affect the shear layer near the wall. And for the second resolution, they found that the union of the two swirl zones downstream of the first cube and upstream of the second makes the flow more turbulent.

P.f. zhang (2006) : Performed a numerical study of two-dimensional laminar flow around an ascending stem and a circular cylinder. The results obtained using fluent simulation Software demonstrate that the average drag coefficient and the coefficient of lift fluctuation of the cylinder can be reduced by an ascending rod.

R. Belakrom (2007) : Established the LES model (large Eddy simulation) which is studied by the finite element method to simulate the unsteady and turbulent flow of an incompressible fluid around a cylinder. they found that the phenomenon of vortex burst is clearly highlighted.

K.Lam -L. Zou (2009) : Digitally and experimentally studied turbulent flows around four cylinders in an integrated square configuration with different spacing ratios selected. The results obtained from the speed fluctuation are found to be similar to the numerical results.

F.Meddane (2007) : Have studied the flow of fluid around an obstacle, such as a profile wing NACA 0021. They have shown that the use of the shutter has a direct fluence on the pressure distribution and thus on the aerodynamic forces of the profile.

Joslin (1999) : studied adaptable structures that will also reduce the weight and number of aircraft parts. The combination of adaptable structure, actuators, sensors, and controllers contemplates active and responsive control of flow around the wings which could also increase the safety of the devices.

Bolonkin et al. (1999) : argued that a significant reduction in the cost of operations could be achieved by using an adaptable wing configuration capable of meeting a number of requirements for both take-off and landing. The configuration of the wing could also be used as fins and be able to modify its arch during the cruising flight. The adaptable structure is here described as being a structure that modifies its shape while maintaining a smooth and continuous geometry.

Prandtl (1904) : exhibited several flow control experiments.

He was the first to study the suction effect of the boundary layer around a cylinder.

Comolet (1976) : made a description of the phenomena of delamination of the boundary layers on two-dimensional profiles which are classified into three categories: thick, medium and thin.

Stanewsky (2000) : realize configurations mixing firstly, a suction and a geometrical adaptation of the wall, secondly a ventilation (blowing and aspiration) and thirdly a ventilation and a suction at the back. Since the context here is a transonic flow, the geometric adaptation of the wall is local (in the form of a hump) and makes it possible to minimize the shock wave. Ventilation also plays a role in minimizing the shock wave by rebalancing the pressure on both sides of the shock. The suction allows a direct control of the boundary layer either to stabilize it upstream of the impact or to minimize the detachment when the suction is located behind the shock.

Tilman (2000) : showed on a NACA 4412 profile that an excessive increase in the speed ratio V R results in a premature dissipation of the verticality and a loss of efficiency of the jets. Ideally, for maximum efficiency, the jets must remain confined in the boundary layer to ensure good persistence of vorticity.

McManus (1997) : showed that the coefficient of lift increases monotonically with the speed of the jets when the ratio V R is greater than two. However, this trend is no longer valid for jet speeds too high.

Batchelor (1967): concluded that the only possibility of behavior of the flow arriving at a sharp angle is the tangential echo tangential to the upstream flow. The external flow then undergoes no deceleration before the separation as in the case of pressure gradient separation: the discontinuity of the wall can be likened to the line of separation for the flow. Nevertheless, Batcheloremet has a reservation with respect to this configuration: "The flow will separate at the level of the discontinuity of the solid wall unless the change of direction of the tangents is small." There is therefore a possibility for the flow to circumvent the sharp angle without taking off.

Hafida.N and Mohamed.S (2010) : conducted a numerical study of the effects of square obstacles on the wind profile. In this project, they found that the influence of the obstacle on the flow of the wind clearly appeared. Also, they showed the evolution of the strong speed gradients that form around the building and close to the ground.

M.M Ouestati (2010) : was interested in the simulation by the different formulations of the function-vorticity to solve the Navir-stoks equation. Studied the flow of regular and transient fluids around a square obstacle. They concluded that this formulation has several advantages, since the pressure limit is eliminated from the government equations and automatically satisfies the continuity equation.

Prandtl (1921) : highlighted the destabilizing effect of vorticity. The instability at transition begins with the growth of sinusoidal perturbations whose existence was demonstrated by Schubart and Skramstad for incompressible flows.

Sendstad & Moin (1992) : Have studied using DNS the generation of these vortices from a vorticity web. They examined the contribution of the different terms of the instantaneous equation of longitudinal vorticity to the creation and evolution of such a sheet towards a longitudinal vortex.

Labarthe (1992) : has studied transitional phenomena in two-dimensional and three-dimensional flows, recalling that the transition is a natural phenomenon that appears in the boundary layer. The transition zone delimits two regions of flow with distinct characteristics:

- The laminar part, located upstream of the transition where the particles have predictable trajectories that can be followed individually,

- The turbulent part, downstream of the transition, where the flow is more disordered, Arnal (1988) : has also conducted studies on boundary layer transition. He has shown that the disturbances responsible for the transition to the turbulent state can be stationary waves (Gortler vortices) or progressive waves (Tollmien-Schlichting instabilities).

Klebanoff et Tidstrom (1972) : studied the influence of a two-dimensional roughness created by a wire of circular section glued to the wall perpendicular to the flow. The role of the wire in the transition process is to locally modify the flow by causing delamination so that the boundary layer does not return to its smooth flat plate characteristics at a distance downstream from the wire. It follows that the transition is advanced to such a point that if the conditions of destabilization are sufficient (for example by placing a wire of large size) the transition point is in the vicinity of the wire. Klebanoff and Tidstrom also indicate that the instability mechanism is of the TS waves type.

Mc Cauley (1966): defined a critical roughness size below which roughness no longer influences the position of the transition. When it is exceeded, the transition moves upstream and reaches roughness.

Savil (1986), Anders & Watson (1985) : showed that the insertion of a thin lamella into the boundary layer resulted in a reduction of friction drag. Originally, most of the studies made by his two researchers were conducted on optimizing the characteristics of these lamellae and their position in the turbulent boundary layer.

Tiederman (1985) : further showed that if the polymer remains confined in the viscous sub-layer, friction reduction does not occur, thus proving that the viscous sub-layer has a passive role in this reduction process. According to these authors, the effects of the injection of the polymer on the flow structure are manifested essentially in the area $<y + <100$.

Bradshaw & Terrel (1969) : studied the three-dimensional boundary layer that develops on a plate placed downstream of a wing in arrow. The measurements revealed a strong anisotropy of the turbulent viscosity: the vector composed by Reynolds voltages -uv and -vw does not align with the direction of the vector formed by the velocity gradients đ U / đ y and đ W / đ y. This result made it possible to make modifications to the classical three-dimensional boundary layer models, which badly represented the physics of the phenomena.

Sicot (2008) : examined the fluctuations of the separation point for a high turbulence flow profile for the study of unsteady static drop characteristics, using pressure measurements and flow visualizations. He proposed a method based on the standard pressure deviation for studying the length of the swirl zone. He concluded that the degree of turbulence has no effect on the generation frequency of Van Karman's alleys. The same result was obtained by spectral analyzes.

Guerri et al [8] : simulated the laminar flow of an incompressible viscous fluid around a wing profile in forced oscillations. The Navier Stokes equations are formulated in ALE and resolved with moving mesh, the hysteresis cycle of the dynamic stall phenomenon has been verified. As it has been shown that the evolution of lift and drag varies with the reduced frequency imposed on the profile.

G. Martinat et al. (2008) : studied the effect of turbulence models of a flow around an oscillating naca00012 profile at a Reynolds number 105 and 106 being made a comparison between 2D and 3D simulation

Windte et al. [5] : studied the laminar flow around a NACA 4402 profile for Reynolds number Re = 6000. The FLOWEer code, based on the resolution of the averaged Navier Stokes equations (RANS methods) is used. This numerical study is carried out with a mobile and deformable mesh. They concluded that propulsive efficiency decreases with increasing reduced frequency

Liu Zhen et al. (2008) : used an experimental device (PIV) Particle Image Velocimetry of water flow around an oscillating NACA0012 aerodynamic profile, and by numerical simulation they used the turbulence model k -ε to study the effect of attack angle

N.Marco (1999) : carried out wing profile shape optimization experiments according to a single criterion: minimization of the drag in the conditions of the cruise or maximization of the lift in the conditions of the take-off.

Tebbiche et al (2014) : used the -Fluent- code and also carried out an experimental test with a wind tunnel to study the effect of V-shaped vortex generators on the airplane profile (NACA 0015), with an air speed of 25 m / s and a Re = 2.6 105. He took into account several parameters, such as position and size, and also the angle of attack. The results obtained showed that there is a drag force reduction of up to 10%.

2- Definition of wings

A wing is a type of fin that produces lift, while moving through air or some other fluid. As such, wings have streamlined cross-sections that are subject to aerodynamic forces and act as an airfoils. A wing's aerodynamic efficiency is expressed as its lift-to-drag ratio. The lift a wing generates at a given speed and angle of attack can be one to two orders of magnitude greater than the total drag on the wing. A high lift-to-drag ratio requires a significantly smaller thrust to propel the wings through the air at sufficient lift.

2-1-The characteristics of the wing

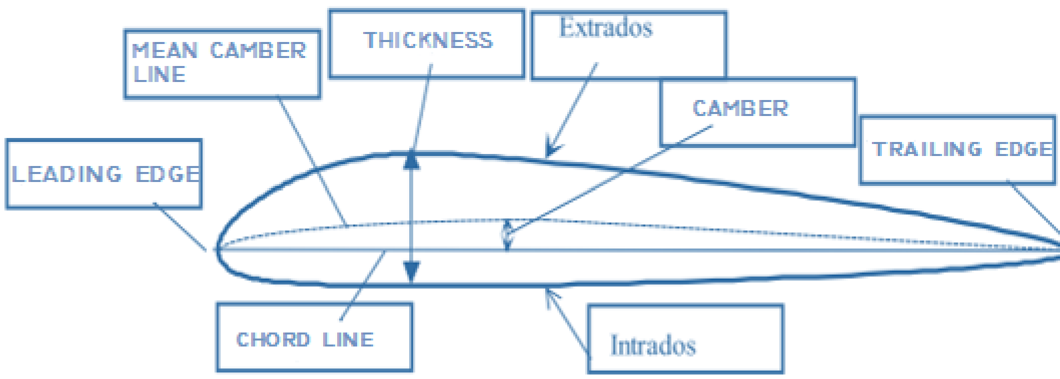


Figure 1: The characteristics of the wing profiles

Leading Edge: Is the part of the wing that first contacts the air. Alternatively it is the foremost edge of an airfoil section

Trailing Edge: The trailing edge designates the rear part of the aerodynamic profile.

Chord Line: Right connecting the leading edge to the trailing edge.

Extrados: This is the line joining the leading edge to the trailing edge from above.

Intrados: This is the line joining the leading edge to the trailing edge from below.

Mean Camber Line: Is the place of the equidistant points of the extrados and the intrados. This rope is usually curved.

Camber: is the maximum distance between the rope and the average line of the profile.

Thickness: The maximum thickness of the wing is the maximum distance between the upper surface and the lower surface.

2-2- Forces exerted on an airplane wing

The forces acting on an airplane wing are:

- The weight.

- The thrust.

- The lift.

-The drag.

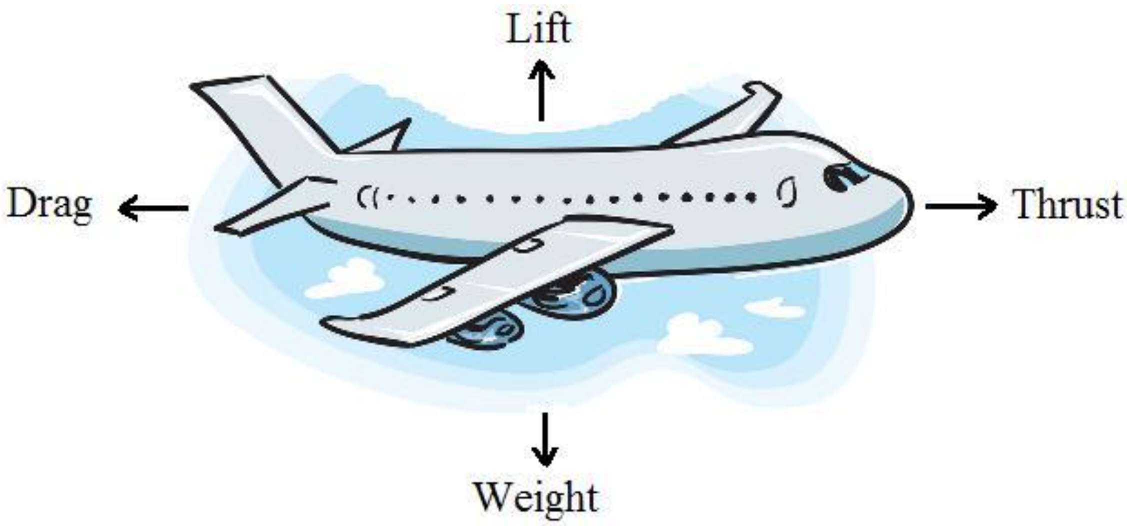


Figure 2 : The forces exerted on an airplane

2-2-1- Weight

This is the effect of gravity on the aircraft, represented by a downwardly directed vector. This weight varies according to the mass of the aircraft. It can be calculated according to the following formula:

$$P=m.g$$

P : weight in N

m : mass in Kg

g : acceleration of gravity in m/s^2 ($g = 9{,}81\ m/s^2$)

2-2-2-The thrust (or traction)

It is the force that makes the airplane move in the air that is traction in the case of a propeller airplane either of propulsion in the case of an airplane with reactors.

The thrust causes the aircraft to move at a certain speed in the air, but this velocity is the velocity relative to the fluid, and not the speed relative to the ground.

This is the reason that planes land and take off in the wind, because it allows to have a relatively high relative speed for a rather low ground speed.

The thrust is represented by a vector directed in the direction of the movement and in parallel. It is transmitted to the aircraft by reactors or propellers and is opposed to drag. It can be calculated according to the formula:

$$\mathbf{F_{p} = v_e \cdot q_m + A_s \cdot (P_s - P_a)}$$

Fp: thrust in newton (N)

Ve: gas ejection speed in (m / s)

qm: mass flow in (kg / s)

As: area of the outlet section of the nozzle in (m2)

Ps: pressure at the outlet of the nozzle in (Pa)

Pa: ambient pressure or outside pressure in (Pa)

2-2-3- Lift

This is the force that allows an aircraft to climb and maintain altitude.

