

**RÉPUBLIQUE ALGÉRIENNE DÉMOCRATIQUE ET POPULAIRE
MINISTÈRE DE L'ENSEIGNEMENT SUPÉRIEUR
ET DE LA RECHERCHE SCIENTIFIQUE
UNIVERSITÉ MOSTEFA BENBOULAIID BATNA 2**



FACULTE DE TECHNOLOGIE

Département de Génie Mécanique

Filière : Génie Mécanique

MEMOIRE DE FIN D'ETUDES

Présenté
pour Obtenir le Diplôme de **Master**

Spécialité :

CONSTRUCTION MÉCANIQUE

Par

ouffai iyad

Thème

***Etude de l'influence du maillage sur l'écoulement tridimensionnel
autour d'un avion touristique.***

Soutenu le 26/06/2023

Encadré par: Dr. L. MESSAOUDI

ANNEE UNIVERSITAIRE 2022/2023

ملخص

نقوم في هذا العمل بدراسة تأثير الشبكة على خصائص التدفق للطائرة في حالة الطيران. قمنا بإنشاء ثماني نماذج مختلفة للشبكة وفحص جودتها. اخترنا أربع طرق لإجراء المحاكاة ومقارنة نتائجها. قمنا بمقارنة نتائج حساب قوة الرفع والسحب ومعاملاتهما، وتوزيع سرعة الهواء وتوزيع الضغط حول الطائرة وتوزيع الضغط حول الجناح. ذلك لدراسة تأثير جودة الشبكة على نتائج المحاكاة، مع مراعاة الواقعية في وقت القياسي باستخدام برامج حرة.

summary

In this work, we study the effect of the network on the flow characteristics of the aircraft in flight. We created eight different network models and checked their quality. We chose four methods to perform the simulation and compare their results. We compared the results of calculating lift and drag forces and their coefficients, air velocity distribution, pressure distribution around the aircraft and pressure distribution around the wing. This is done to study the effect of network quality on the simulation results, taking into account realism in a time benchmark using free software.

résumé

Dans ce travail, nous étudions l'effet du réseau sur les caractéristiques d'écoulement de l'aéronef en vol. Nous avons créé huit modèles de réseau différents et vérifié leur qualité. Nous avons choisi quatre méthodes pour effectuer la simulation et comparer leurs résultats. Nous avons comparé les résultats du calcul des forces de portance et de traînée et leurs coefficients, la distribution de la vitesse de l'air, la distribution de la pression autour de l'avion et la distribution de la pression autour de l'aile. Ceci est fait pour étudier l'effet de la qualité du réseau sur les résultats de la simulation, en tenant compte du réalisme dans un temps de référence à l'aide de logiciels libres.

THANKS

I would like to express my gratitude for the remarkable memory stimulant, Mr. Eid Masoudi. I thank him for the guidance, guidance, assistance and advice he gave me. I would like to extend my sincere thanks to all the professors, speakers and all the people who have guided my thinking and agreed to meet and respond to my questions during my research. I thank my dear parents, who have always been by my side. Finally, I thank my friends and neighbors who have always been by my side. Their unconditional support and encouragement was very helpful. To all these contributors, I extend my thanks, respect and deep gratitude.

Thanks

Dedications

Table of content

General Introduction

Table of Contents

Chapter I: Aerodynamics and CFD concepts	2
I.1- Introduction	3
I.2.2-Drag force	4
I.3- Notions of meshing.....	5
I.3.1- Mesh types	5
I.3.2- Mesh generation techniques.....	7
I.3.3- Mesh quality.....	8
I.3.4- Mesh adaptation	8
I.4- Simulation tools	9
I.5- Conclusion	9
Chapter II: Meshing of the EV-55 aircraft	10
II.1- Introduction	11
II.2- Different meshing methods	12
I.2.1- Computational domain.....	12
II.2.3- Method 2.....	15
II.2.4- Method 3.....	19
II.2.5- Method 4.....	22
II.2.6- Method 5.....	25
II.2.7- Method 6.....	27
II.2.8- Method 7.....	31
II.2.9- Method 8.....	33
II.2.10- Comparison of different methods	37
II.3- Conclusion.....	40
Chapitre III: Simulation numerique	41
III.1- Introduction.....	42
III.2- Boundary conditions	42
III.3- Simulation residuals.....	44
III.4- Speed distribution	46
III.5- Pressure distribution.....	48
III.6- Streamlines.....	53
III.7- Aerodynamic forces	55
III.7.1- Lift.....	55
III.7.2- Drag.....	56
III.6 Conclusion.....	58
References	59
Annexes	60

General Introduction

Over the past two decades, computational fluid dynamics (CFD), which involves numerical simulation of fluid flows, has experienced significant growth across various fields, particularly in the aerospace industry. Nowadays, it is extensively used in the aeronautical industry from the pre-project phase to production to assess the aerodynamic performance of aircraft. Simultaneously, the computational power of computers has increased, enabling CFD simulations to solve the averaged Navier-Stokes equations (RANS) within an acceptable timeframe for industrial purposes. However, the geometric complexity of the simulated configurations has also increased, incorporating the relative motion of bodies, such as propellers on airplanes or rotors on helicopter fuselages, and incorporating intricate geometric details.

Our goal in this work is to study the impact of the mesh on the flow characteristics is to use the free "OpenSource" software available to solve an aerodynamic problem involving air flow analysis around a tourist plane known as the Outback EV-55. During the flight of this study, we chose to use different software tools. FreeCAD has been selected for 3D design, cfMesh for network generation, OpenFoam as a store that uses the limited size method, and ParaView as a visualization tool to analyze the obtained results.

The document is organized in the following manner:

Chapter 1 provides an introduction to the basic concepts of aerodynamics and CFD simulation, Chapter 2 addresses the challenges posed by highly complex aircraft geometry, mesh adaptation and methods for optimizing them, Chapter 3 covers the use of a solver to digitally solve a problem and presents simulation results Finally, the document concludes with a comprehensive summary and future prospects .

Chapter I: **Aerodynamics and CFD concepts**

I.1- Introduction

In this chapter, we will present a brief overview of fundamental concepts in aerodynamics that will serve as the foundation for our simulation work. These concepts will be essential for understanding and applying them in subsequent sections of our study.

I.2- Aerodynamics

Aerodynamics is a specialized branch of fluid dynamics that primarily deals with the study and analysis of airflow and its impact on solid objects within the surrounding environment. These effects can arise from either a stationary body subjected to a flowing air with a certain velocity or a moving body within the atmosphere. Additionally, aerodynamic phenomena can also occur due to the combined effects of both scenarios mentioned above [14].

I.2.1- Lift force

Lift is the force that acts perpendicular to the direction of the oncoming airflow and allows the aircraft to stay airborne. This force is generated due to the interaction of the wing of the aircraft with the air flow, as the air accelerates over the wing, which leads to a decrease in pressure and a low pressure area above the wing. At the same time, air flows at a lower speed under the wing and has a higher pressure, and this difference in pressure creates lift that helps the plane stay airborne [1].

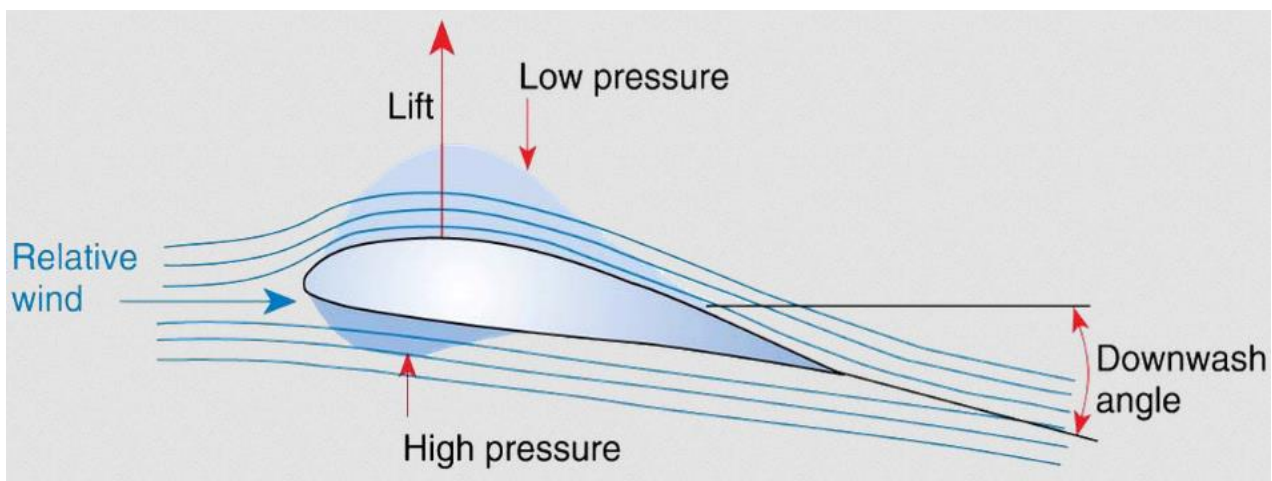


Figure I.1- Lift force [8].

Lift is defined by the formul

$$F_L = \frac{1}{2} \rho S V^2 C_L \quad (I.1)$$

ρ :is the density of air (kg/m³)

S: is the wing surface of the wing. It is the total surface of one of its faces.

V: is the velocity of the flow of solid air.

C_z : is the lift coefficient

$$C_L = 2 F_L / \rho V^2 S \quad (I.2)$$

I.2.2-Drag force

Drag force on an aircraft consists of parasite drag and induced drag. Parasite drag is caused by different parts of the aircraft, such as the fuselage, tail, and landing gear. Manufacturers aim to minimize parasite drag during the design process. The drag contribution from each component needs to be estimated to determine the overall drag and its line of action, which can vary with changes in the angle of incidence.

Induced drag is associated with the generation of lift. As an aircraft wing produces lift, there is a downward flow of air behind the wing, creating an upward force called induced lift. However, this lift is accompanied by a backward force known as induced drag.

The magnitude of drag depends on factors like the object's shape, size, speed, air density, and viscosity. Minimizing drag is crucial for optimizing the performance and efficiency of various objects, including aircraft, automobiles, and streamlined structures [1].

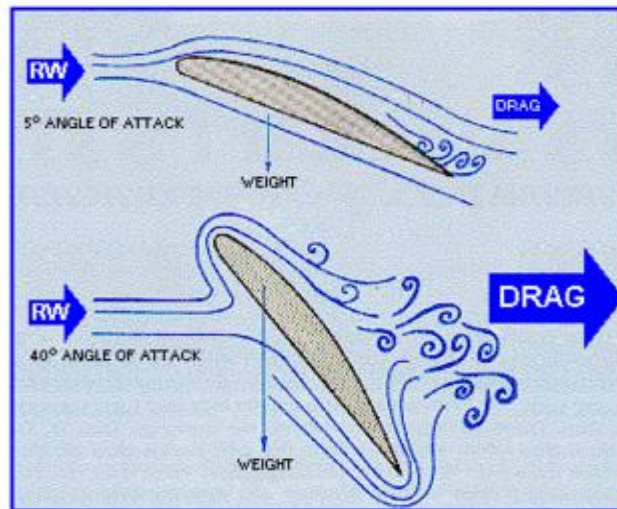


Figure I.2- Drag Force [16].

Drag force is defined by

$$F_D = \frac{1}{2} \rho S V^2 C_D \quad (I.3)$$

The drag coefficient is :

$$C_D = 2 F_D / \rho V^2 S \quad (I.4)$$

I.3- Notions of meshing

Meshing is a crucial step in the numerical modeling and simulation of fluid flows using the Computational Fluid Dynamics (CFD) method. It consists in discretizing the study domain into a structure composed of finite elements or finite volumes. Here are some important notions about meshing:

I.3.1- Mesh types

There are different types of mesh used in CFD, such as structured mesh and unstructured mesh. Structured mesh is made up of regular cells and is used for simple, regular geometries. Unstructured meshing is more flexible and can be used for complex, irregular geometries [3].

a- Structured mesh

A structured mesh is a type of mesh used in CFD analysis that has a regular and organized structure. It is important to generate a high-quality mesh for accurate and reliable results in CFD simulations. The quality of the mesh affects convergence, solution accuracy, and computational efficiency. A good mesh minimizes distortions and ensures proper resolution in regions with high gradients. It should also have a smooth transition between elements. The mesh represents the fluid and solid surfaces in the computational domain. Overall, structured meshes provide accuracy and stability, particularly for problems with regular or symmetric geometries, but may be more challenging for complex geometries or regions requiring local refinement [10].

b- Unstructured mesh

Unstructured meshing is a flexible technique used in computational simulations like CFD and FEA. It accurately represents complex geometries by connecting nodes and cells in a flexible manner. While unstructured meshes enable realistic simulations, they require more memory and computational resources due to their higher mesh count. The choice between structured and unstructured meshing depends on the trade-offs between accuracy and computational efficiency [10].

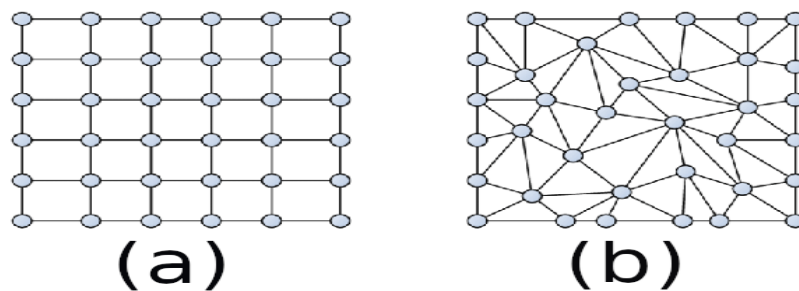


Figure I.3-Structured (a) and unstructured (b) mesh [8].

c-Adaptive meshing

Adaptive meshing is a method in which the mesh resolution is adjusted according to the flow characteristics. Regions where solution gradients are high or where significant phenomena occur can be meshed more finely, while coarser meshes can be used in regions where variations are low. This optimizes computational efficiency while guaranteeing sufficient accuracy [2].

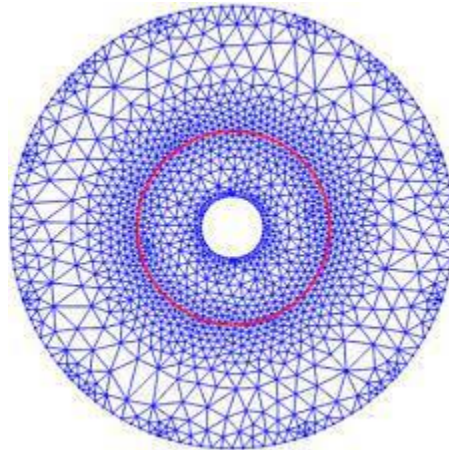


Figure I.4-Adaptive meshing[15].

d-Hexahedral mesh

Hexahedral meshes are used to represent three-dimensional geometries using hexahedral cells. These meshes offer better element quality and numerical accuracy than tetrahedral meshes, but can be more difficult to generate and manipulate, particularly for complex shapes [2].

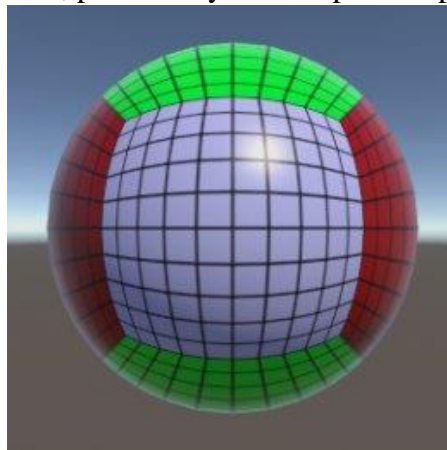


Figure I.5-Hexahedral mesh[17].

Delaunay Triangulation

Delaunay triangulation is a popular technique for generating triangular meshes. It ensures that no points lie within the circumcircle of any triangle in the mesh, resulting in a well-shaped mesh. Delaunay triangulation can be extended to higher dimensions [13]

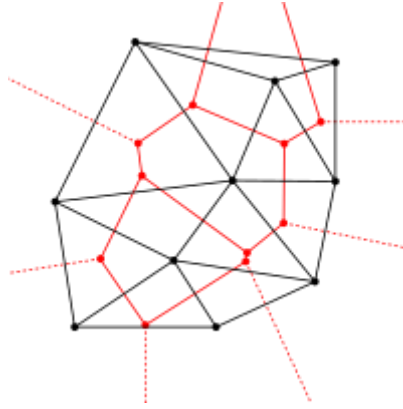


Figure I.6-Delaunay Triangulation[18].

f-Hybrid mesh

A hybrid mesh is a mesh that is generated by combining elements of different types, such as triangles or quadrilaterals in 2D, and tetrahedra, prisms, or pyramids in 3D. It brings together the advantages of both structured and unstructured meshes [13].

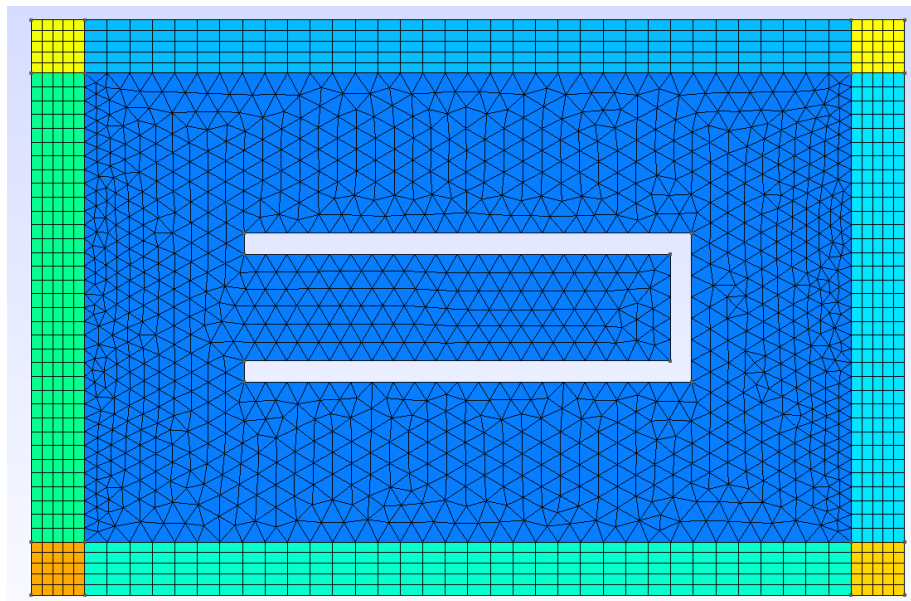


Figure I.7-Hybrid mesh [7].

I.3.2- Mesh generation techniques

There are several techniques for generating a mesh. Among the most common are the Delaunay method, the finite element method and the finite volume method. These techniques can be used to generate high-quality meshes tailored to specific simulation requirements [8].

I.3.3- Mesh quality

Mesh quality plays a crucial role in the accuracy and stability of CFD simulations. Mesh quality criteria include cell regularity, element size, cell shape and proximity of cells to surfaces and walls. High-quality meshing provides more accurate and reliable results [3].

Quality	excellent	Good	Acceptable	Poor	Very poor	Bad
F d	0 – 0.25	0.25 – 0.50	0.50 – 0.80	0.80 – 0.95	0.95 – 0.99	0.99 – 1.00

Table I.1-Quality associated with the distortion factor [13].

I.3.4- Mesh adaptation

mesh density in areas of interest. This allows computational resources to be concentrated more where they are needed, saving time and resources in less critical areas [3].

I.4- Simulation tools

The simulation tools mentioned, FreeCAD, OpenFOAM, cfMesh and cfdOF, are widely used in the field of numerical simulation and CFD (Computational Fluid Dynamics). Here is a brief description of each of these tools:

- FreeCAD is a parametric 3D computer-aided design (CAD) program for modeling and creating complex geometries. It is free and open source, offering great flexibility for the creation of 3D models. FreeCAD is often used as a user-friendly graphical interface for preparing geometric models prior to simulation [5,19].
- OpenFOAM is an open source software package for numerical CFD simulation. It offers a wide range of functions for solving fluid conservation equations and turbulence models. OpenFOAM is known for its flexibility and ability to handle complex problems, and is widely used in the CFD community for simulations in research and industry [4].
- cfMesh is an open source software package dedicated to mesh generation for CFD. It is often used in conjunction with OpenFOAM to prepare the meshes required for simulations. cfMesh offers advanced features for the creation of high-quality structured and unstructured meshes, as well as options for mesh adaptation [6;19].
- cfdOF is a workshop or specific extension of the FreeCAD environment, which enables the use of OpenFOAM to be integrated directly into FreeCAD. This facilitates the process of preparing geometric models, generating meshes and configuring simulations using OpenFOAM. cfdOF offers a user-friendly graphical interface to simplify the CFD workflow with FreeCAD and OpenFOAM [9,19].

I.5- Conclusion

In this chapter, we have given some general definitions of Aerodynamics, some Meshing concepts, simulation tools and also the basis of CFD simulation. The next chapter will focus on the aircraft Meshing and the methods used with OpenFoam and cfMesh.

Chapter II: **Meshing of the EV-55 aircraft**

II.1- Introduction

In this chapter we will highlight the methods and tools used to divide the computational field and design the EV-55 aircraft, taking into account its complex form to generate a high-quality mesh.

As for the design of a complex aircraft, the design has already been completed [11].

