Escuela Politécnica Superior de Jaén

Grado en Ingeniería Mecánica

Universidad de Jaén
Escuela Politécnica Superior

Trabajo Fin de Grado

ESTUDIO AERODINÁMICO DE LA ZONA FRONTAL DE UNA AERONAVE

Alumno: **Ben Hamad Aiman**

Tutor: Prof. D. Miguel Angel Rubio Paramio

Dpto: departamento de Ingenieria Grafica, Diseño y proyectos

Universidad de Jaén

Escuela Politécnica Superior de Jaén
Departamento de Informática

Don Miguel Angel Rubio Paramio , tutor del Proyecto Fin de Carrera titulado: ESTUDIO AERODINÁMICO DE LA ZONA FRONTAL DE UNA AERONAVE, que presenta Ben Hamad Aiman, autoriza su presentación para defensa y evaluación en la Escuela Politécnica Superior de Jaén.

Jaén, Junio de 2018

El alumno: Los tutores:

Ben Hamad Aiman Miguel Angel Rubio Paramio

RESUME

L'objectif de ce travail consiste à analyser les solutions de contrôle permettant de réduire la traînée aérodynamique et donc de diminuer la consommation d'un airplane.

Ce travail à dominante numérique se décompose en deux parties : la première a pour objectif de valider les résultats de calculs avec solution de contrôle et des mesures expérimentales, la seconde d'explorer les résultats numériques pour créer un nouveau modèle.

Mots clés : trainée, turbulence, maillage, nombre de Reynolds, forme Aérodynamique

ABSTRACT

The objective of this work is to analyze control solutions that reduce aerodynamic drag and thus reduce the consumption of an airplane.

This numerical work is divided into two parts: the first aims to validate the results of calculations with control solution and experimental measures, the second to explore numerical results to create a new model.

Keywords: drag, turbulence, mesh, Reynolds number, aerodynamic shape

acknowledgments

*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. Especially **Mr Miguel Angel Rubio Paramio**, who accepted to frame me during this research.*

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

This work is a very modest gift that I give to my parents, my friends for their continued support and deep affection.

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

Contents

Figure List

Table List

Introduction

Aerodynamics was first used to increase vehicle performance in fighter jet during the 1940s. Fighter jet engineers realized that air flowing around the vehicle could be used to increase downforce and reduce aerodynamic drag on the plane. As fuel economy became a strong factor in road vehicle design, engineers soon realized that the methods of reducing aerodynamic drag on fighter jet could be transferred to cruise plane in order to improve fuel economy.

To decrease the amount of drag created by a vehicle, manufacturers began incorporating vehicle body designs that would allow the vehicle to be more streamlined. Methods of decreasing the drag coefficient of a vehicle include re-shaping the tail, covering the underside of the vehicles, and reducing the amount of protrusions on the surface of the plane.

Bibliographic studies

Introduction:

This part consists of an approximate study of our problematic. We will begin with a general definition and then present the main factors that will affect on the Aerodynamic.

I. What is aerodynamics ?

Aerodynamics is a part of physics that corresponds to the study of phenomena accompanying any relative movement between a body and the air where it is bathed. It comes into play fluid conditions (humidity, humidity, pressure ...), materials, and the shape of the model.

1. What is his interest ?

The goal of aerodynamics is to allow all machines to reduce fuel consumption (due to the reduction of wind resistance) and improve its capabilities by slipping the air on the device. This allows a reduction of pollution and the cost of the trip. This is a necessary study for aerodynamic advances as well as ecology.

2. Concrete Example :

To understand the role of aerodynamics in aviation and energy consumption, we will draw a comparative table between two similar aircraft but from a different time:

We will study the case of Caravelle 12 (which made its first flight on October 29, 1970) and the Airbus A320:

Fig 1: Example of Aircrafts

Appareil	Caravelle 12	Airbus A320
Longueur	36,23 m	37,57 m
Envergure	34,30 m	34,10 m
Hauteur	8,72 m	12,11 m
Poids à vide	32,8 t	42,4 t
Capacité de carburant	19 000 l (soit 15,2 t)	23 860 l
Poids maxi au décollage	58 t	77 t
Poussée totale	13,4 t	15 t
Vitesse de croisière	825 km/h	952 km/h
Altitude maxi	12 000 m	12500 m
Nombre maxi de passagers	139	150

Fig 2:Aircrafts Characteristics

Following this comparison, we can see that the Airbus A320, has characteristics similar to those of the Caravelle 12. However, the Airbus has a more efficient aerodynamics. This is felt when comparing the cruising speed of both aircraft. Airbus A320 is ahead of the Caravelle 100 km / h. In addition, after averaging all the characteristics (number of passengers carried, maximum take-off weight ...), the Airbus consumes less fuel than the Caravelle.

We also note that this Airbus has wings curved at its end allowing a displacement of air that generates a swirling trail at the ends of the wings, and that is called "Vortex" (as seen in the notions of aerodynamics)

II. Definition of Radome :

A radome (radar and dome) is an impervious protective shelter used to protect an antenna from weather and / or manholes, so as not to divulge the orientation of the antenna (as part of listening / interception of communications). Various materials can be used for construction but they have in common to minimize the signal transmitted and received. There are rigid and other flexible radomes made of fabric and inflatable. They can have different forms, varying according to the use that can be terrestrial, maritime or aerial. In particular, on aircraft, they must have an aerodynamic shape, it can be constructed in several shapes - spherical, geodesic, planar, etc. - depending on the particular application.

Fig 3:airplane Radome

1. Necessities of optimization the Radome Shape:

On airliners the nose cone is also a radome protecting the weather radar from aerodynamic forces.

The shape of the nose cone must be chosen for minimum drag so a solid of revolution is used that gives least resistance to motion. The article on nose cone design contains possible shapes and formulas.

2. Nose cône design :

Given the problem of the aerodynamic design of the nose cone section of any vehicle or body meant to travel through a compressible fluid medium (such as a rocket, aircraft or missile…), an important problem is the determination of the nose cone geometrical shape for optimum performance. For many applications, such a task requires the definition of a solid of revolution shape that experiences minimal resistance to rapid motion through such a fluid medium, which consists of elastic particles.

3. Types of Radome shape :

over time, the engineers always try to improve the aerodynamic shape of Radome and this is some types of Radome:

- ✓ **Classic radome :**

Classic propeller plane front view on airshow display

Fig 4: Classic radome

4. The Aerodynamic Radome :

Aerodynamic drag is the resistance of the air to the movement of an object.

✓ **The pressure drag :**

Thus named because it is due to the pressure that the air exerts on the front face of the moving object.

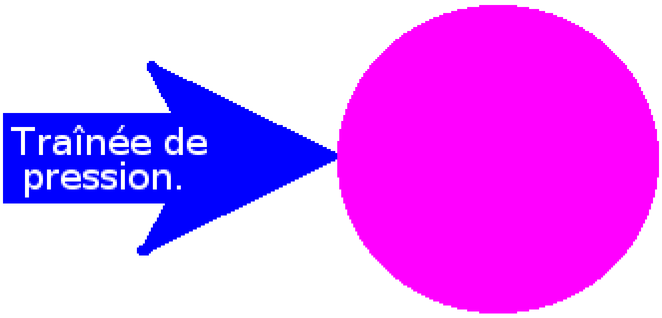


Fig 5: The pressure drags

One may wonder why classic planes, are not supersonic, have a nose like this:

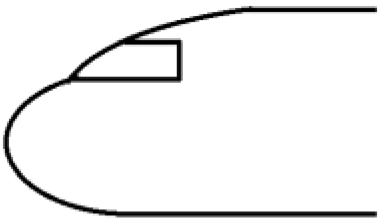


Fig 6: subsonic Radome

And not of this kind:

Fig 7: Supersonic Radome

In other words, why are not all planes sharp like supersonic?

The fact is that when an object moves in the air, it produces a "pressure wave", that is to say a slight overpressure that propagates ahead of the object at the speed of sound, decreasing as it moves away and "spreads out".

This overpressure causes a pre-separation of the air upstream of the object.

The phenomenon is easily visible in winter, when it snows and when one moves by car. We can then see the snowflakes go up to the approach of the windshield and even sometimes go over without touching it.

Fig 8: Car Air Flow

Transposed to the case of an airplane, this means that the air bypasses the nose of the plane like this:

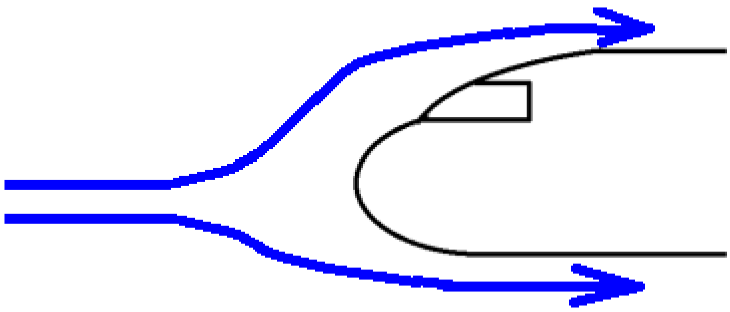


Fig 9: Aircraft Air Flow

On the other hand, when the speed of the airplane becomes equal to or greater than the speed of the sound, the overpressure can no longer move away from the aircraft to "warn" the air molecules since the latter is also propagated to this plane. same speed of sound.

"Caught up" by the plane, the pressure wave that can no longer move away, accumulates and then forms a shock wave, It is therefore mainly because they are deprived of the benefit of this pressure wave that supersonic aircraft are forced to be better "profiled".

Moreover, it should be known that the pressure drag is increased by the depression that is formed behind a mobile when it is poorly profiled, it is then called drag base.

III. Goals and objectives :

The main goal of the project is the development of efficient numerical methods for shape Radome design and optimization. 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 to decrease the drag of new model
- Establish a set of criteria to determine the best results.
- Obtain a robust form analysis model to optimize it.

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 Radome and their progression over time.

The following chapter has been retained to present the different parts of Aerodynamics.

State of Art

Introduction:

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

I. Different types of flow air :

Characteristics of the air:

These features will be studied in more detail in meteorology, the main definitions of the ambient air state are:

• Density: noted p expressed in Kg / m3

• The pressure, noted P, expressed in Pascal

• The temperature, noted T, expressed in degrees Kelvin (T in K = T in ° C + 273)

Temperature	T	K	217
Density	ρ	$Kg.m^{-3}$	0.413
Dynamic viscosity	μ	$Kg.m^{-1}.\ s^{-1}$	0.996×10^{-5}
Kinematic viscosity	v	$m^{2}.s^{-1}$	2.413×10^{-5}
Pressure	P	Pa	2644.4

Tab 1: the main characteristics of the Air

The flight of the aircraft is achieved through the interactions between the lift elements and the air mass in which the aircraft moves. During a flight, it is the plane that is moving relative to the air, however to study the aerodynamic phenomena related to the lift, we place ourselves in the reference frame of the plane where we consider the plane like motionless and moving air: this is the way to proceed in the wind tunnel. There are three main types of flow [4]:

1. Laminar flow:

We speak of a laminar flow when the set of air streams are parallel and follow a perfectly straight trajectory or marry perfectly the shapes of the solid.

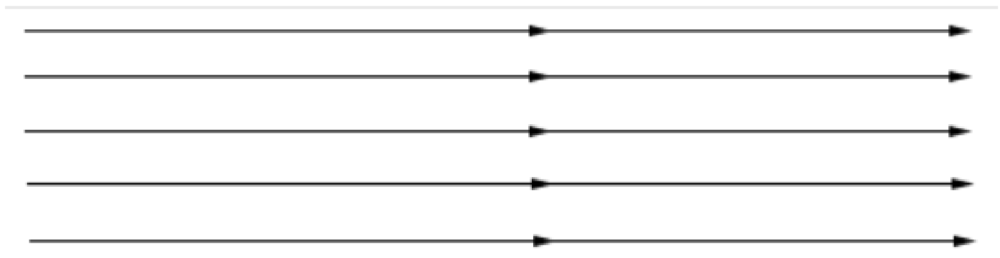


Fig 10: Laminar flow

2. Turbulent flow:

Turbulent flow occurs when the air streams follow almost all the same movement but no longer have a regular trajectory. This type of flow will tend to increase the frequency of friction between air and solid.

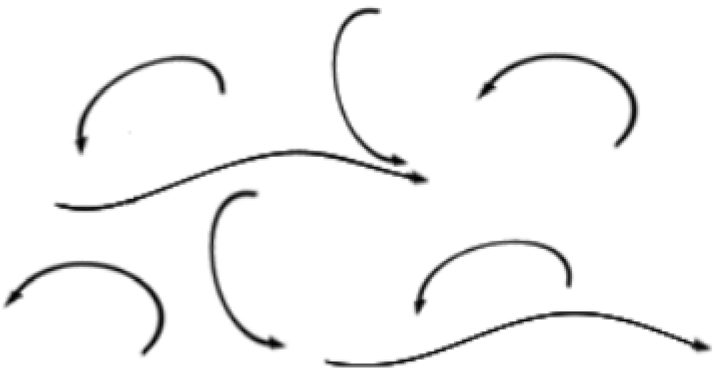


Fig 11: Turbulent flow

3. Swirling flow:

When the air streams take off completely from the solid, if it is positioned, for example, in a very inclined manner, there is a total disorganization of the air streams, which tend to form irregular swirls and no longer allow the solid to keep your balance ...