Aerodynamic lift is the component of the force experienced by a moving body in a fluid that is perpendicular to the direction of movement.

The lift Fz in newtons (N) of a wing is:

$$\mathbf{Fz = 1/2 \times S \times \rho \times V^2 \times Cz}$$

P: density of the fluid in kg / m3

V: speed in m / s

S: reference area in m2

CZ: lift coefficient

1/2. ρ. V^2: is the dynamic pressure exerted by the flow and S the frontal surface of the body.

2-2-4- The drag

It is the aerodynamic result which opposes the thrust (see figure 2), and therefore brakes the plane and of following formula

$$\mathbf{Rx = 1/2.\rho.S.V^2.C_x}$$

P: density of the fluid in kg / m3

V: speed in m / s

S: reference area in m2

Cx: coefficient of drag

3-The shape of the wings

The shape of the wings of an airplane is what makes it capable of flying. The wings of the planes are curved at the top and flatter at the bottom. This shape allows air to circulate faster than under the bottom. So, less air pressure is above the wing. This condition causes the wing and airplane to which it is attached to ascend.

- **Right wing**

The straight wings form a right angle with the axis of the fuselage. This type of wing is used only for aerodynes having a relatively low speed (less than Mach 0.7).

Example: MH-1521 Broussard

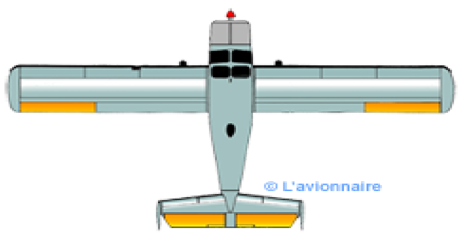


Figure 3 : Right wing

❖ **Elliptical wing**

The advantage of the elliptical wing is the reduction of the curvature at its end resulting in a uniform aerodynamic flow over its entire length and minimal drag in subsonic flight.

The disadvantage of such a wing is flying at low speeds with a faster and more brutal stall, as well as the complexity of the structure.

Example: Speed Spitfire

Figure 4 : Elliptical wing

❖ **Trapezoidal wing**

This high performance wing is characterized by a low elongation with a positive arrow leading edge while the trailing edge has a negative arrow. Like the elliptical wing, the trapezoidal wing has a uniform aerodynamic flow over its entire length but at low speeds the stall is distributed over its entire surface.

Example: Lockheed F-22 Raptor

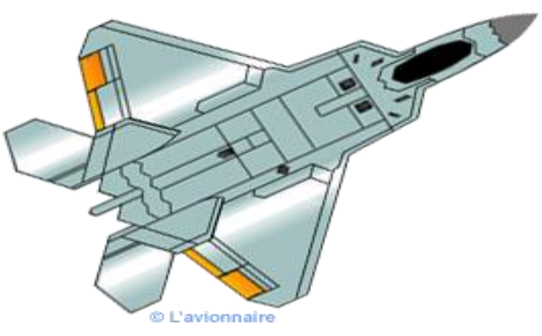


Figure 5 : Trapezoidal wing

❖ **Flying wing**

The flying wing has neither fuselage nor horizontal tail. Certainly as the strategic bomber Northop B2 has more drift. Computer-assisted steering is done through the control surfaces, all located in the wing. Although the concept is old, the flying wing poses problems of stability notably in pitch and yaw.

Example: Northop B2

Figure 6 : Flying wing

4- Winglet

Figure 7: Winglet

The winglet was developed at the NASA Research Center in Langley, United States, in 1974 by American aerodynamicist Richard Whitcomb.

Ailerons are more or less vertical surfaces installed at the end of the wings of an airplane or glider. Its effect is to weaken the vortex that forms at this location and, therefore, reduce drag, so the resistance of the air. The reduction is usually 2 to 3%. The winglet aims to recover some of the swirling energy induced by the elevator. At the end of the wing, the flow of the intrados under relative overpressure tends to pass on the extrados in relative depression, generating a marginal vortex. The latter not only increases the drag of the aircraft but causes turbulence behind the aircraft that persist over long distances and decreases the lift.

The winglet phenomenon decreases lift and increases drag. It creates an extra "wake turbulence". This tourbillon is very dangerous for light aircraft that land or take off on the runway that just used a jumbo jet. It can even be dangerous in flight for a small plane or an ultralight that would pass in the wake of a jumbo jet that has just taken off or landing.

However, a vertical structure at the end of the wing also has its disadvantages. It adds weight, can create vibrations or even create a trail if it is poorly designed.

5- Work purpose

The purpose of this work is to study a numerical simulation that was done using the FLUENT calculation code. For comparison The different fields of velocity, and pressure thus the phenomenon of van-karman between the flow around obstacles square and cylindrical In the three-dimensional case and in particular the flow around the wing. A comparative study was made between the experimental study and our numerical simulation of turbulent flow.

Numerical simulation was done using the FLUENT calculation code. The effect of turbulence on flow has been taken into account using the K-Ω model.

In all the cases considered, the pressure and velocity fields satisfying the physical phenomena such as the development of the boundary layers.

6- Goals and objectives

The main objective of the project is the development of effective numerical methods for the design and optimization of the wing. In order to:

❖ To create multiple shapes, considering the mesh problems.

❖ Make a aerodynamics study of space to compare between two model, to look for solutions for the new model

❖ Compare the models that I did to determine the best results.

Conclusion

This chapter served to put our project in its frame. It allowed us to know the importance of the aerodynamic effect on the shape of the wing and their progression over time.

The following chapter was chosen to present the different wings profiles and the different air flows.

Chapter 2 :

State of art

Introduction

In this chapter makes it possible to contextualize the work. The aerodynamic prerequisites are exposed, and then the flow around a body is described with precision.

1-Definition of the profile and different types of profiles

The profile of an aerodynamic element is the contour of this element in a fluid in relative motion. In the case of an aircraft wing, a propeller blade or a main rotor, the profile is the contour of the cross-section or "section", which may be constant or variable from one end to the other of the element.

There are many profiles that are classified according to their forms by families.

1-1- Symmetrical biconvex

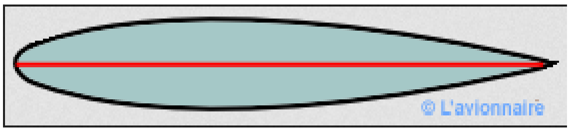


Figure 8 :Symmetrical biconvex

The convex intrados and extrados are symmetrical with respect to the rope.

The middle line is straight and is confused with the rope.

These profiles are used for vertical and horizontal empennages.

1-2- Biconvex asymmetrical

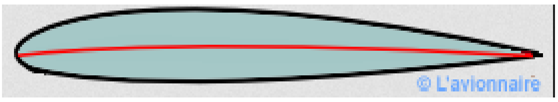


Figure 9 : Biconvex asymmetrical

The curvature of the extrados is more pronounced than that of the intrados.

The average line is single curvature (intrados and extrados convex).

These profiles are the most used for recreational aircraft wings.

1-3- Convex plane

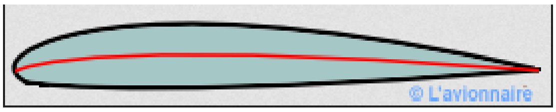


Figure 10 :Convex plane

It is a profile with a relatively flat intrados and a curved extrados (convex). The average line is single curvature.

1-4-Hollow plane

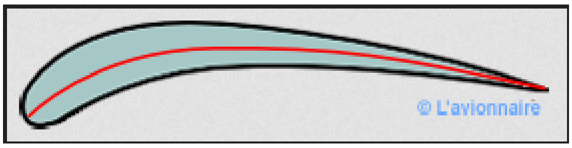


Figure 11 : Hollow plane

The extrados is convex and the concave intrados.

These profiles are very buoyant but generate a significant drag.

This type of profile was once used for gliders.

The average line is single curvature.

1-5-Double curvature

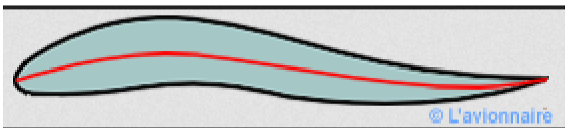


Figure 12 : Double curvature

The average line is double curvature.

The second curvature of the middle line gives stability qualities, hence the term AUTOSTABLE.

This unusual profile is mainly used for flying wings.

1-6-Supercritical

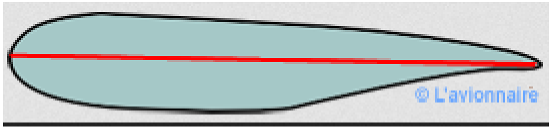


Figure 13 : Supercritical

Double inverted curvature. Extrados very tight (large radius of curvature).

In the zone of the leading edge of the extrados mini radius of curvature.

Strong relative thickness of the order of 17%.

1-7-Laminar profile

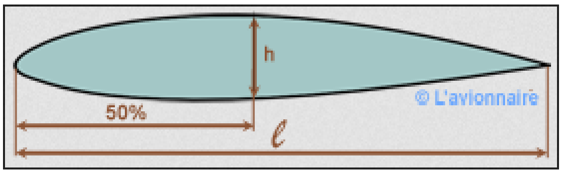


Figure 14 : Laminar profile

It is doubly symmetrical in thickness and length. Its greater thickness instead of being located in the front third of the wing is substantially in the middle of the depth, which delays the appearance of the stall.

He's stability is medium and he's fairly buoyant.

2-Flow of the air

2-1-Introduction

The flight of an airplane is achieved through the interactions between the lift elements (such as wings) and the air mass in which the aircraft moves.

Normally, the plane is moving relative to the air. However, to study the phenomenon of aerodynamics, we will place ourselves in the frame of reference of the plane , that is to say that we consider the plane as immobile and the air on the move: this is the way to proceed in the wind tunnel.

2-2- Definition of the flow

The flow is the movement of air in relation to an object.

2-3-Differents flows

There are three main types of flow:

-Laminar flow

-Turbulent flow

-Swirling flow

2-3-1-Laminar flow

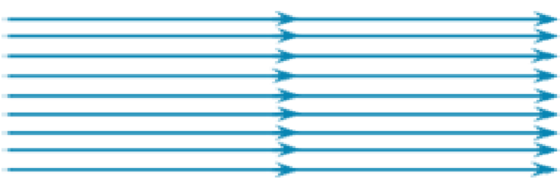


Figure 15 : Laminar flow

The particles of air slide perfectly on each other without exchanges of particles between them. They follow a rectilinear and parallel movement.

Laminar flow is usually the one that is used when you want to fly a plane (because it is more stable and predictable by the equations).

To obtain low air resistance and thus increase aerodynamics, the flow must be as laminar as possible.

2-3-2-Turbulent flow

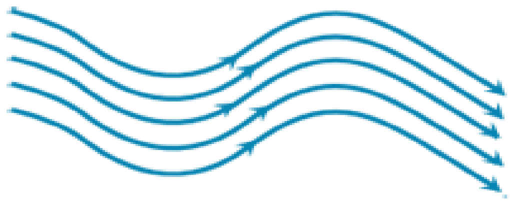


Figure 16 : Turbulent flow

The air particles have almost parallel paths between them, but they are no longer rectilinear, while moving globally in the same direction at the same speed.

2-3-3-Swirling flow

Figure 17 : Swirling flow

The flow is very disordered, the particles mix and do not follow a straight or parallel path, and some particles can go upstream and thus form eddies.

2-4-The flow of air on different objects

2-4-1-Flow of air on a plate

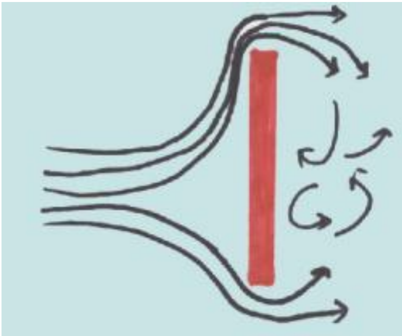


Figure 18 : Flow of air on a plate

The flow is at first laminar, when it arrives on the object, the flow is turbulent, and behind the plate, it is swirling. It creates a depression behind the plate that will hold it backwards. It is this depression that we call air resistance.

2-4-2-Flow of air on an hemisphere

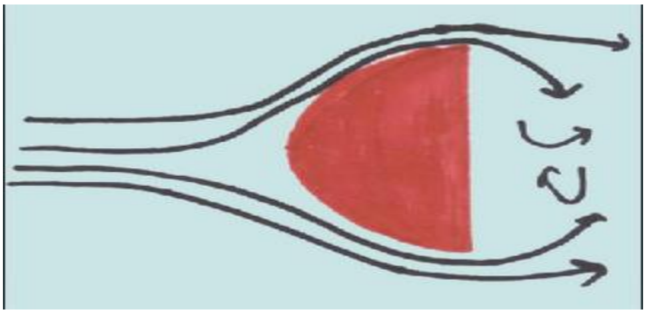


Figure 19 : Flow of air on an hemisphere

We find that the air better circumvents the object, the overpressure decreases but there is still a depression and a swirling area at the back.

2-4-3-Flow of air on a sphere

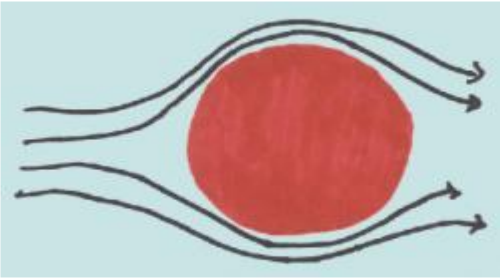


Figure 20 : Flow of air on a sphere

The flow is more easily on the surface of the object, but there is still a slight depression at the back of the sphere.

2-4-4-Flow of air on a beveled object

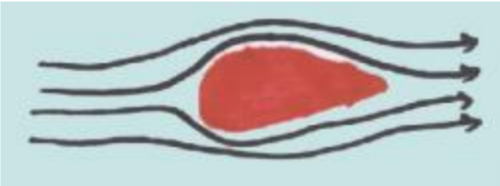


Figure 21 : Flow of air on a beveled object

The air resistance is even lower than the sphere, it is also very similar to the profile of an aircraft wing.

2-4-5- Flow of air on a car

Figure 22 : Flow of air on a car

We see that the air streams are at first very turbulent, then we see in the photograph that a depression forms at the rear of the vehicle.

3- Reynolds number

The Reynolds number (R e) is used in fluid mechanics. It was highlighted in 1883 by Osborne Reynolds. It characterizes a flow, in particular the nature of its regime (laminar, transient and turbulent).