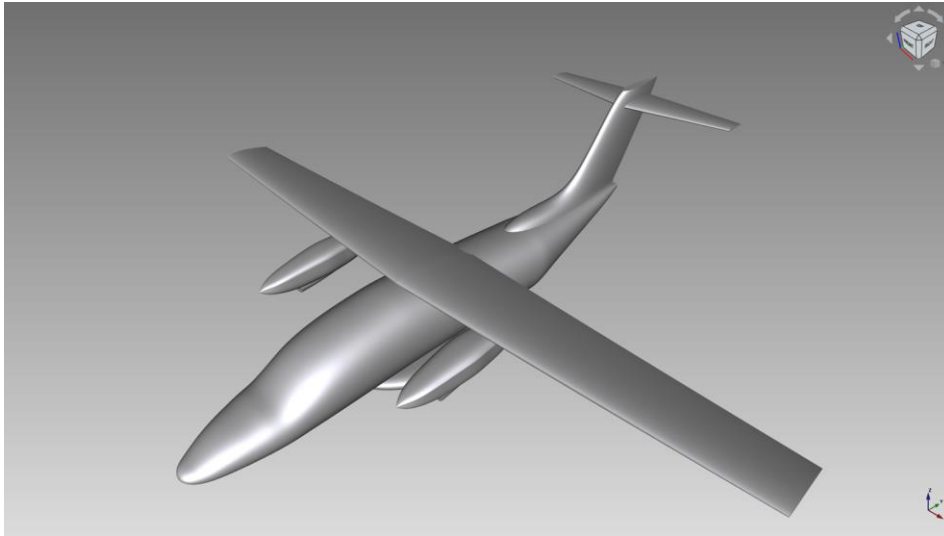


Figure II.1- The aircraft in isometric 3D view.

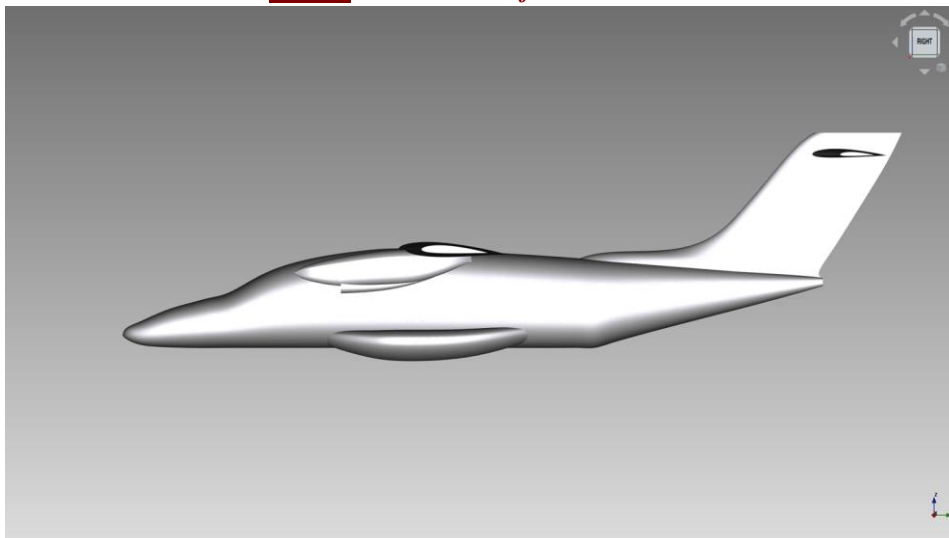


Figure II.2- Side view of an airplane.

II.2- Different meshing methods

I.2.1- Computational domain

The computational domain is defined as the physical area in which the computational simulation is performed. Simulation limits and extent of area are determined. The domain is segmented into a mesh of numerical accounts. Choosing a suitable area of calculation is crucial for obtaining accurate simulation results. In our study, a fixed area of calculation was used for all cases, and its dimensions were explained in the figure below. We used the base body length as a reference for analysis: $L = 14.34 \text{ m}$

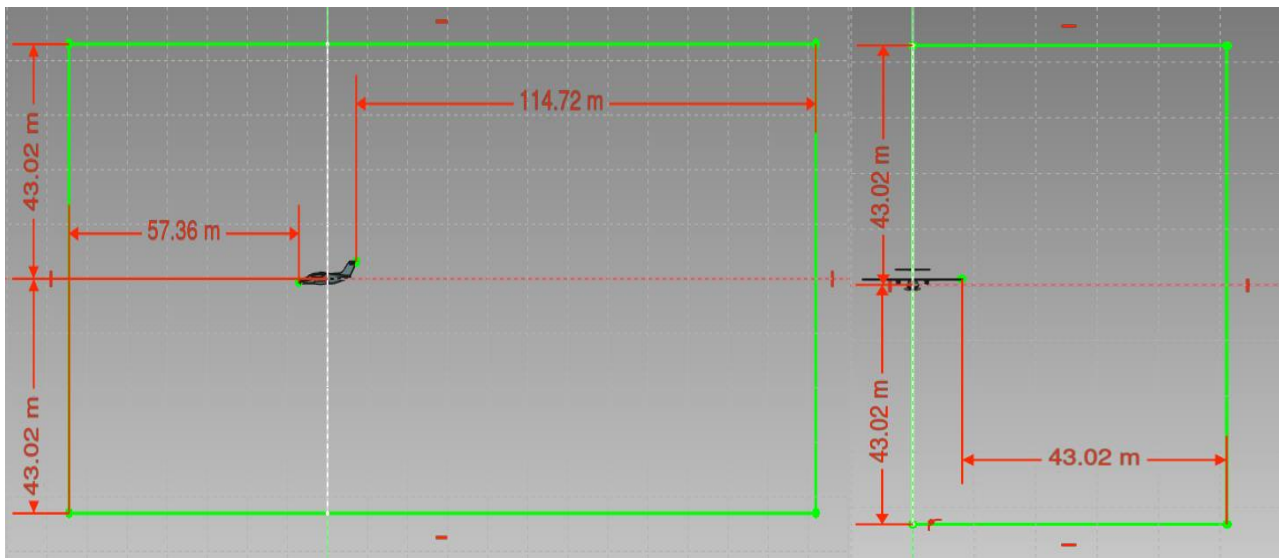
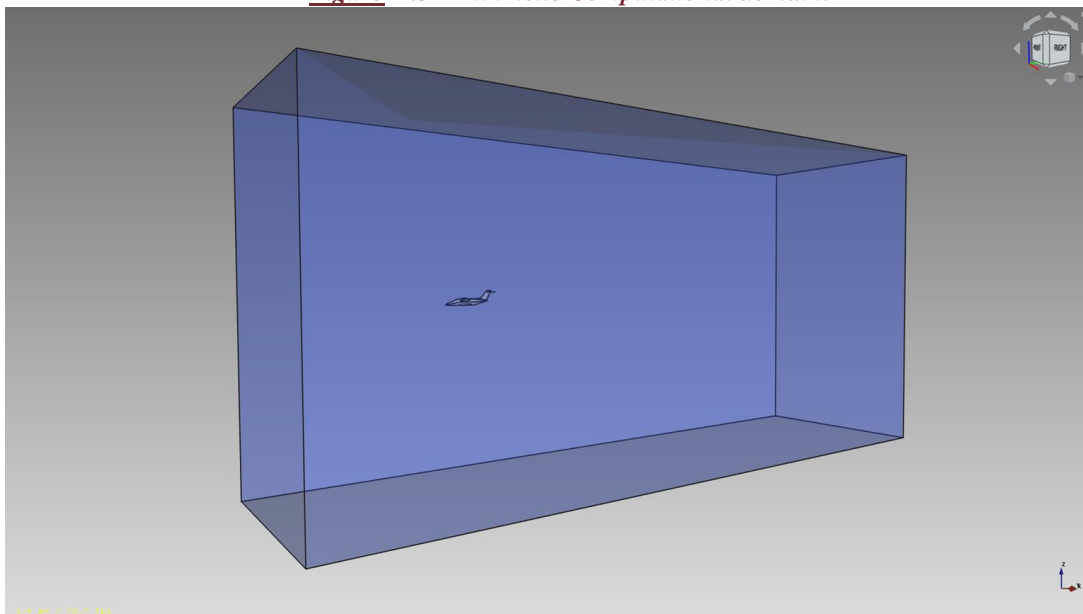


Figure II.3- Arithmetic Computational domain.

II.4-

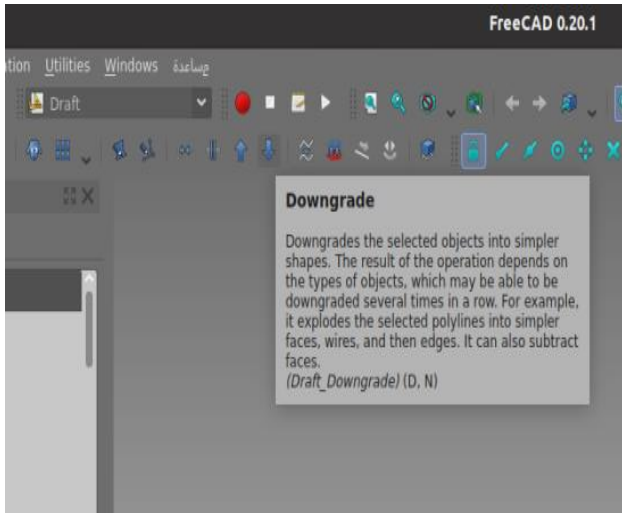


Figure

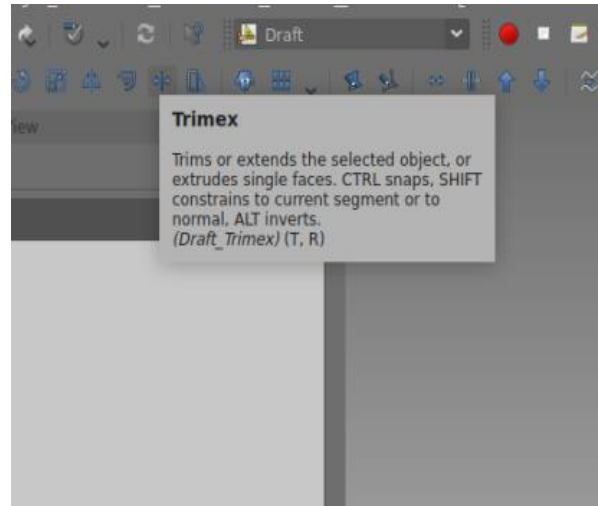
Computational domain

II.2.2- Methode 1

It is an advantage in the workbench draft we in this way divide flight surfaces by pressing the downgrade button and fixing a surface from the surfaces and pressing the trimex feature you extrude the surface and we choose the direction of the extrusion to suit the geometry [12]



FigureII.5- Picture of downgrade feature



FigureII.6- Picture of Trimex feature

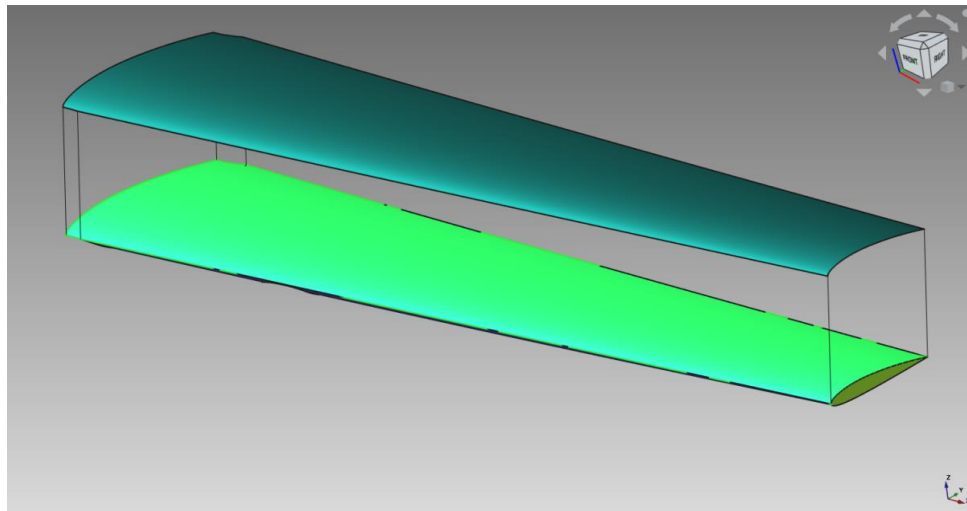


Figure II.7- downgrade plane surface

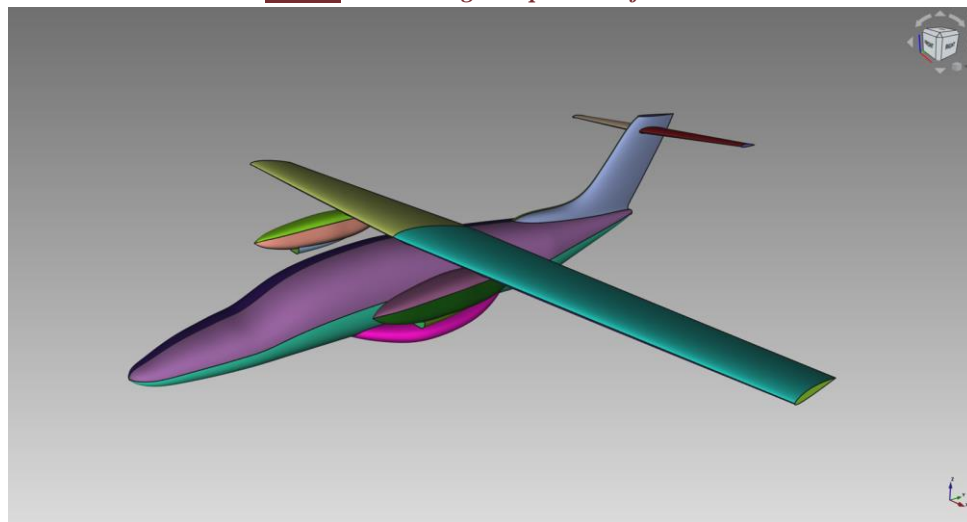
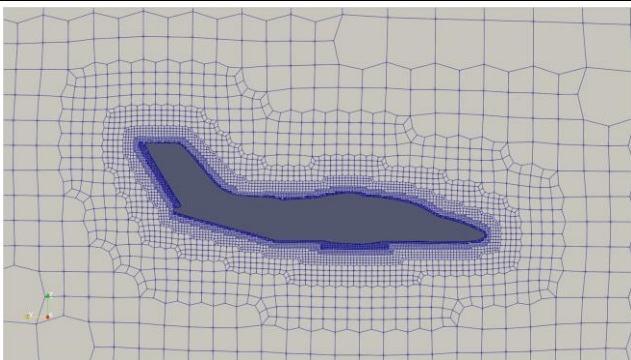
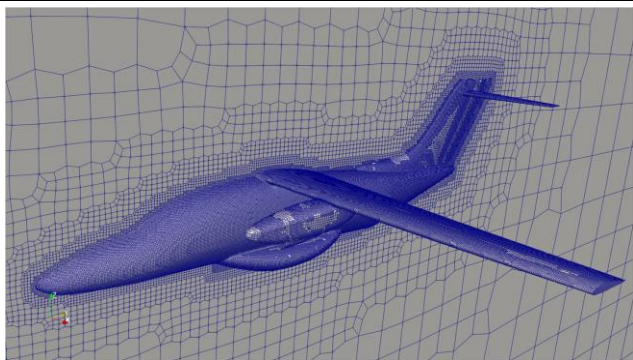
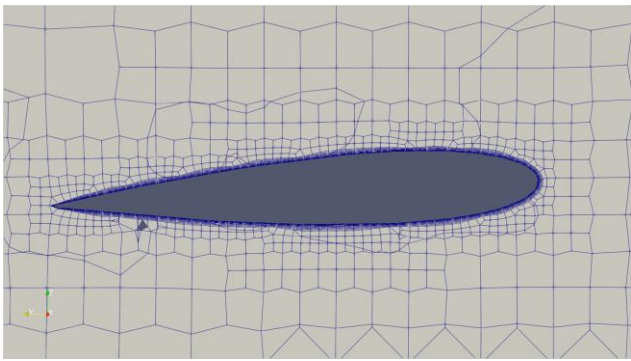
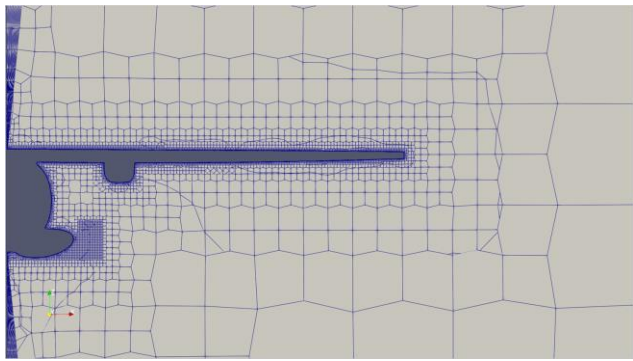


Figure II.8- Image showing extrusion surface of aircraft

We do this step on the entire surface of an airplane that requires a lot of effort, time and concentration. When we finish, we combine all the surfaces and get an airplane-shaped template. Extrusion thickness=50mm.

Mesher	Cartesian
Base Element Size [m]	3 m
Boundary layer number (surface)	20
Mesh time [min]	1min 47s
Disk size [MB]	435.096MB
Total number of nodes	2766302

Table II.1- Mesh Parameters method 1.

	
X=0.001m	3D
	
x=3m	Y=-1.5m
Mesh method 1	

Note

We stop at this point because of the very large number of items without Refine the good around the plane so we tried the second method.

II.2.3- Method 2

In this method, we create volume around the fuselage, the tail and the engine, the front and rear parts of the aircraft, the landing gear, and the wing

- . In the Part Design workbench of our CAD software, we select a level or plane. We create a 2D sketch outlining the desired shape around the part. Then, we extrude the sketch by specifying the direction and distance for the extrusion. The software generates the 3D geometry by extending the sketch in the specified direction, adding depth to the design
- . After selecting a different level in the Part Design workbench, we draw a rectangle around a plane part. Then, using the "cavity" command, we remove the material within the rectangle, creating a void or cavity in the desired location of the solid model.

Method shown in the following images

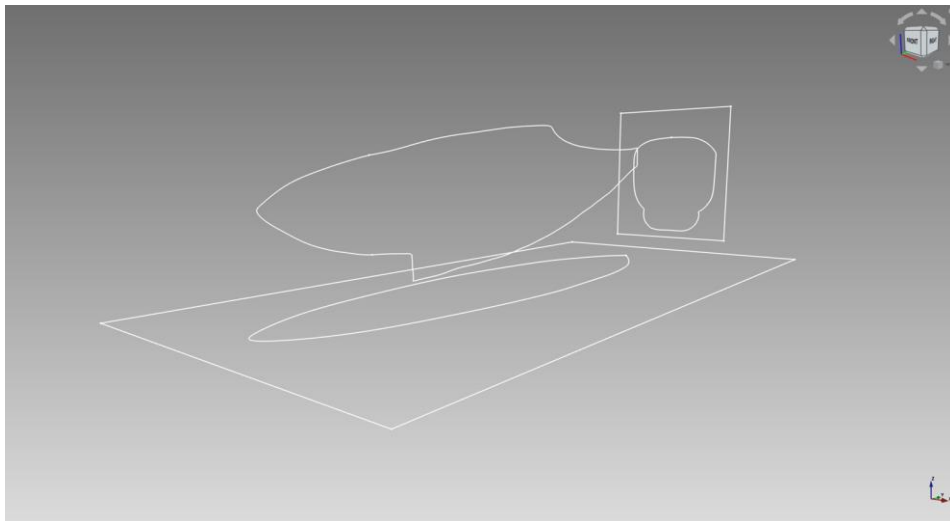


Figure II.9- Three-plane drawing.

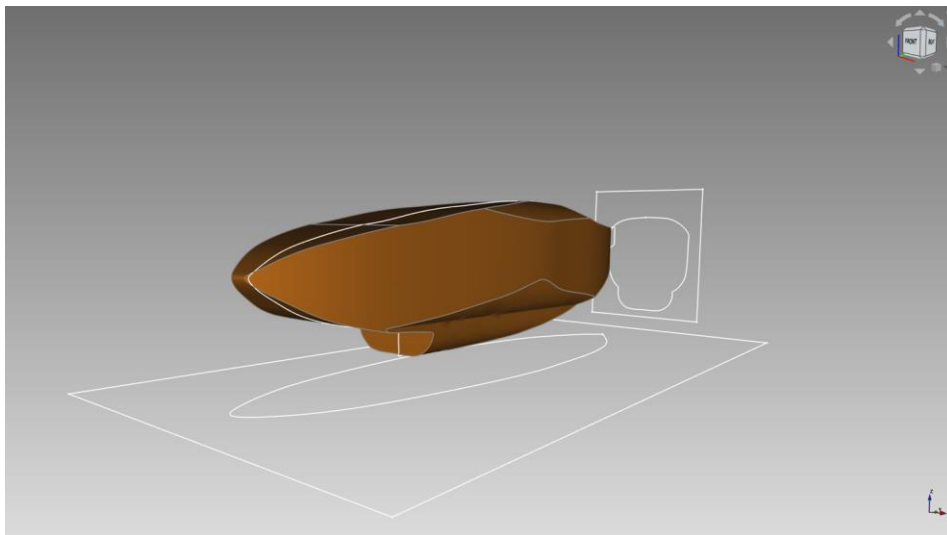


Figure II.10- Image after extrusion and cavity.