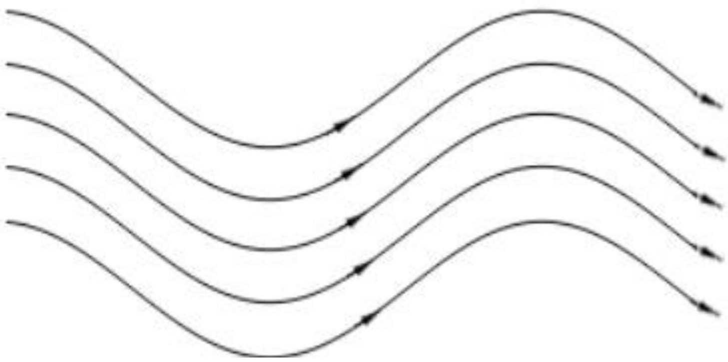


Fig 12: Swirling flow

II. Reynolds number:

The Reynolds number is the ratio of inertial forces to viscous forces within a fluid which is subjected to relative internal movement due to different fluid velocities, in which is known as a boundary layer in the case of a bounding surface such as the interior of a space.

With respect to laminar and turbulent flow regimes:

- laminar flow occurs at low Reynolds numbers, where viscous forces are dominant, and is characterized by smooth, constant fluid motion;
- turbulent flow occurs at high Reynolds numbers and is dominated by inertial forces, which tend to produce chaotic eddies, vortex and other flow instabilities.

The Reynolds number is defined as follows:

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

Where:

- ρ 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)
- μ is the dynamic viscosity of the fluid (Pa·s or N·s/m2 or kg/m·s)

ν is the kinematic viscosity of the fluid (m2/s).

III. THE AERODYNAMIC FORCE :

Before we dive into how wings keep airplanes up in the air, it is important that we take a quick look at four basic aerodynamic forces [3]: **lift**, **weight, thrust** and **drag**.

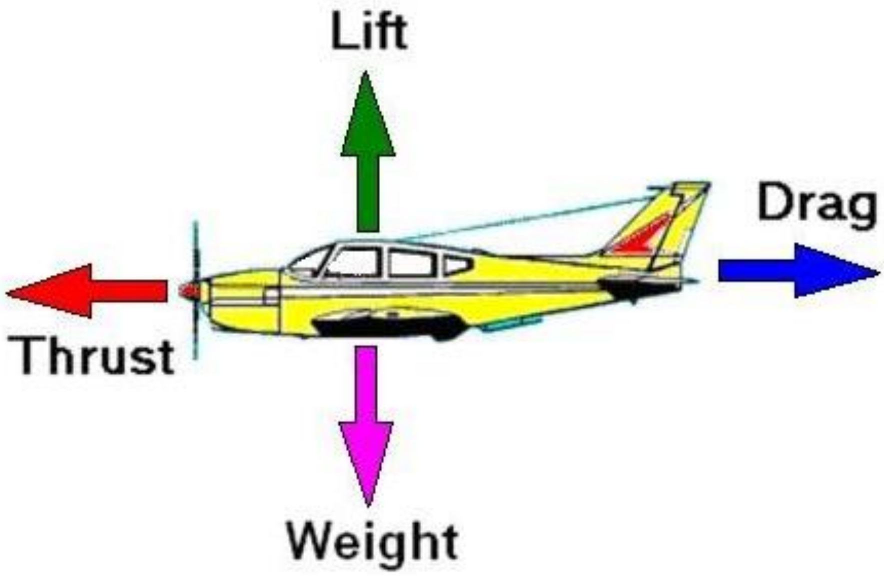


Fig 13: The Aerodynamic Forces

1. Straight and Level Flight:

In order to make an airplane fly straight and level, the following relationships must be true:

• Thrust = Drag

• Lift = Weight

If, for any reason, the amount of drag becomes larger than the amount of thrust, the plane will slow down. If the thrust is increased so that it is greater than the drag, the plane will speed up.

Similarly, if the amount of lift drops below the weight of the airplane, the plane will descend. By increasing the lift, the pilot can make the airplane climb.

a. Thrust:

Thrust is an aerodynamic force that must be created by an airplane in order to overcome the drag (notice that thrust and drag act in opposite directions in the figure above). Airplanes create thrust using propellers, jet engines or rockets. In the figure above, the thrust is being created with a propeller, which acts like a very powerful version of a household fan, pulling air past the blades.

Characteristics of the thrust:

- Point of application: On engines
- Direction: Towards the direction of movement of the plane
- Standard: **Thrust = m × (V^2 output motors - V^2 input motors)**

- **m**: mass in kg

- **V**: velocity in m.s-1

- **Thrust**: force in N (Newton)

b. Drag:

Drag is an aerodynamic force that resists the motion of an object moving through a fluid (air and water are both fluids). If you stick your hand out of a car window while moving, you will experience a very simple demonstration of this effect. The amount of drag that your hand creates depends on a few factors, such as the size of your hand, the speed of the car and the density of the air. If you were to slow down, you would notice that the drag on your hand would decrease. We see another example of drag reduction when we watch downhill skiers in the Olympics. You'll notice that, whenever they get the chance, they will squeeze down into a tight crouch. By making themselves "smaller," they decrease the drag they create, which allows them to move faster down the hill.

If you've ever wondered why, after takeoff, a passenger jet always retracts its landing gear (wheels) into the body of the airplane, the answer (as you may have already guessed) is to reduce drag. Just like the downhill skier, the pilot wants to make the aircraft as small as possible to reduce drag. The amount of drag produced by the landing gear of a jet is so great that, at cruising speeds, the gear would be ripped right off from the plane.

Features of the Drag:

- Application point: On the upper surface of the wing
- Direction: Horizontal, opposed to the direction of movement of the plane
- Standard:

$$\mathbf{Tr = 1/2 \times S \times \rho \times V^2 \times Cx}$$

- **ρ**: density of the air in kg / m 3, It varies with temperature and pressure.

- **V**: velocity in m / s

- **S**: area in m²

- **Cx**: coefficient of drag (without units)

- Drag: force in N (Newton)

c. Lift:

Lift is the aerodynamic force that holds an airplane in the air and is probably the trickiest of the four aerodynamic forces to explain without using a lot of math. On airplanes, most of the lift required to keep the plane aloft is created by the wings (although some is created by other parts of the structure).

Features of the Lift:

- Application point: On the upper surface of the wing
- Direction: Vertical Up
- Standard:

$$Fa = 1/2 \times S \times \rho \times V^2 \times Cz$$

- **ρ**: density of the air in kg / m 3, It varies with temperature and pressure.

- **V**: speed in m / s

- **S**: area in m²

- **Cz**: lift coefficient (without units)

- **Lift**: force in N (Newton)

d. Weight:

Weight, which 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:

$$P = m \times g$$

features of the weight:

- Point of application: Center of gravity of the aircraft
- Direction: Vertical and down

with **P** in Newton, **m** in Kg and **g** gravity constant = 9.81 N / Kg.

2. Types of Drag:

In reality there are several Drags:

- lift-induced drag, induced drag or vortex drag is a lift-resistance force created by lift.

- The parasite drags composed of:

+ Friction drag related to velocity differences between fluid streams (viscosity).

+ Form drag appears as soon as an object has a certain thickness.

- Interference drag is due to the junction between the different components.

i. lift-induced drag:

Induced drag is a lift-induced resistance force that depends on certain characteristics of the wing (elongation, shape, arrow, etc.). [1]
The mechanism of induced drag has been theorized by Ludwig Prandtl (1918). To have a lift, there must be a relative overpressure on the underside of the wing and / or a depression relative to the upper surface of the wing. Under the effect of this pressure difference, the air passes directly from the intrados to the extrados, by-passing the end of the wing and creates a deflection of the air flow.

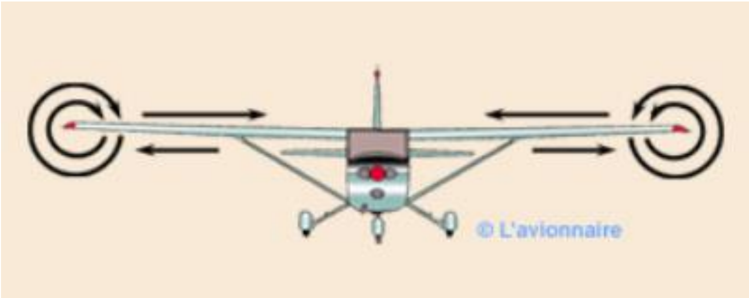


Fig 14: lift-induced drag

ii. Drag friction:

Friction drag is due to the viscosity of the air. Air molecules in contact with the surface of a body are braked by friction forces. These forces are such that the velocity of the air streams is zero in contact with the body. Friction drag affects all parts of the aircraft. Its size varies with:

- the total surface area of the aircraft liner (wing, fuselage, empennage etc...)

- the boundary layer (laminar, turbulent).

- the roughness of the walls.

- the relative wind speed.

- the shape and thickness of the profile.

- the angle of attack.

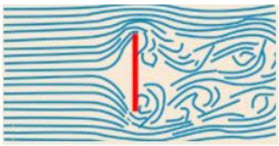
- pressure difference between the extrados and the intrados

Friction drag is currently under intensive investigation. One solution is to modify the wall of the aircraft by applying longitudinal grooves or riblets that mimic the shark skin. Another solution to achieve this is the suction of the boundary layer to stabilize or the blowing of the extrados. For a transport aircraft of the Airbus or Boeing type, the friction drag contributes about 50% to the total drag and the wing is responsible for about half of the friction drag. In other words, the friction of the boundary layer on the wings is responsible for a quarter of the total drag.

iii. The Form drag:

As the name suggests it is related to the shape of the profile. It is the part of the resistance that is caused by the fact that when a viscous fluid flows around a solid object, vortex is formed and the flow threads are no longer smooth. It is essential that shape drag is minimized on all parts of the aircraft that are exposed to the air. For example, a perpendicular plate exposed to relative wind has a resistance of 100%, a cylinder has 50%, a tapered shape has 15%, a profile has 5%.

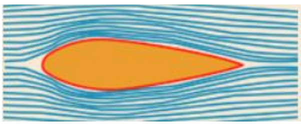
Airflow on a thin flat disc arranged parallel to the air streams is the seat of minimal resistance due simply to the friction of the air on both sides of the plate.
Nearly zero resistance



By placing a flat disk perpendicular to the airflow of a wind tunnel, we find that the flow of the air is difficult to circumvent the obstacle and that an overpressure is formed at the front and a depression at the rear with swirling effect.
100% resistance

In the case of sphere, the flow is improved, the rear swirl zone is reduced, but not completely resorbed.
50% resistance

By stretching the back of the sphere to obtain substantially the shape of an egg, it is found that the air threads meet at the rear without creating swirls. This result in a streamlined body.
15% resistance



A tapered body "pointed" at the back (trailing edge), create a circulation.

5% resistance

The air resistance of a solid can be written using this equation:

$$\mathbf{R = K.\ \rho.\ V^2.\ S}$$

K = coefficient of the shape and surface of the solid, without unit.

ρ = density of the air in Kg m-3

V = air particle velocity in m s-1

S = area of solid in m2

iv. shock waves Drag:

Finally, the wave length, although low, can be further reduced by an optimization of the shapes or by the use of local control devices of the shock waves.

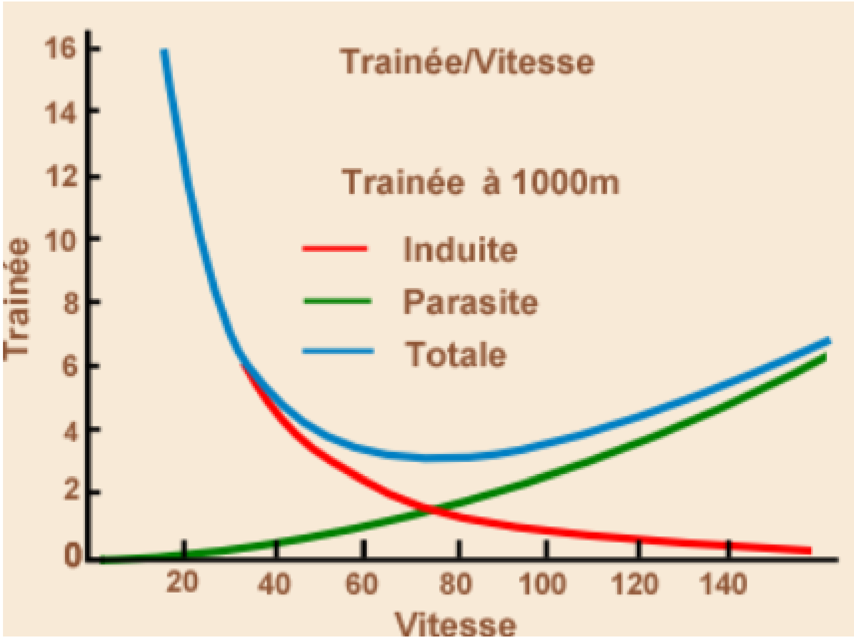


Fig 15: The different Types of drag according to the velocity

While the drag of a slow object is controlled by viscosity, that of a fast object is mainly due to the pressure force induced by the flow, called dynamic pressure. As we have seen, this pressure makes it possible to evacuate laterally the surrounding fluid from upstream to downstream, so that the moving body can take its place. This difference in pressure generates an overall thrust on the body, which opposes its movement.

At high speed the effect of the viscosity becomes negligible, and this overpressure at the stopping point can be estimated at ρfU2 / 2 using the Bernouilli relation. By multiplying this overpressure by the transverse area S, we obtain the law of turbulent drag stated above. In reality, it is the transverse area of the turbulent wake, of the order of Cx rather than the area S of the object itself, which controls the drag.

In the case of an asymmetric body, such as an airplane wing or a sail, another dynamic pressure force appears, perpendicular to the speed, which is called lift (read the article Archimedes thrust and lift). It adds to the drag, which is aligned with the speed and opposite direction. These two forces are proportional to the density of the fluid and the square of the velocity, and their ratio is therefore constant. As we will see later this one characterizes the capacity of a plane to glide: this is called the lift-to-drag ratio **“finesse”.**

3. the lift-to-drag ratio “finesse”.