The Reynolds number represents the ratio of inertial forces to viscous forces. This dimensionless number naturally appears by adimensioning the Navier-Stokes equations.

The Reynolds number is defined as follows:

$$Re = \frac{\rho\, u\, L}{\mu} = \frac{u\, L}{\nu}$$

Where:

- $\rho$is the density of the fluid (SI units: kg/m3)
- u is the velocity of the fluid with respect to the object (m/s)
- L is a characteristic linear dimension (m)
- $\mu$is the dynamic viscosity of the fluid (Pa·s or N·s/m² or kg/m·s)
- $\nu$is the kinematic viscosity of the fluid (m²/s).

4- Boundary layer

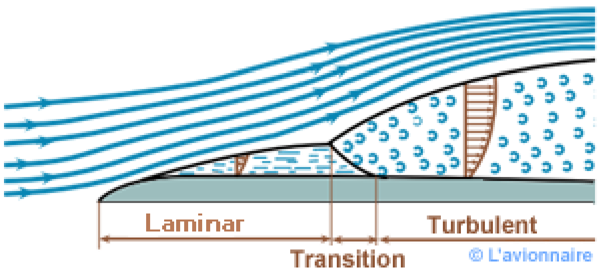


Figure 23 : The boundary layer

The boundary layer is the interface zone between a body and the surrounding fluid during a relative movement between the two, consequence of its viscosity. It is an important element in fluid mechanics, (aerodynamics, hydrodynamics), in meteorology, in oceanography.

It plays a major role in the performance of a bearing surface. Its detachment occurs when the angle of incidence of the wing becomes too great, which corresponds practically to a pitched attitude of the aircraft (landing by example). If this angle is too great, the stall phenomenon occurs: the boundary layer is severely peeled off and the lift can drop significantly, more or less abruptly. This phenomenon is at the origin of many air accidents, the loss of lift which can lead to the loss of control of the aircraft.

- **Description of the boundary layer**

When a real fluid flows along a supposedly fixed wall, the velocities on the wall are zero whereas at infinity (that is to say, far from the obstacle) they are equal to the speed of undisturbed flow. On a normal to the wall, the speed must therefore in all cases vary between 0 and a maximum. The law of variation depends on the viscosity of the fluid which induces a friction between the adjacent layers: the slowest layer tends to slow down the fastest layer which, in return, tends to accelerate it. Under these conditions, a high viscosity equalizes the maximum speeds. On the contrary, if the fluid is not very viscous, the different layers are much more independent: the speed at infinity is maintained until a short distance from the obstacle and there is a greater variation of the speeds in the small thickness boundary layer.

In the first case, it is necessary to use the general equations of the viscous fluid. In the second, simplified boundary equations supplemented by experimental results can be used in the boundary layer. Equations, also simpler, of the perfect fluid applied beyond the wall "fattened" by the boundary layer provide the boundary conditions for the calculation.

In fact, it is not the viscosity itself that intervenes. As always in fluid mechanics, it is a dimensionless number that characterizes the phenomenon: the Reynolds number. This describes the ratio of forces related to speed to friction forces. Thus, instead of increasing the viscosity, a similar phenomenon can be obtained by reducing the speed or dimensions of the obstacle.

- **Side effects of the boundary layer**

The boundary layer can seriously disturb the operation of a jet engine, firstly because of turbulence in the air flow ingested by the engine, and secondly by reducing its efficiency because of the low speed of the engine. The air at the level of the layer.

On the other hand, when the air intake is located along the fuselage (especially in the case of military aircraft), it is most often slightly separated from it to be placed out of the boundary layer.

Conclusion

In this chapter makes it possible to contextualize the work. The aerodynamic prerequisites are exposed, and then the flow around a body is described with precision.

The following chapter was chosen to present the mathematical equations.

Chapter 3 :

Mathematical modeling

Introduction

This chapter makes it possible to contextualize the work .For the study of a problem of flow of a fluid around the wing profile, we present in this chapter the mathematical equations.

1- Navier Stokes equation

The Navier-Stokes equation, established in the 19th century by Frenchman Navier and British Stokes, is an equation that describes the velocity field of a fluid. More precisely, it is a differential equation whose velocity field is the unknown.

The main objective of this equation is to describe the movement of fluids. Since a fluid can be a liquid or a gas, we understand that the Navier-Stokes equation concerns a whole lot of things around us. It can be used, for example, to understand the movements of currents in the oceans, as well as those of large air masses in the atmosphere. It is also used to study the flow of blood in our arteries, and to simulate the path of air around an airplane wing or a car body. This equation is even used in video games to improve the realism of certain scenes.

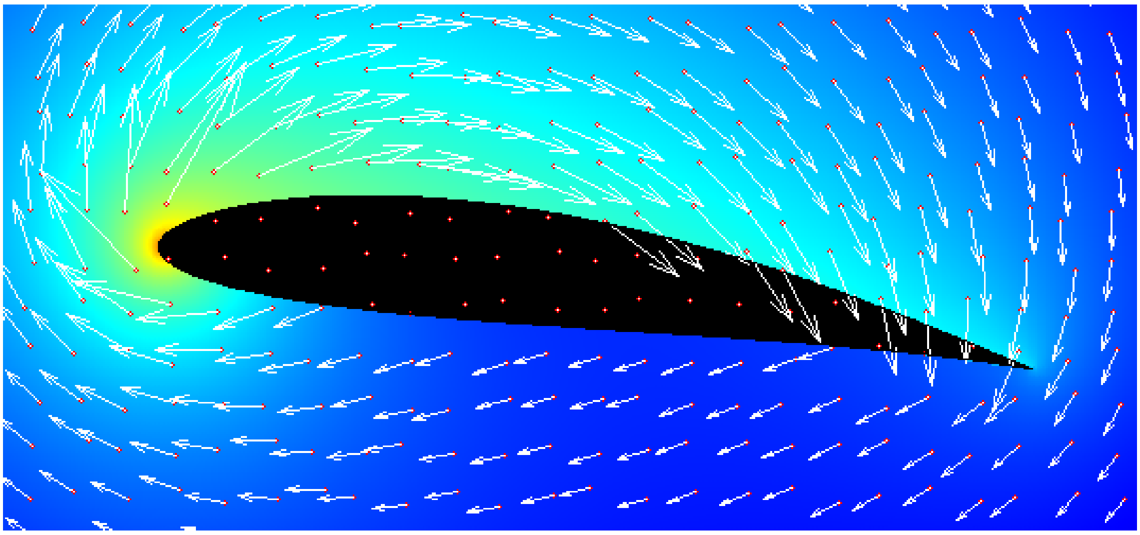


Figure 24 : Air movement around the airplane wing

To correctly describe a fluid in motion, you must know its speed in all points of space. This is called his speed field. The image below shows an example of a velocity field in a fluid around an airplane wing, a field that is traditionally represented with small arrows more or less long, proportional to the speed and oriented in the direction of the flow (here in the soil reference)

❖ **What does the equation look like?**

In a fluid, we will consider two types of forces: pressure forces and viscous forces. The pressure forces are those that come from the fact that a small piece of the fluid is pushed by all the rest of the fluid that surrounds it. The viscous forces are the equivalent of the friction forces for a skier. When a piece of fluid slides on another, there is a friction which brakes it and which is all the more important that the fluid is viscous.

Here is the Navier-Stokes equationfor a fluid:

$$\rho \frac{D\vec{v}}{dt} = \rho(\frac{\partial\vec{v}}{\partial t} + \vec{v}.\vec{\nabla}\vec{v}) = \eta\Delta\vec{v} + \vec{\nabla}P + \vec{f}$$

The Navier-Stokes equation is used to describe the movement of fluids. It means that it applies as well to the movement of olive oil poured from a bottle, as to that of the air when passing an airplane.

In the case of the bottle of oil, the movement of the fluid is all that is more quiet and regular. In the case of air in the passage of the plane, it is total chaos. This is shown in the image opposite, which allows to visualize with the help of colored smoke the kind of movements of the air that takes place on takeoff of a small plane.

Figure 25 : Air motion around an airplane Wing

This chaotic behavior is called turbulence. And this turbulence is the physical counterpart of the non-linearity of the Navier-Stokes equation.

As we said, in some situations (the oil in the bottle), the flow of fluids is done quietly. This is called the laminar diet. It occurs when the fluids are viscous, slow and rather confined.

On the other hand, when the fluids are not very viscous, fast and move over great distances, the flows occur in a chaotic manner and have many eddies: this is called the turbulent regime. The transition between the two can be quite sudden (see my note on the Reynolds number) and experimentally, for a given flow, one knows quite well whether one will be laminar or turbulent.

We would very much like to be able to demonstrate rigorously that for certain values of velocity or viscosity, the solutions of the Navier-Stokes equation will be turbulent, and for others they will be laminar.

2- Continuity equation

Laminar In fluid mechanics, the principle of conservation of mass can be described by the continuity equation in several different forms: conservative local (derived in normal time), local non-conservative (the time derivative follows the particle in its motion), or integral.

Depending on the problems, one or the other of these equations may be used, all being equivalent.

The continuity equation means that the amount of fluid entering is equal to the amount of fluid exiting.

The conservation equation of the mass (the continuity equation) is written in the form:

$$\frac{\partial \rho}{\partial t} + div\vec{v} = 0$$

The local shape of the conservation of the mass applies only on a point M of the volume element đ V of fluid.

We recall the integral form:

$$\frac{\mathrm{d}m}{\mathrm{d}t} = \iiint_V \left(\frac{\partial \rho}{\partial t} + \operatorname{div}\left(\rho\, \overrightarrow{v}\right) \right) \mathrm{d}V = \iiint_V \left(\frac{\mathrm{d}\rho}{\mathrm{d}t} + \rho \operatorname{div} \overrightarrow{v} \right) \mathrm{d}V = 0.$$

We then obtain the continuity equation in two equivalent forms:

* **The conservative form:**

$$\frac{\partial \rho}{\partial t} + \operatorname{div}(\rho \overrightarrow{v}) = 0 \,;$$

* **The non-conservative form:**

$$\frac{\mathrm{d}\rho}{\mathrm{d}t} + \rho \operatorname{div} \overrightarrow{v}.$$

Conservation of the mass and continuity equation: consequence of the continuity equation.

We will apply these equations for particular flows: permanent flows and incompressible flows.

❖ **Case of a permanent flow:**

It is recalled that a permanent flow is a flow whose Euler variables do not depend on time. The expressions of the continuity equations will thus be simplified (we give below the simplified local form, then the simplified global form).

$$\frac{\partial \rho}{\partial t} = 0 \Leftrightarrow \mathrm{div}(\rho \overrightarrow{v}) = 0 \Leftrightarrow \oiint_S \rho \overrightarrow{v} \cdot \overrightarrow{\mathrm{d}S} = 0$$

Either a current tube in which there is a permanent flow. The current tube does not change over time. Simplified global continuity equation allows to write:

$$\oiint_S \rho \overrightarrow{v} \cdot \overrightarrow{n} \,\mathrm{d}S = \underbrace{\iint_{S_1} \rho \overrightarrow{v_1} \cdot \overrightarrow{n_1} \,\mathrm{d}S}_{\text{Entrée}} + \underbrace{\iint_{S_L} \rho \overrightarrow{v_L} \cdot \overrightarrow{n_L} \,\mathrm{d}S}_{\text{Surfaces latérales}} + \underbrace{\iint_{S_2} \rho \overrightarrow{v_2} \cdot \overrightarrow{n_2} \,\mathrm{d}S}_{\text{Sortie}} = 0.$$

So :

$$\oiint_S \rho \overrightarrow{v} \cdot \overrightarrow{n} \,\mathrm{d}S = \iint_{S_1} \rho \overrightarrow{v_1} \cdot \overrightarrow{n_1} \,\mathrm{d}S + \iint_{S_2} \rho \overrightarrow{v_2} \cdot \overrightarrow{n_2} \,\mathrm{d}S = 0 \Rightarrow q_{m_{\text{entrée}}} = q_{m_{\text{sortie}}}$$

In a permanent flow, mass flow is maintained in the sense that mass flow is constant on all sections of a current tube.

❖ **Case of an incompressible flow:**

An incompressible flow is a flow whose fluid has a density constant over time and uniform in space ρ = cte therefore $\partial \rho = 0$

The conservation equation is reduced to:

$div\vec{v} = 0$ (volume conservation)

3- Bernoulli's equation

Bernoulli's theorem, which was established in 1738 by Daniel Bernoulli, is the mathematical formulation of Bernoulli's principle which states that in the flow of a fluid, an acceleration occurs simultaneously with the decrease of the pressure. In a fluid flow without viscosity and therefore in which a pressure difference is the only force of acceleration, the velocity is equivalent to that given by the laws of Newton's motion. It is very common that the Bernoulli effect is quoted to assert that a change of speed causes a change of pressure; however, Bernoulli's principle does not make this report and this is not the case. He laid the foundations for fluid dynamics and, more generally, for fluid mechanics. Initially used for fluids circulating in a pipe, he found a large scope in aerodynamics (lift).

Daniel Bernoulli's equation is an integral equation of the Navier Stockes equation and translates the conservation of energy.

$$\rho \frac{v^2}{2} + \rho g z + P = Cte$$

or :

P is the pressure at a point (in Pa or N / m²)

ρ is the density at a point (in kg / m³)

v is the velocity of the fluid at a point (in m / s)

g is the acceleration of gravity (in N / kg or m / s²)

z is the altitude of the considered point (in m)

The Bernoulli equation relates the pressures, velocities and altitudes of two points of a flow line of a density fluid ρ to a continuous laminar flow. Applying the previous equation between A and B along a stream line, we obtain:

$$\rho \frac{v_A^{\ 2}}{2} + \rho g z_A + P_A = \rho \frac{v_B^2}{2} + \rho g z_B + P_B$$

The variables PA, VA, ZA represent the pressure, the velocity and the altitude of the fluid at point A while the variables PB, VB, ZB refer to the pressure, velocity and altitude of the fluid at point B.

4- Finite element method

The finite element method is part of applied mathematics tools. It is a question of setting up, using the principles inherited from the variational formulation or weak formulation, a discrete mathematical algorithm making it possible to look for an approximate solution of a partial differential equation (or EDP) on a compact domain with conditions at the edges.

It is characterized by its interest in satisfying the conservation of mass, momentum and energy in all the finite elements as well as in the whole of the calculation.