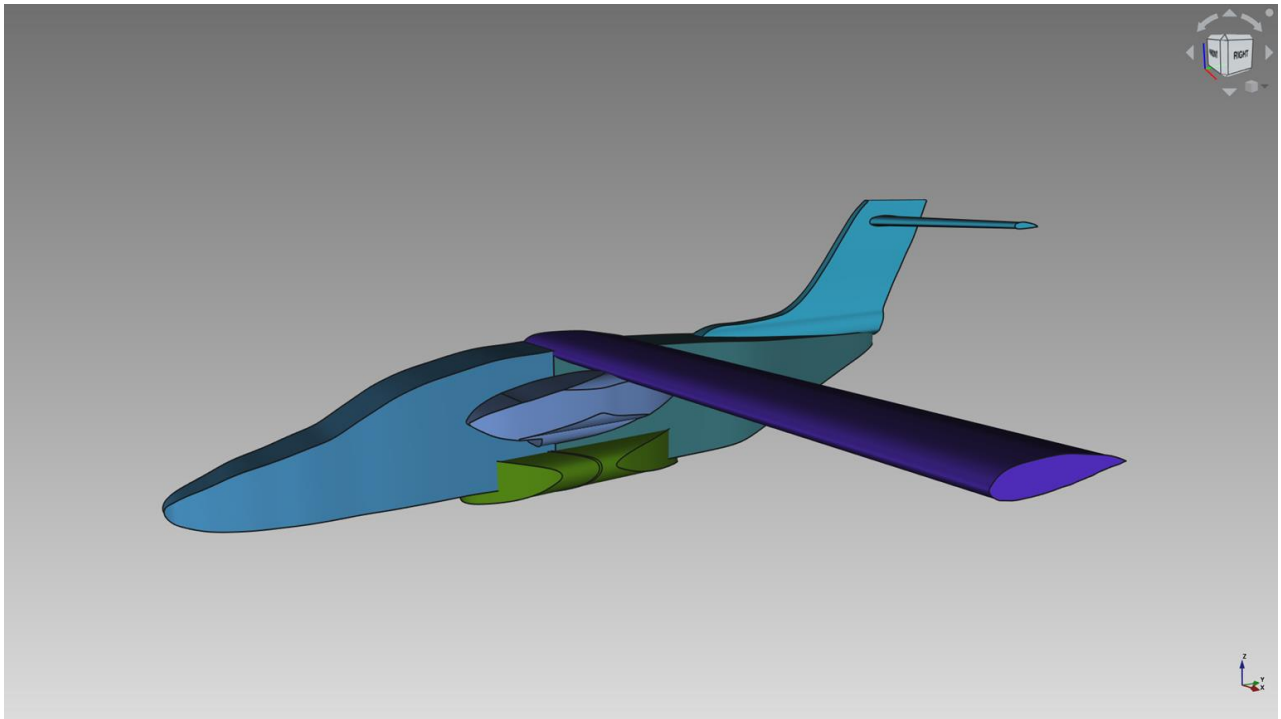


Figure II.11- Image of a method 2.

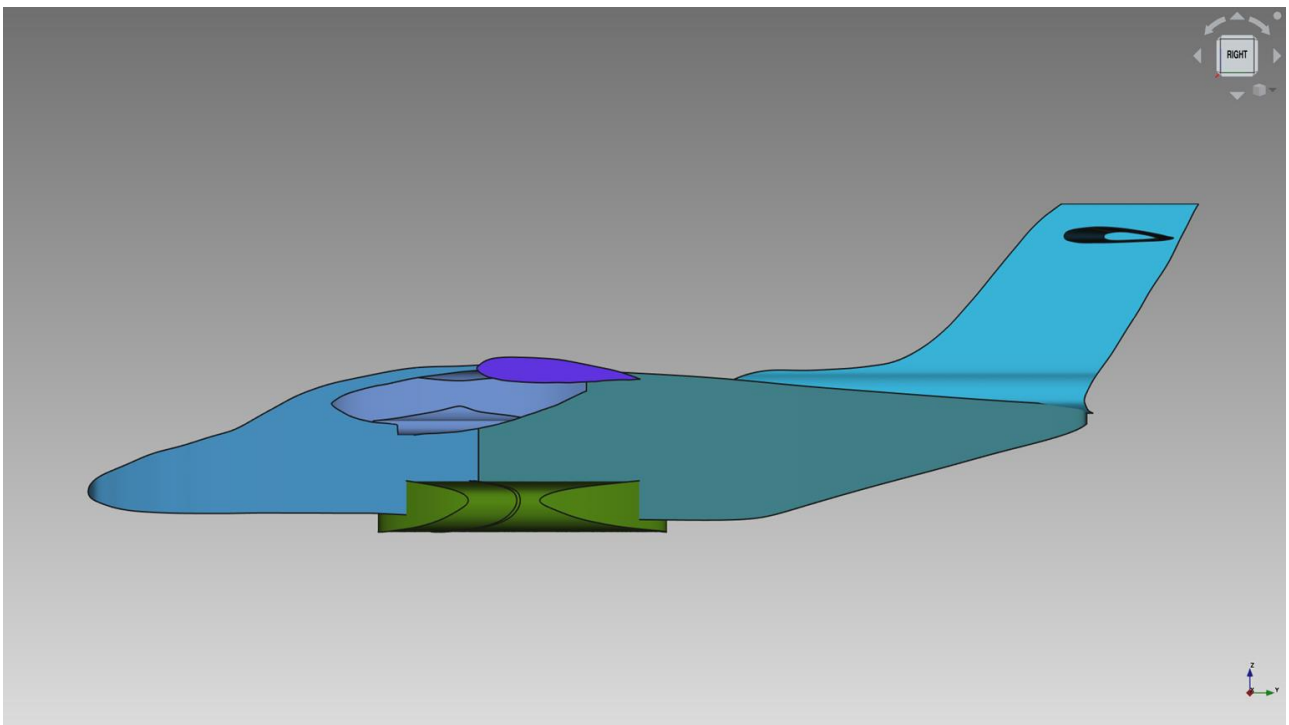
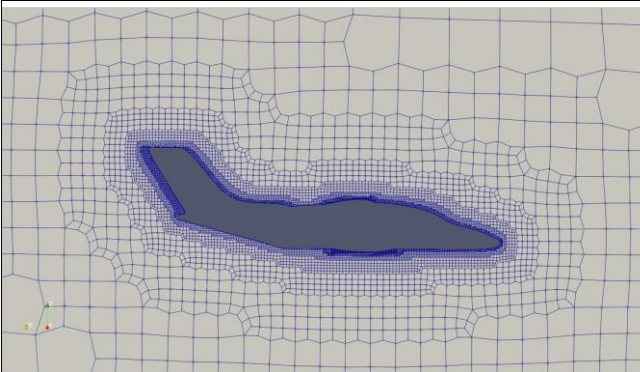
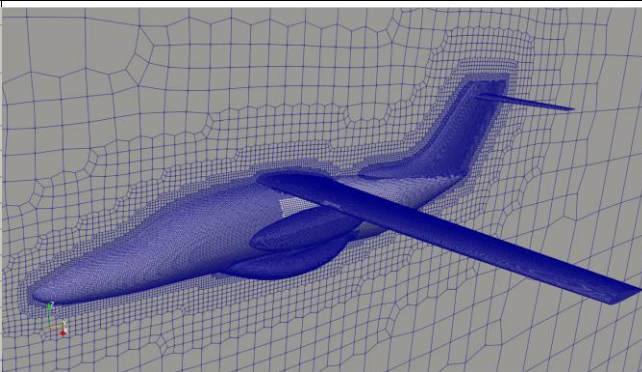
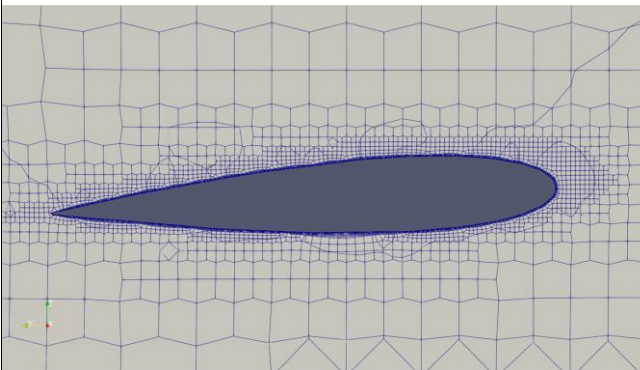
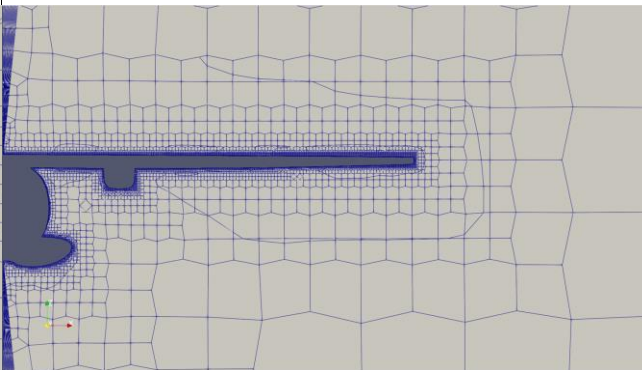


Figure II.12-Side view of the method 2.

mesher		Cartesian
Base Element Size [m]		2m
scale	Wing + engine + box landing	0.015
	fuselage face + tail	0.020
	fuselage behind	0.040
Boundary layer number (surface)		20
Mesh time [min]		09min 49s
Disk size [MB]		592.904 MB
Total number of nodes		6,747,625

Table II.2-Mesh Parameters method 2.

	
X=0.001m	3 D
	
X=3m	Y=-1.5m
Mesh method 2	

Note : We stop at this point because of the very large number of items.

II.2.4- Method 3

By employing a meticulous approach, we direct our attention towards a concentrated gradient encompassing an aircraft, dedicating our entire focus to thoroughly understand and optimize its performance.

Since I am a complex and unbalanced geometry aircraft, we are strapping the fuselage and opening the workbench draft, By selecting the clone function, we generate a duplicate that grants us the flexibility to proportionally scale along the X, Y, and Z axes as needed, ensuring precise adjustments.

we generate four additional copies of the aircraft. With careful consideration, we apply the necessary scaling to each copy, ensuring appropriate grading. This process enables us to enlarge the aircraft in a visually appealing manner, resulting in the desired illustrated format as depicted in the image(figure).

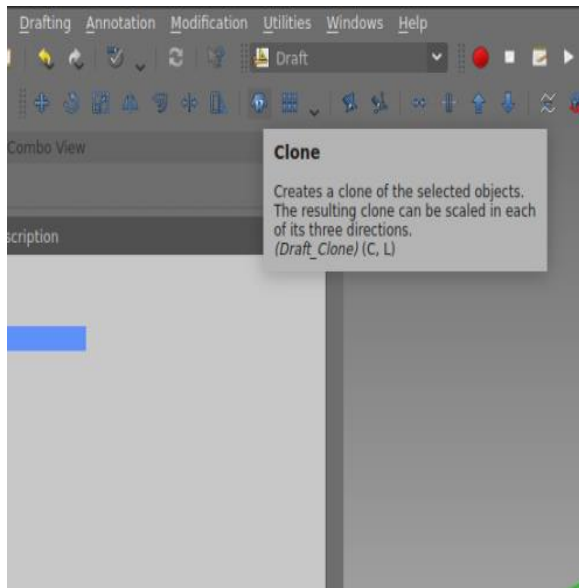


Figure II.13- Feature image clone.

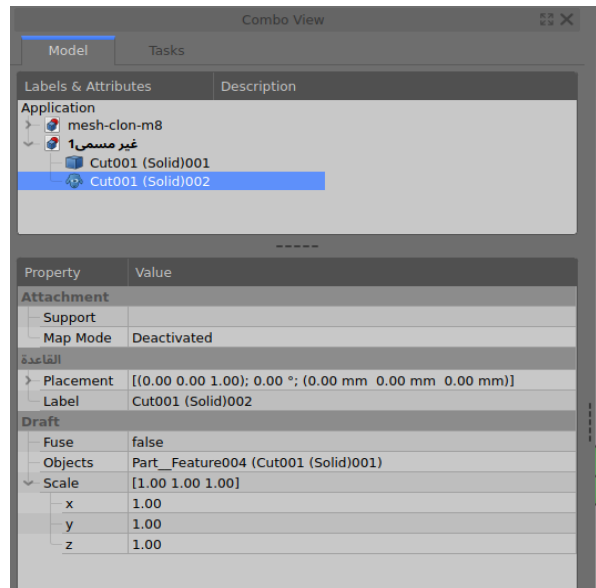


Figure II.14 Feature image scale.

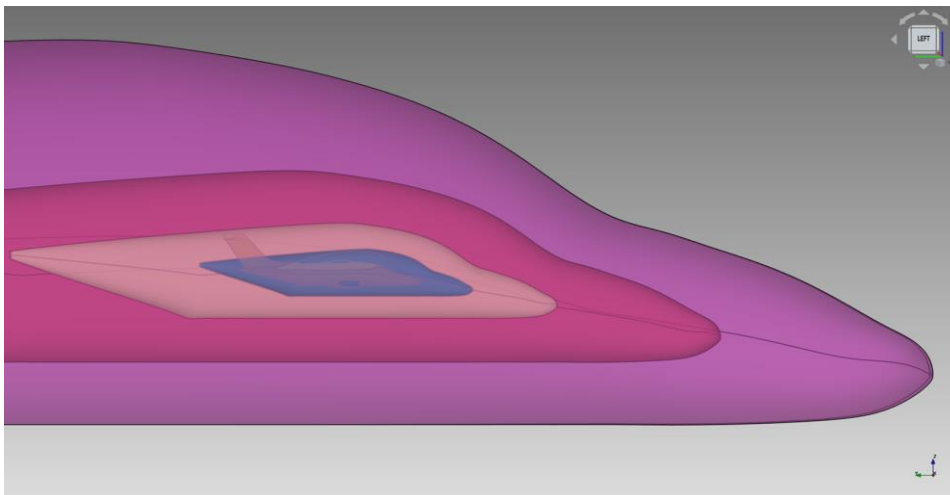


Figure II.15- Image of a method clone.

We call the proximity of the plane scale0 and scale3 and scale

mesher		Cartesian
Base Element Size [m]		2m
Surface refinement airplane		0.020
scale	Scale0 / Scale wing	0.050
	Scale1	0.100
	Scale2	0.200
	Scale3	0.300
Boundary layer number (surface)		20
Mesh time [min]		03min 20s
Disk size [MB]		655.659 MB
Total number of nodes		4,673,620

Table II.3- Mesh Parameters Method 3.

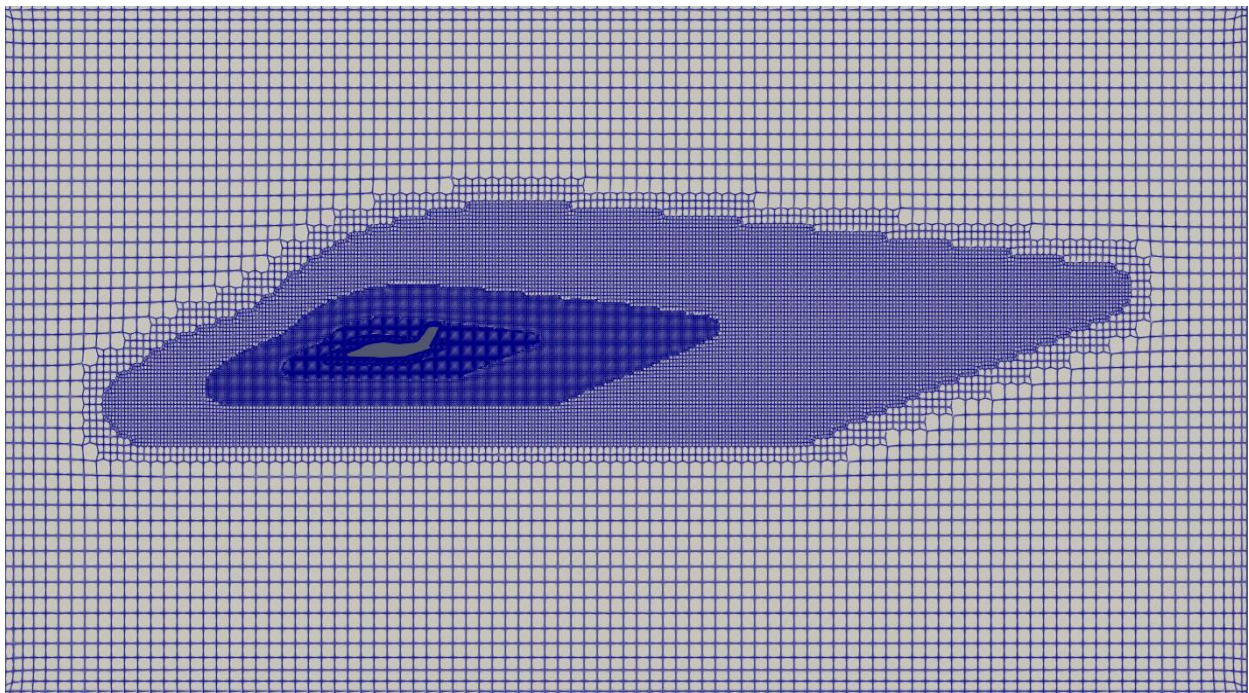
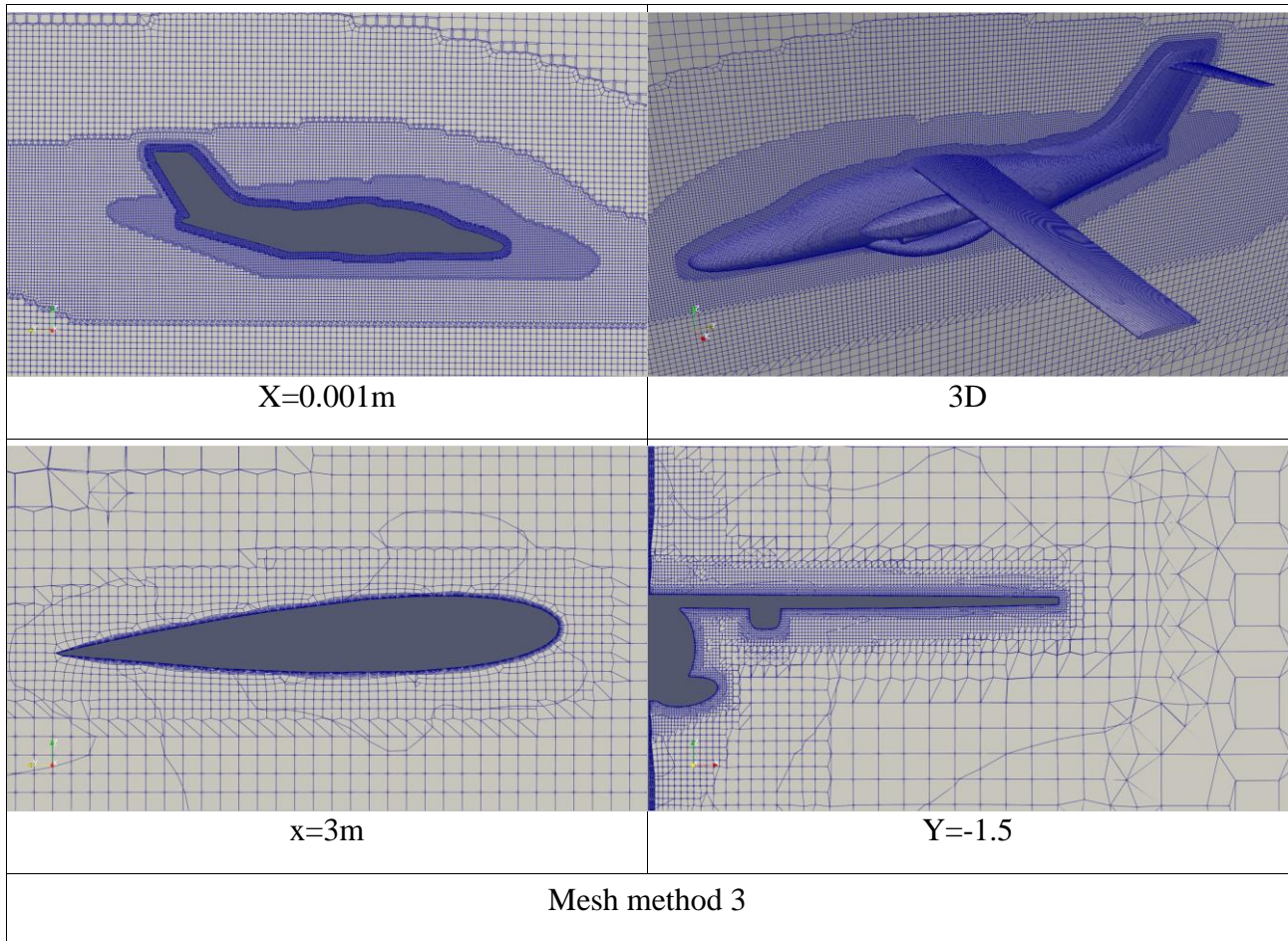


Figure II.16- Mesh profile picture.



Note

With a well-graded and well-reasoned mesh we stop at this point because of the very large number of elements in undesirable areas.

II.2.5- Method 4

In this way we turn an airplane into a solid object and then we create plates with the appropriate length and number and put them in the right place to cut and press slice apart and we do the cutting showing us the cut-off penalty we export the required parts of the pieces in the format of the stip file.

Cut off parts (the front part of an aircraft, the engine, the front part of the landing gear and the air inlet of the engine) and (the front and rear parts of the wing and tail).

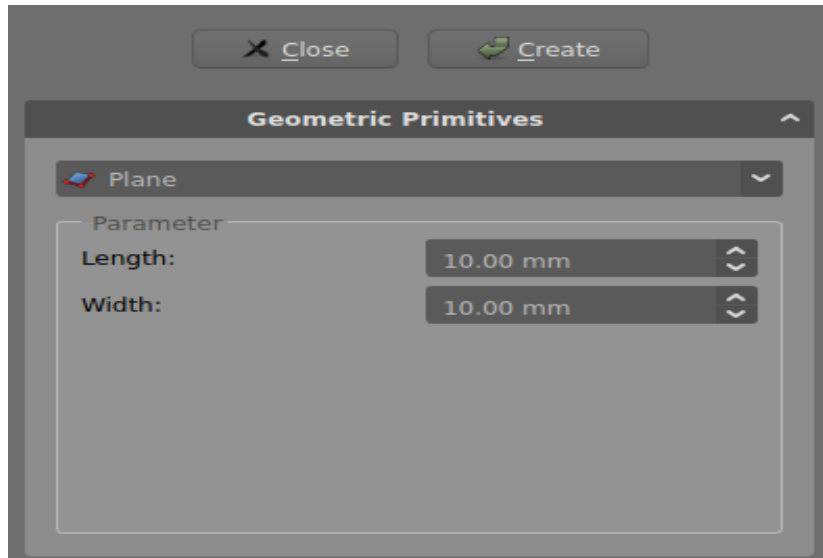


Figure II.17- Image showing how to create a plate.