In aerodynamics, the lift-to-drag ratio, or L/D ratio, is the amount of lift generated by a wing or vehicle, divided by the aerodynamic drag it creates by moving through the air. A higher or more favorable L/D ratio is typically one of the major goals in aircraft design; since a particular aircraft's required lift is set by its weight, delivering that lift with lower drag leads directly to better fuel economy in aircraft, climb performance, and glide ratio.

$$\frac{C_z}{C_x}$$

4. Definition of the boundary layer:

The boundary layer is generally defined as a region of small thickness produced by the flow of a viscous fluid in the presence of an obstacle. A boundary layer is described as laminar, when the value of the Reynolds number Re (order of magnitude of the ratio of the inertia forces on the viscous forces) is less than a critical value, of about 3 x 10; otherwise it is described as turbulent.

The first measurements made in the turbulent boundary layer (Townsend 1951, Klebanoff 1954, Laufer 1954, Kovaszney 1967) made it possible to distinguish different regions, described by different characteristic kinematic quantities.

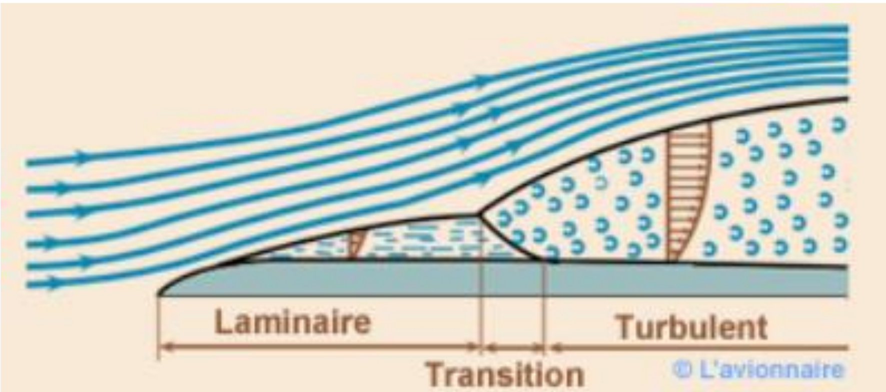


Fig 16: the boundary Layer

In the near-wall region, the kinematic quantities are generally related to the boundary layer internal parameters, ur, the wall friction rate and the kinematic viscosity of the fluid. We define nondimensional quantities such as average velocity $\overline{U}^{+}$ and distance **y+** by: $\overline{U}^{+}$ = $\overline{U}$ **/ u_r** and **y+ = y u_r / υ**. In the outer region, other parameters may occur such as **δ** the thickness of the boundary layer and **U□** the flow velocity outside. Thus, the average speed profile in the turbulent boundary layer makes it possible to distinguish three regions.

- The viscous sub-layer, valid for **O ≤ y+ ≤ 5** where the viscous stress is greater than turbulent stress. In this region the average speed profile is linear, $\overline{U}^{+}$ = y+.
- A second region, where the production and dissipation of turbulent kinetic energy are dominant. It consists of a "buffer zone" also known as "buffer layer" valid for **5 ≤ y+ ≤ 30**, and a logarithmic zone defined by:

  $$\overline{U}^{+} = \frac{1}{k} \ln y^{+} + B,$$

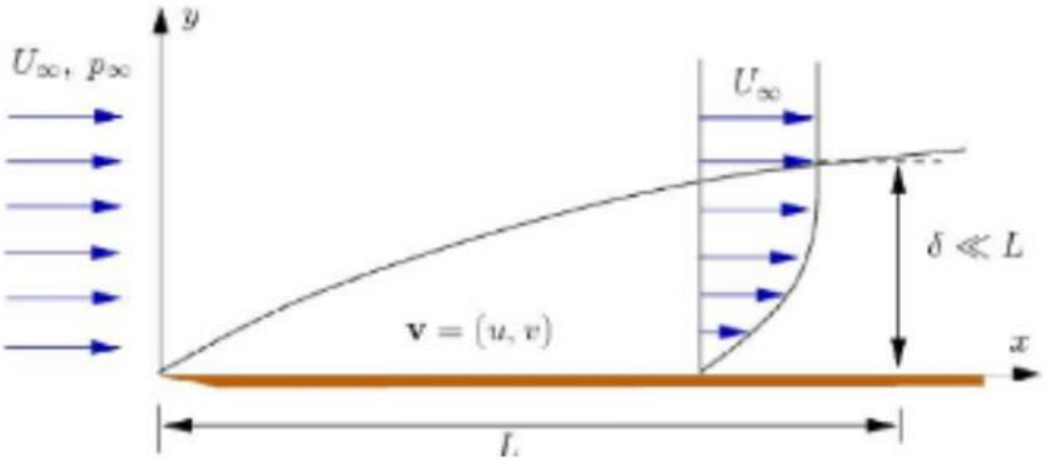
  valid for **30 ≤ y+ ≤ 200**, **k** being the universal constant of Von Karman and B a constant close to 5. The viscous sub-layer and this second region constitute the internal region of the turbulent boundary layer and corresponding to the zone y ≤ 0,2 **δ**.

- An outer region where the flow structure is entirely controlled by turbulence. It is valid for **y/δ ≥ 0,2**. In this region, we talk about the law of speed deficit which is given by:

  $$\frac{U_{\infty} - \overline{U}}{u_{\tau}} = \Phi\left(\frac{y}{\delta}\right)$$

  where **U□** - $\overline{U}$, represents the speed deficit with respect to the external speed. The function **ɸ** is not universal because it can be influenced by many parameters such as the pressure gradient, the Reynolds number or the development conditions of the boundary layer upstream of the abscissa x.

  It should be noted that the transition between these different regions requires a recovery of speed profiles valid in two neighboring regions.



5. Turbulent structure of the boundary layer:

Since the late sixties, many studies have demonstrated the existence of different types of turbulent structures in a boundary layer flow [2]. Currently, there are no models capable of clearly establishing the spatio-temporal relationships between these different structures, nor the dynamics that animate them and the links of cause and effect that connect them. The only sure results are the detection and recognition of well-marked events that play a key role in the mechanism of turbulence generation and maintenance. Thus, several works have been realized with the aim of a better understanding of the links between these structures, in order to elaborate models of calculation more and more faithful to the phenomenological reality. These works are on the one hand, of an experimental nature (spatial correlations of speed, conditional analyzes, visualizations) and on the other hand numerical with the help of the exploitation of the results of direct simulations of the Navier Stokes equations.

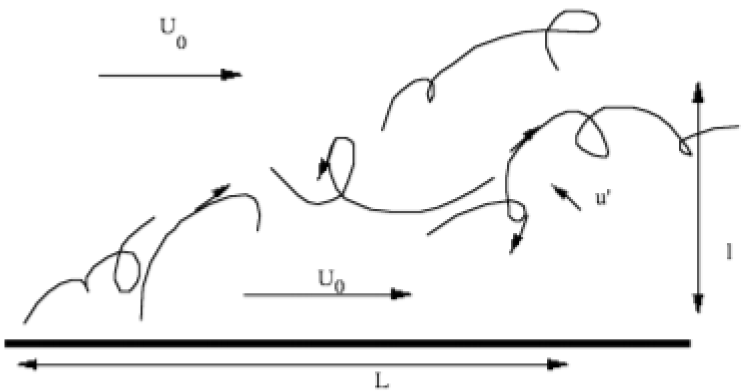


Fig 17: artistic view of a turbulent boundary layer.

At the origin of this work, the visualizations carried out by the Stanford University group (Kline et al., 1967, Kim et al, 1971) revealed the existence in the near-wall zone of a sequence of organized "events" contributing predominantly to the production of energy turbulent kinetics. This sequence has been called "burst" is divided into three phases:

- ✓ emergence in the near-wall zone of highly decelerated fluid zones, referred to as "low speed streaks" (Training Streaks),
- ✓ progressive uprising accompanied by weak oscillations of these "streaks" (streaks lifting),
- ✓ amplification of oscillations and appearance of more disordered movements (breakup).

These observations completely changed the idea of the structure of the turbulent boundary layer at the time, and since then studies in this type of flow have focused on the "organized" aspect of the turbulent boundary layer flow.

IV. Navier Stokes equations :

The Navier-Stokes equation is one of the most important of all physics. If it does not have the chance to be as well-known as E = mc2, it still serves us to predict the weather, simulate the oceans, optimize the wings of planes and even improve the realism of video games.

In fluid mechanics, the Navier-Stokes equations are nonlinear partial differential equations that describe the movement of Newtonian fluids (hence gases and most liquids). The resolution of these equations modeling a fluid as a single phase continuous medium is difficult. [4]

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. We can use it for example to understand the currents movements in the oceans as well as those of large air masses in the atmosphere.

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)

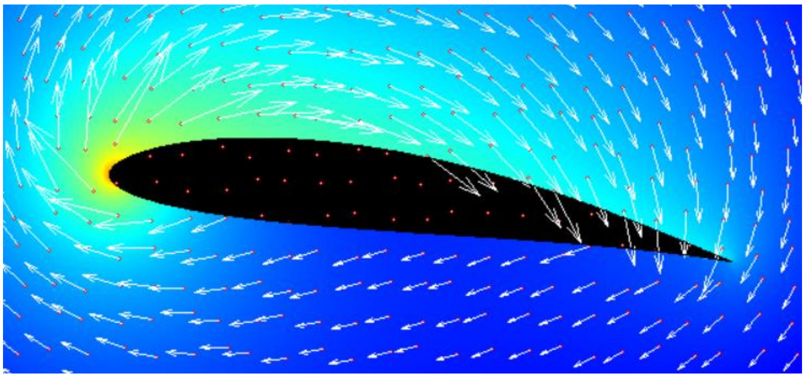


Fig: Air motion around an airplane Wing

V. Divergence

The divergence at a point (x, y, z) is the measure of the flow of the vector on a surface surrounding this point. According to its sign, the divergence expresses the dispersion or the concentration of a quantity (such as a mass for example) and the previous theorem indicates that a dispersion within a volume is necessarily accompanied by an equivalent total flow out of its border.

If a fluid is circulating in a certain area, then the rate at which the fluid flows from a certain region within that area can be calculated by adding the sources within the region and subtracting the wells.

The fluid flow is represented by a vector field, and the divergence of the vector field at a given point describes the strength of the source or sink. Therefore, the integration of the divergence of the field within the region should be equal to the integral of the vector field on the boundary of the region.

The divergence theorem says that it is true. [4] The divergence theorem is used in any conservation law that states that the total volume of all wells and sources, the volume integral of the divergence, is equal to the net flow through the volume limit.

conclusion:

In this chapter we contextualize the work and we expose the aerodynamic prerequisites, then we explain the flow around a body and the drag.

Material and Methods

Introduction:

In this chapter, we are interested in the numerical simulation software used to solve the problems, import the results to create our new model

I. What's CATIA :

CATIA (an acronym of computer-aided three-dimensional interactive application) is a multi-platform software suite for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), PLM and 3D, developed by the French company Dassault System's.

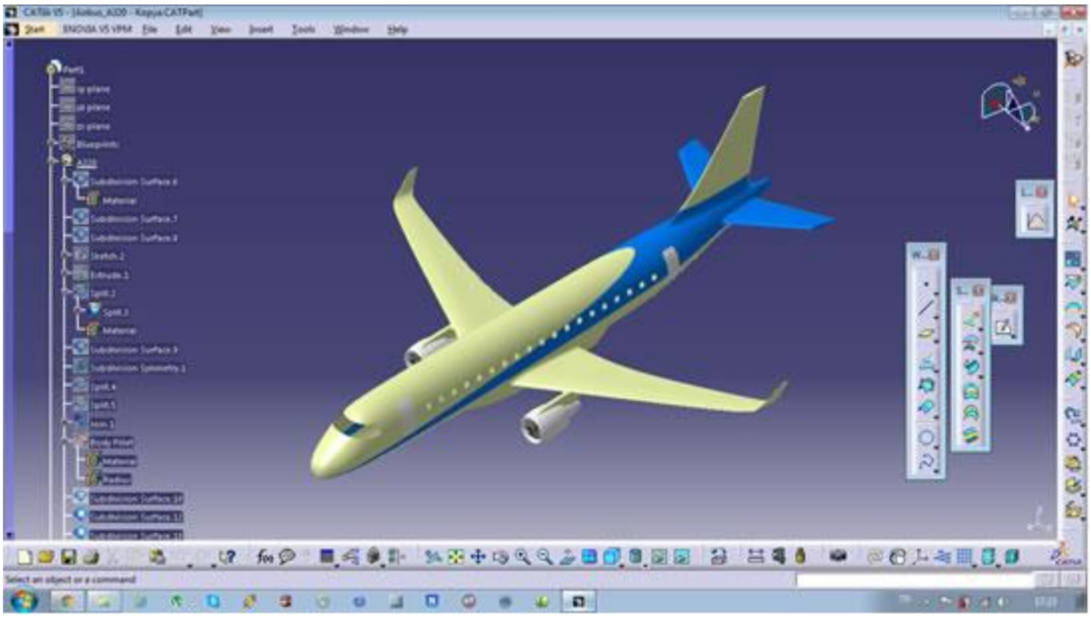


Fig 18: Airplane model in Catia

CATIA enables the creation of 3D parts, from 2D sketches, sheet metal, composites, molded, forged or tooling parts up to the definition of mechanical assemblies. The software provides advanced technologies for mechanical surfacing & BIW. It provides tools to complete product definition, including functional tolerances as well as kinematics definition. CATIA provides a wide range of applications for tooling design, for both generic tooling and mold & die. In the case of Aerospace engineering an additional module named the aerospace sheet metal design offers the user combine the capabilities of generative sheet metal design and generative surface design.