It is, therefore, above all the approximate resolution of a problem, where, thanks to the variational formulation, the solutions of the problem satisfy conditions of existence that are weaker than those of the solutions of the starting problem and where a discretization makes it possible to find an approximate solution.

It allows the modeling of some problems with a high complexity. It is used in particular in solid mechanics for the calculation of structures, the spatial domain in subdomains or element on which the unknown magnitude is approximated by a finite dimensional function with local support. This technique allows a very wide choice in terms of relates to the geometrical form of the element (triangle, quadratic, ...).

It is a method based on the variational formulation of equations rather than their differential form.

The finite element method makes it possible to solve this problem in a discrete and approximate manner; we are looking for an approximate solution that is "sufficiently" reliable.

The discretization consists of "cutting" the domain Ω, that is to say to seek a solution of the problem on a polygonal or polyhedral domain in pieces; there is a redefinition of geometry. Once the geometry has been approached, it is necessary to choose a space of approximation of the solution of the problem. In the MEF, this space is defined using the mesh of the domain (which also explains why it is necessary to approach the geometry). The mesh of the domain makes it possible to define a paving whose pavements are the finite elements.

The EDP is resolved at the nodes of the mesh, that is to say that the solution is computed at given points (discrete resolution) and not at each point of the domain Ω. This requires being able to interpolate, that is to say to determine the values at any point from the known values at certain points. Polynomial functions are generally used.

Conclusion

Mathematical equations can help us solve the problems encountered in practical work.

The following chapter was chosen to present the work on CATIA and ANSYS Software

Chapter 4 :

Results and design

Introduction

Digital Software can simulate the problem to give a proper solution close to reality. In this chapter, we will draw a real model of the wing of the aircraft to compare the aerodynamic effect and after the comparison, we will create a new model with less drag and friction and more lift depending on the results.

1- What's CATIA

Figure 26 : CATIA

CATIA ("Interactive Applied Three-dimensional Assisted Design") is computer-aided design (CAD) Software originally created by Dassault Aviation for its own purposes under the name of CATI (Acronym for Interactive Three-dimensional Assisted Design). Dassault Systèmes was founded in 1981 to develop and maintain it under the name of CATIA, IBM by marketing it.

Now, we are wanting to know if there is one drawing program better than the other? we should, think and imagin and then build a design in our mind either drawing program will help us to turn these graphics on paper, for that we prefer CATIA because it provides us all the tools we need .

Before we start learning 3D modeling Software, we need to know their classification as a drawing program, where CATIA is classified under the following Softwares:

-CAD (computer-aided design)

-CAM (Computer Aided Manufacturing)

-CAE (Computer Aided Engineering)

CATIA consists of modules with each module specialized in specific design field , and this is the following models presentation :

- Sketcher : this module is responsable for the implementation of two-dimensional shapes ; in preparation for make there-dimensional commands on it.
- Part desing : this module is responsible for converting two-dimensional graphics to three-dimensional objects which is most famous in CATIA and is closely linked with sketcher module . The part desing module it is considered from most important modules ,that used by the designer to get additional advantage from cad programs, which is stereotaxic drawing or three-dimensional drawing .
- Assembly: this module is responsible for assembling the parts previously produced in Part Design , and it is most important for those who work in the field of machinery desing or design in general , because it is the one who shows the inter-relationships between the parts of the machine or any mechanical establishment .
- Stress Analysis : this module is responsible for testing parts desingned to withstand the loads expected occurrence on it , and shows how the mechancial parts are affected by the colors , where they can learn the most dangerous points in terms of emotion through the distribution of colors.
- Drafting : this module is responsible for converting what you see on the screen to standard engineering drawings can be traded in the workshop for manufacturing or save them for documentation

- Surface and wiremframe: with this module surface can be drawing with zero size and weight and has its used in the aerospace , automotive, ships and mold design .

2- What's ANSYS

Figure 27 : ANSYS

Ansys is an American public company based in Canonsburg, Pennsylvania. It develops and markets engineering simulation Software. Ansys Software is used to design products and semiconductors, as well as to create simulations that test a product's durability, temperature distribution, fluid movements, and electromagnetic properties.

Ansys was founded in 1970 by John Swanson.

Simullation-Driven product development takes engineering simulation to another level – the unequalled depth and breadth of our Software coupled with its unmatched engineered scalability, comprehensive multi physics foundation and adaptive architecture set our technology apart from other CAE tools. These ANSYS advantages add value to the engineering design process by delivering efficiency.

❖ **Fluid mechanics (FLUENT)**

ANSYS Fluent Software is the most-powerful computational fluid dynamics (CFD) tool available, empowering you to go further and faster as you optimize your product's performance. Fluent includes well-validated physical modeling capabilities to deliver fast, accurate results across the widest range of CFD and multiphysics applications.

❖ **Optimization**

The first step in an analysis process is simulating the performance of a base design. Then you can re-use the model to investigate design parameters , different loading , and changes in environmental conditions and variations in manufacturing .ANSYS Design Xplorer enables you perform all your simulation in a single environment, ensuring the best design for any condition.

❖ **Solution**

ANSYS delivers solutions-engineering simulation solutions for your product design challenges. Whatever your industry, application or role in your organization's engineering and design team ; we have just the solution that will make you successful .

ANSYS WORKBENCH PLATFORM :

* **ANSYS Space Claim :**

ANSYS Space Claim Direct Modeler is the easiest and fastest solution to prepare geometry for simulation . Space Claim Direct Modeling solution speeds up time to analysis by allowing engineers to simplify models during Pre-processing. Space Claim also provides a CAD neutral environment freeing engineers to focus on the physics and modeling .

* **ANSYS Design Modeler Mesh Generation**

All engineering simulation starts with geometry to represent the design this could be a solid component for a structural analysis or the air volume for a fluid or electromagnetic study .This geometry has been produced in a computer aided design (CAD) system or built from scratch .ANSYS Design Modeler Software is the gateway to geometry handling for analysis with Software from ANSYS.

3- The design of CATIA Software

3-1- Model 1

The first step to design the model is to select the part design then you choose the plane which you must work then draw on the sketch number 1 the wing profile. First of all you need to make points then you must connect them with the outline tool 

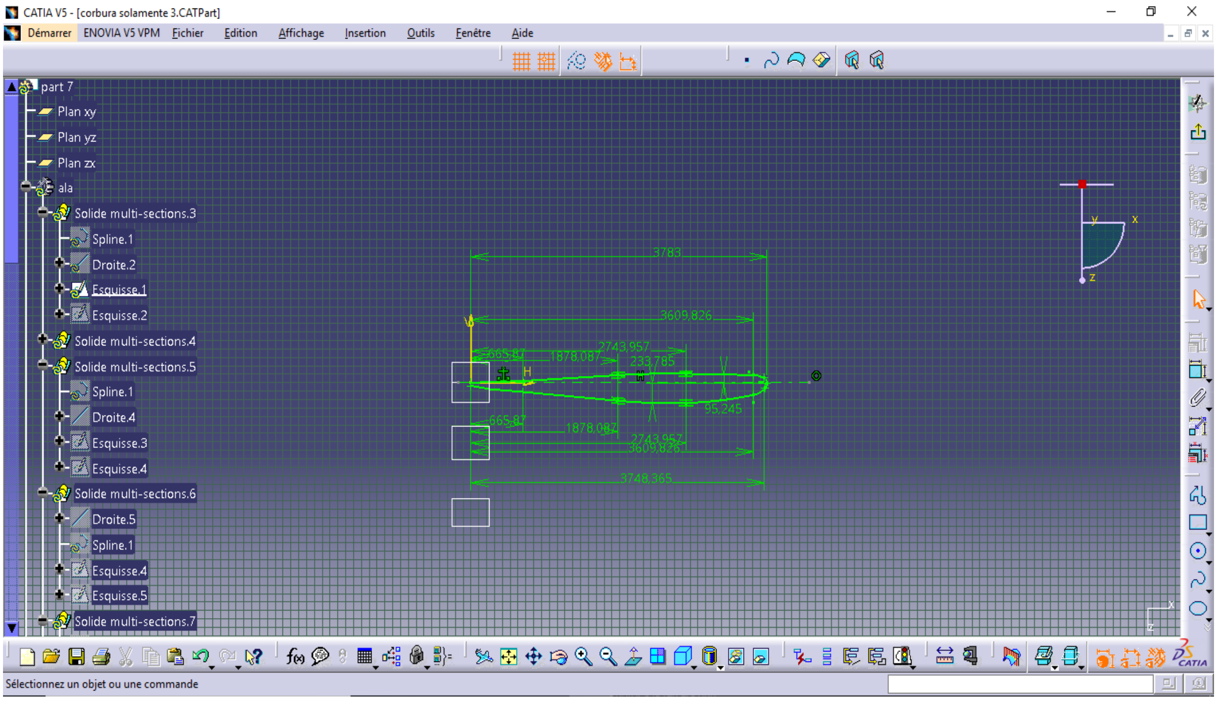


Figure 28 : First sketch obtained

By repeating the same work on sketch number 2, we obtain the following form:

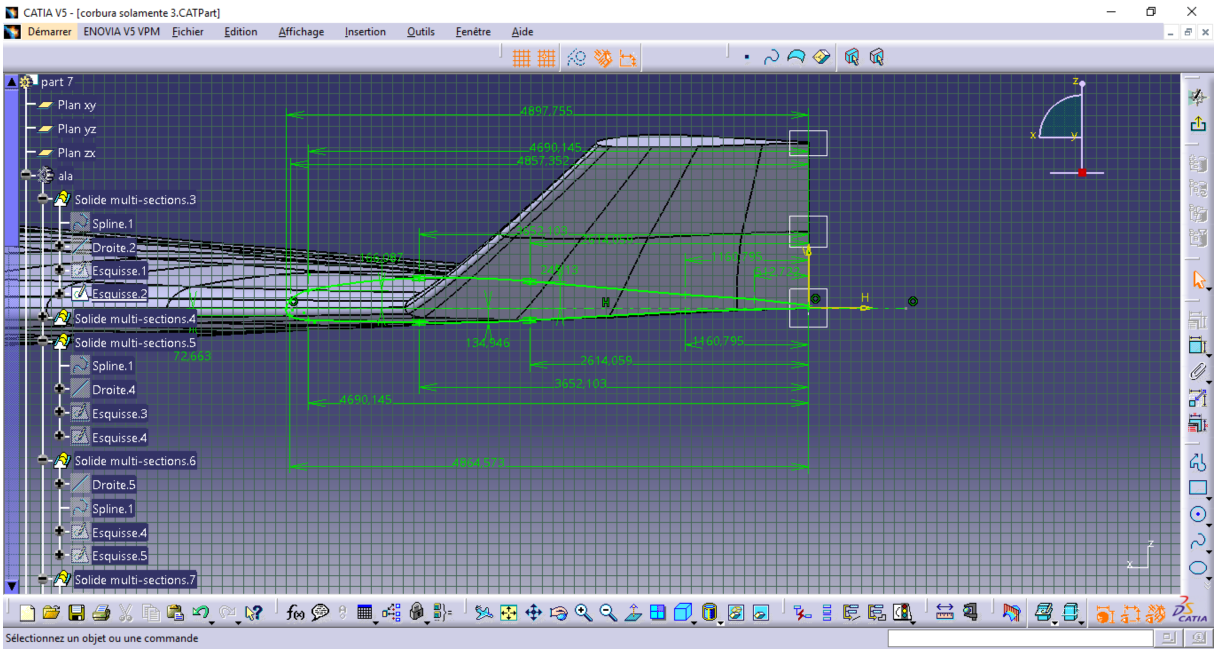


Figure 29 : Second sketch obtained

When we complete drawing the first step , we can use the tool multi section solid  and thanks to that we can obtain our first volume by thinking, beforehand, to define a guide for the extrusion and to select the sketches corresponding to the profiles.

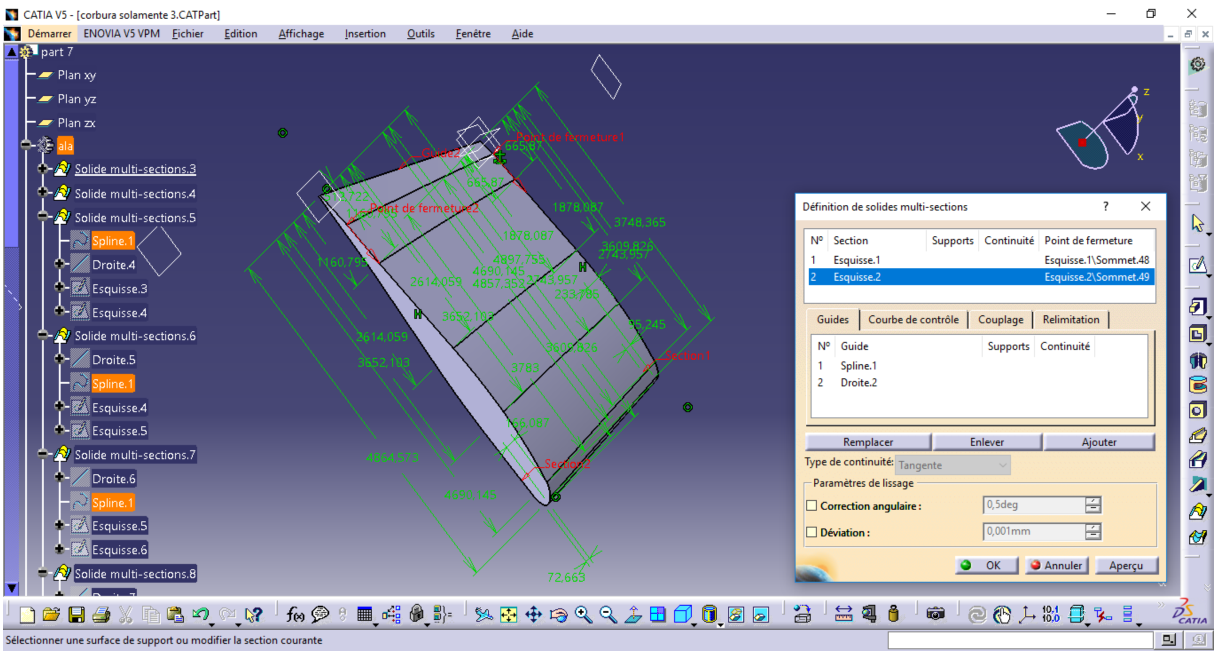


Figure 30 : Multi section of the two parts

After repeating this action several times, we get the following form:

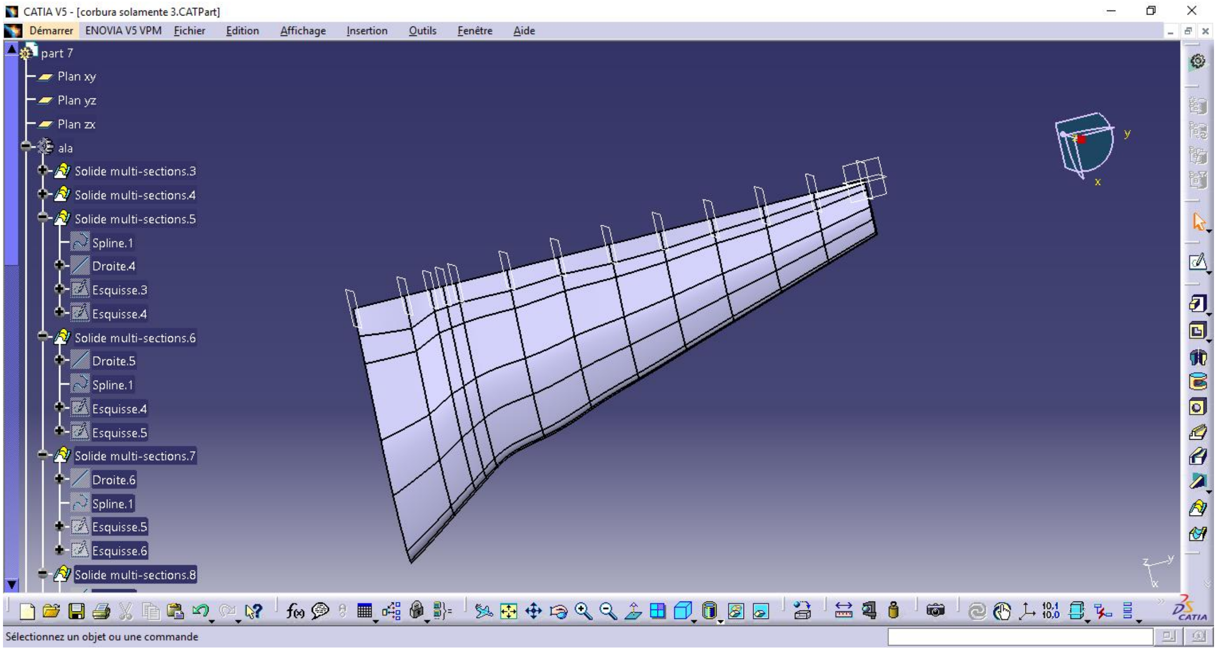


Figure 31 : The wing of the plane

Then we have to select another plan perpendicular of the first plan choosing to draw the next part of the design with the following geometry.

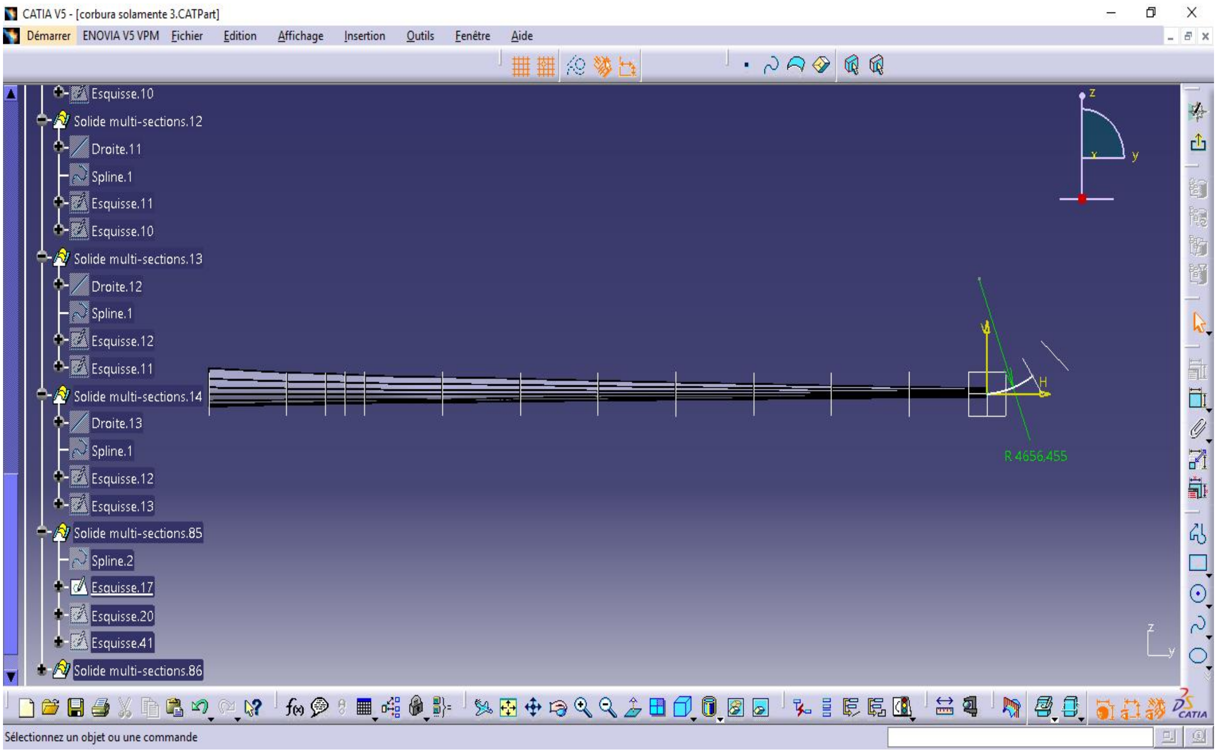


Figure 32 : Draw the new geometry

To complete the next part, we should create a plan normal to the curve.

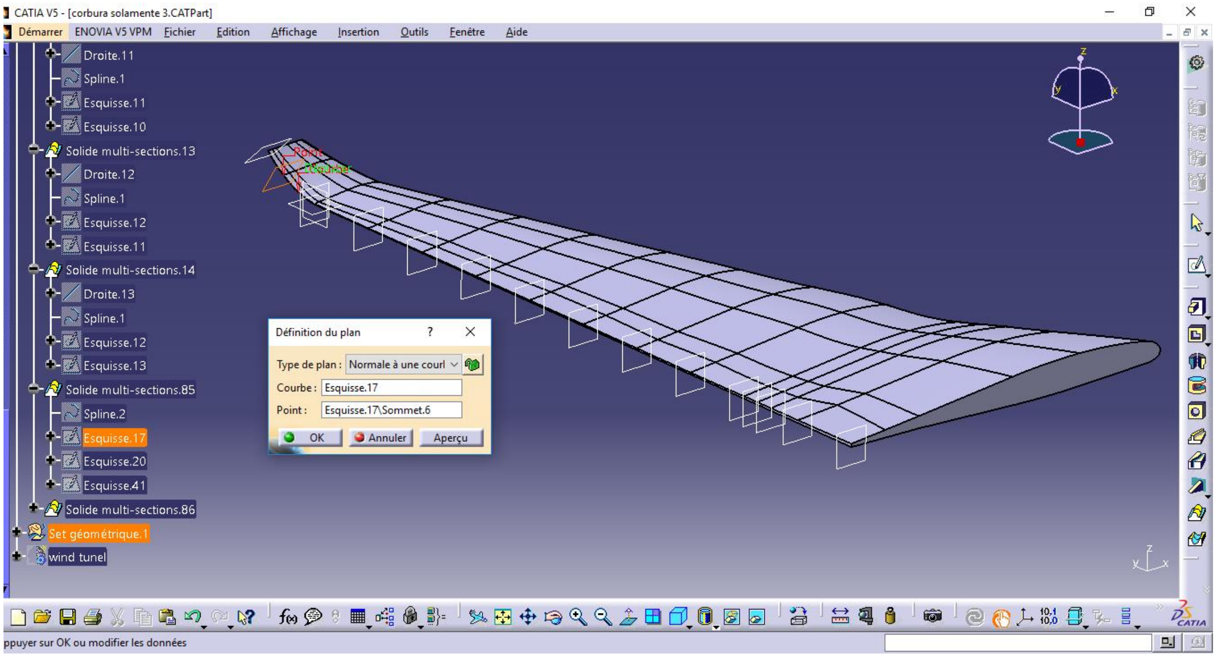


Figure 33 : The creation of a plan

For the next step, we draw the sketch below.

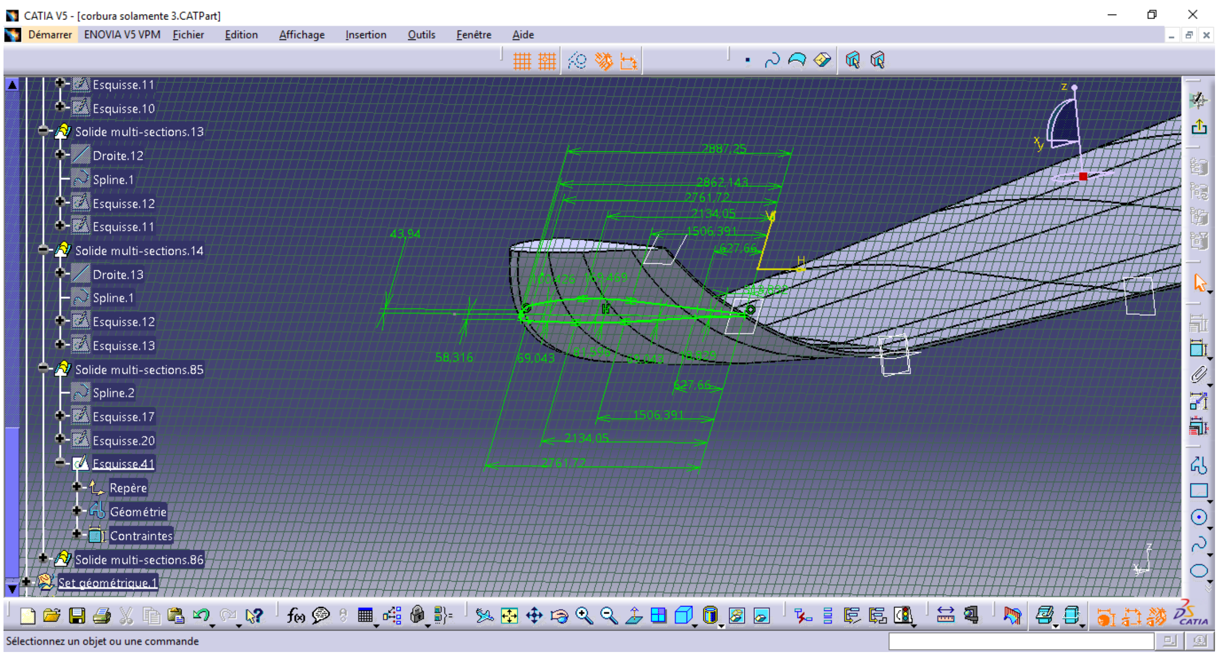


Figure 34 : Draw the shape of the wing profile

After repeating the previous step another time. We have to create multi-sections solid to connect between the two parts.

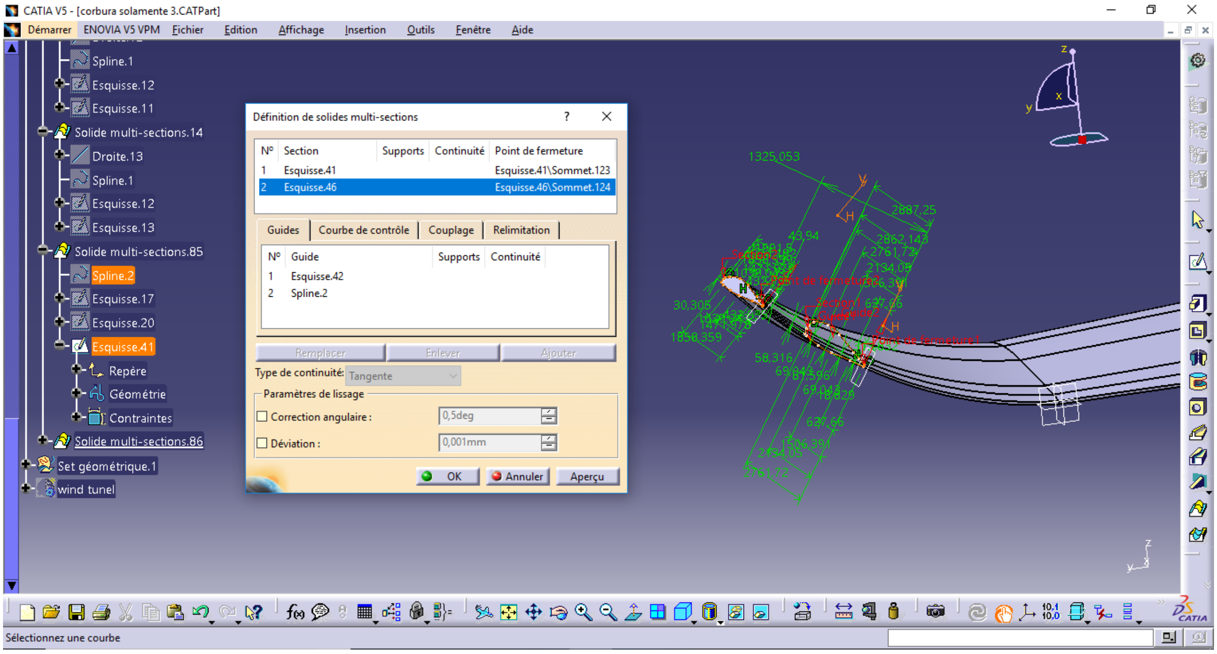


Figure 35 : Multi section of two sketches

Why we create a tunnel around the corps?

We create a tunnel around the corps to study the aerodynamic specifications that will help us to improve the form of the shape and get a new model more efficient.