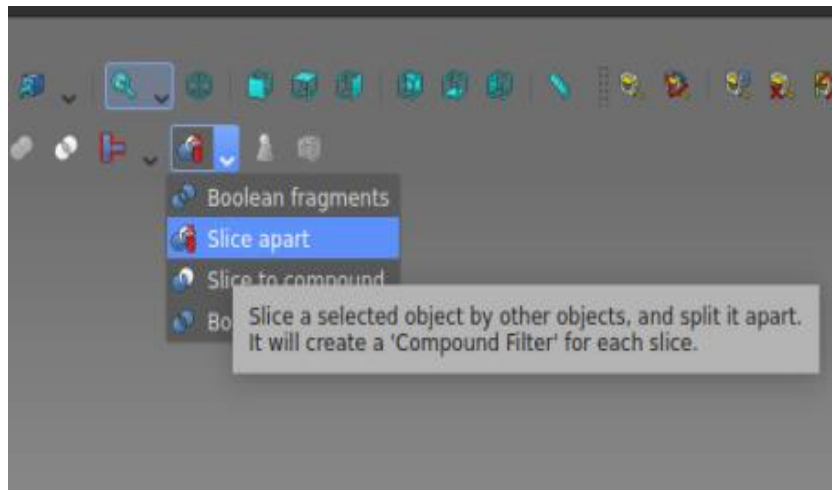


Figure II.18- Illustration of cutting function.

Put a plane in a cube and polish it low to organize the mesh because the sanding in large parts distorts the mesh around the plane.

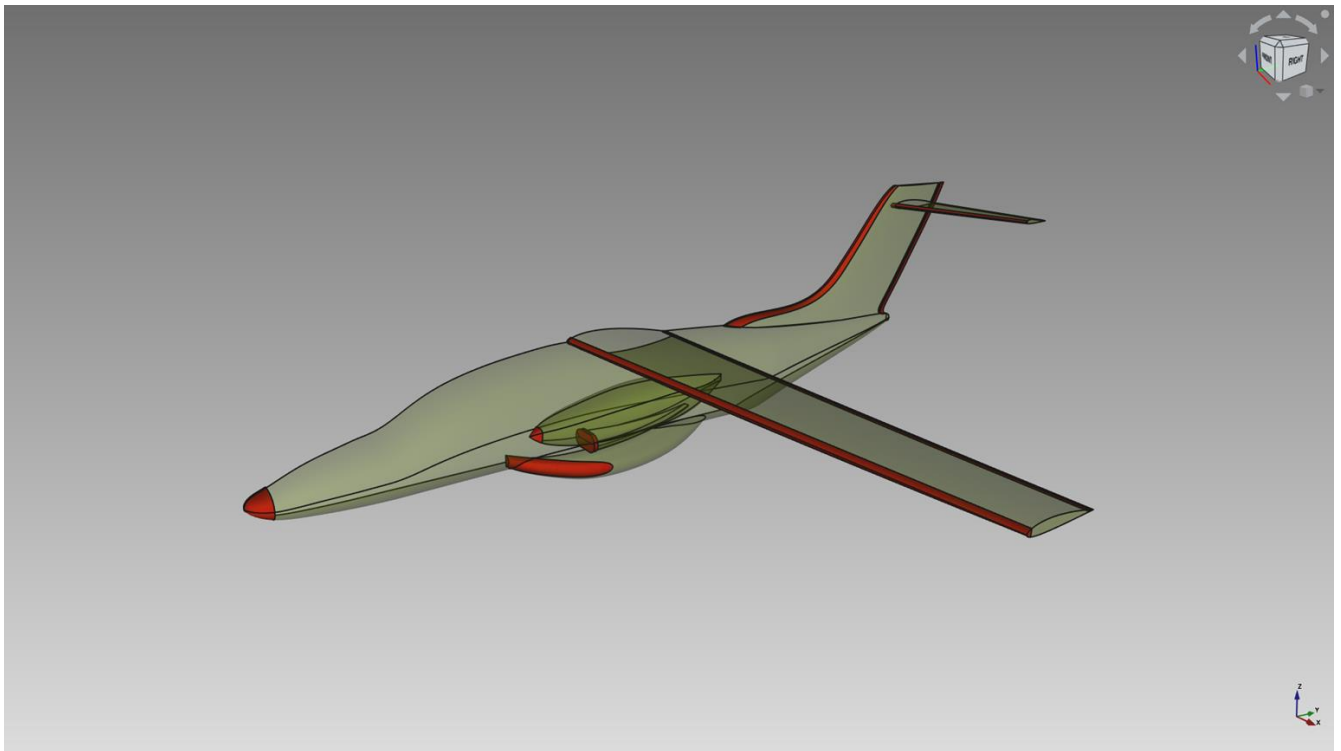
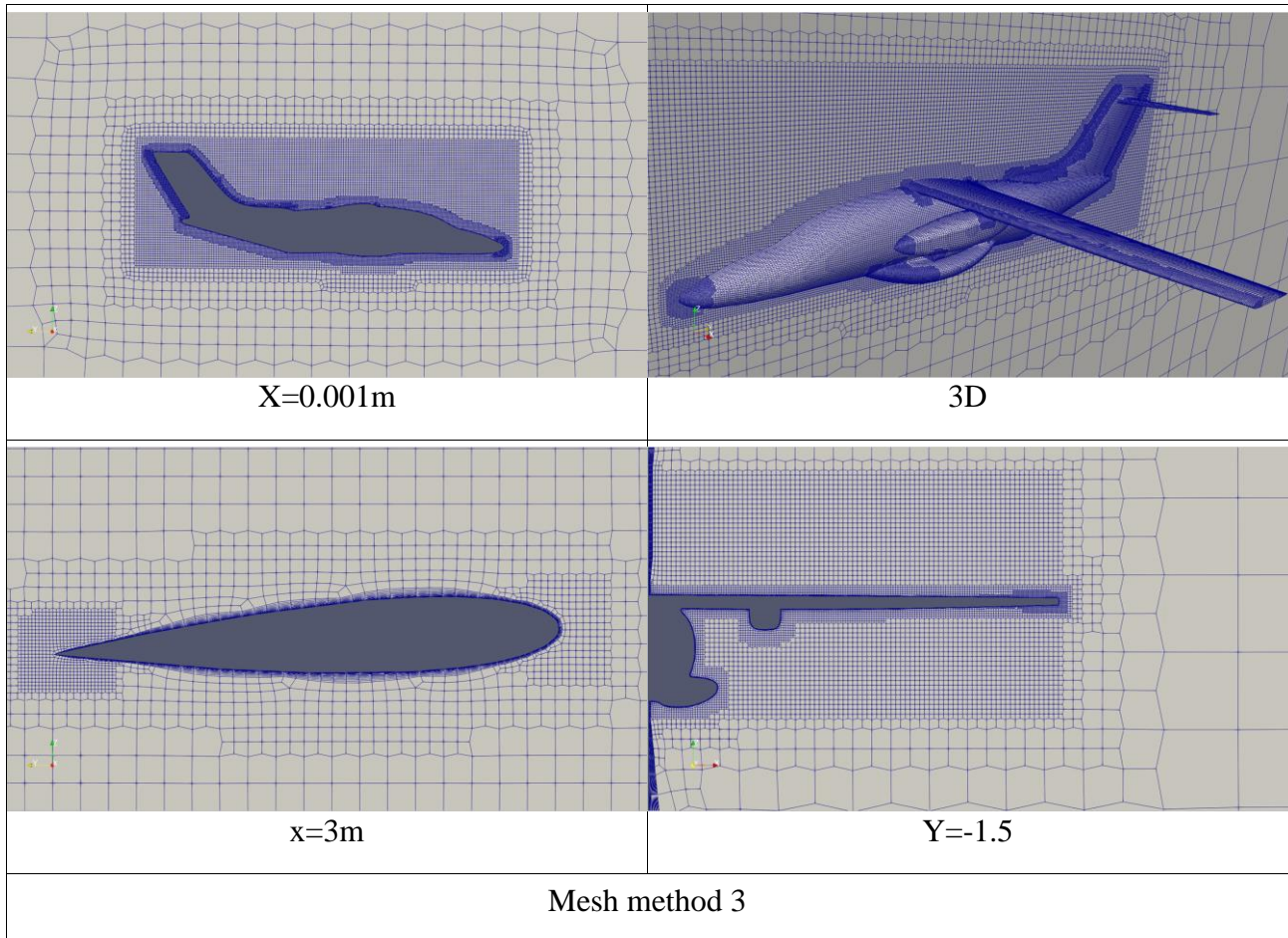


Figure II.19- Image showing the parts cut in a red plane.

mesher		Cartesian
Base Element Size [m]		3m
Surface refinement		0.020
Refinement thickness		100mm
volume	box	0.100
	The back of the wing	0.005
	other parts	0.010
Boundary layer number (surface)		20
Mesh time [min]		6min 47s
Disk size [MB]		469.99 MB
Total number of nodes		5222639

Table II.4- Mesh Parameters method 4.



Note

The results obtained thus far are commendable, displaying a high level of quality with their polished appearance, sharp edges, and regular grid structure. However, we have decided to pause our progress at this juncture due to the significant volume of elements involved. In order to overcome this challenge.

II.2.6- Method 5

method simple mesh: We call it a simple mesh method In this way we import half an aircraft and a simulated field (airplane subtraction tunnel) in format step in freecad we click on the field of simulation and dab to workbenches cfd of And then you click on CfdAnalysis An then you click on CFD mesh we select half a plane.

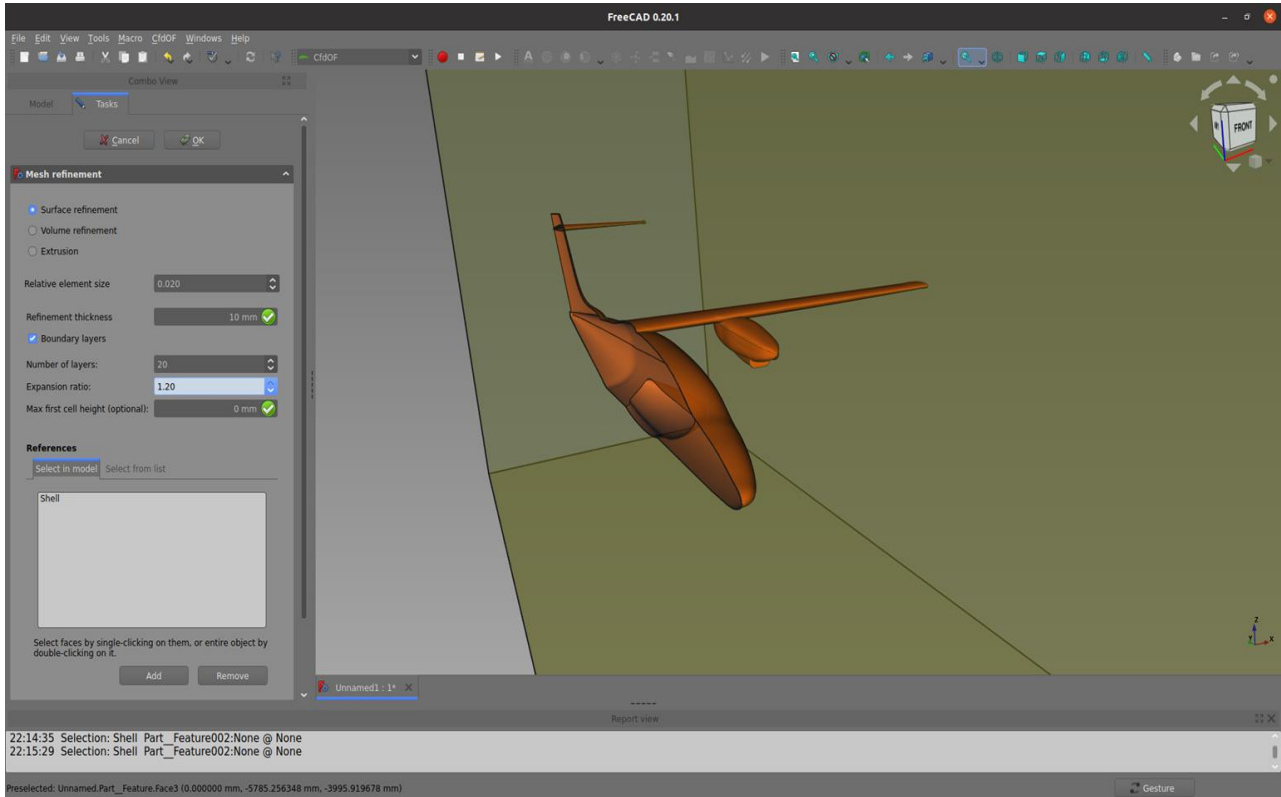
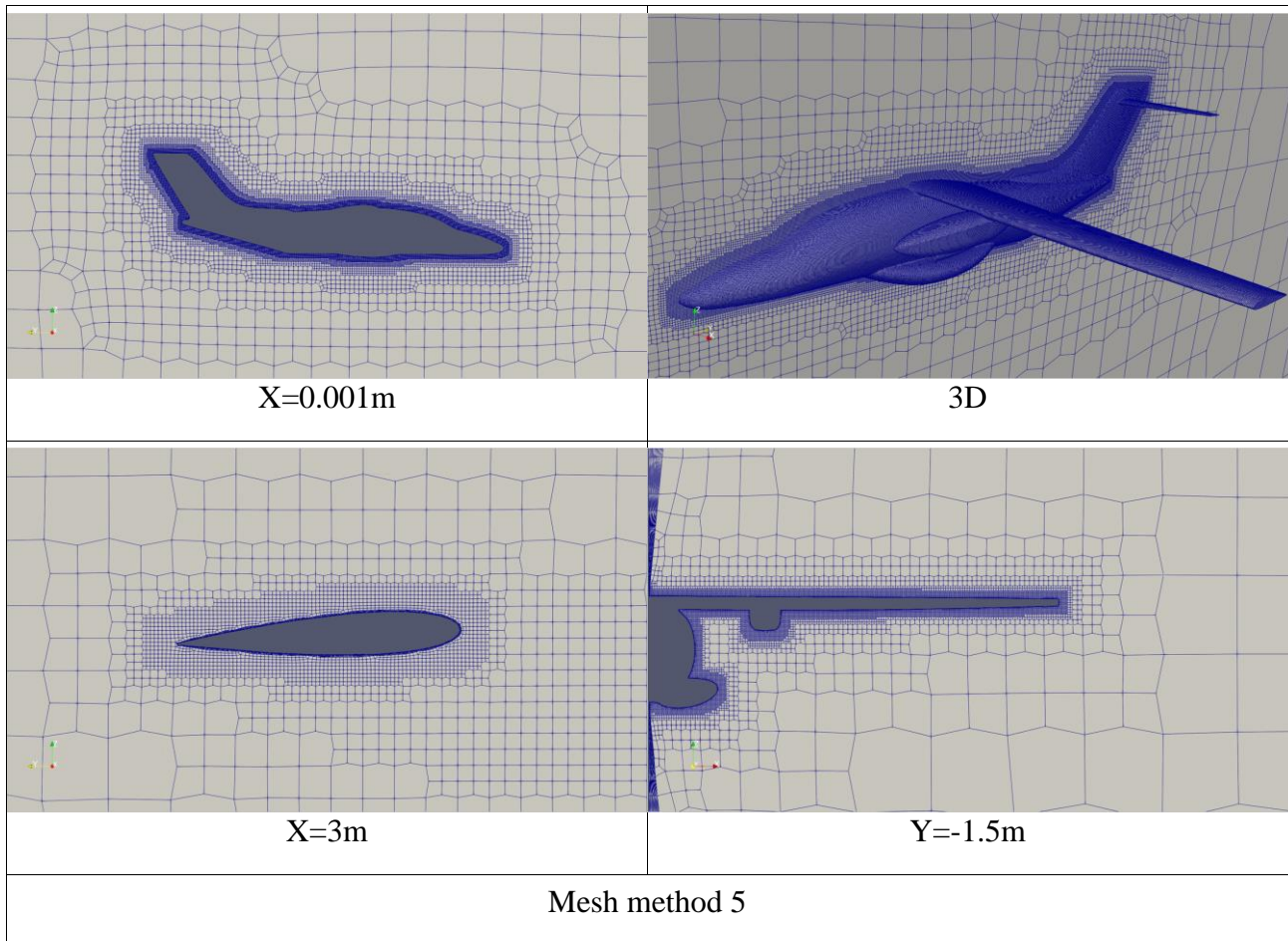


Figure II.20- Mesh refinement.

mesher	Cartesian
Base Element Size [m]	3 m
Surface refinement	0.015
Boundary layer number (surface)	20
Mesh time [min]	0.40 min
Disk size [MB]	216 MB
Total number of nodes	1369246

Table II.5- Mesh Parameters method 5.



Note

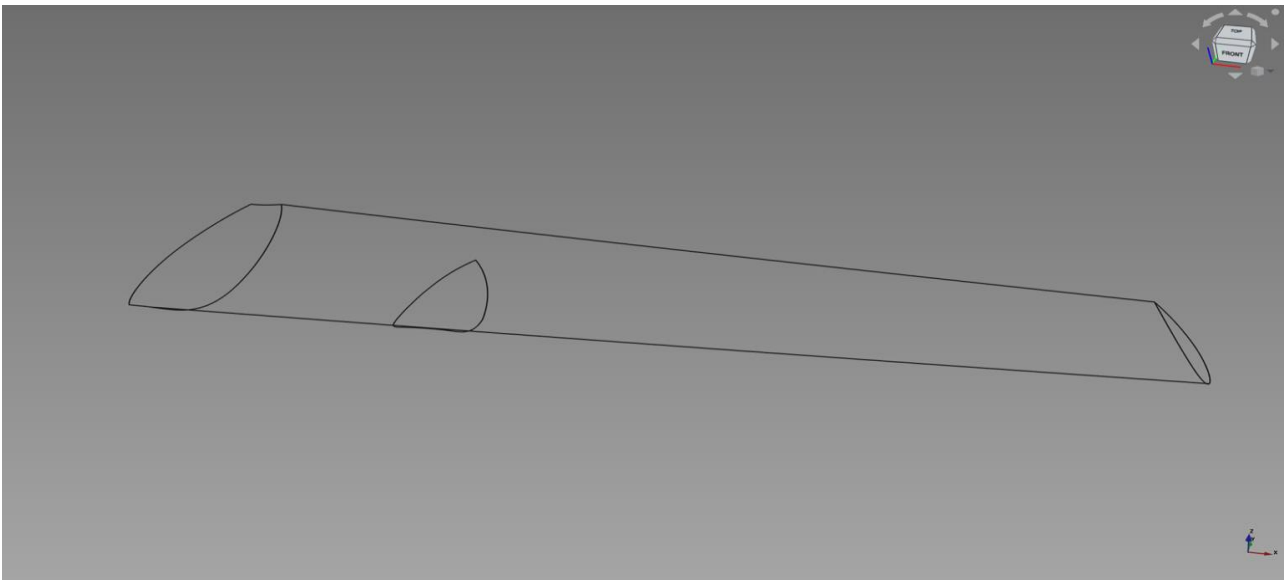
Note that the mesh looks kind of good with a moderate number of elements we come to choose another different method and notice the difference.

II.2.7- Method 6

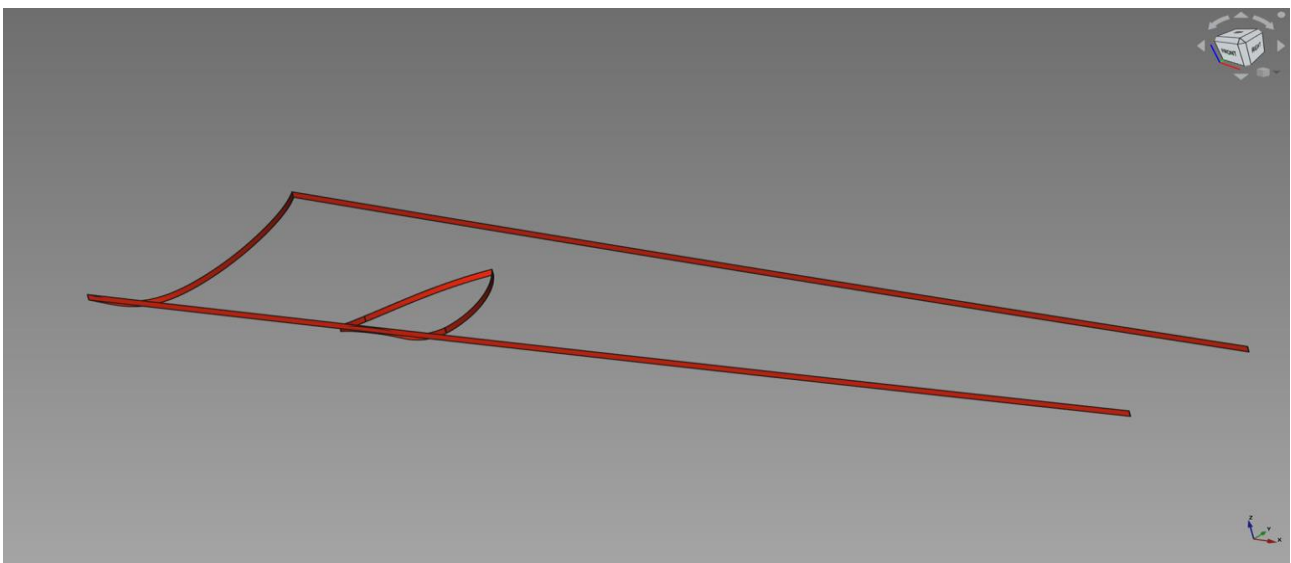
Strips Method The naming method refers to a particular technique where 50 mm wide strips are created along the edges of the plane. This technique is detailed below.