CATIA offers a solution to shape design, styling, surfacing workflow and visualization to create, modify, and validate complex innovative shapes from industrial design to Class-A surfacing with the ICEM surfacing technologies. CATIA supports multiple stages of product design whether started from scratch or from 2D sketches(blueprints).

II. What's Ansys:

ANSYS structural analysis software enables you to solve complex structural engineering problems and make better, faster design decisions. With the finite element analysis (FEA) tools available in the suite, you can customize and automate solutions for your structural mechanics problems and parameterize them to analyze multiple design scenarios. You can connect easily to other physics analysis tools for even greater fidelity. ANSYS structural analysis software is used throughout the industry to enable engineers to optimize their product designs and reduce the costs of physical testing.

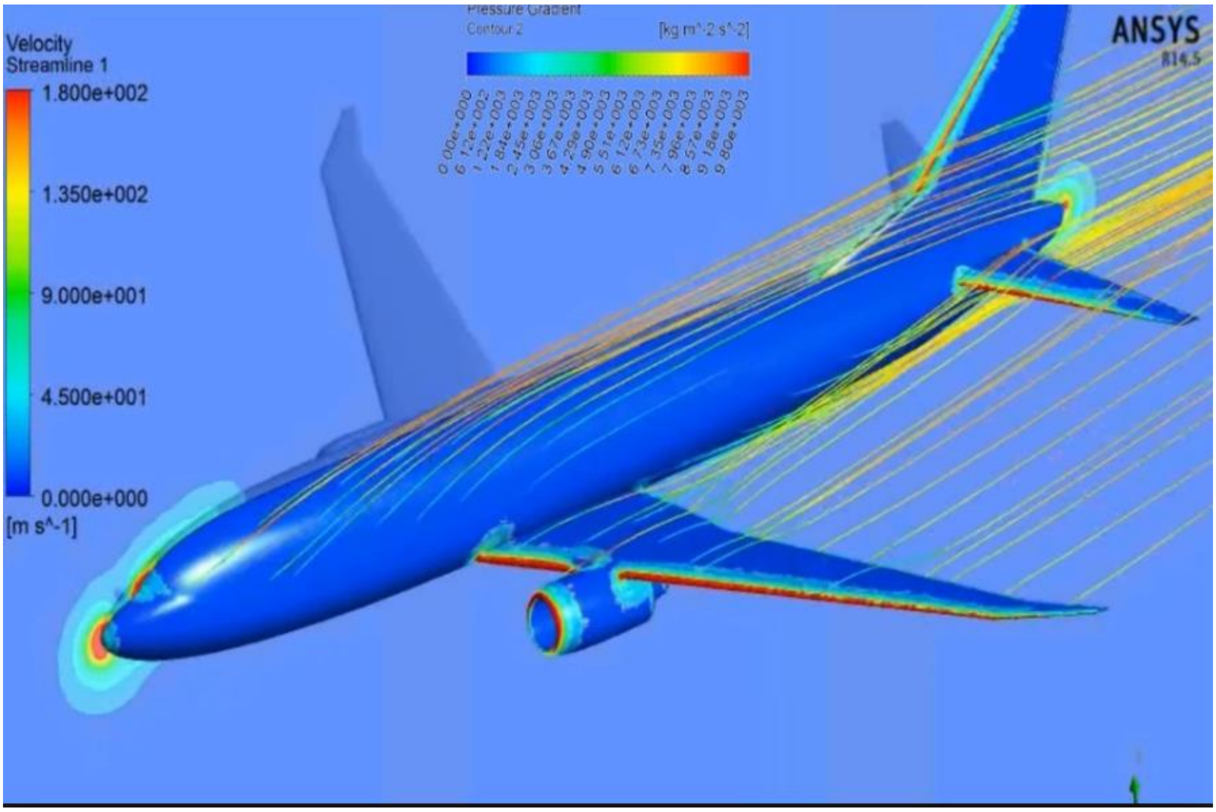


Fig 19: Airplane model in Ansys

1. Mesh definition:

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.

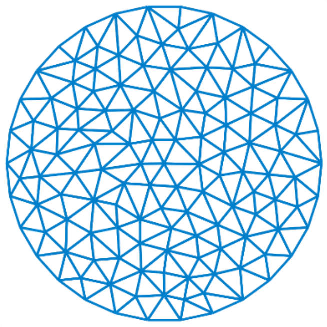


Fig 20: Example of Mesh

A mesh size is defined by:

- his landmark;
- points the constituent, characterized by their co-ordinates;
- cells, constituting polytopes linking up n of these points;

and is possible be notably characterized by:

- his dimension: typically, 2D or 3D;
- her volume (covered complete dimension);
- his delicacy: surface or medium volume of cells composing mesh size;
- the geometry of cells: triangles, quadrilaterals (parallelograms, rectangles, squares), polygons, there 2D; tetrahedrons, prisms, hexaèdres (parallelepipeds, cubes), polyhedrons their 3D;
- the degree of the element: it is the degree of the polynomial serving for describing sides or bones of elements, an element of degree 1 has sides or straight bones; in the case of the finished elements, it is also the degree of polynomials of interpolation

Conclusion:

in this chapter, we make a presentation of the numerical simulation software used in the project

Description for realization:

Introduction :

We will focus in this chapter on the presentation of the design of the different parts leading to the realization of our objective which is summed up in the creation of a new model with less air resistance.

I. Development environment:

As we said in the previous chapter, digital software can simulate the problem to give an adequate solution close to reality.
in this chapter, we will draw a real model of the radome A320 aircraft and B787 aircraft to compare the aerodynamic effect on these two aircraft. and after the comparison, we will create a new model with less drag and friction according to the results between two aircrafts.

1. CATIA:

a. A320:

Before drawing the model, we must know the true geometry and, in the drawing, below, there is a complete drawing of all the dimensions of the plane.