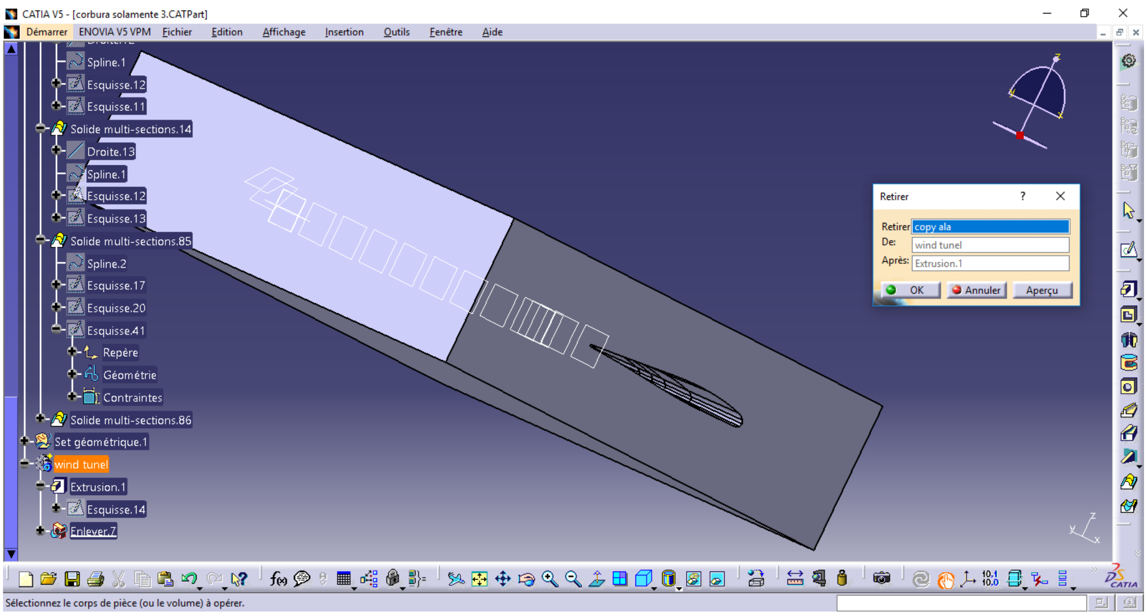


Figure 36 : Creation of wind tunnel

3-2- model 2

After finishing drawing the first part, we have to reapet drawing the second part but with other dimension on the winglet and different geometry.

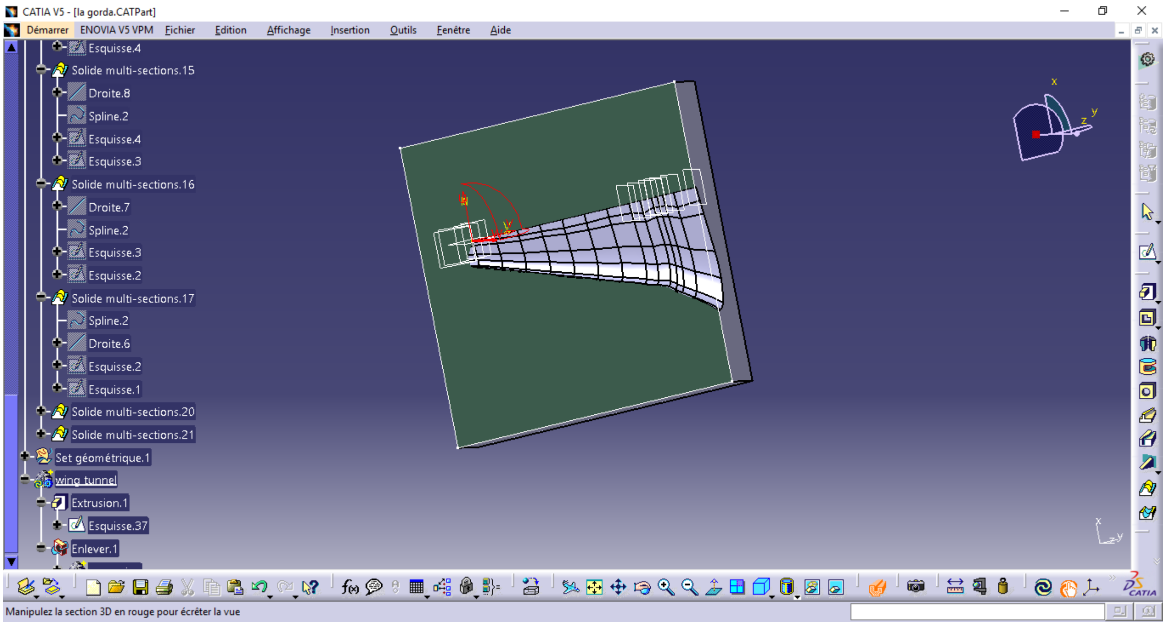


Figure 37 : Second model of the wing

4- The analysis of ANSYS Software

When a viscous fluid flows on a wall, the velocities of the fluid particles at the levels of this wall are canceled because of the adhesion condition. The speed increases as one moves away from the wall until reaching its value at infinity upstream: this zone is called boundary layer.

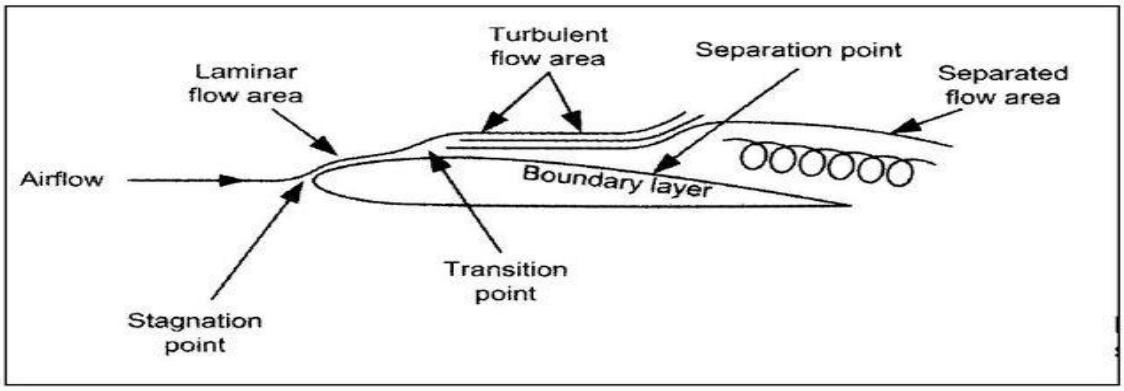


Figure 38 : Formation of a Boundary Layer

This flow will be modeled using the ANSYS-Fluent system in the ANSYS-Workbench 17.0 suite.

❖ **Numerical method :**

Launch the "ANSYS-Workbench" suite and open the analysis system "Fluid mechanics (FLUENT)"

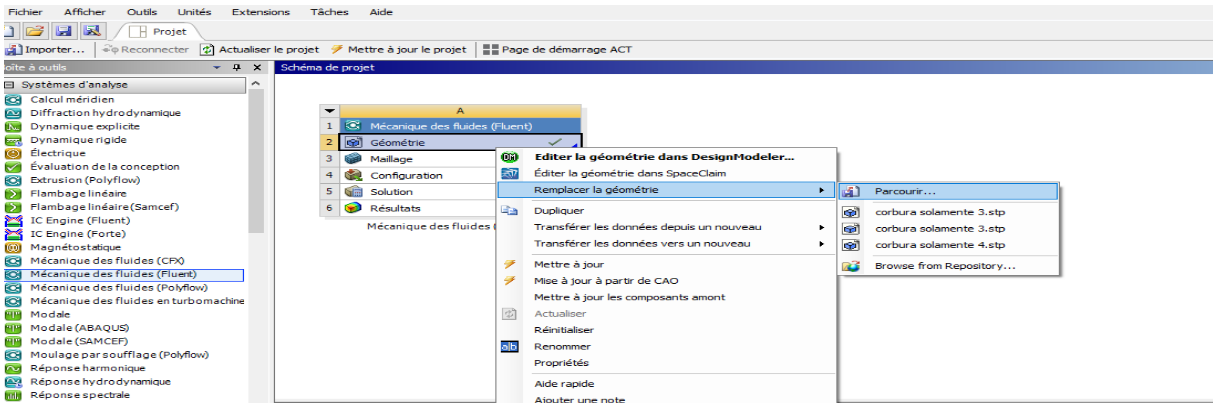


Figure 39 : The Import of the geometry

4-1- Mesh

A mesh is the spatial discretization of a continuous medium, or also a geometric modeling of a domain by finite and well-defined proportioned elements. The object of a mesh is to simplify a system by a model representing this system and, possibly, its environment (the environment), for the purpose of simulations of calculations or graphical representations.

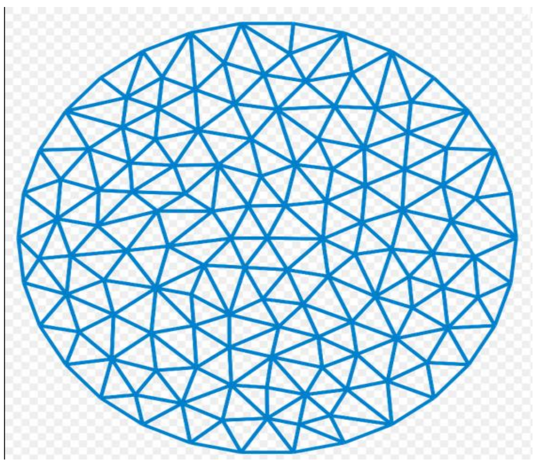


Figure 40 : Example of mesh

To obtain a good result, it is necessary that the mesh is with a great precision for this and thanks to the numerous tests which we carried out, the physical properties are displayed in the following figure.

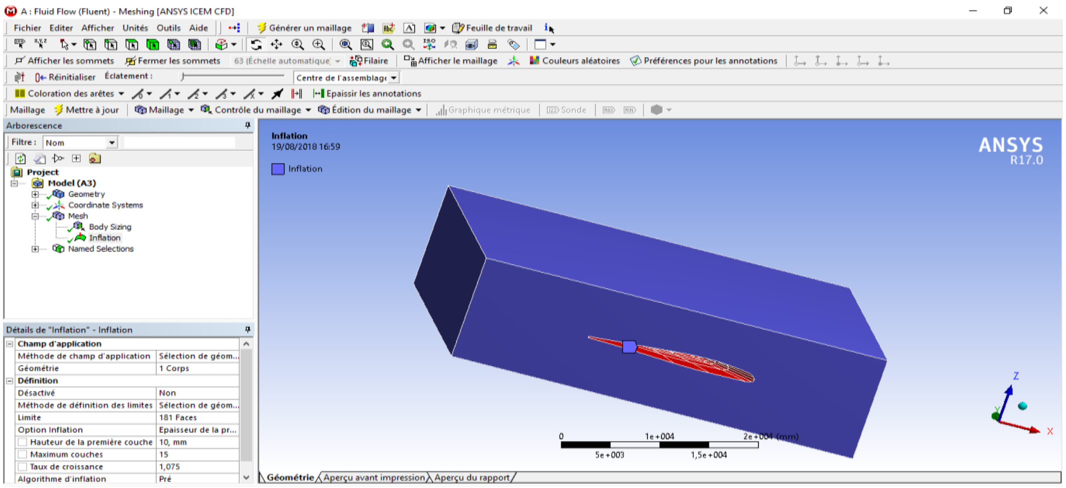
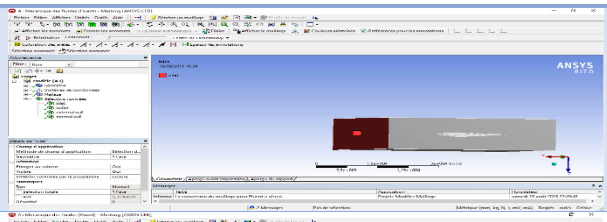
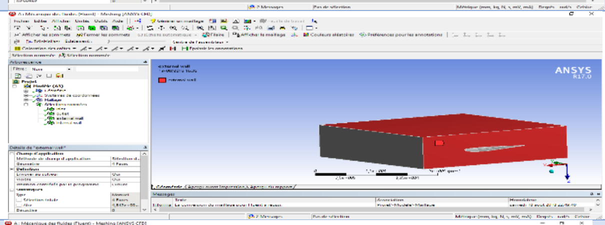
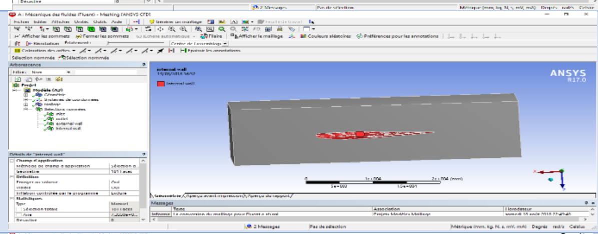
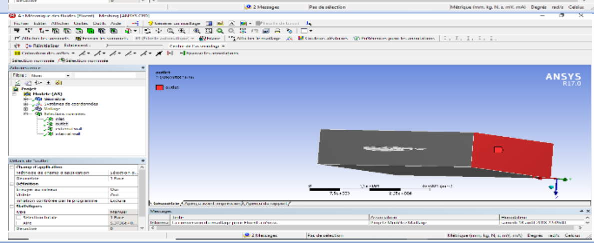


Figure 41 : The mesh characteristics

Then, we must name the different workspaces:

Table 1: The different workspaces

The part		The name
the forward		inlet
the side		external wall
the inner		internal wall
the back		outlet

When, we finish all the instruction, we have update the mesh to get the final result with good precision.