We open draft workbenches and click downgrade we extract the plane parts and then we choose the wing faces and we return the ball with one wing facet and it turns into wires we choose a wire from the wire and we return the ball and disintegrate the wire to the edges we choose the edges and the wheel to the workbench part and click on the extrusion we give the value of 50mm and click on the symmetry

And then we repeat the process on the rest of the plane.



***Figure II.21-** Picture of the pavilion in the shape of the edges.*



***Figure II.22-** Picture of the pavilion in the form of strips.*

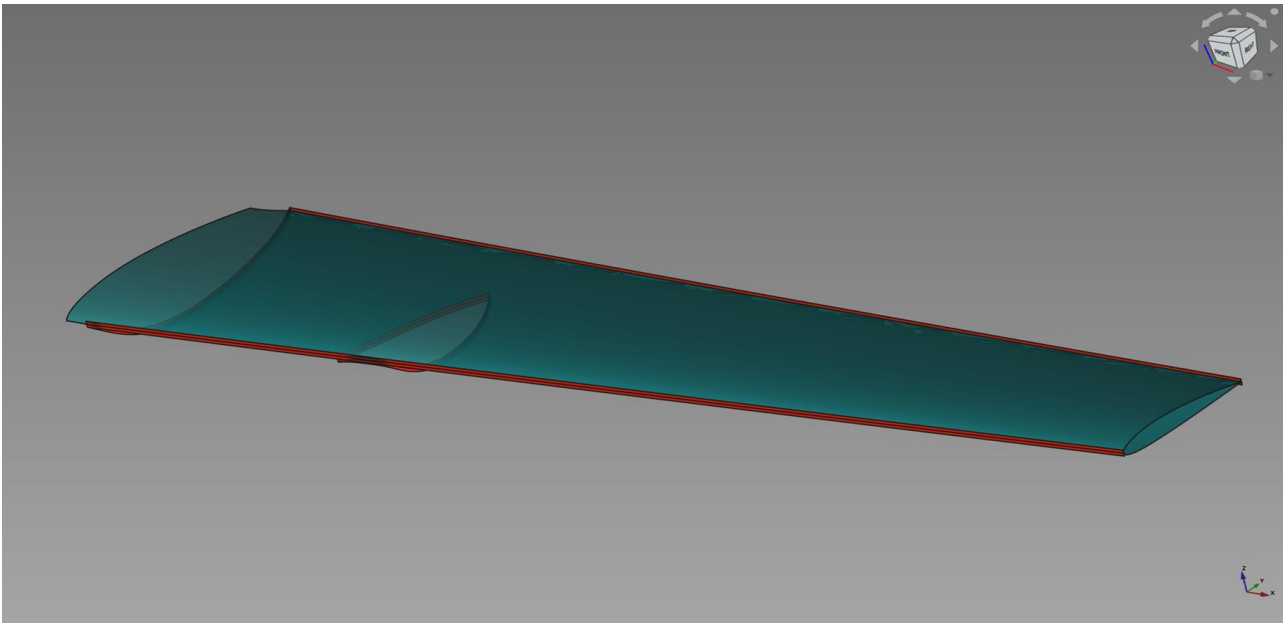


Figure II.23- Picture of the pavilion and strips with some.

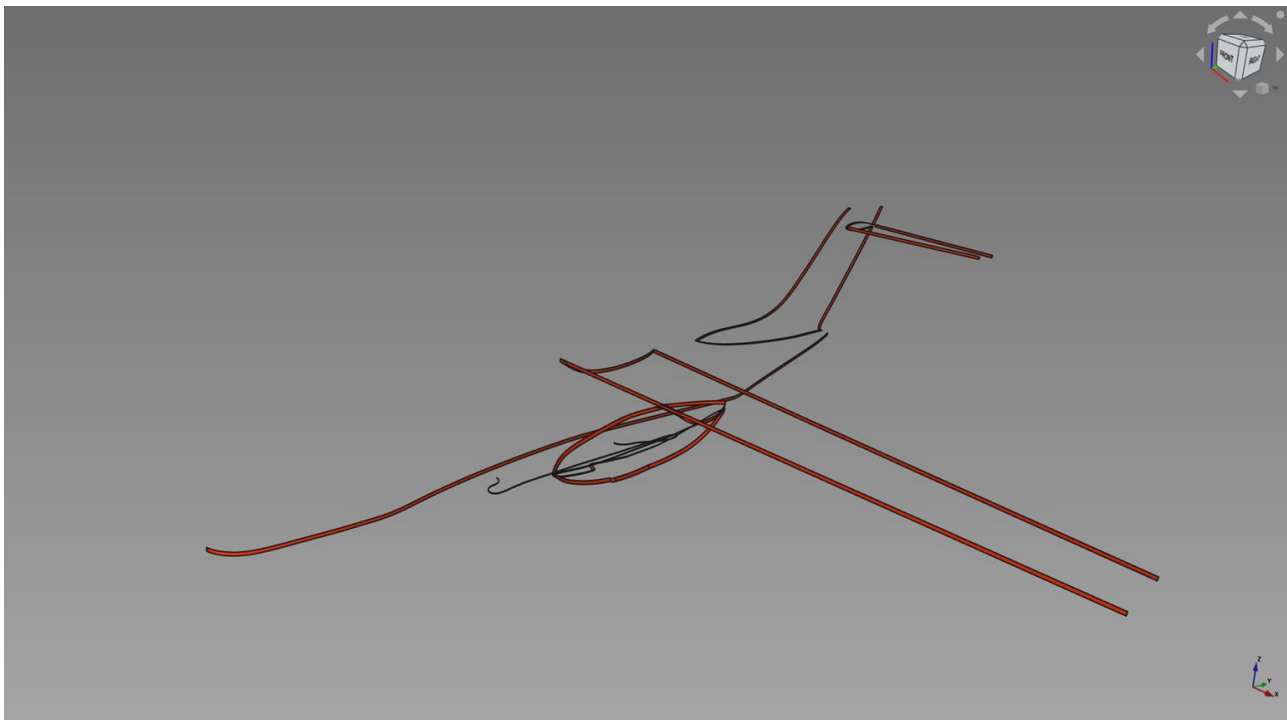
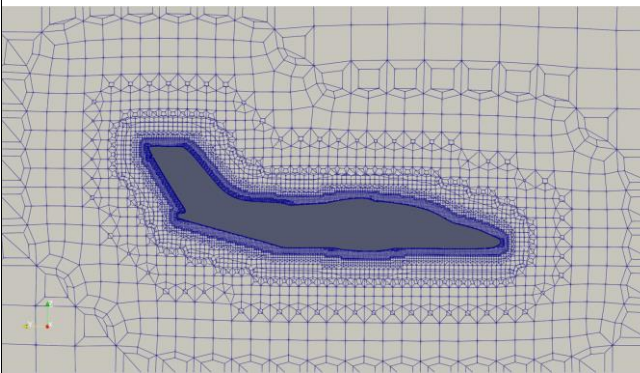
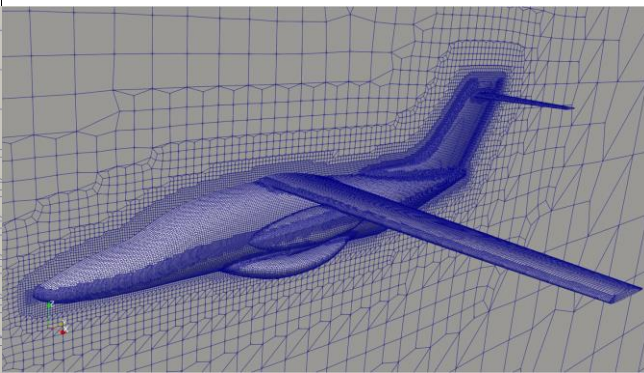
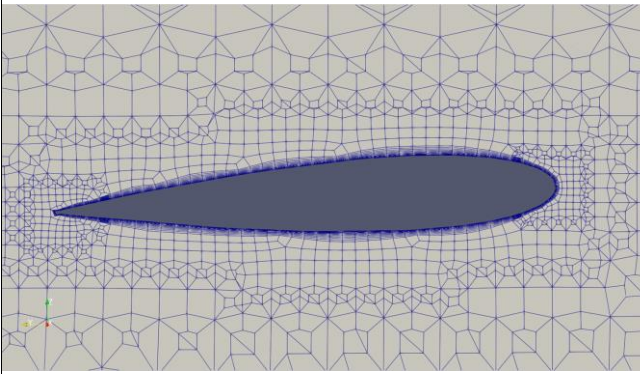
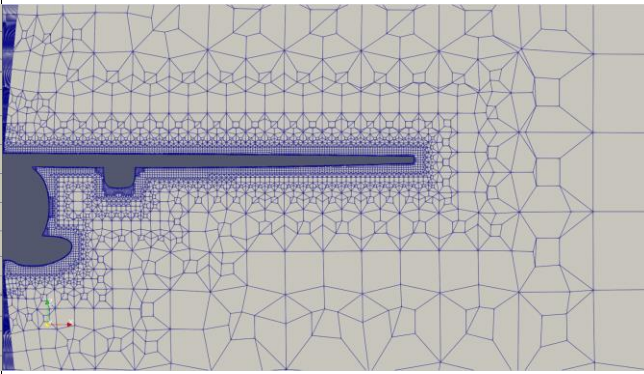


Figure II.24- Picture of full plane strips.

mesher		Cartesian
Base Element Size [m]		3 m
Surface refinement		0.030
Refinement thickness		10mm
Strips Surface refinement	All strips	0.010
	Strips wing	0.005
Boundary layer number (surface)		20
Mesh time [min]		1Min 49s
Disk size [MB]		410 MB
Total number of nodes		2409878

Table II.6- Mesh Parameters method 6.

	
X=0.001m	3D
	
x=3m	Y=-1.5m
Mesh method 6	

Not: This is a way that I was given a good mesh with excellent refinement and is included in a mesh around flight surfaces with a focus on the fuselage and a relatively few acceptable elements.

II.2.8- Method 7

Limb-focused mesh method and edges In this way we focus on effective areas in an airplane We focus on the front and steam of the wing, the Dell, the engine and the plane

We cut a small part of the front of the plane and an entrance engine for air for the engine Cut a small part Wing front, broth and same step with tail. The goal is to focus the mesh in the regionsIn this particular approach, our emphasis is placed on key areas of an airplane that are crucial for its functionality We pay particular attention to the front and rear sections of the wings, the fuselage, the engine, and the tail of the aircraft. To achieve this, we selectively isolate and analyze specific regions.

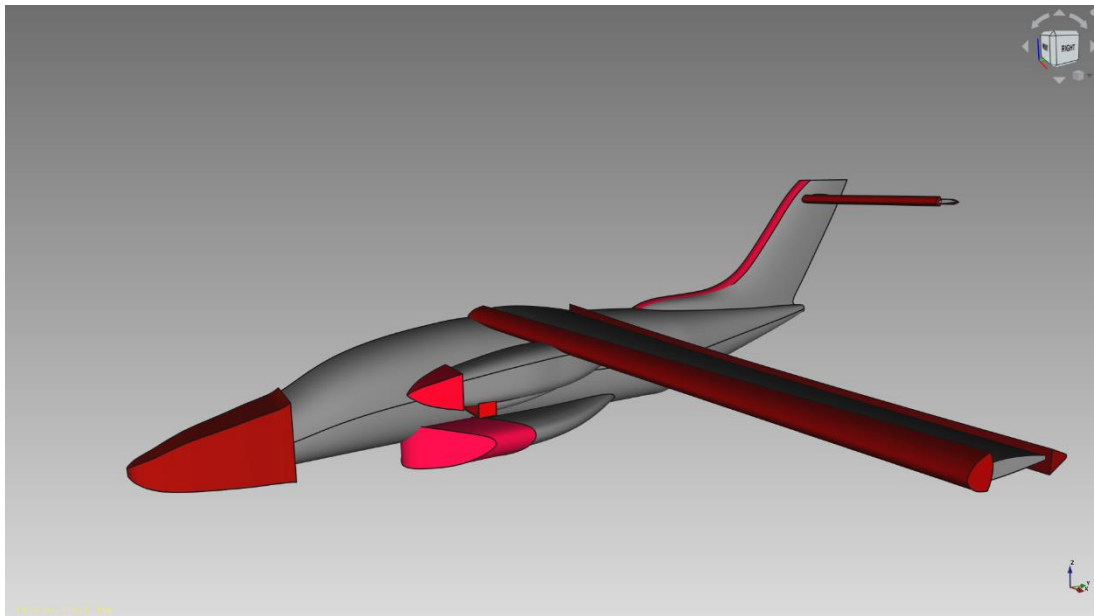


Figure II.25-- View of an airplane.

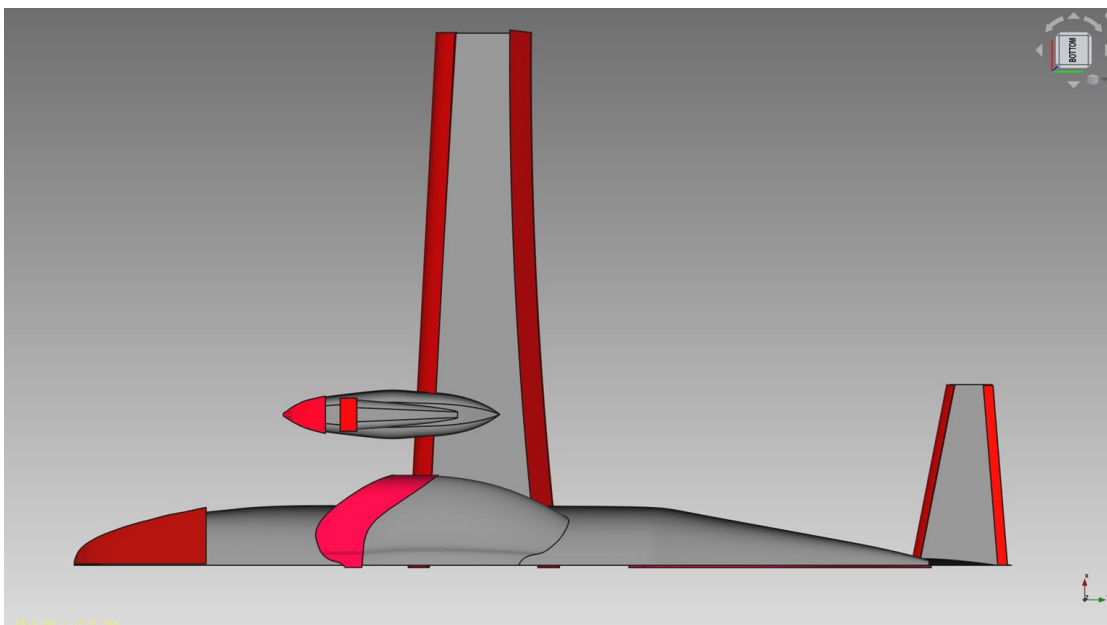
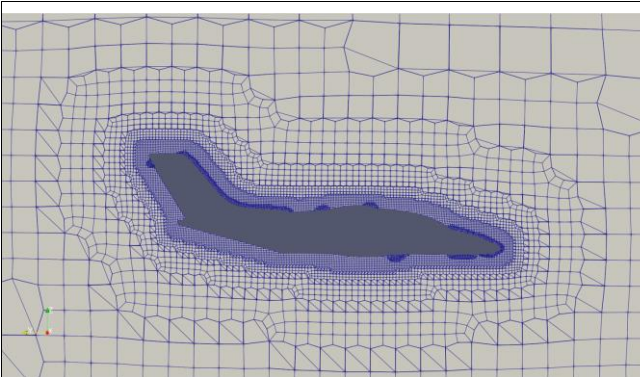
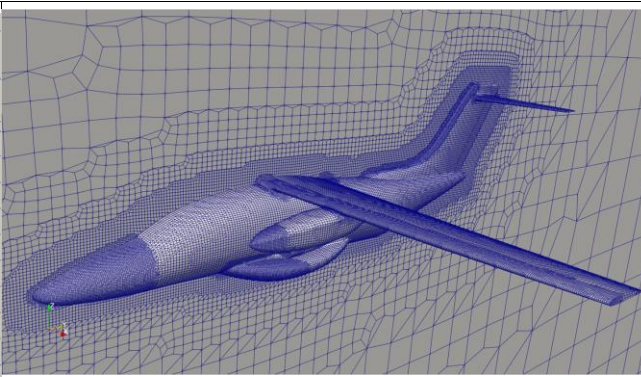
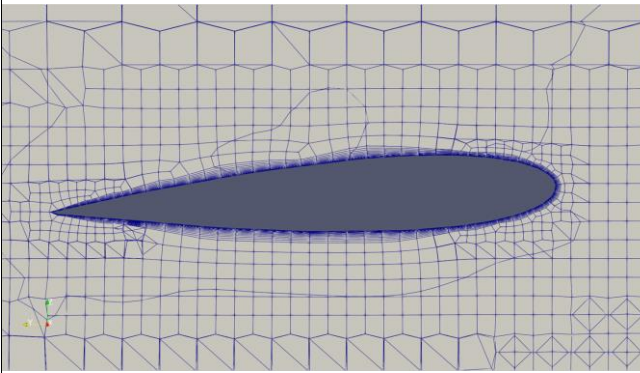
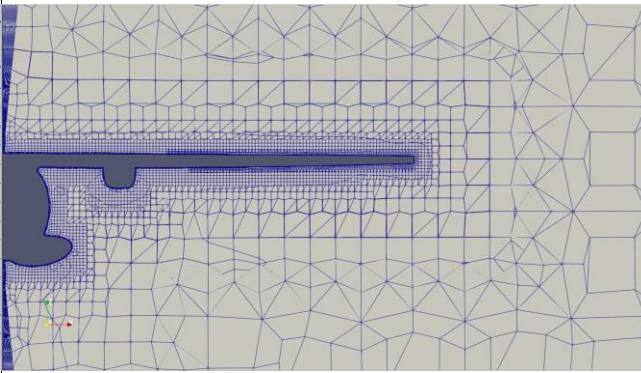


Figure II.26- Top view of an airplane.

Areas of refinement Front part of the fuselage, engine, landing gear box and air input in the engine front and rear part of the wing, front part of the tail.

mesher	Cartesian
Base Element Size [m]	3m
Surface refinement airplane	0.050
Volume refinement	0.020
Boundary layer number (surface)	20
Mesh time [min]	5min
Disk size [MB]	240 MB
Total number of nodes	1500000

Table II.7- Mesh Parameters method 7.

	
X=0.001m	3D
	
x=3m	Y=-1.5m
Mesh method 7	

Note

We notice that I am a well-distributed mesh in the effective regions and graded around a plane.

II.2.9- Method 8

In this approach, we generate a three-dimensional volume by initially creating a plane. Then, we proceed to outline the desired shape around this plane and finally extrude the drawing to give it depth and form. This method allows us to transform a flat plane into a solid object by adding thickness and dimension to the original drawing.

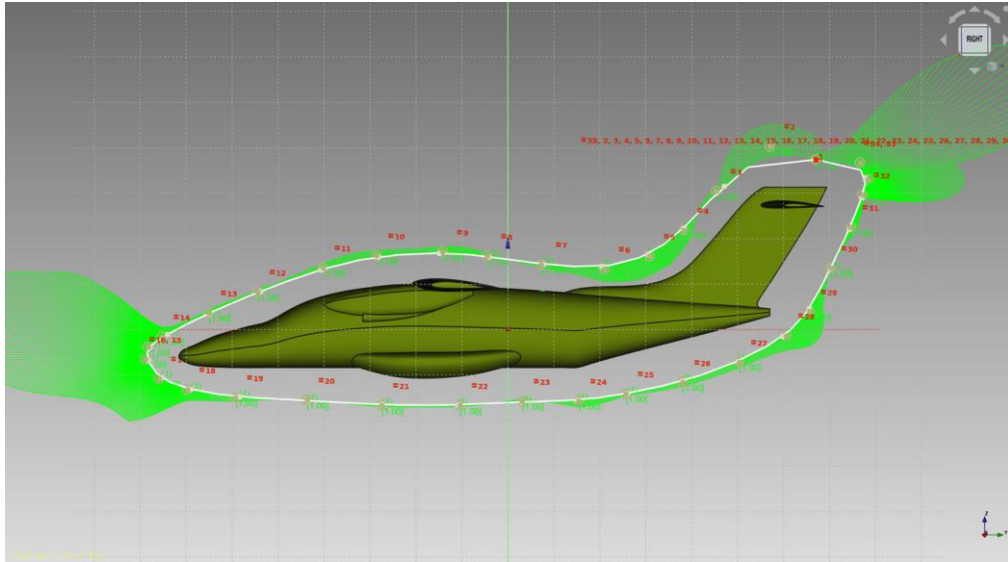


Figure II.27-Image of sketch about the plane.

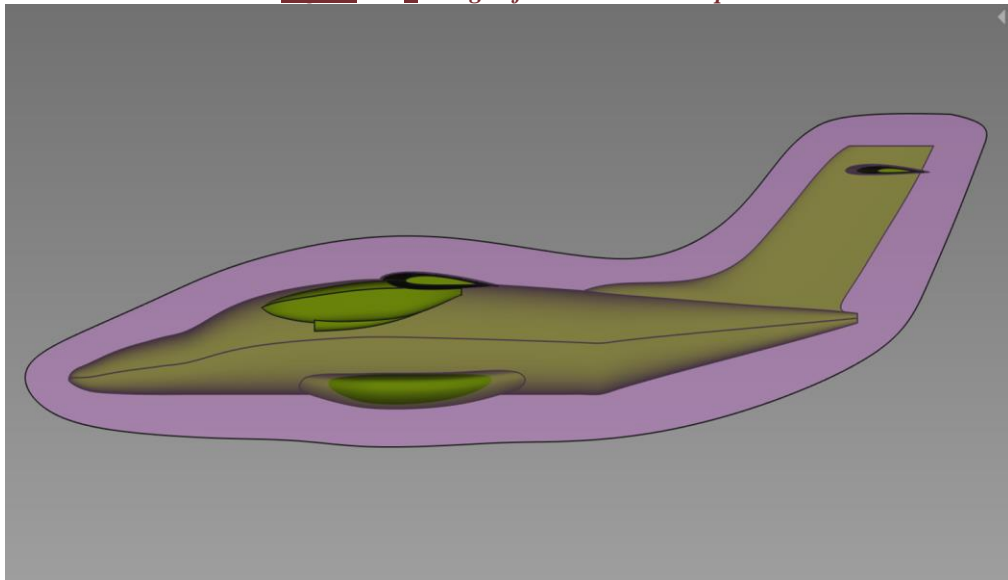


Figure II.28-Photo of me to extrude around the plane.