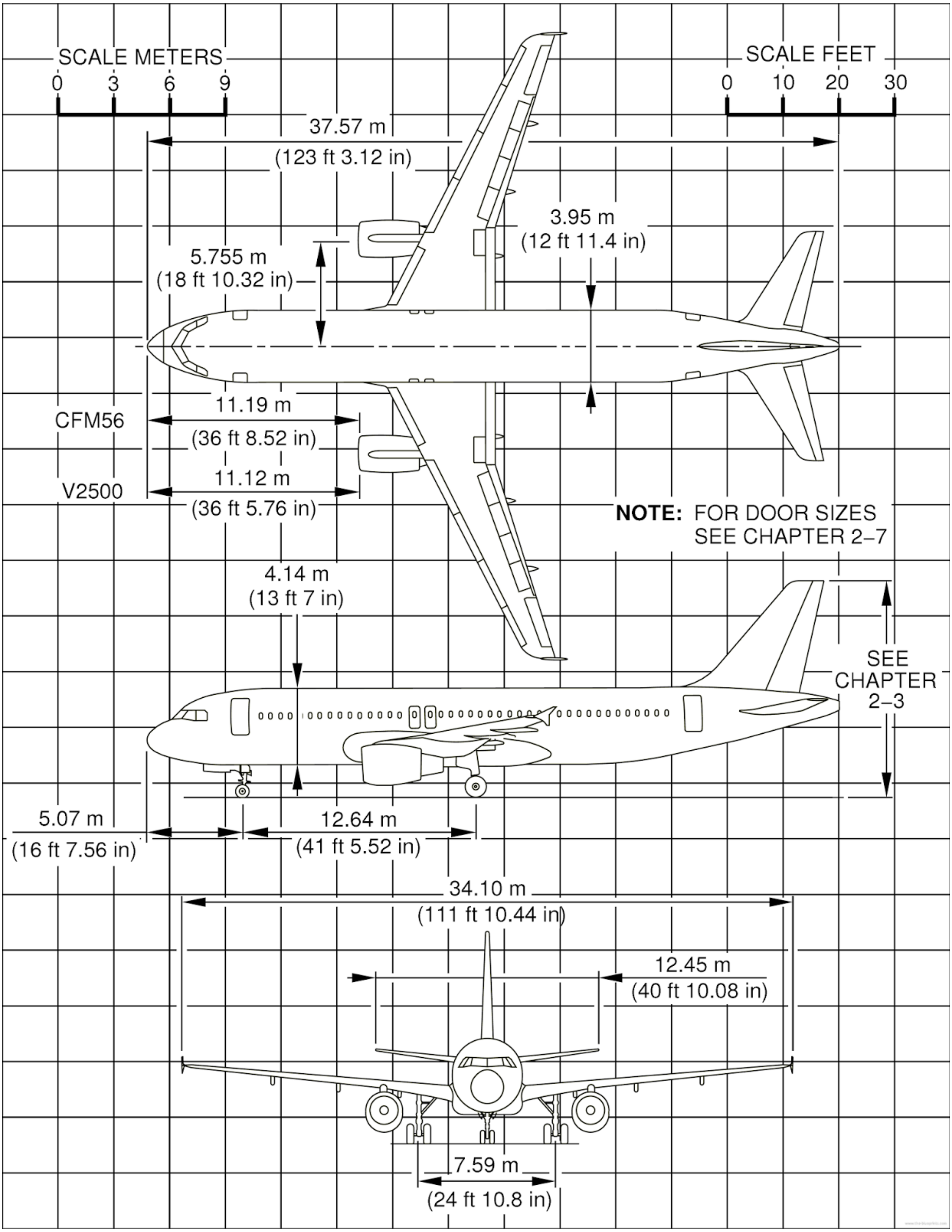


Fig 21: A320 geometry 1

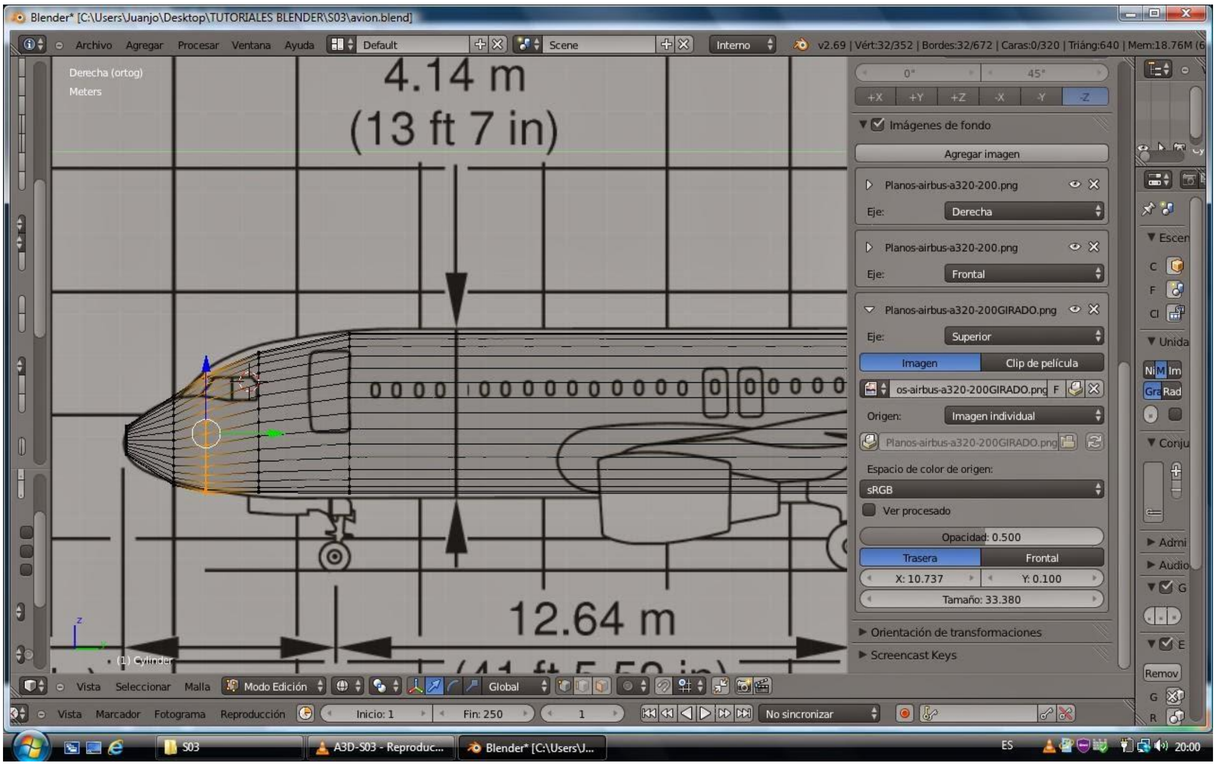
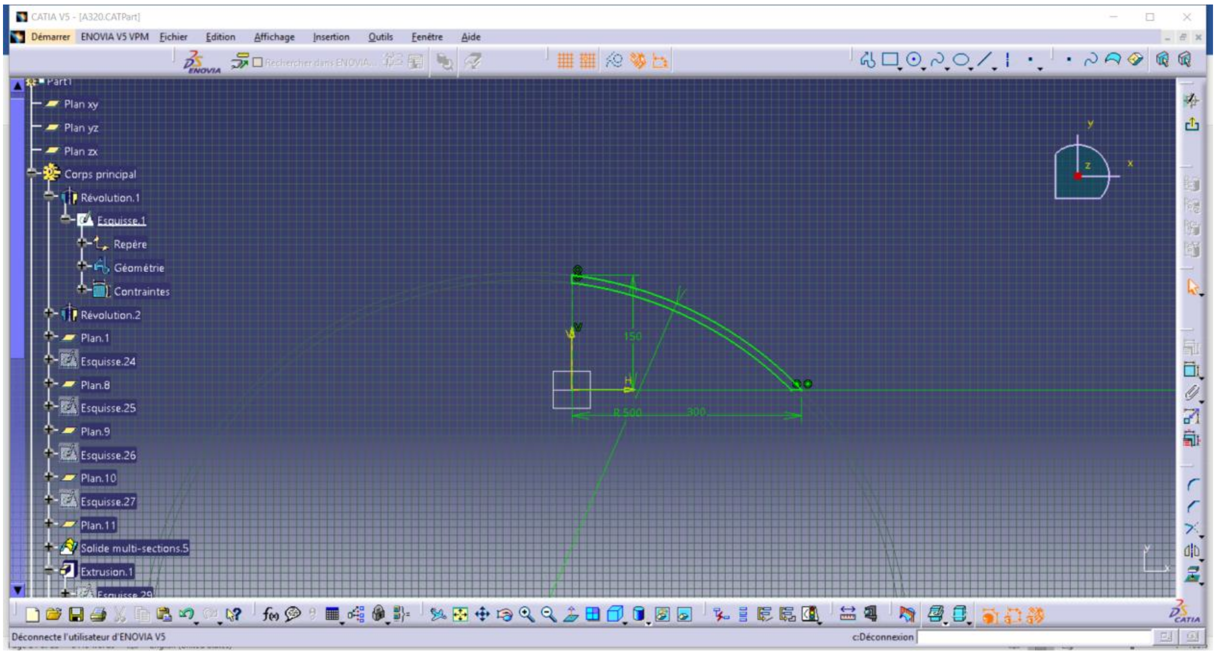
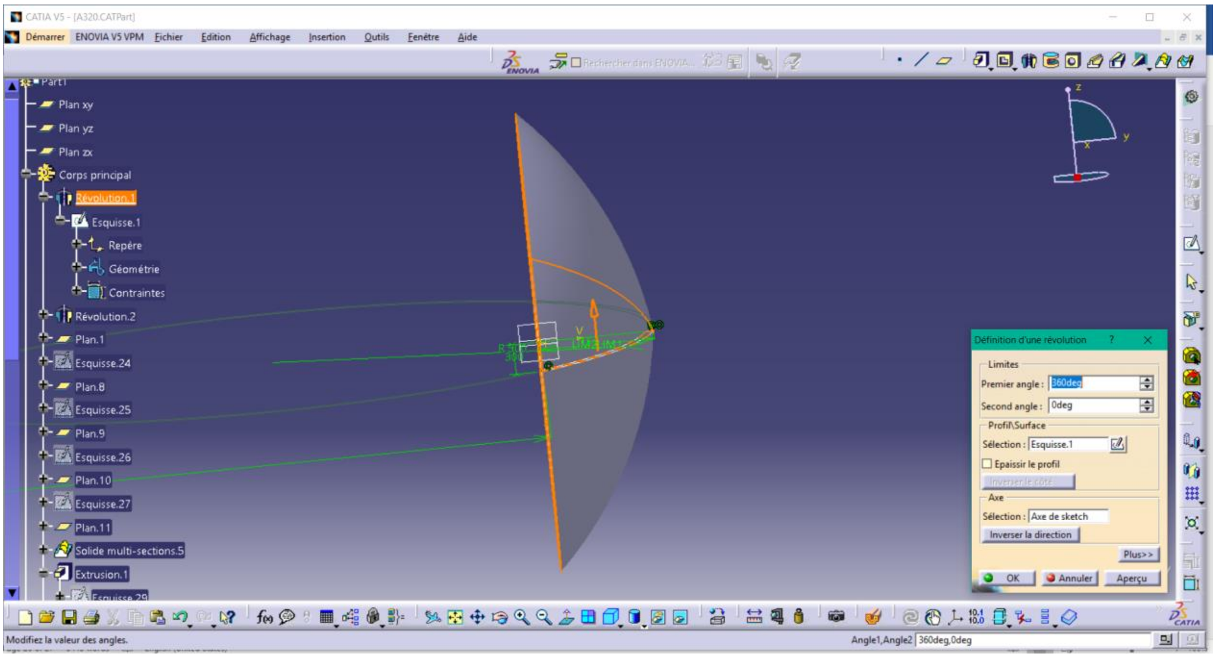


Fig 22: A320 geometry 2

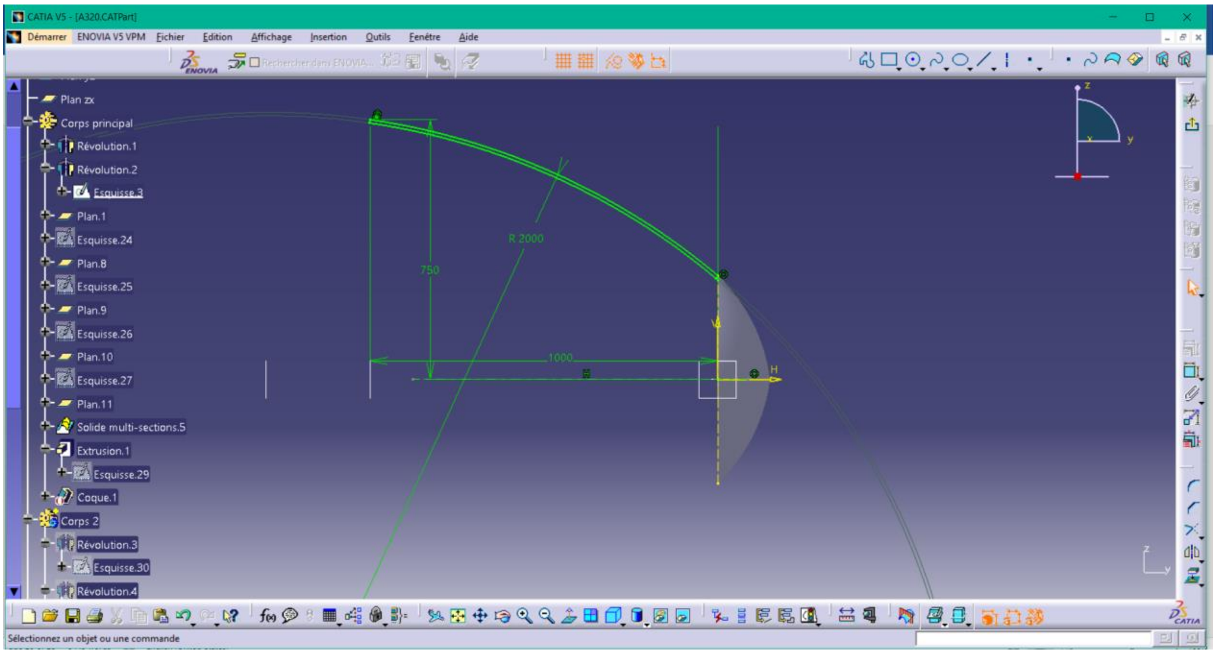
The first step to design the A320 model is to select the part design then you have to choose one of three plans to draw on it.



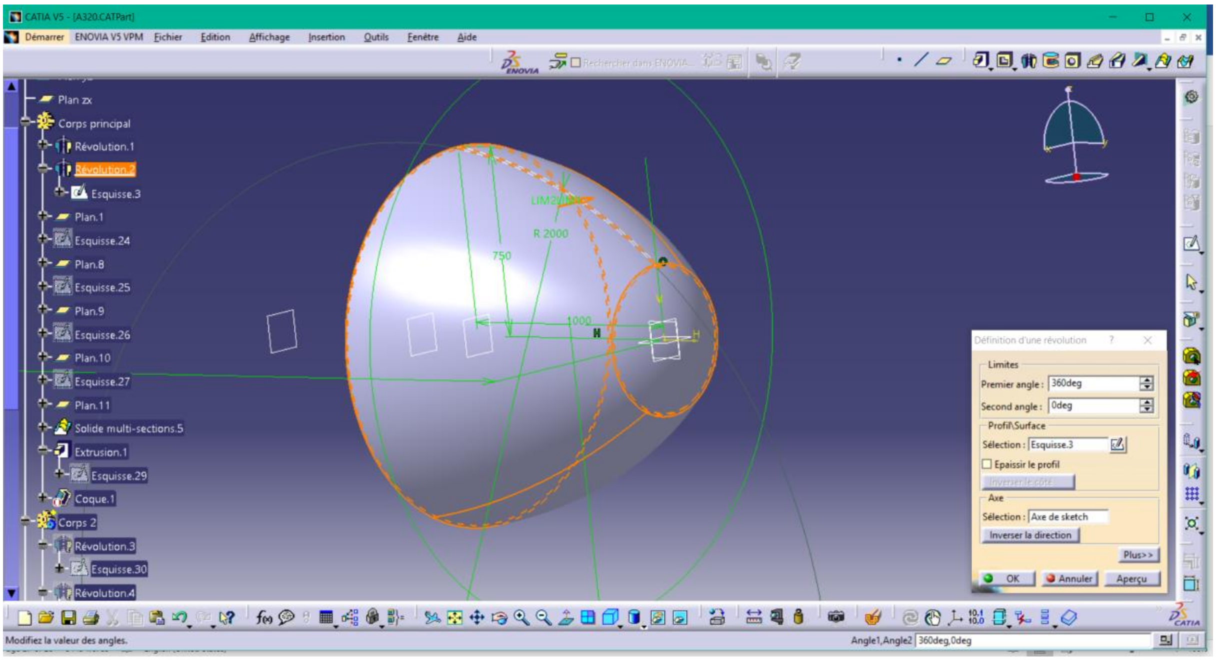
When we complete drawing the first step, we can use the tool shaft  around 360°



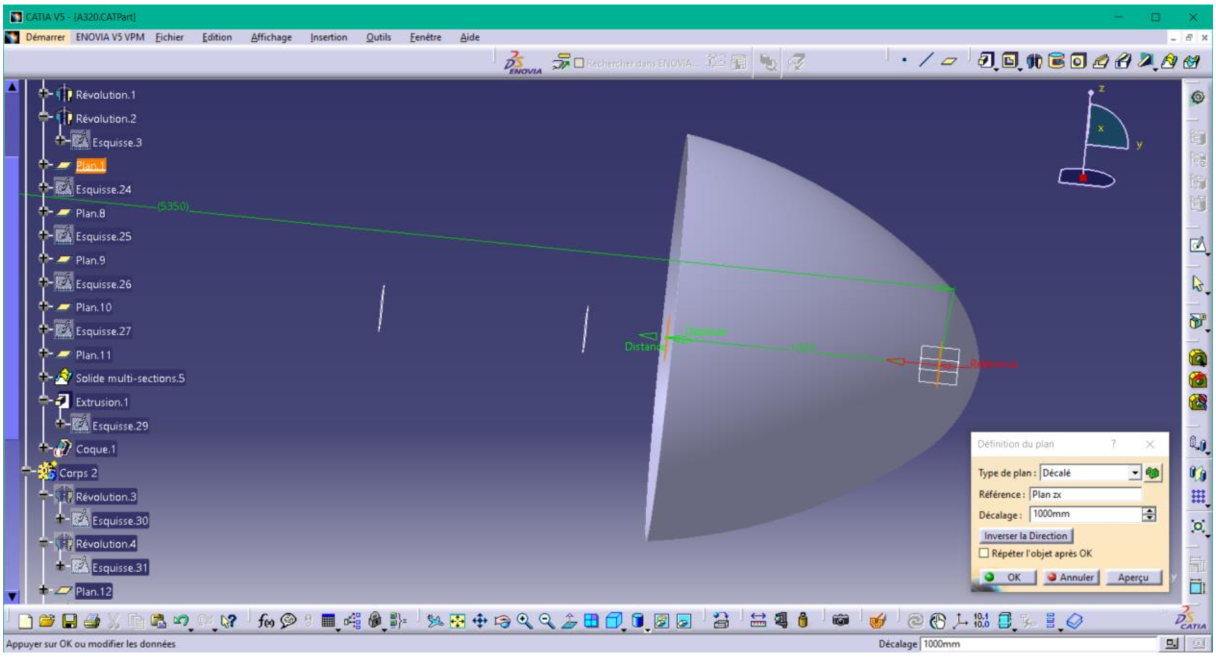
Then we have to select another plan perpendicular of the first plan choosing to draw the next part of the design with the following geometry.



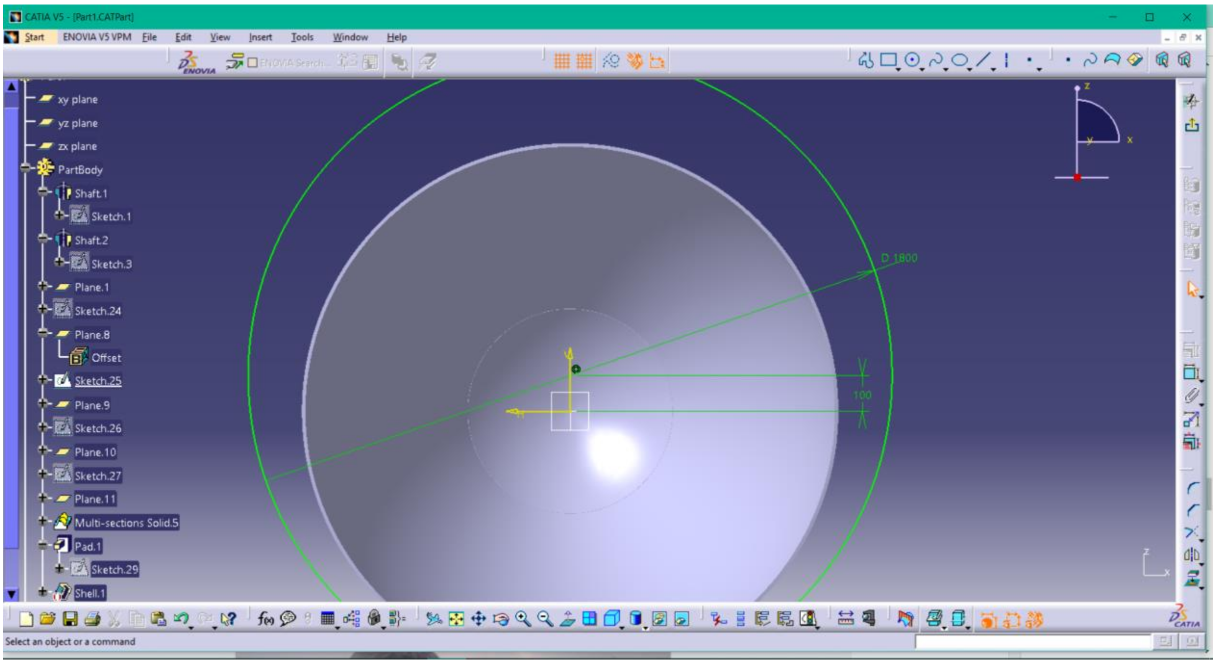
Now, we create another shaft around 360°.