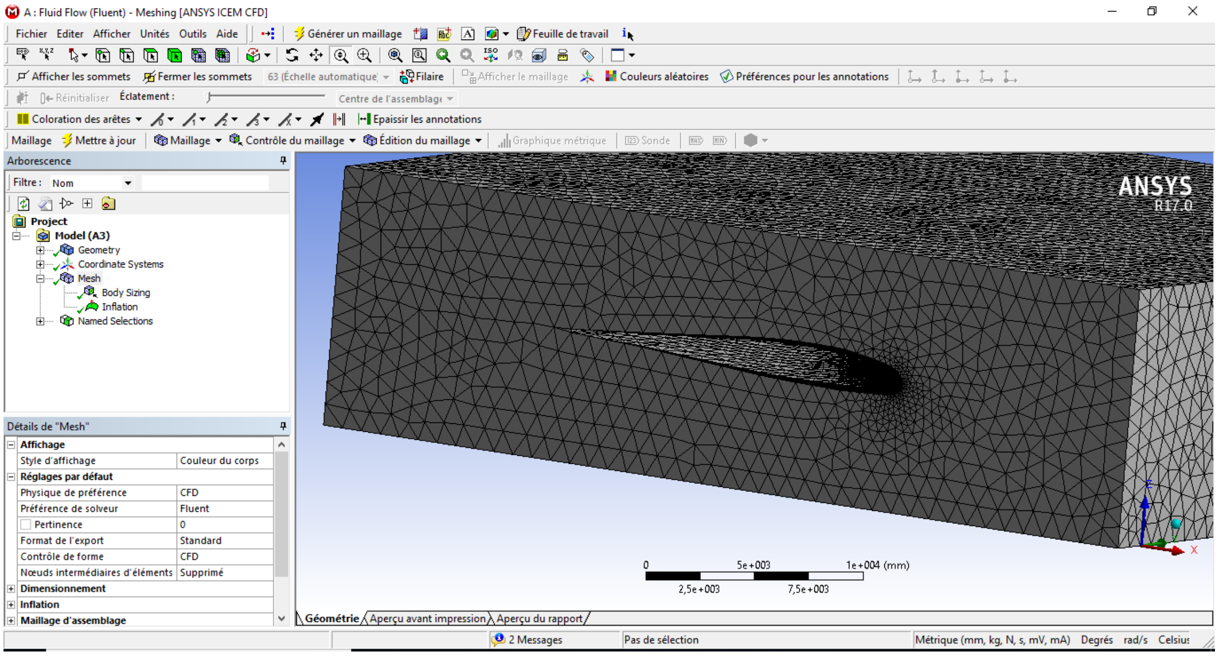


Figure 42 : Final mesh

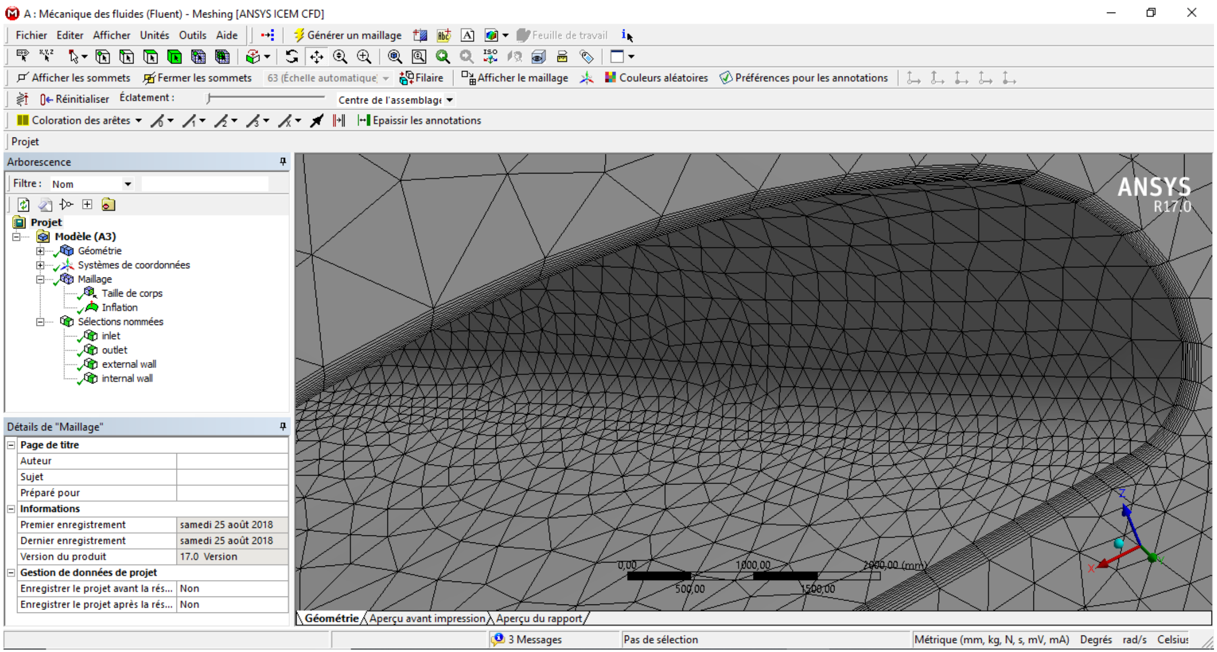


Figure 43 : The inflation

4-2- Configuration

This configuration step is very important because it defines the workplace and simulates the true reality .In this step we will calculate the drag and the lift around the wing.

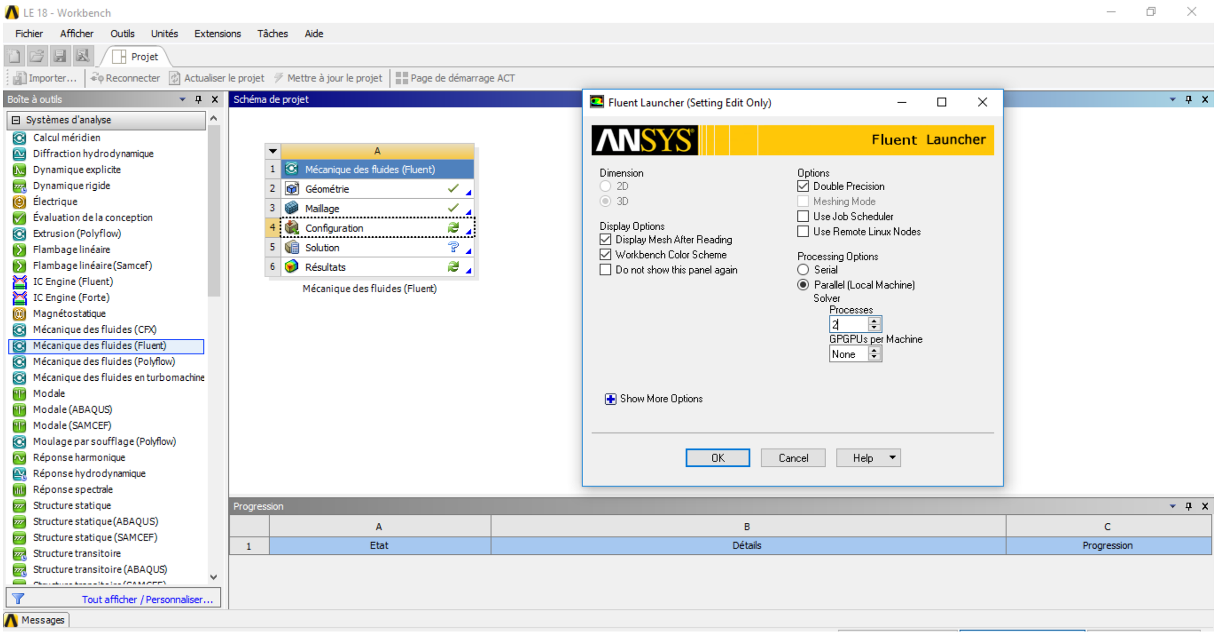


Figure 44 : ANSYS configuration interface

❖ K omega SST

The SST k-ω turbulence model is a two-equation eddy-viscosity model which has become very popular. The shear stress transport (SST) formulation combines the best of two worlds. The use of a k-ω formulation in the inner parts of the boundary layer makes the model directly usable all the way down to the wall through the viscous sub-layer, the transport variable k determines the energy in turbulence and ω determines the scale of turbulence.

The SST k-ω model can be used as a Low turbulence model without any extra damping functions. The turbulent energy k is given by :

$$k=(3/2)*(UI)^2$$

Where U is the mean flow velocity and I is the turbulence intensity.

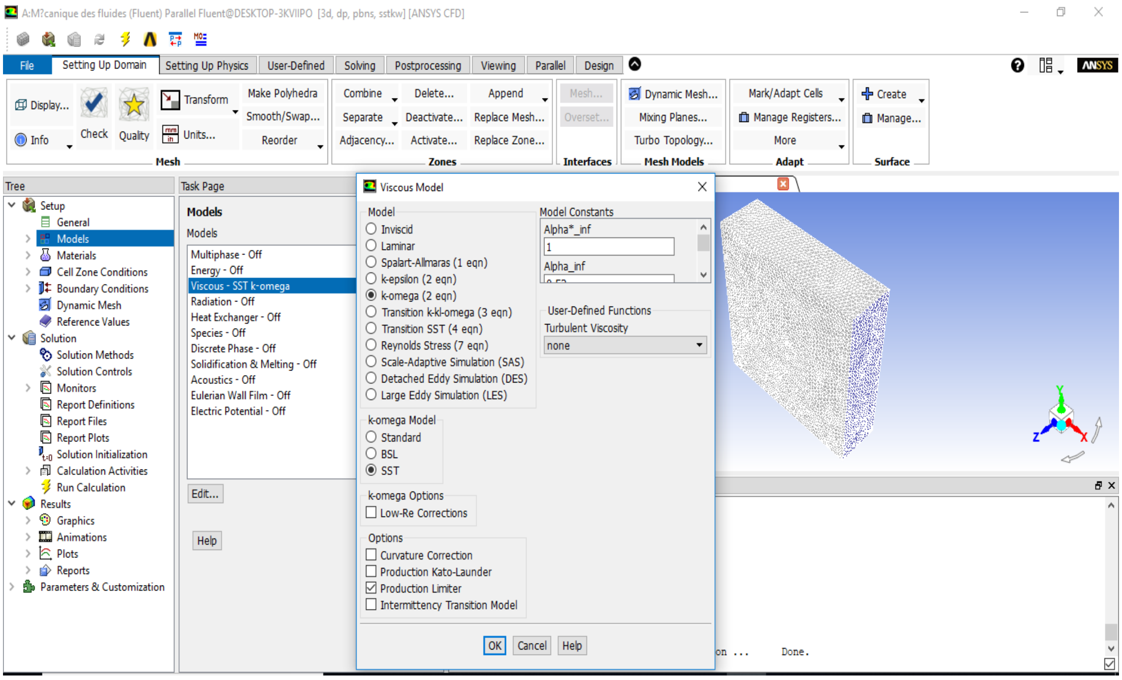


Figure 45 : K-omega SST

Finally, when we finish entering the physical property of the workspace, we start the calculation of the result.

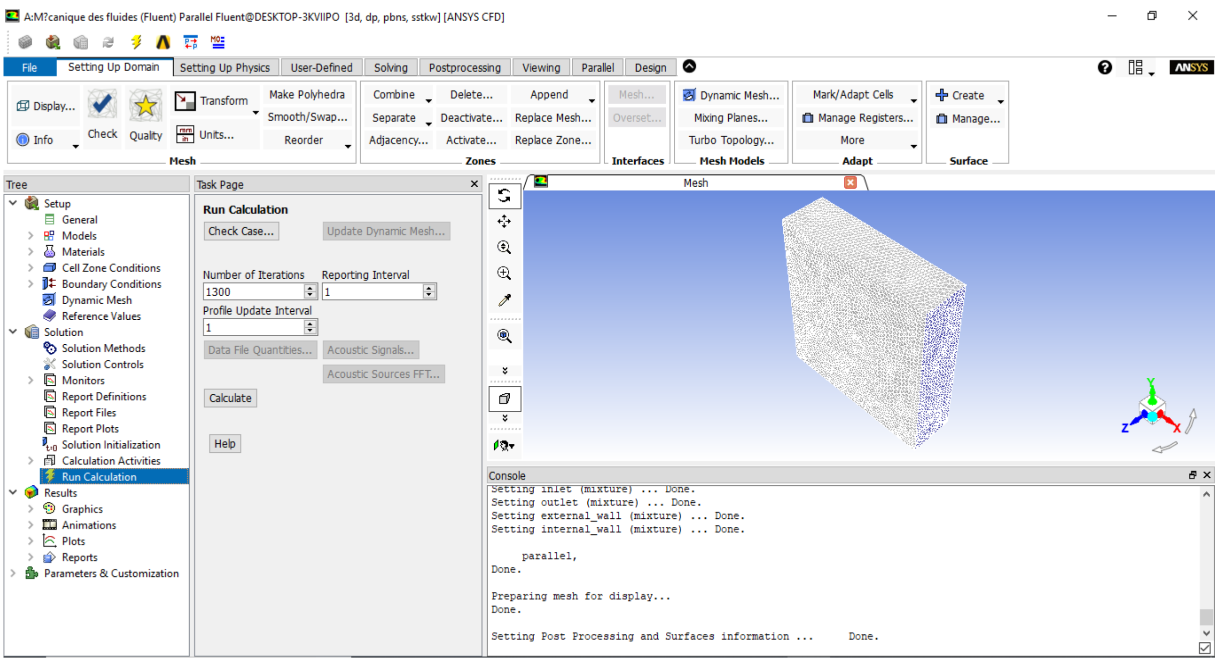


Figure 46 : Run calculation of 1300 number of iteration

4-3- Result

When the execution of the calculation is completed and the solution converges, we display the results.

There are many of the results that can be displayed like the velocity of air, the pressure and the turbulence.