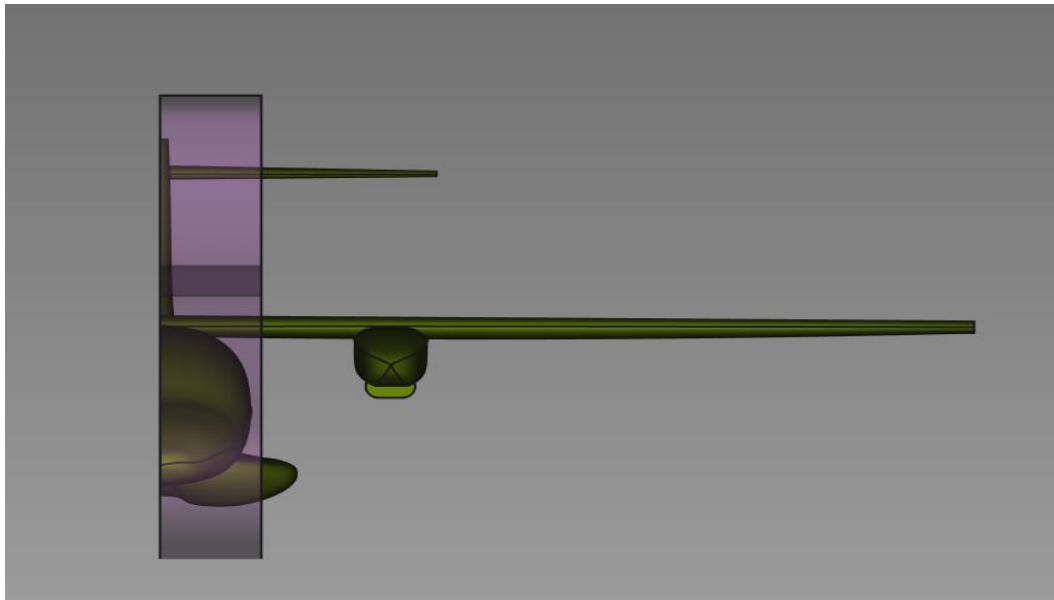
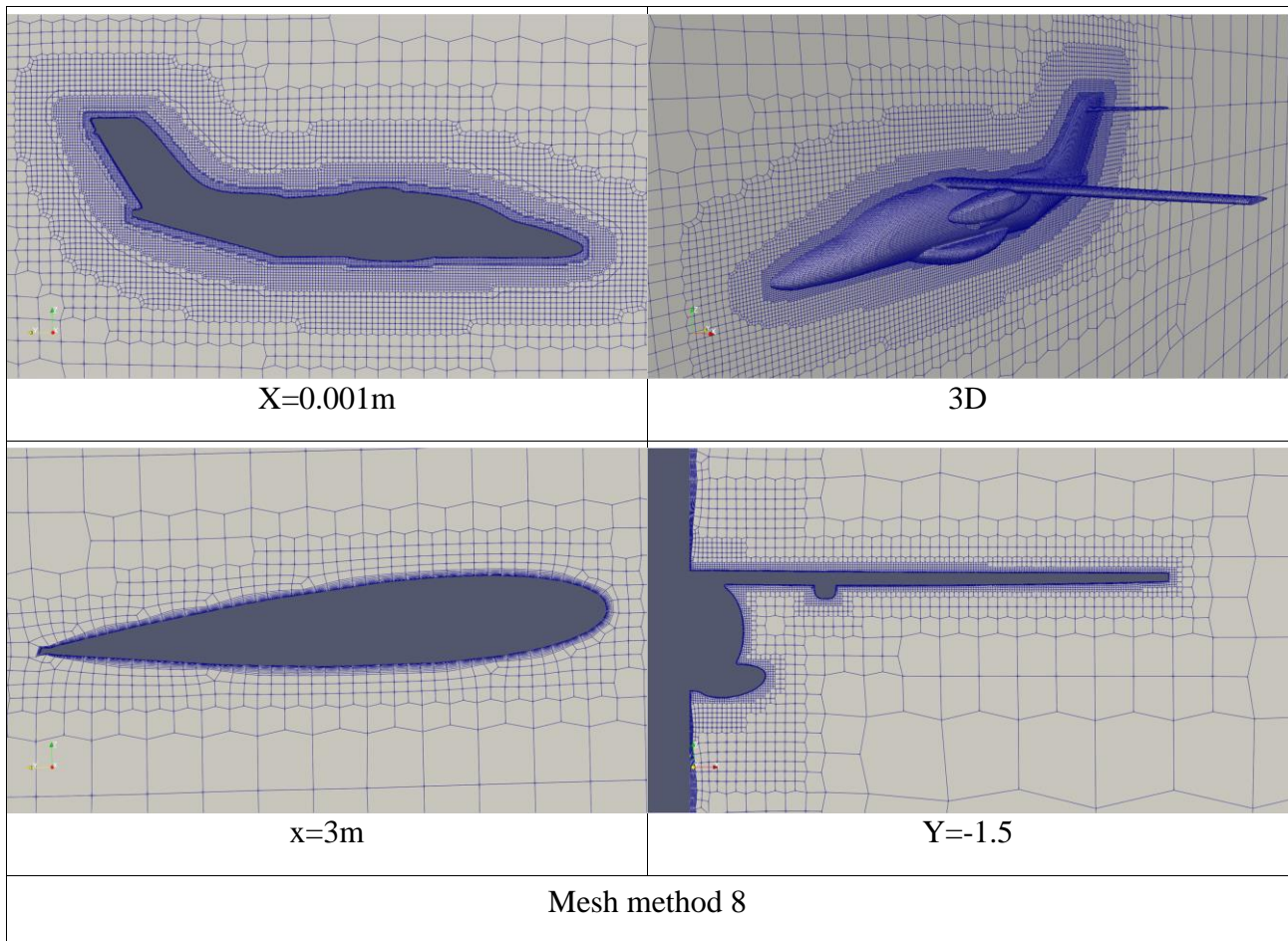


Figure II.29-Front-face image of plane extrusion.

We refine the entire plane surface by 0.020 from the base with 10mm thickness and 14 boundary layer and also refine the size of the shape we created by 0.040 and refine the body surroundings surface by 0.080 with 500mm thickness

mesher		Cartesian
Base Element Size [m]		2.6m
Surface	Surface refinement airplane	0.020
	Refinement thickniss airplane	10mm
volume	Volume refinement	0.040
	Surface refinement	0.080
Refinement thickniss form		500mm
Boundary layer number (surface)		14
Mesh time [min]		0.35min
Disk size [MB]		229.791 MB
Total number of nodes		1471379

Table II.8- Mesh Parameters method 8.



Note

this is a way that I was given a good mesh with excellent refinement and is included in a mesh around flight surfaces with a focus on the fuselage and a relatively few acceptable elements.

II.2.10- Comparison of different methods

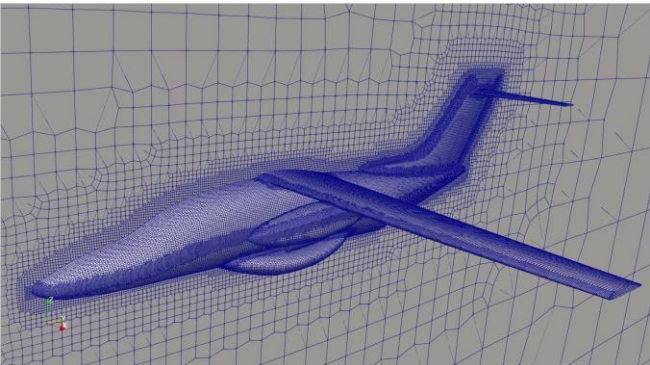
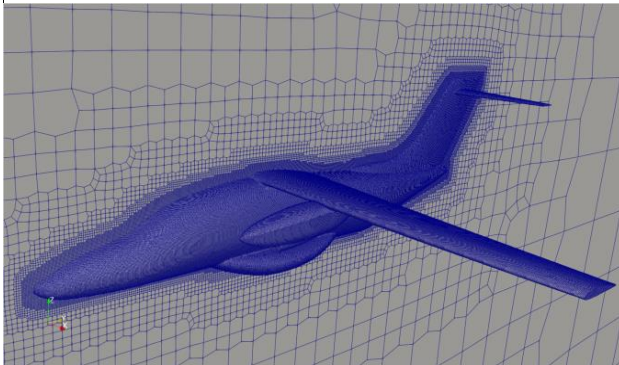
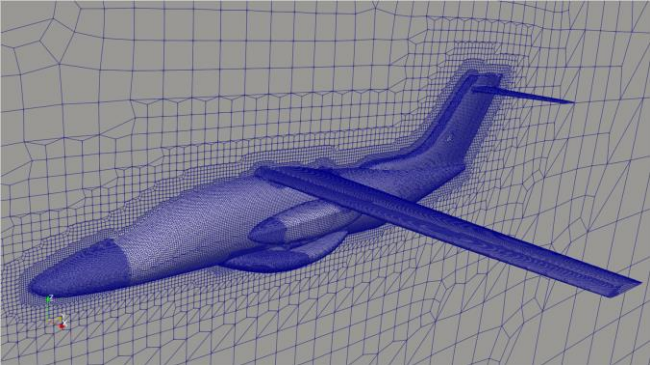
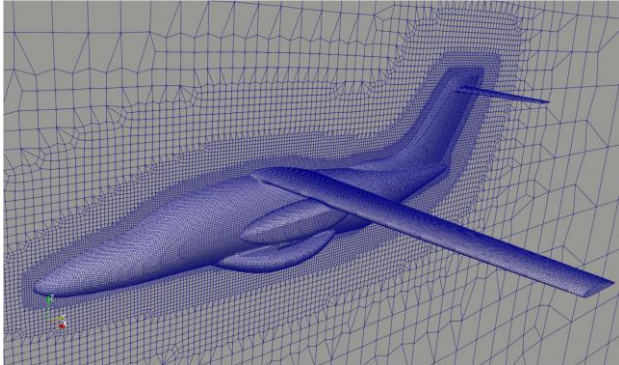
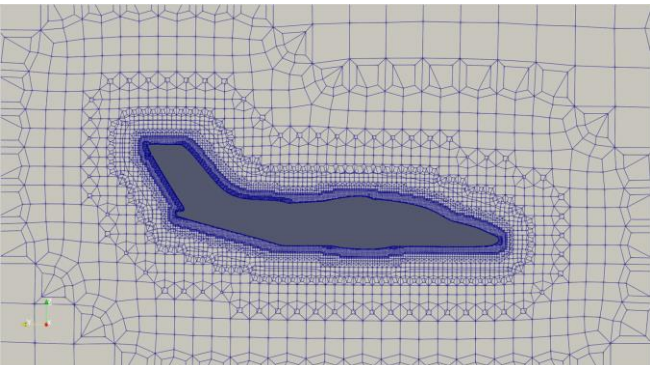
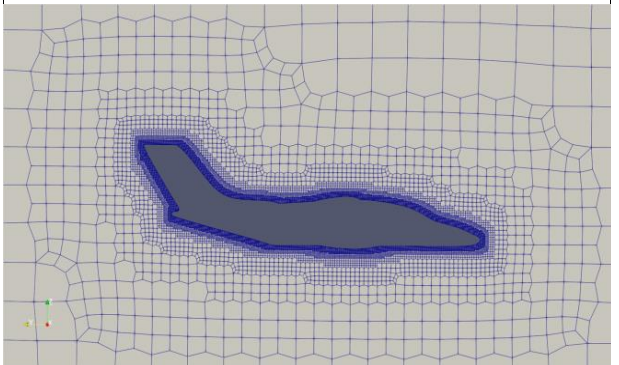
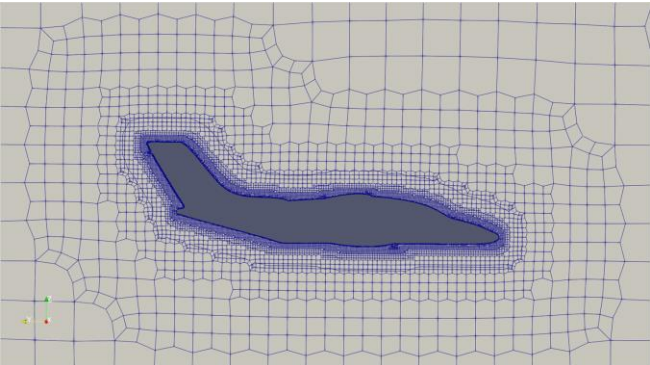
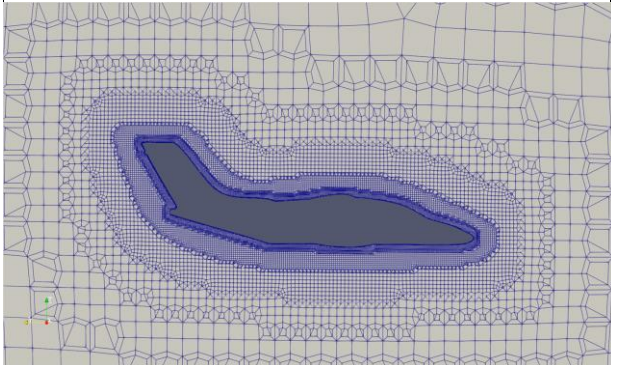
At this stage four ways have been selected to continue with us in the next stages:

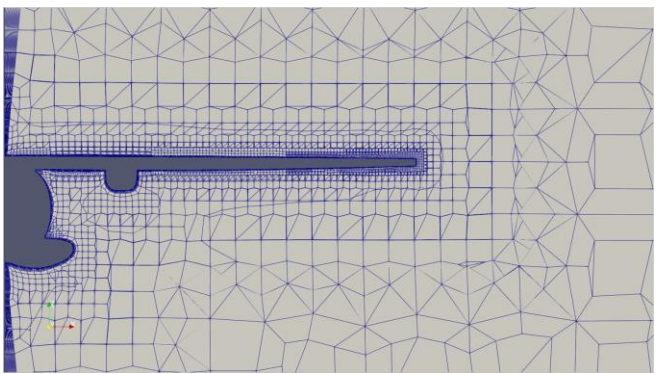
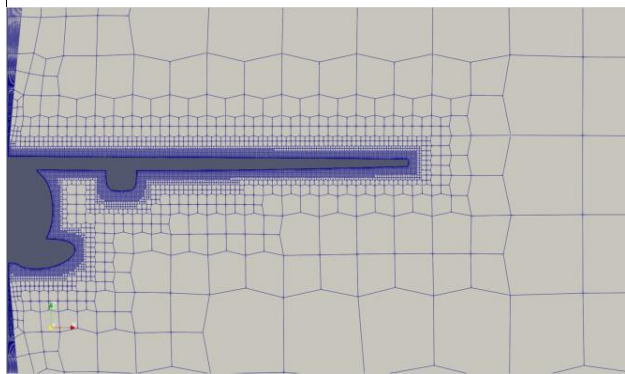
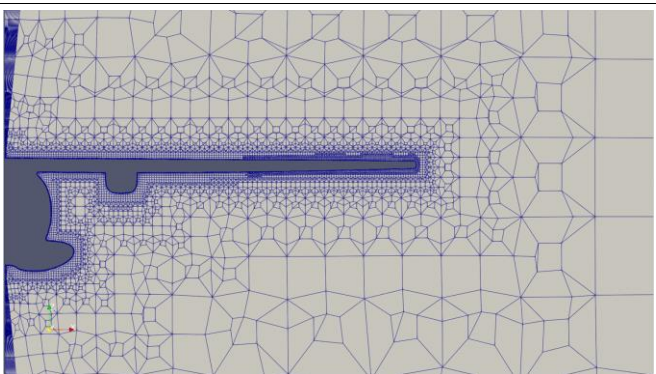
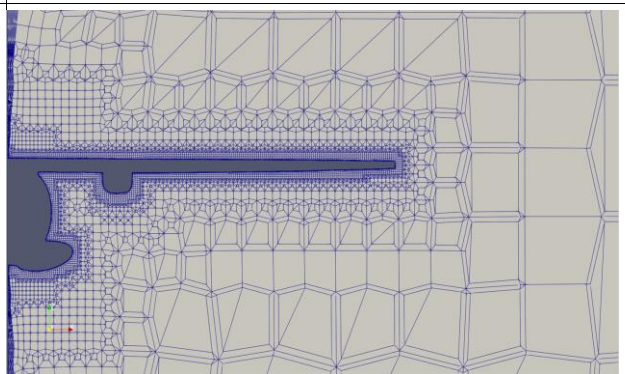
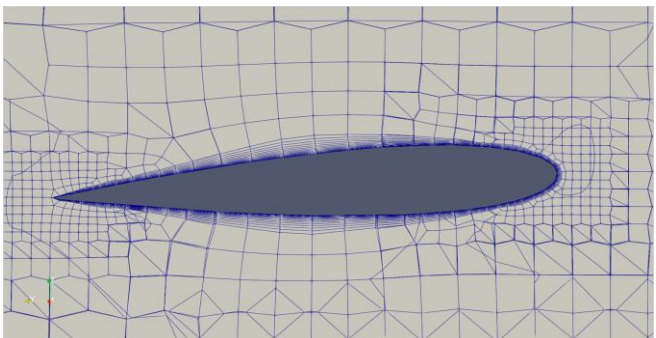
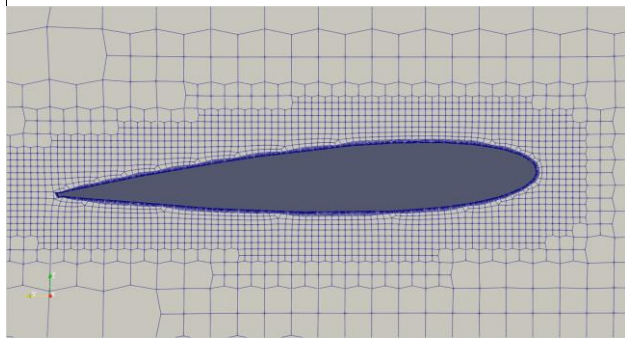
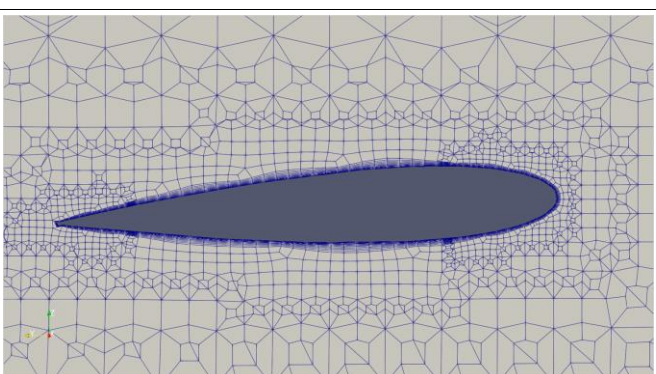
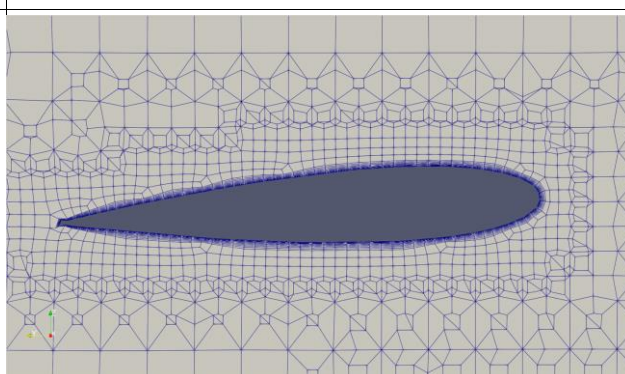
method5/method6/method7/method8 We change the name to case1/case2/case3/case4

These were chosen based on the good polishing, the mesh gradation around the plane surface and the small number of elements.

	Case 1	Case 2	Case 3	Case 4
Mesh time [min]	40s	1min 49s	4min8s	35s
Disk size [MB]	216MB	410MB	360.938 MB	229.791 MB
Total number of nodes	1369246	2409878	2303751	1471379
badCells	285	295	200	70
checkMesh	Failed	ok	Failed	Failed

Table II.9-Showing the difference between mesh methods.

	
Case 2	Case 1
	
Case 3	Case 4
Comparison of methods 3D	
	
Case2	Case1
	
Case3	Case4
Comparison of methods X=0.001m	

	
Case2	Case1
	
Case3	Case 4
Comparison of methods $Y=-1.5m$	
	
Case 2	Case 1
	
Case 3	Case 4
Comparison of methods $x=3m$	

Note

We note that I am approaching the quarter that have been compared to compare the quality of a good mesh worth the next stage traffic.

Eye Mesh Inspection.

In this examination, we relied on the inspection of a mesh on the structure of a full-edged airplane. The wing is well polished. The boundary layers are graded. The mesh around the aircraft is intense.

II.3- Conclusion

In the end, after all the efforts to do eight ways and compare them we achieved acceptable results, and we were able to choose four ways To do the simulation in the next chapter.

Chapitre III: **Simulation numerique**

III.1- Introduction

In this chapter, we will verify the result of the network quality study through the simulation results obtained after performing calculations that require all system resources to work at maximum capacity.