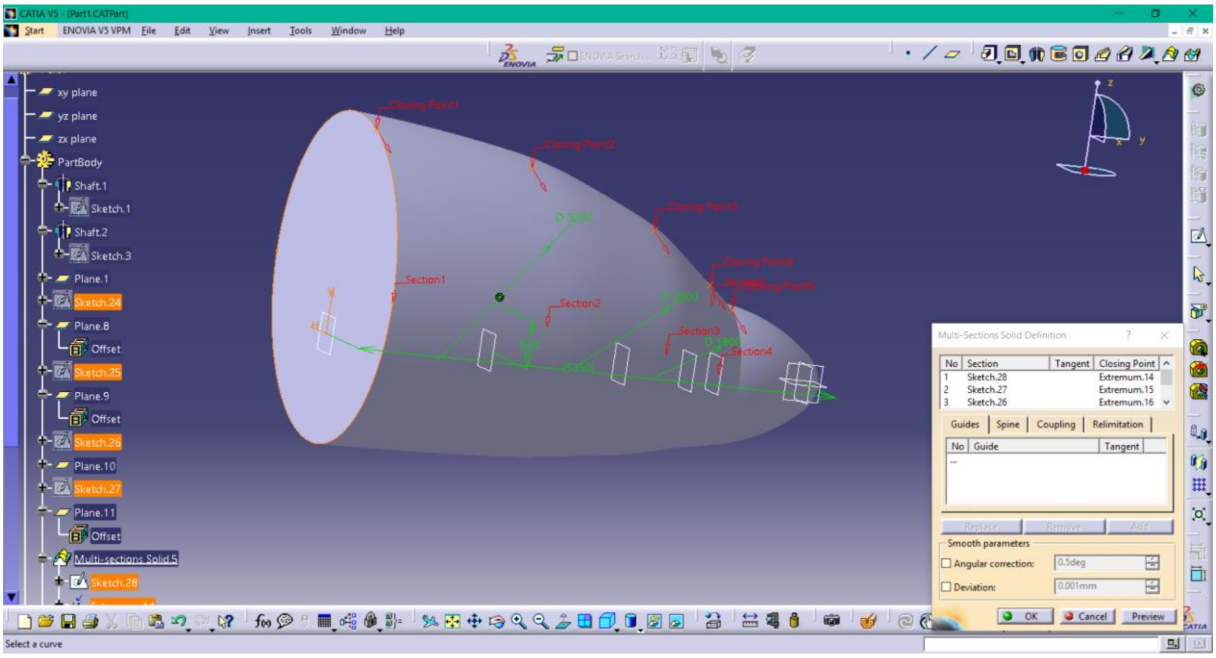
To complete the next part, we should create many plans .



For the next step, we draw the sketch below

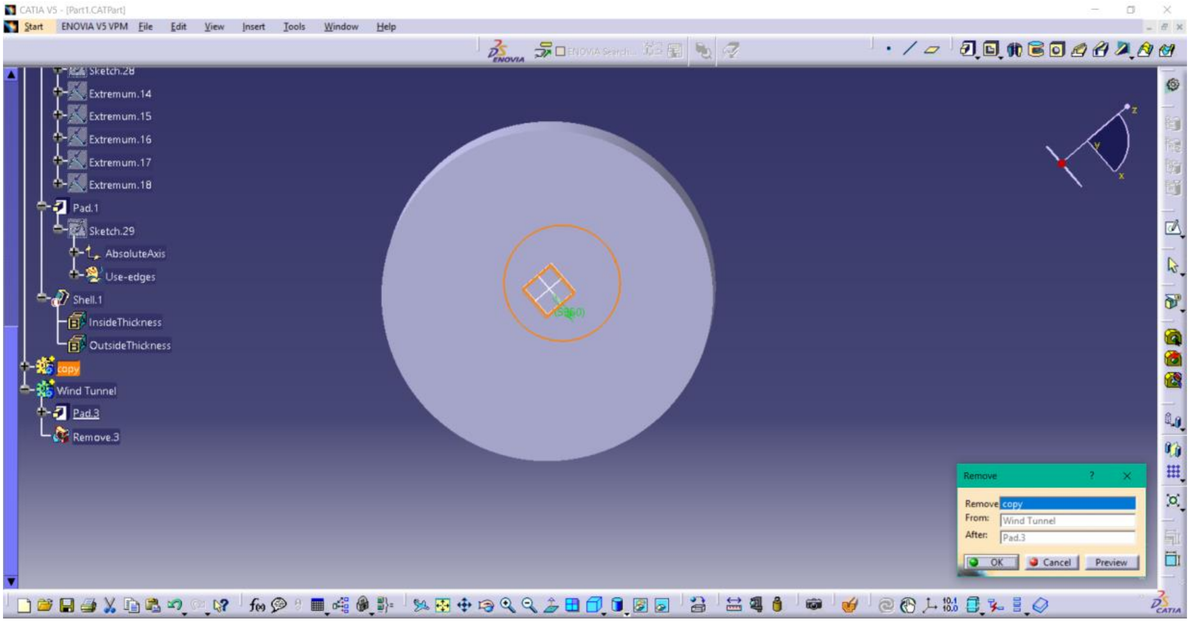


After repeating the previous step tree times. We have to create multi-sections solid to connect between all the parts.



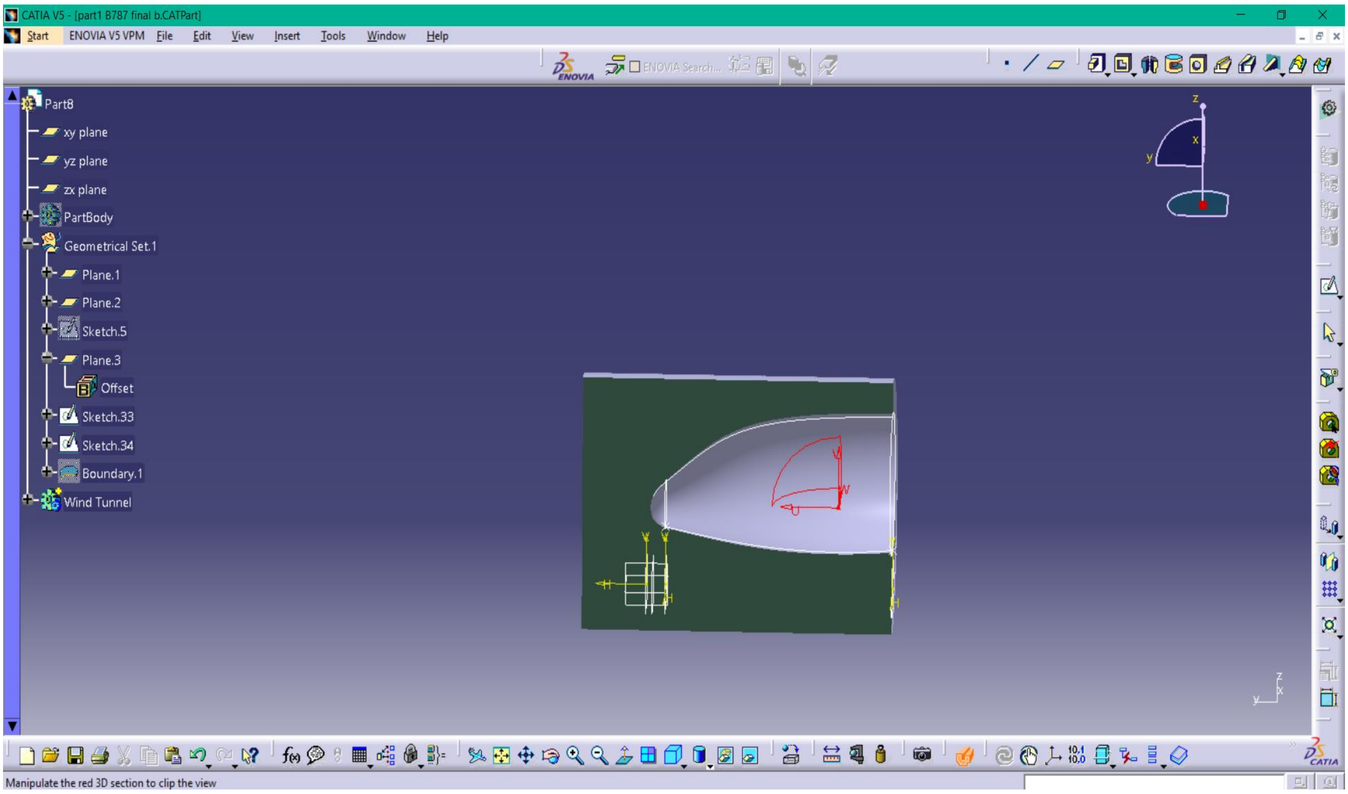
Why we create a tunnel around the corps?

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.



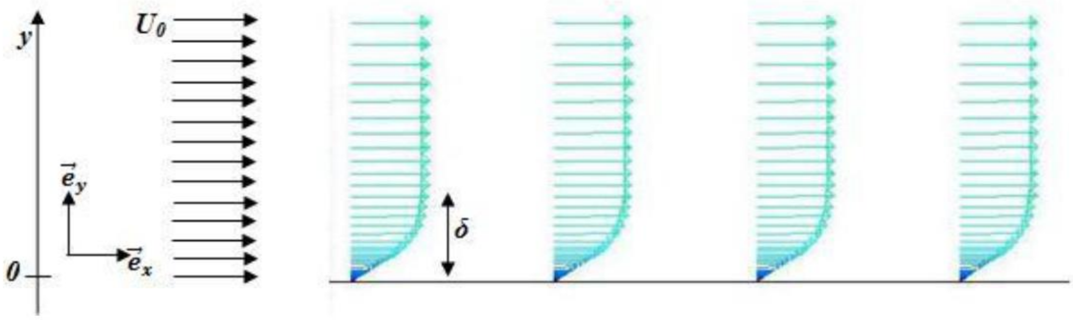
b. B787:

After finish drawing the A320 part, we have to repeat drawing the second part (B787) but with other dimension and different geometry.



2. Ansys:

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.



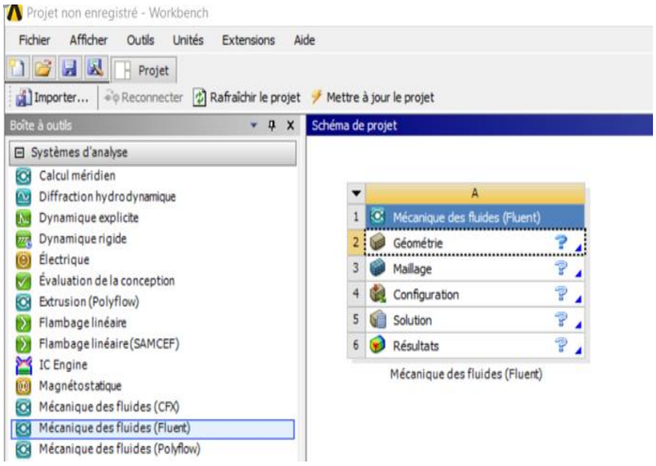
The velocity of the flow on the plate $V = u\ x, y\ e\ x$ where u (x, y≥δ) = U0. With: U0 the initial velocity of the flow and δ the thickness of the boundary layer.