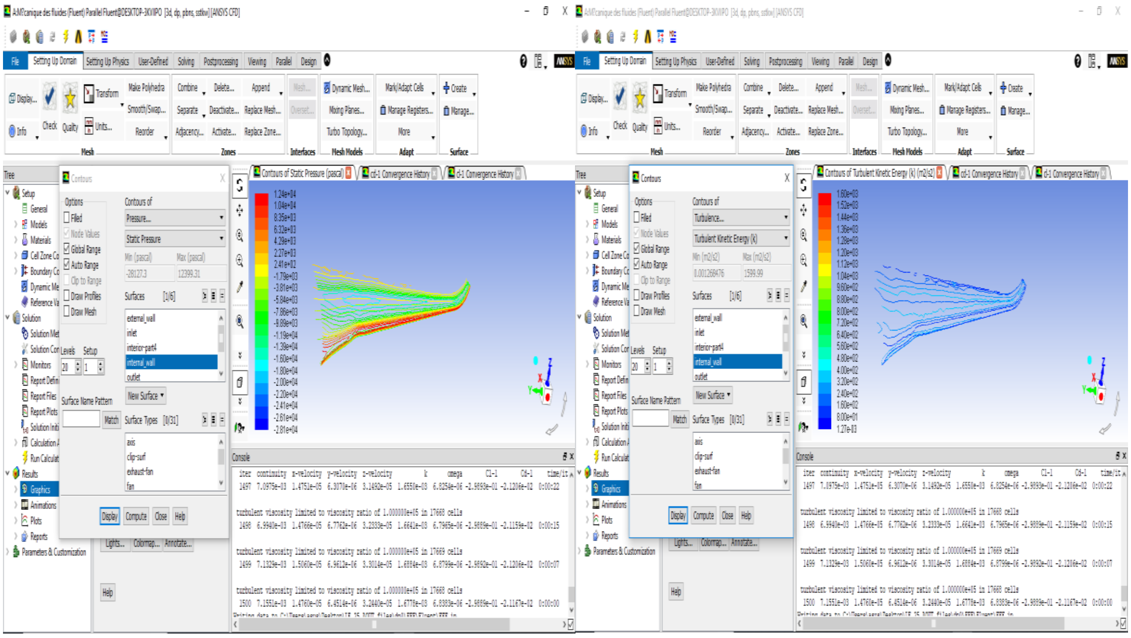


Figure 47: The results of pressure and turbulence

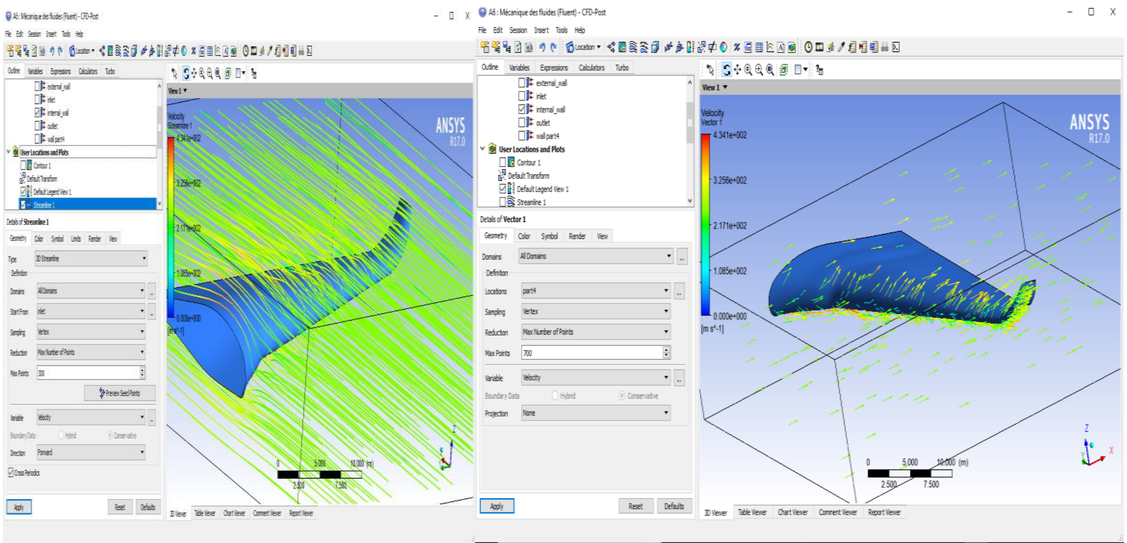


Figure 48: The results of the speed and air flow

According the results, it can be concluded that the wing shape has a great influence on the aerodynamicity flow.

Table 2: The results obtained between the three models

	MODEL 1	MODEL 2	MODEL 3
Velocity inlet	250 m/s	250 m/s	250 m/s
Max velocity	296,397 m/s	339,355 m/s	389,466 m/s
Static pressure	26500 Pa	26500 Pa	26500 Pa
Pressure	-5527,51 Pa	-10837 Pa	-17926,3 Pa
Density	0,413 kg.m-3	0,413 kg.m-3	0,413 kg.m-3
Coefficient Drag	0.012875	0.0209166	0.0250698
Coefficient Lift	0.109329	0.197641	0.298908

According to these results, the highest aerodynamic forces are detected in the third model. We therefore conclude that the thickest surface , the greater the dynamic force created.

From where the lift and the drag are proportional to the thickness of the wing.

The third model has the biggest lift and the last one has the highest coefficient numbers.

- **Solutions to reduce the drag:**

Engineers reduce friction drag by making the airplane more streamlined, the wings narrower, or by using new materials that make the surface more smooth, decreasing the ability for the force of drag to effect it. As the roughness and surface area of the airplane decreases the friction drag will decrease.

Generally, pressure drag is more of a concern than friction drag.

Following the results that we obteined and research, we have reached to create a model with good aerodynamic performance.

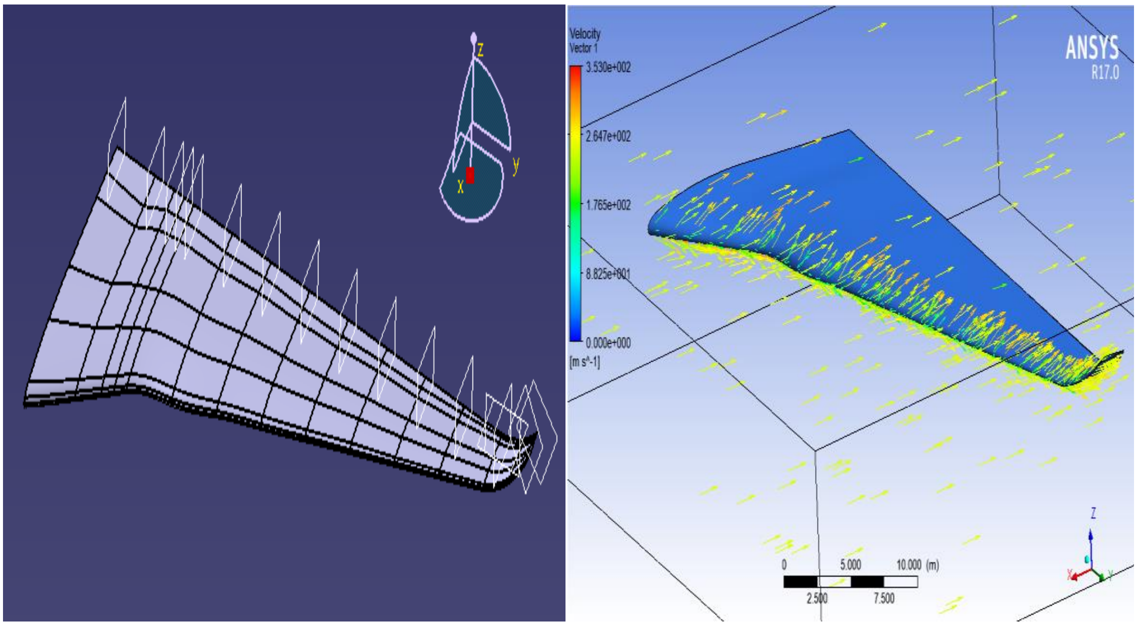


Figure 49 : Final results

❖ **Drag improvement benefits:**

-Better aerodynamics.

-Low fuel consumption.

-The noise reduce.

-Double-flow reactors allow better performance.

-More stability.

-Less vibration.

❖ **Disadvantage :**

-Decrease the number of passengers.

-High manufacturing cost.

❖ **Solutions to increase the lift:**

The wing of an aircraft is designed for high or cruise speed where lift is mainly created by forward speed only. For slow flight we need to increase lift somehow, but there are limits to what can be done by increasing the angle of Attack. Aerodynamicists have devised other ways of increasing the amount of lift generated. These are called lift augmentation devices. Pilots use these devices on almost every flight .

An airliner wing may produce a pound of lift per square inch in level flight. That doesn't seem like much, but over the entire surface of the wings these pounds-per-square-inch add up. The wings of a Boeing 747 have a surface area of about 510 square meters (5,500 square feet) and can produce as much as 390 tons (850,000 pounds) of lift.

Conclusion

As we saw in my project we have achieved an improvement of wing performance more than satisfactory. We are aware that our results are largely based on the use of ANSYS aerodynamic analysis Software.

Our results allow us to conclude that theoretically the improvement of wing performance is limited only by the laws of physics.

General conclusion

These five months of project have allowed us to achieve many things and achieve our goals.

Indeed, the many technical difficulties encountered led us to review our objectives, to finally focus on the essentials.

The purpose of this project was to describe drag and lift control strategies tested by numerical simulation.

Indeed, by associating CATIA and ANSYS Software, we were able to set up a modal analysis and obtain interesting results.

In addition, we managed to create several shapes, considering the problems encountered in the mesh and performing an aerodynamic study of space to compare the models to find solutions to reduce drag and increase lift.

As we have seen with our project, we have achieved a satisfactory improvement in wing performance.

Bibliographic reference

[1] Hamdullah Ozogul , 2006 , « Écoulements de fluides à seuil autour d'un cylindre en milieu confiné : études expérimentale et numérique », préparée au sein du Laboratoire Rhéologie et Procédés dans l'École Doctorale I-MEP2.

[2] BEDJEGHIT EL-HOUAS , 2010 , « Etude expérimentale et numérique d'un écoulement autour d'obstacles carré et cylindrique », Université MENTOURI Constantine, Faculté Des Sciences De L'ingénieur, Département De Génie Mécanique.

[3] Marcin Huptas, Witold Elsner,2008,"Steady and unsteady simulateon of flow structure of two surface-mounted square obstacles "Task Quarterly 12 N 3,197 -207.

[4] P.F. Zhang et al (2006) : Numerical simulation of flow around a circular cylinderin ground effect The seventeenth international symposium on transport phenomena. Toyama JAPAN. 2006

[5] R. Belakrom et al, 2007,Simulation numérique du phénoméne d'éclatement tourbillonnaire dans la zone de sillage d'un obstacle de section circulaire. International conférence on Energetics and pollution constantine

[6] K.Lam -L. Zou, 2009, Experimental study and large eddy simulation for the turbulent flow around four cylinders in an in-line square configuration. International journal of heat and fluid flow vol 30 PP 276-285,2009.

[7] F.Meddane et al (2007) Influence du volet sur les performances du profil NACA 0021. International conférence on Energetics and pollution constantine

[8] JOSLIN,1999, Transitioning active flow control to applications,30th AIAA Fluid Dynamics Conference,Norfolk, VA, AIAA1999-3575

[9] BOLONKIN ,1999,Estimated benefitsof variable-geometry wing camber control for transport aircraft.NASA Technical Memorendum,TM-1999-206586

[10] Prandtl L. (1904) Uber flussigkeitsbewegung bei sehr kleiner reibung.Internationaler Mathematiker -Kongress Heidelberg, vol. Verhandlungen 3, pp.484–491.

[11] Comolet R. (1976) Mécanique expérimentale des fluides. 2nd ed.

[12] STANEWSKY,2000, Aerodynamic benefits of adaptive wing technology,Aerospace Science and Technology, 4, 439–452.

[13] Tillman C, 2000, CHaracterization Of Pulsed Vortex Generators Jets for Active Flow Control. Paper presented at the RTO AVT Symposium on Active Control Technology for Enhanced Performance Operational Capabilities of Military Aircraf t, Land Vehicles and Sea Vehicles », Braunschweig, Germany

[14] McManus K ,1997, Airfoil Performance Enhan cement Using Pul- sed Jet Separation Control , AIAA 97-1971

[15] Batchelor G ,1967 ,An Introduction to Fluid Dynamics , Cambridge University Press, 1st ed.

[16] Hafida .N, Mohamed.S , 2010 , étude numérique des effets d'obstacles sur le profile de vent , revue des énergies renouvelable SMEE 10 Bou Ismail-Tipaza

[17] M.M Ouestati ,2010, numerical simulation of flow around obstacle with stream function-vorticity formulation of navir-stokes equations 2 nd International conference on Energy conversion and conservation 2010

[18] Prandtl L, et Tietjens O , 1921 , Hydro- and Aeromechanik . Erster Band. Berlin, Springer ed

[19] SENDSTAD & Moin , 1992. The near wall mechanics of three-dimensional turbulent boundary layers. Report No. TF-57.

[20] Labarthe ,1992 , Problème de stabilité linéaire et prévision de la transition sur des configurations tridimensionnelles incompressibles et compressibles, Thèse de Doctorat.

[21] Arnal D ,1988 , Recherches sur la transition liées au projet Hermés, OA 18/5005 AYD (DERAT 18/5005.08).

[22] Klebanoff P. et Tidstrom K. ,1972, Mechanism by which a two-dimensionnal roughness element induces boundary layer transition.Phys. Fluids, vol. 15

[23] Cauley W.M., Saydah A. et Bueck J. ,1966, Effect of spherical roughness on hypersonic boundary layer transition.AIAA Journal, vol. 4(12), pp. 2142–8.

[24] SAVILL A. M., TRUONG T. V. & RYHMING I. L, 1986 ,Turbulent drag reduction by passive means, First european drag reduction meeting.

ANDERS J. B. & WATSON R. D. 1985. Airfoil large-eddy breakup devices for turbulent drag reduction. AIAA Shear flow control conference, Boulder, colorado, Paper No. 85-0520.

[25] TIEDERMAN W. G., LUCHIK T. S. & BOGARD D. G. 1985. Wall-layer structure and drag reduction. J. Fluid Mech. 156, 419-437.

[26] BRADSHAW P. & TERREL M. 1969. The response of a turbulent boundary layer on a Infinite swept wing to the sudden removal of pressure gradient. National physical lab. Aero Rept. 1305, Teddington, England.

[27] Sicot F, (2008), Block-Jacobi Implicit Algorithms for the Time Spectral Method, AIAA Journal, vol.46 no.12, pp. 3080-3089

[28] O. Guerri, 2008, 'Numerical Simulation of the Flow Around Oscillating Wind Turbine Airfoils. Part 1: Forced Oscillating Airfoil', International Journal of Multiphysics, Vol. 2, N°4, pp. 367 - 385, 'Fluid Structure Interaction of Wind Turbine Airfoils', Wind Engineering Journal, Vol. 32, N°6, pp. 539 - 557

[29] G. Martinat , 2008,Turbulence modeling of the flow past a pitching NACA0012 airfoil at 105 and 106 Reynolds numbers. Journal of Fluids and Structures 24, 1294–1303.

[30] J. Windte , Aerodynamic Analysis of Flapping Airfoil Propulsion at Low Reynolds Numbers, Université technique de Braunschweig Bienroder Weg.

[31] Liu Zhen ,2008, Square cylinder Large Eddy Simulation based on random inlet boundary condition,Journal Of Applied Fluid Mechanics,Vol.3,No.1,pp.35-45

[32] Marco N ,Désideri J-A , Lanteri S, 1999,Multi-objective Optimization in CFD by Genetic Algorithms INRIA Sophia , RR-3686

[33] H Tebbiche , , 2014, Aerodynamic Drag Reduction by Turbulent Flow Control with Vortex Generators', International Symposium on Aircraft Materials - ACMA 2014 - Marrakech – Morocco; April 23-26

Netography

https://www.lavionnaire.fr/

https://tel.archives-ouvertes.fr/tel-00308715/file/These_finale-Bourgois.pdf

https://www.yumpu.com/fr/document/view/19263326/etude-experimentale-et-numerique-dun-ecoulement-autour-d-

https://hal.inria.fr/inria-00072312/document