Means of calculation

We used a low-cost computer compared to those used in aerodynamics, We used i5-9300H Intel® Core™ CPU at 2.40 GHz 8 processors with 4 cores and 16 GB RAM, Nvidia GTX 1650 with 4 GB, which means only Not suitable for this type of simulation.

III.2- Boundary conditions

physical model	Time	Constant
	flow	Incompressible viscous
	turbulent	RANS
	Model	KOmegaSST
fluid properties	fluid	Air
	density	1,2 kg/m ³
	dynamic viscosity	1,8e-05 kg/m.s
Initialization of the variables of internal flows	k	1,9 m ² /s ²
	Omega	2,5 deg/s
speed	Uy	120 m/s

Table III.1- Boundary conditions.

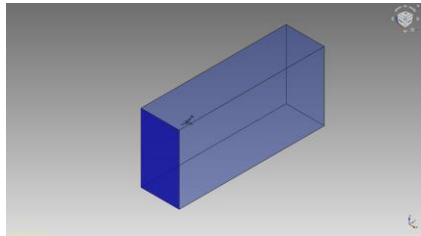
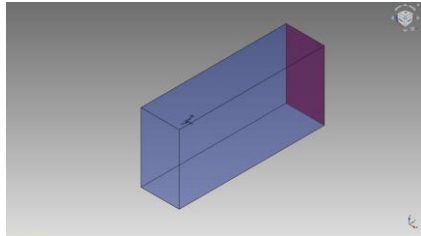
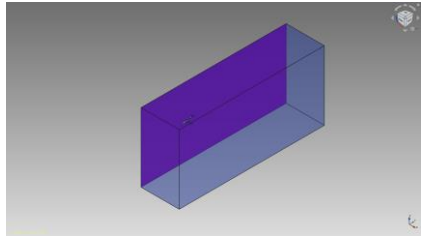
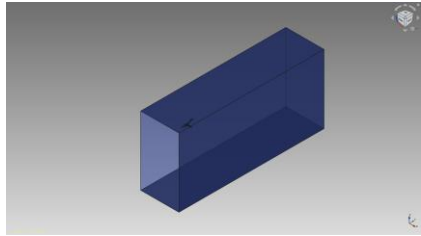
location	Boundary	Type	Face
The faces of the tunnel	Inlet	uniform velocity	
	Outlet	Static pressure	
	Contrail	Symétrie	
	Walls	slip	
The faces of half plane	Wall	noslip	Half plane

Table III.2- Boundary condition.

III.3- Simulation residuals

The calculation begins with an estimate of a potential solution, followed by running the OpenFOAM solver times. Once the affinity is verified, the results are ready for post-processing.

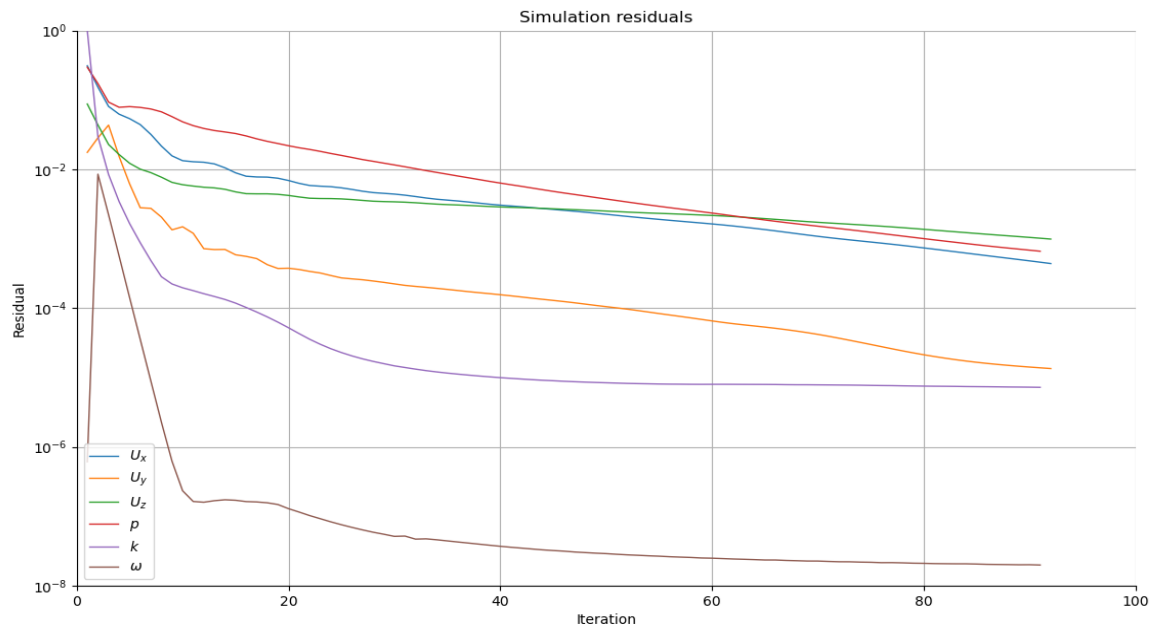


Figure III.1- Résidus de la simulation Case 1.

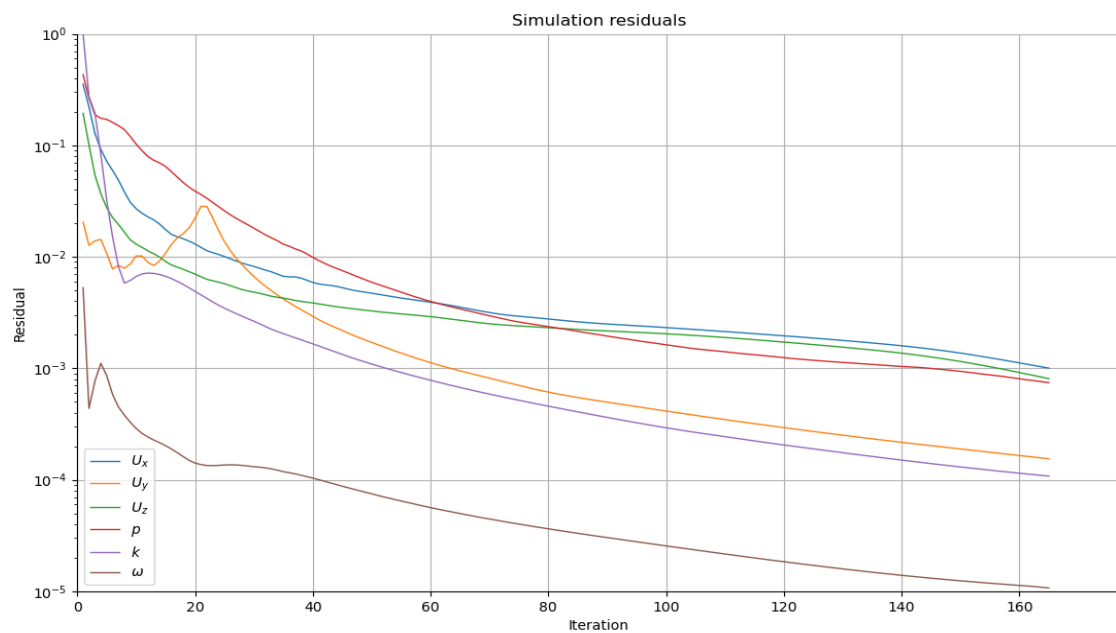


Figure III.2- Résidus de la simulation Case 2.

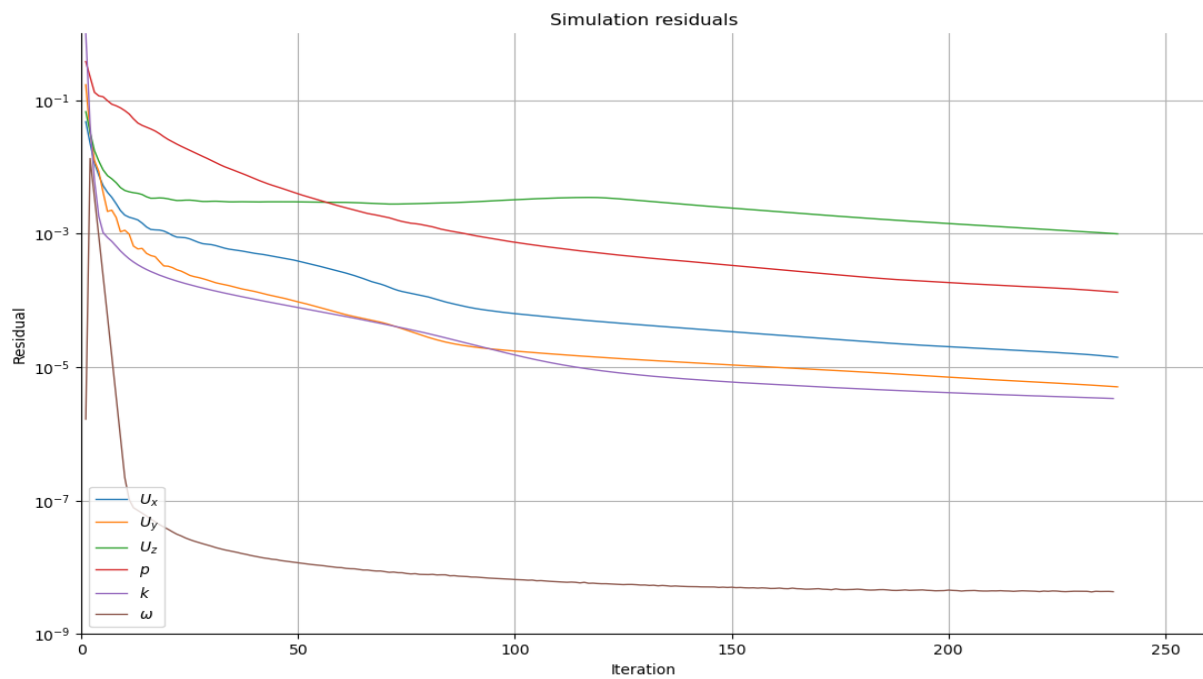


Figure III.3- Résidus de la simulation Case 3.

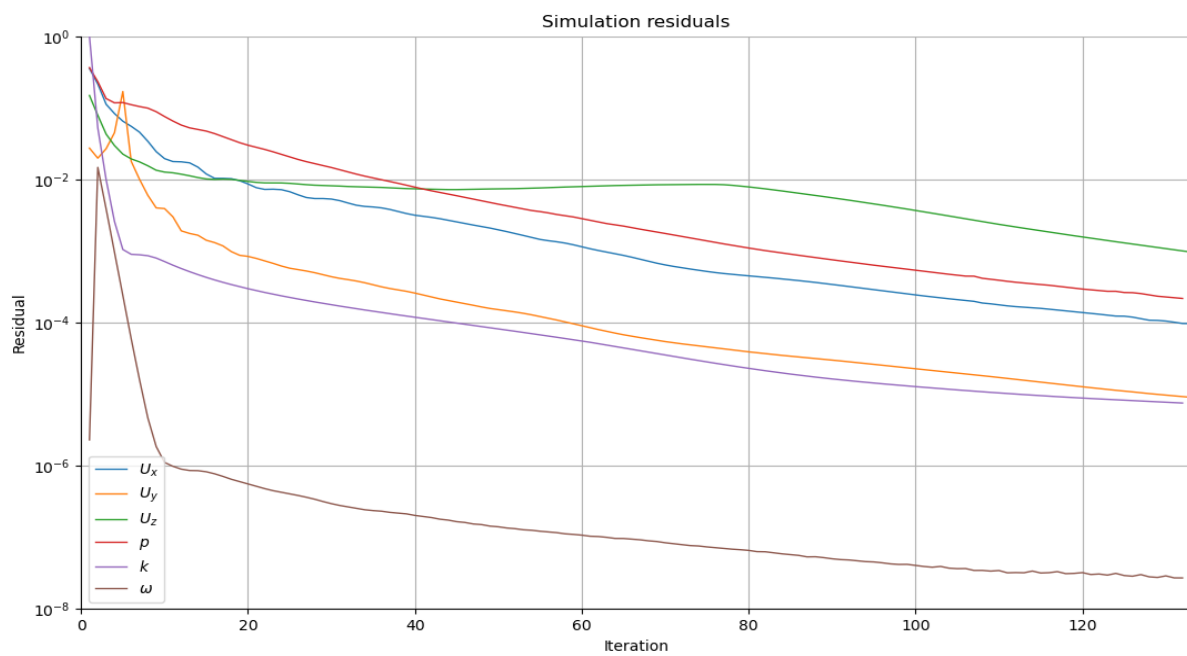


Figure III.4- Résidus de la simulation Case 4.

The time taken to analyze the four methods are shown in the table:

	Case 1	Case 2	Case 3	Case 4
Calculation Time	38 min	32 min33 s	49 min 30s	25 min 24 s

Table III.3- showing the time spent on the analysis.

III.4- Speed distribution

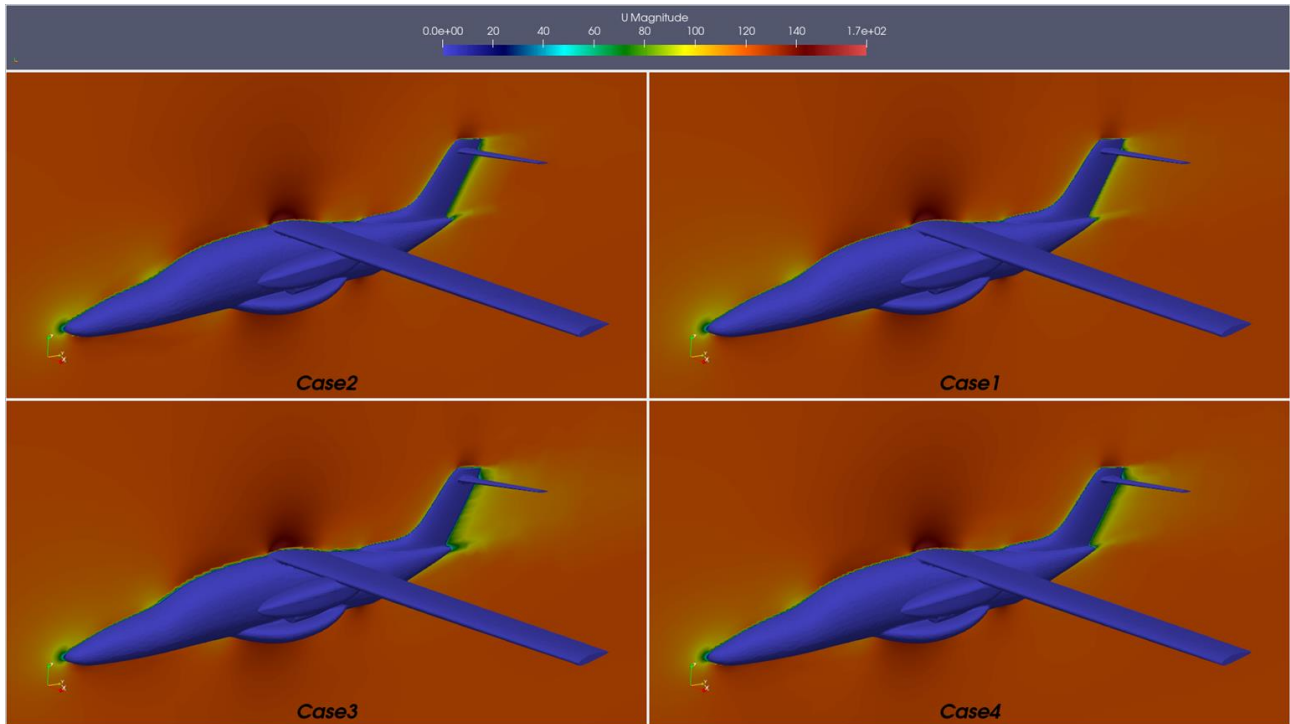


Figure III.5- Airspeed distribution around the aircraft.

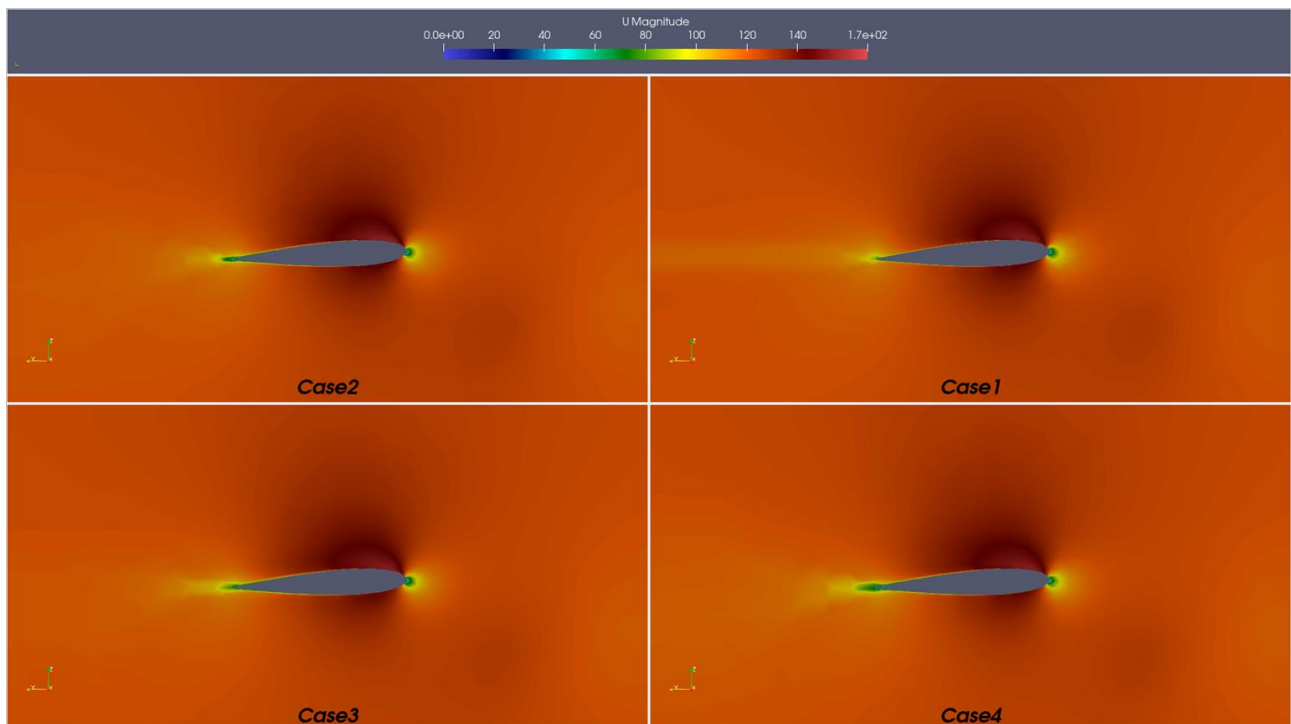


Figure III.6- Airspeed distribution around the aircraft X=3 m.

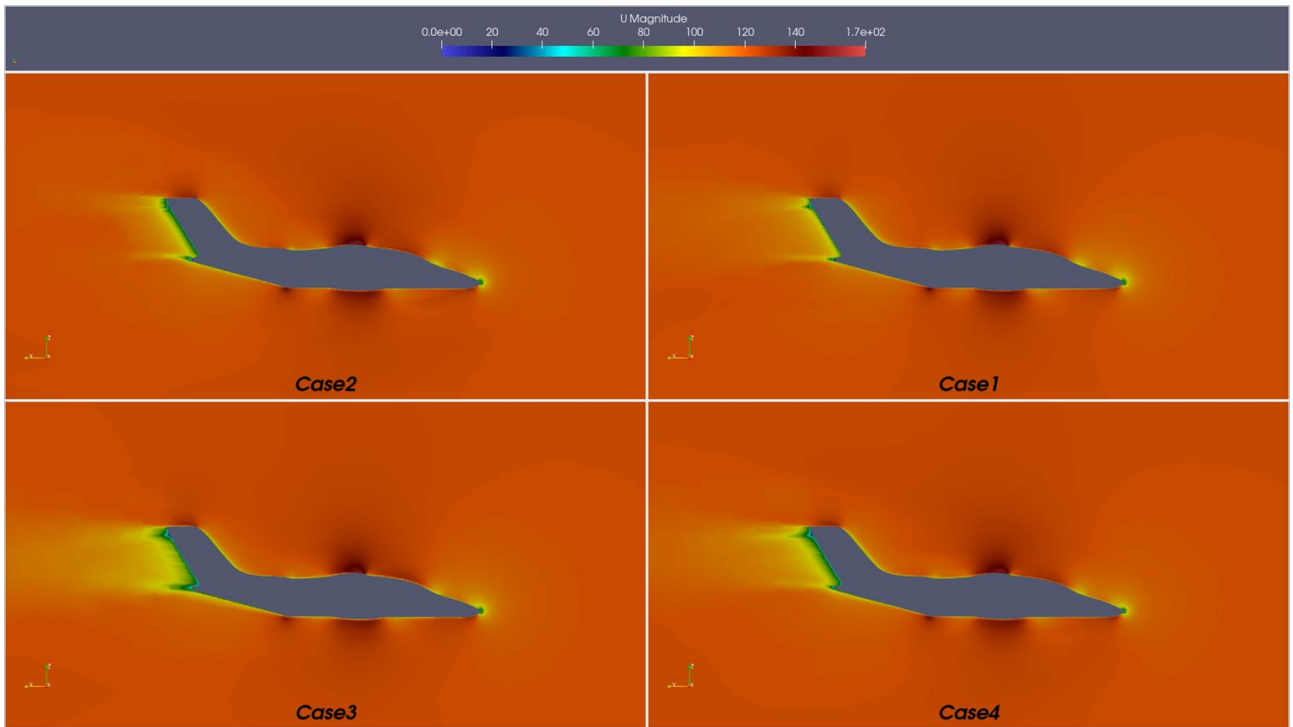


Figure III.7- Airspeed distribution around the aircraft. $X=0.001m$.