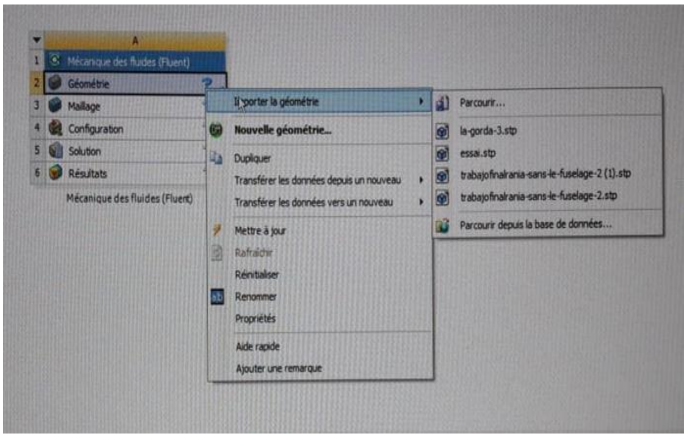
This flow will be modeled using the ANSYS-Fluent system in the ANSYS-Workbench 15.0 suite. The steps followed to study this problem are:

- The design of the fluid domain geometry using the "ANSYS-Design Modeler" system
- Mesh generation of the fluid domain using the "ANSYS-Meshing" system.
- Start the simulation in the "ANSYS-Fluent" system after the determination of: physical properties of fluid, boundary conditions, numerical simulation models ....
- Display the results (pressure contours, speed, ...) by "ANSYS-Fluent" and / or "ANSYS-CFD-Post".

a. Numerical method:

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



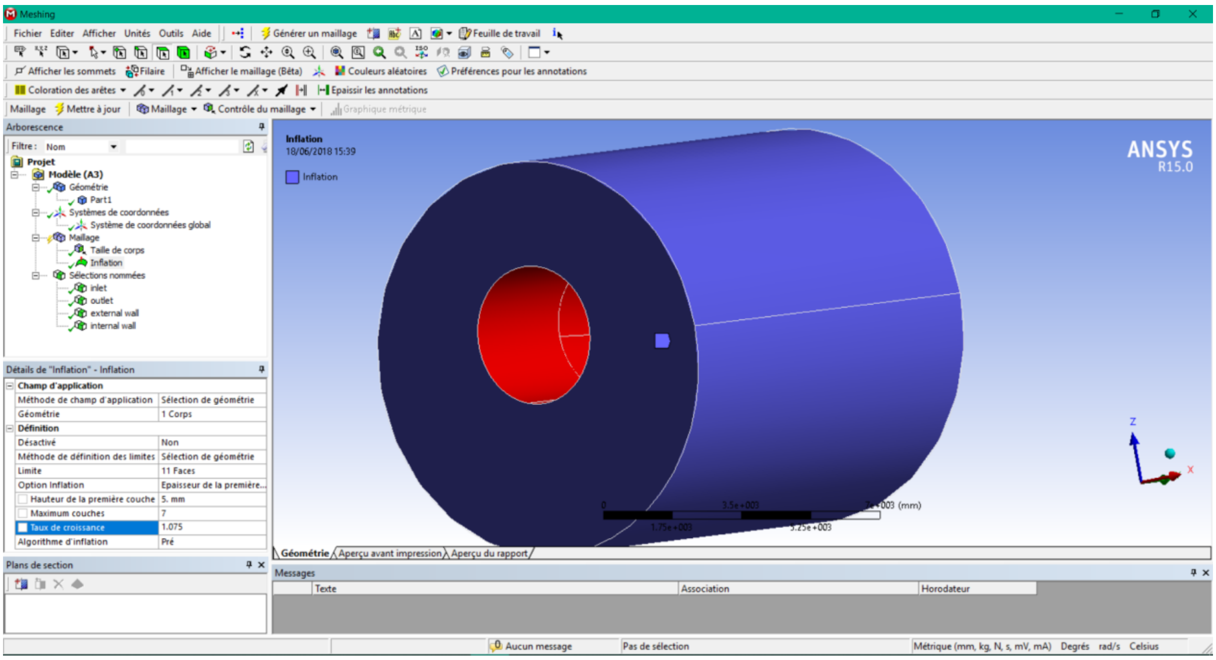


i. Mesh:

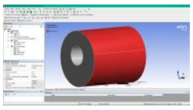
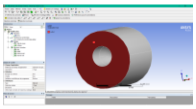
To achieve a good result, it is necessary that the mesh would be with high precision for that and through many tests I made it, the physical properties are given in the table following:

Velocity	**250 m/s**
Size of elements	150 mm
Number of layers	7
Ratio	1.075
Y+	5

Tab 2: the mesh characteristics



Then, we must name the differents workspaces:

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

Tab 3: the differents workspaces

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

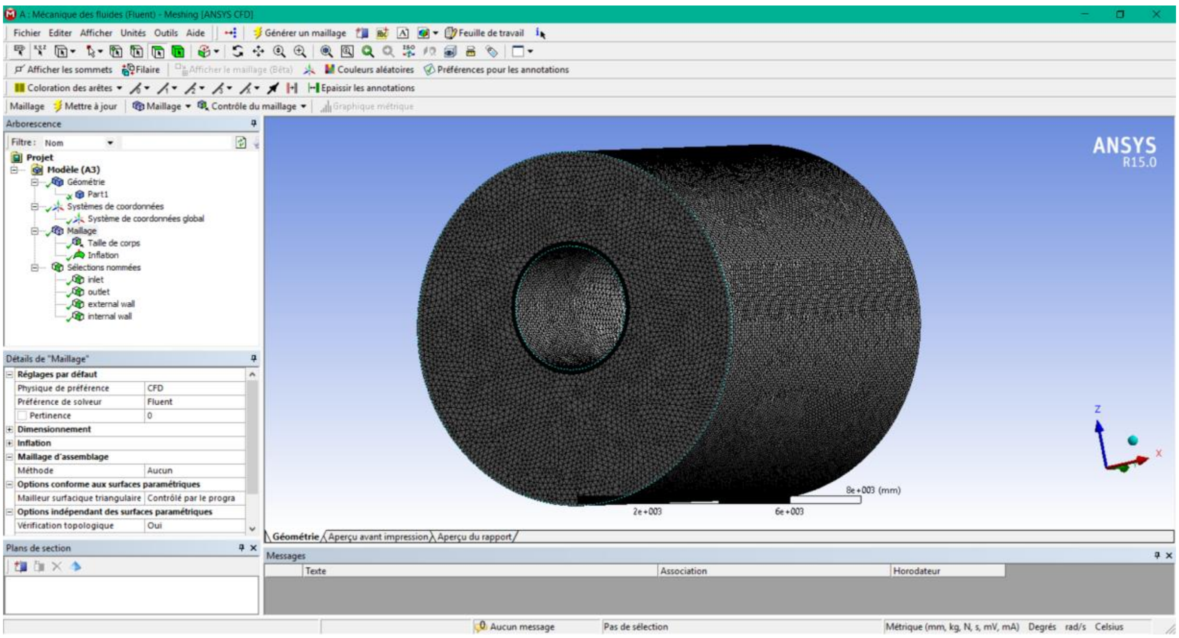


Fig 23: Final mesh

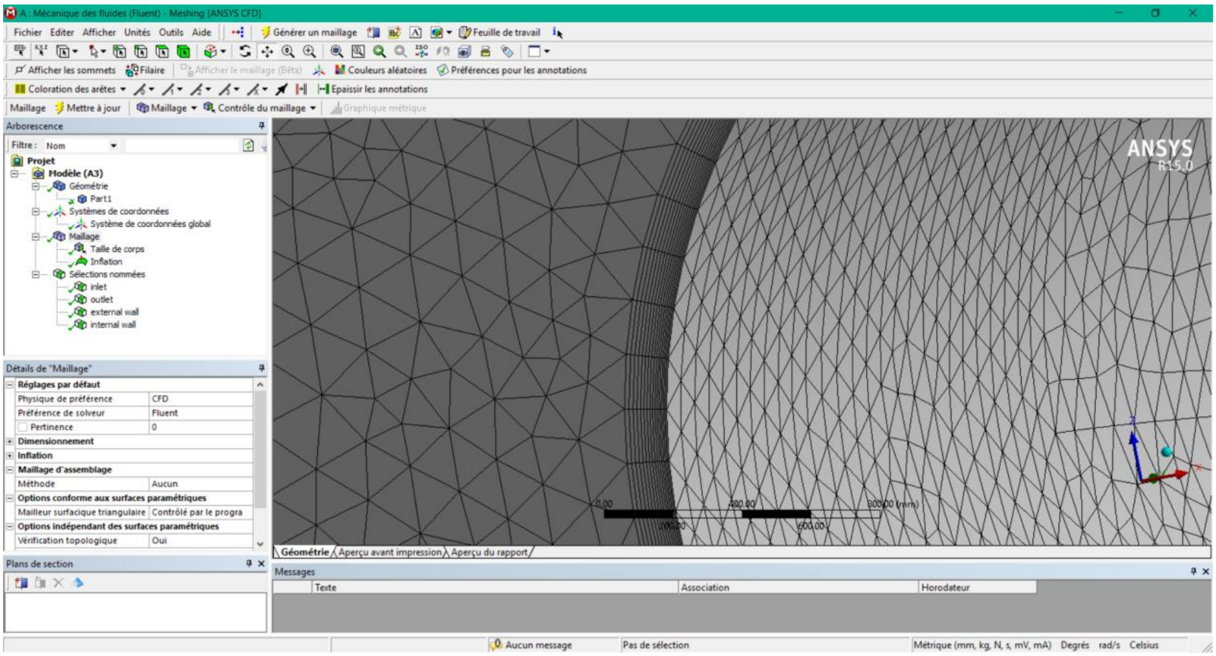


Fig 24: Inflation

ii. Configuration:

This configuration step is very important step which because it defines the workplace and simulates the true reality for this it is necessary to pay attention to the properties that define the environment where we go to work.

Temperature	T	K	217
Density	ρ	Kg.m−3	0.413
Dynamic viscosity	μ	$Kg.m^{-1}\cdot s^{-1}$	0.996×10^{-5}
Kinematic viscosity	v	$m^2\cdot s^{-1}$	2.413×10^{-5}
Pressure	P	Pa	2644.4

Tab 4: the Air characteristics used in Ansys

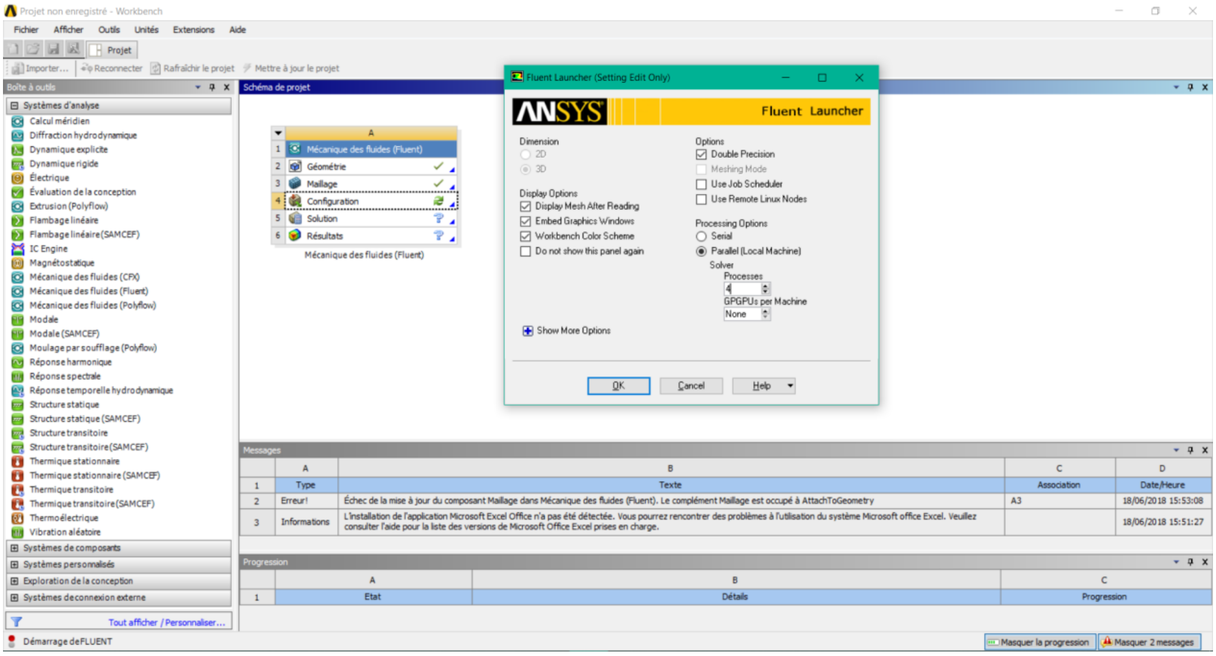


Fig 25: ANSYS configuration interface

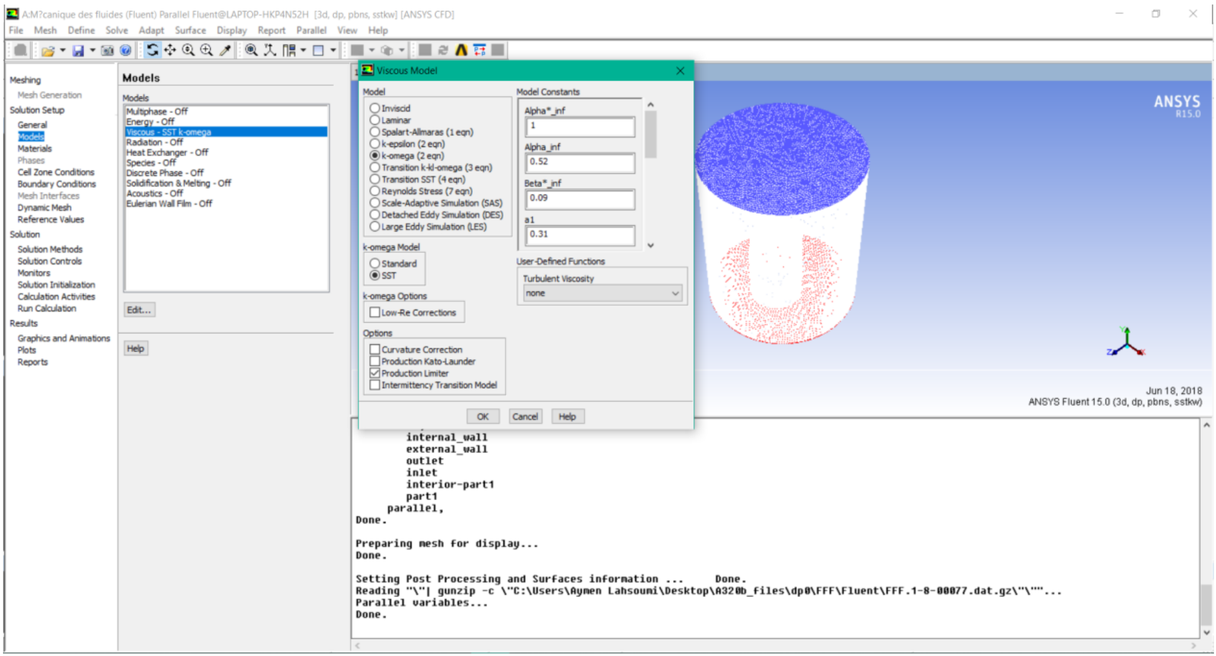
K omega SST:

K-omega (k-ω) model is one of the most commonly used models. This include 2 additional transport equations to represent turbulent properties of flow - to account for history effects like convection and diffusion of turbulent energy. The transport variable k determines the energy in turbulence and ω determines the scale of turbulence.

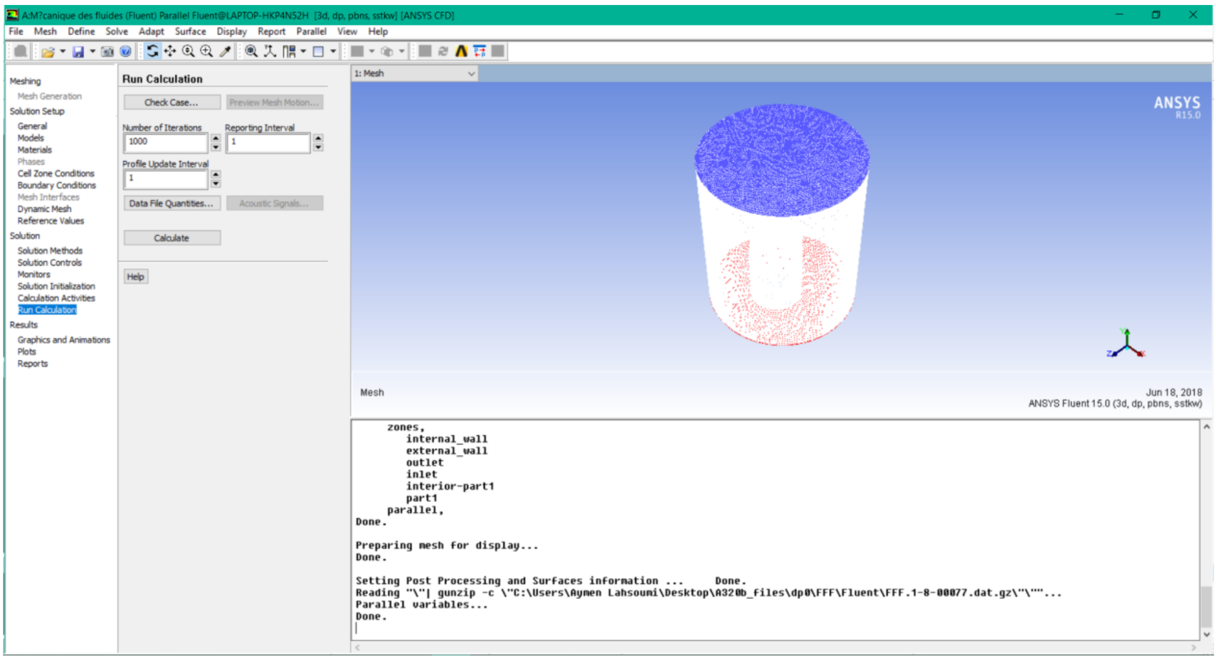
The turbulent energy ***k*** is given by:

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

where ***U*** is the mean flow velocity and ***I*** is the turbulence intensity.

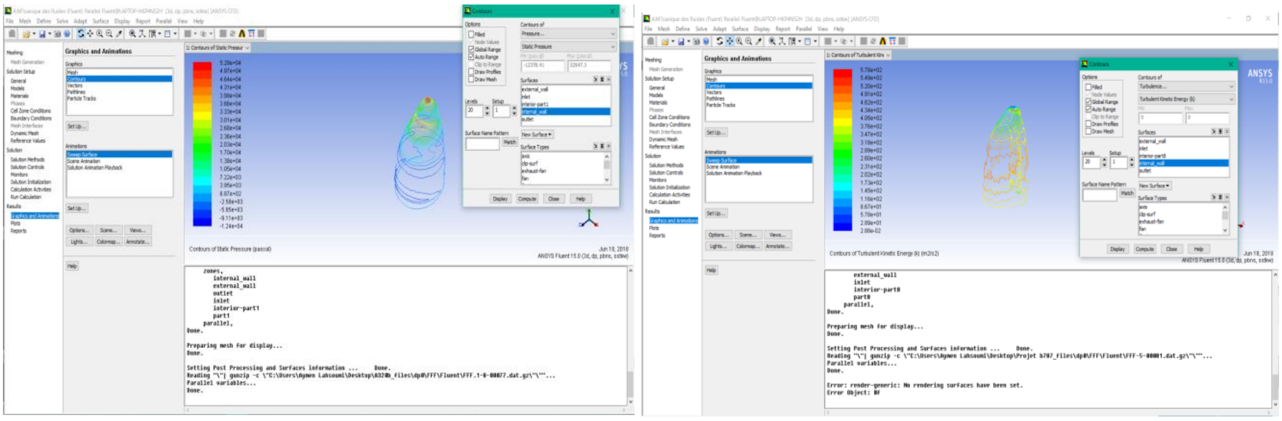


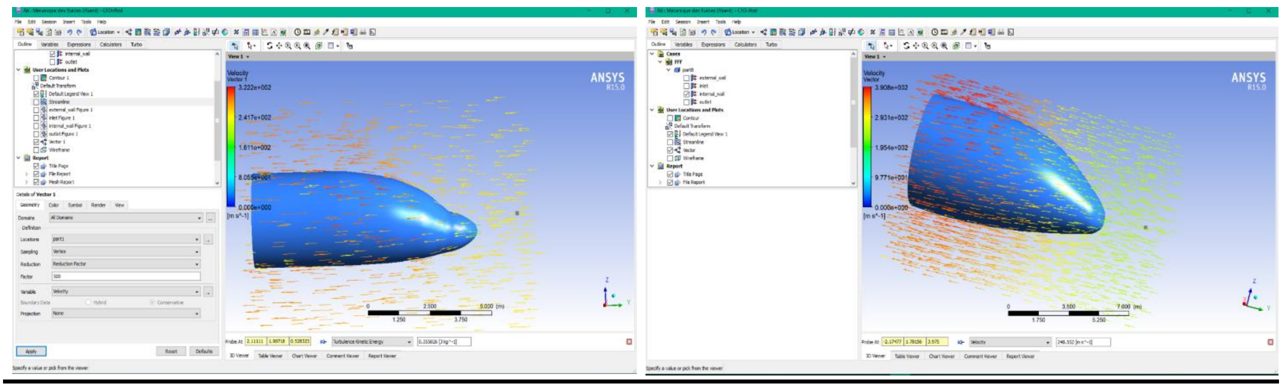
Finally, when we finish entering the physical property of the workspace, we start the calculation of the result.



iii. Result

when the calculation execution end and the solution converged. we display the results. there are many of the results that can be displayed, for example we can display the pressure in all points of the model, the velocity of air, the turbulence...





	A320	B787
Velocity inlet	250 m/s	250 m/s
Max velocity	305.37m/s	382.28m/s
Static pression	2644.2 Pa	2644.2 Pa
Turbulence kinetic energy	451.7 J/Kg	296.9 J/Kg
Density	0.413 Kg.m−3	0.413 Kg.m−3
Coefficient Drag	**0.0013**	**0.00061**

Tab 5: the results obtained between A320 and B787

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

One of the main things that increases the drag of the plane is the cockpit shoulder (A320) (a shoulder is a sudden change in diameter of a cylindrical part) which allows to create a wall of air flow so create a turbulence zone.

On the other hand, in the second model (B787), there is no such sudden change in the radome, that's why the resistance of the air is very small compared to the first model (A320).

To reduce the drag we did a complete study and a lot of experience to build another model with a few less air resistance.

3. Solutions to reduce the drag:

Reducing aircraft consumption is an absolute necessity for the aviation industry and a fight on all fronts. This includes reducing the various drag that slow down the aircraft.

- Reduce the turbulence:

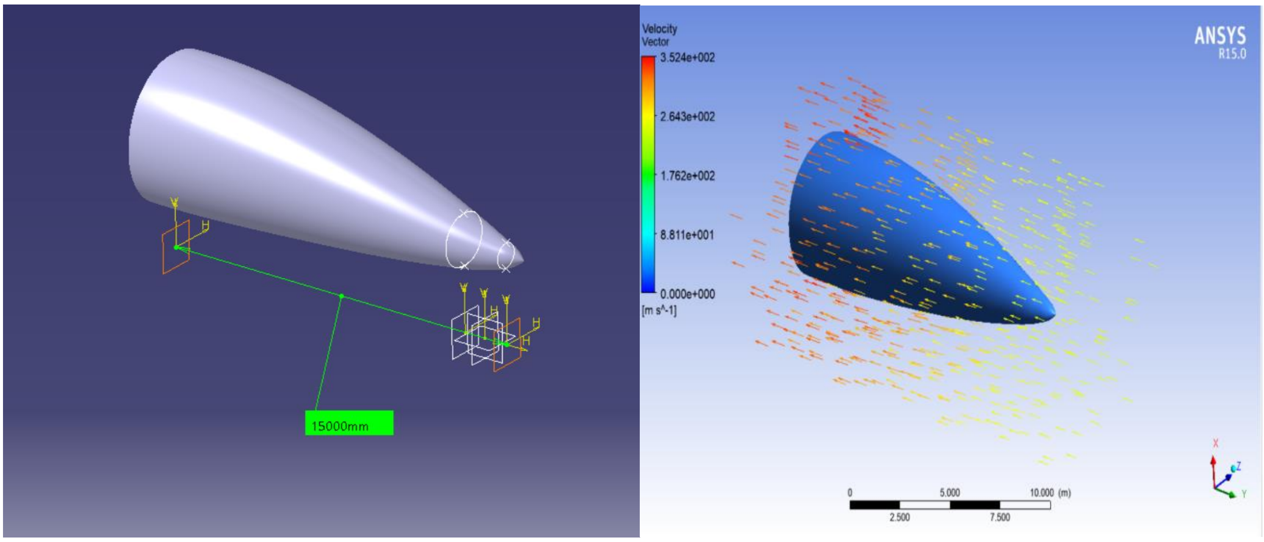
When a body of large size moves in a fluid as little viscous as air or water, the flow becomes unstable when there is a creation of turbulence and to reduce turbulence, it is necessary to reduce the number of Reynolds and make the flow more laminar.

- Improve the form:

According the experience, to decease the drag form, is to decrease the diameter of the plane and increase the length of the radome to decrease the turbulence and the Reynolds number.

following the results (B787, A320) and research, we have reached to create a new model with good aerodynamic performance.

- minimize friction and the air resistance.
- Improve the surface roughness.



	A320	B787	Aiman Model
Velocity inlet	250 m/s	250 m/s	250 m/s
Max velocity	305.37 m/s	332.28 m/s	341.78 m/s
Static pression	2644.2 Pa	2644.2 Pa	2644.2 Pa
Turbulence kinetic energy	451.7 J/Kg	296.9 J/Kg	287.12 J/kg
Density	0.413 Kg.m−3	0.413 Kg.m−3	0.413 Kg.m−3
Coefficient Drag	**0.0013**	**0.00061**	**0.00012**

Tab 6: the results obtained between the three models.

The new model reduces drag by 9% from A320 and 19% from B787. These are very satisfactory values that show that the new model is capable of reproducing the main phenomena of flow.

Drag improvement benefits:

- low fuel consumption.
- better aerodynamics.
- Double-flow reactors allow better performance.
- The noise reduce.
- More stability.
- Less vibration.

disadvantage

- decrease the number of passengers.
- high manufacturing cost.

Conclusion

These five months of project allowed me to achieve many things and eventually reach my goals.

Indeed, the many technical difficulties encountered I pushed to revise my objectives, to finally concentrate on the essential.

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

In addition, I succeeded to create multiple shapes, considering the mesh problems and make a aerodynamics study of space to compare between two model, to look for solutions to decrease the drag of new model

Among all the configurations made, the results in terms of drag are satisfactory since all the proposed new configurations are accompanied by a drop in the drag of the new model.

Far from being perfect, this project can be improved by finding a solution for the problem of energy Autonomy.

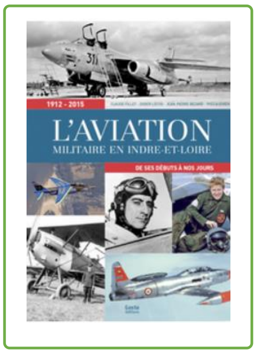
Bibliography

Title: Contribution to the study of drag reduction in turbulent flow

Author: Ferhat Hadri [1]

The publishing house: UNIVERSITE de CAEN/BASSE-NORMANDIE

Year: 2006



Title: L'aviation militaire en Indre-et-Loire

Author: Yves Audren [3]

The publishing house: Grasset

Year: 2015

Title: Aerodynamic study and control of body drag

Author: Yoann Eulalie [2]

The publishing house: UNIVERSITE de BORDEAUX

Year: 2014

Title: Mecanique des fluides

Author : Didier Desjardins, Natalie bonneton [4]

The publishing house: Dunod

Year: 2006

Netography

https://www.cfd-online.com/Wiki/Dimensionless_wall_distance_(y_plus)

https://fr.wikipedia.org

https://www.lavionnaire.fr

http://avionaprespetrole.e-monsite.com

http://optimisationduvoldunavionenpapier.yolasite.com

http://pierre.garde.free.fr