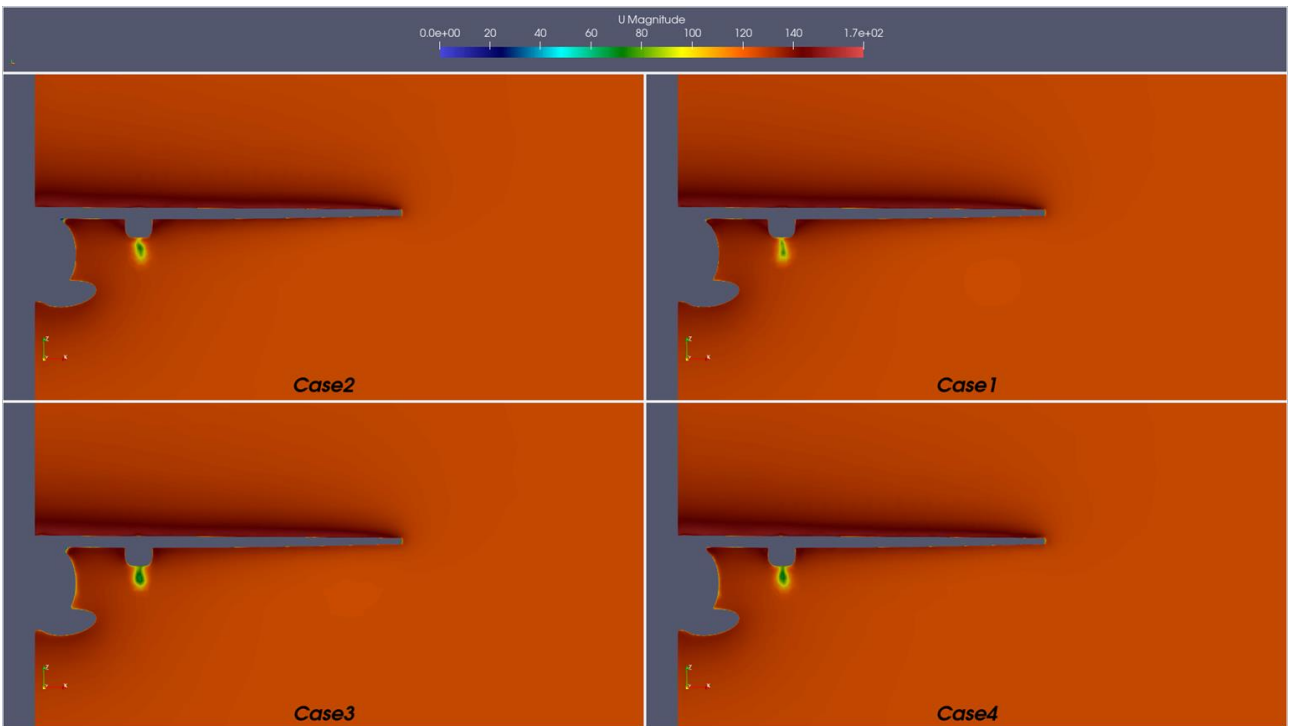


Figure III.8- Airspeed distribution around the aircraft. $Y=-1.5m$.

III.5- Pressure distribution

aircraft

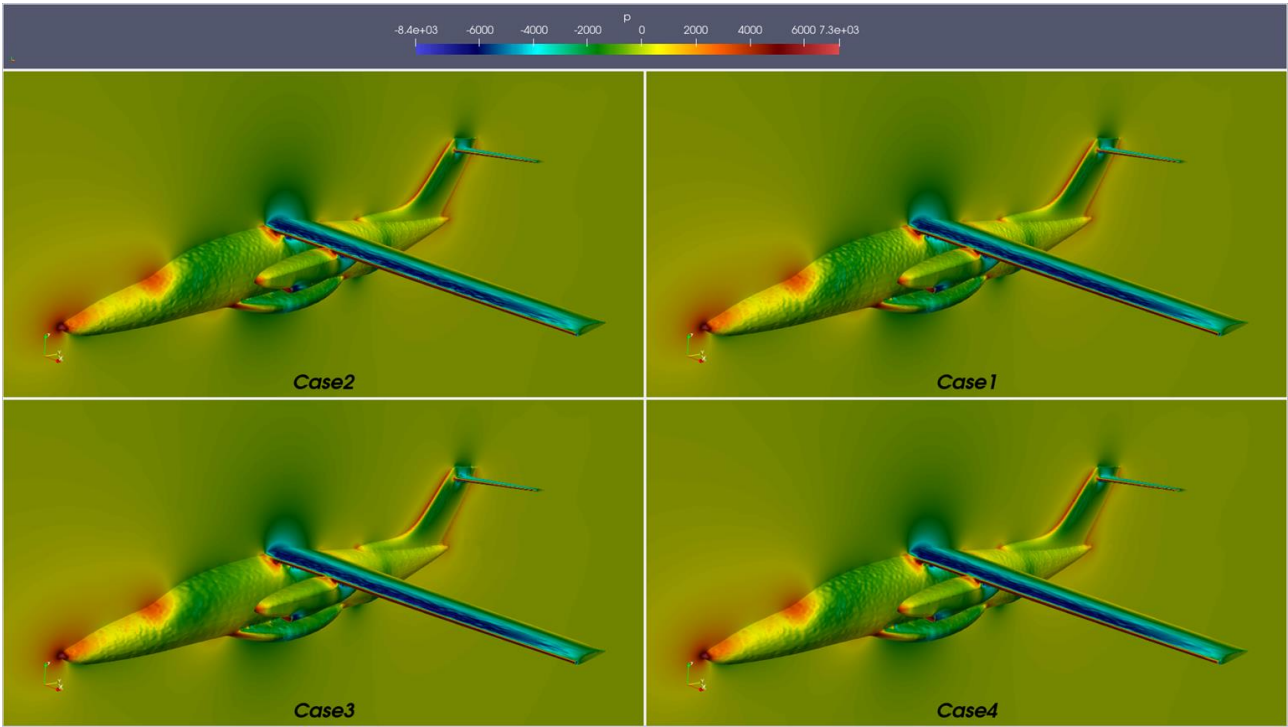


Figure III.9-Distribution of pressure around the plane.

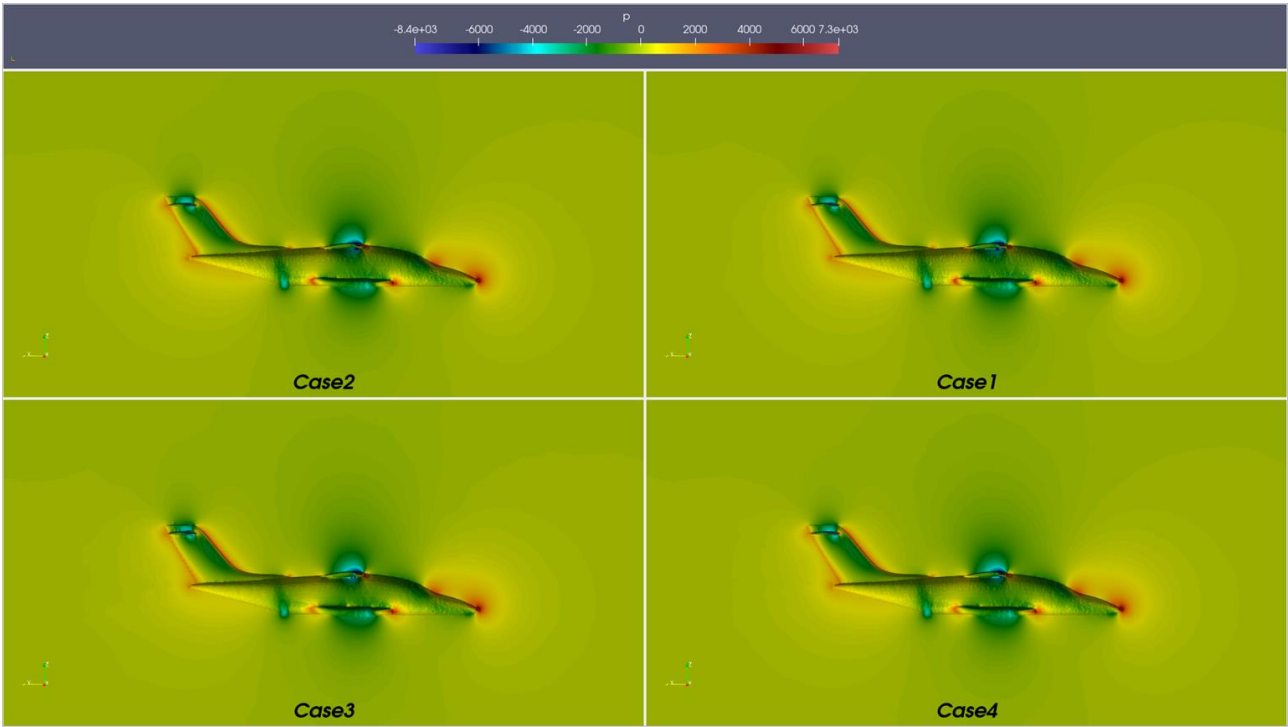


Figure III.10-Pressure distribution around the aircraft X=0.001m.

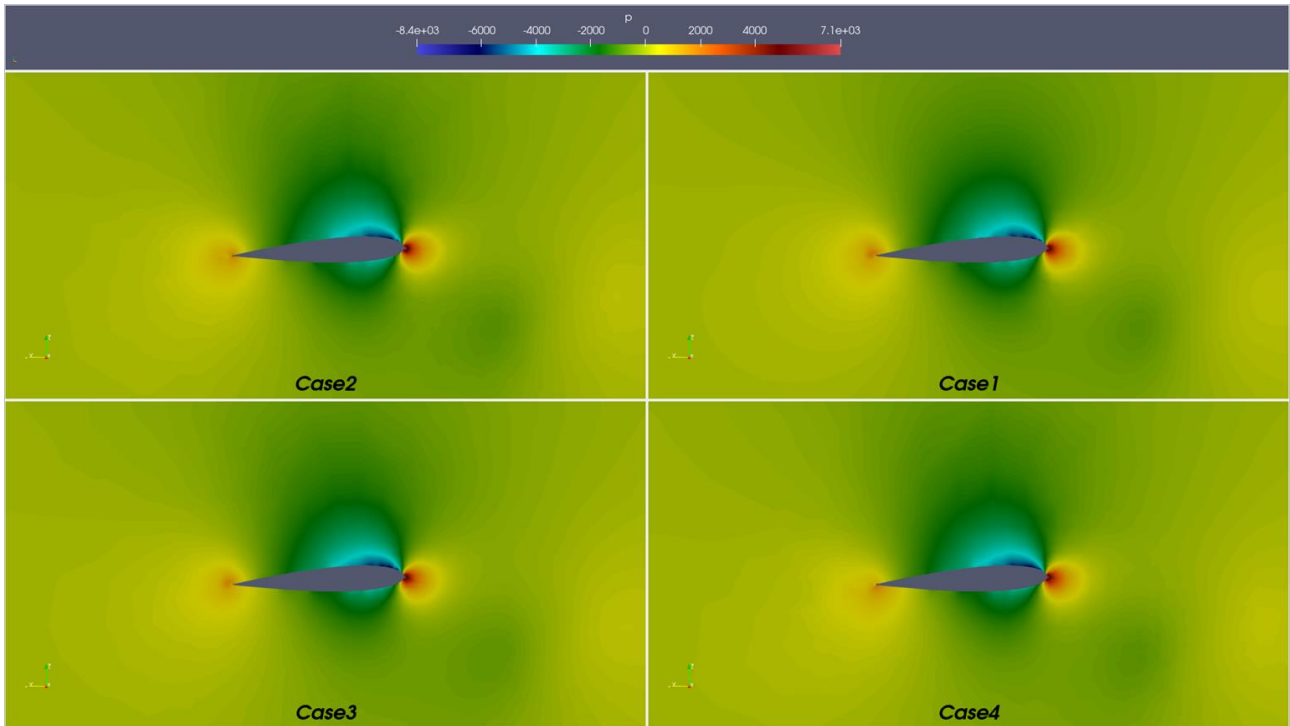


Figure III.11- Pressure distribution around the aircraft $x=3m$.

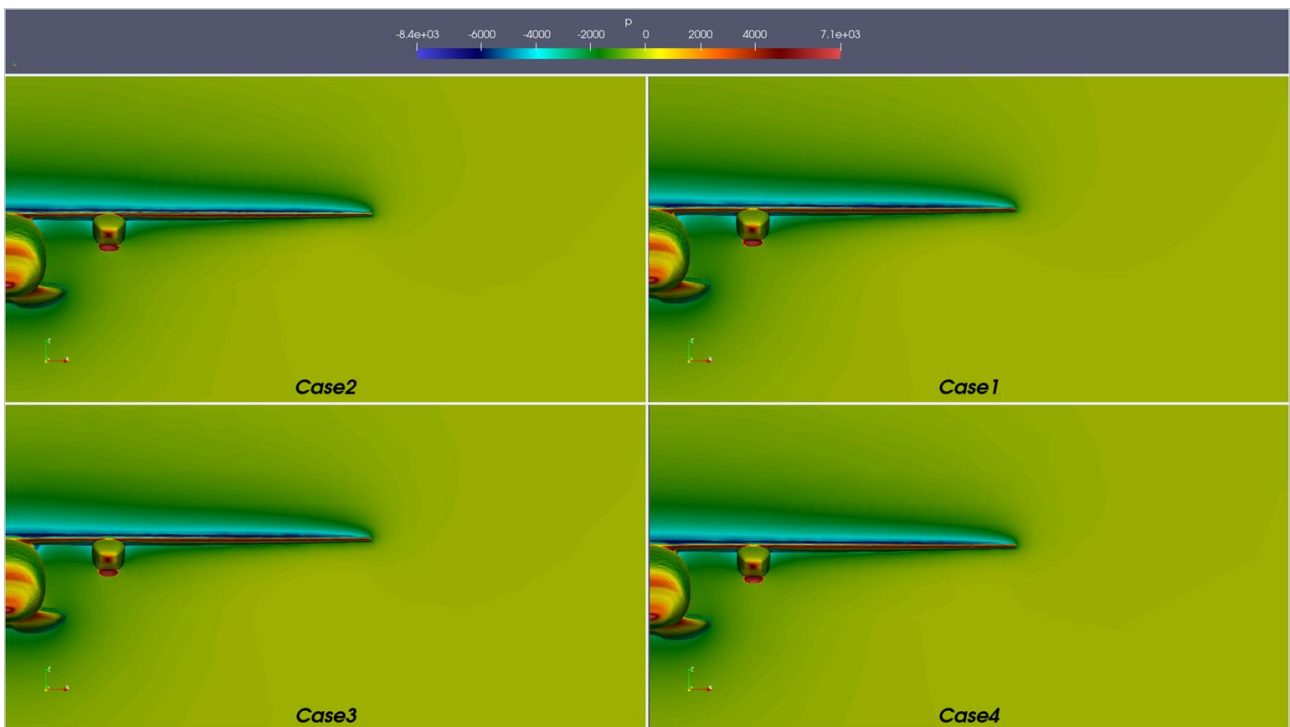


Figure III.12- Pressure distribution around the aircraft $Y=-1.5m$.

Pressure repair

We provide below (Figure 55) the graph of pressure variation as a function of the distance around the aircraft.

The image below shows the distribution of pressure over several segments along the wing ($x=3\text{m}$ / $x=4\text{m}$ / $x=5\text{m}$ / $x=6\text{m}$ / $x=7\text{m}$)

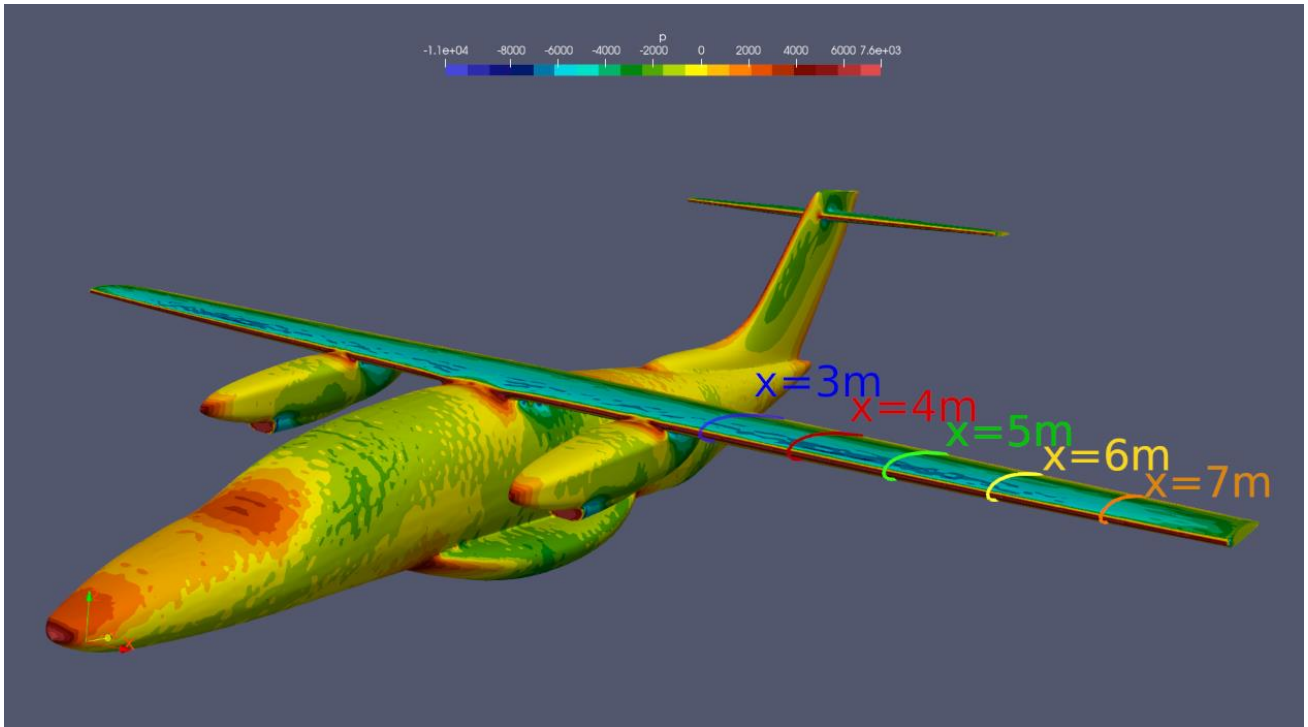


Figure III.13- The image shows sections of pressure distribution along the wing.

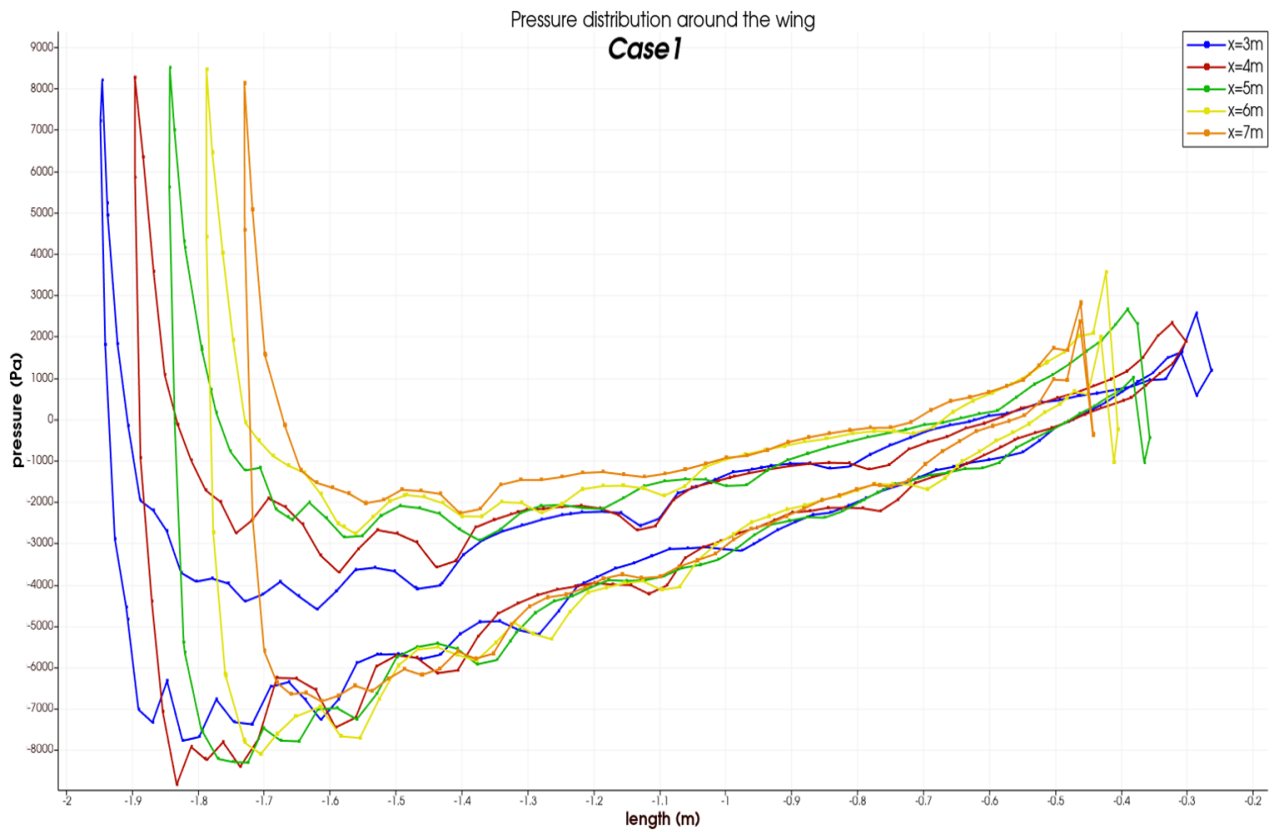


Figure III.14- Pressure distribution around the wing at Case 1.

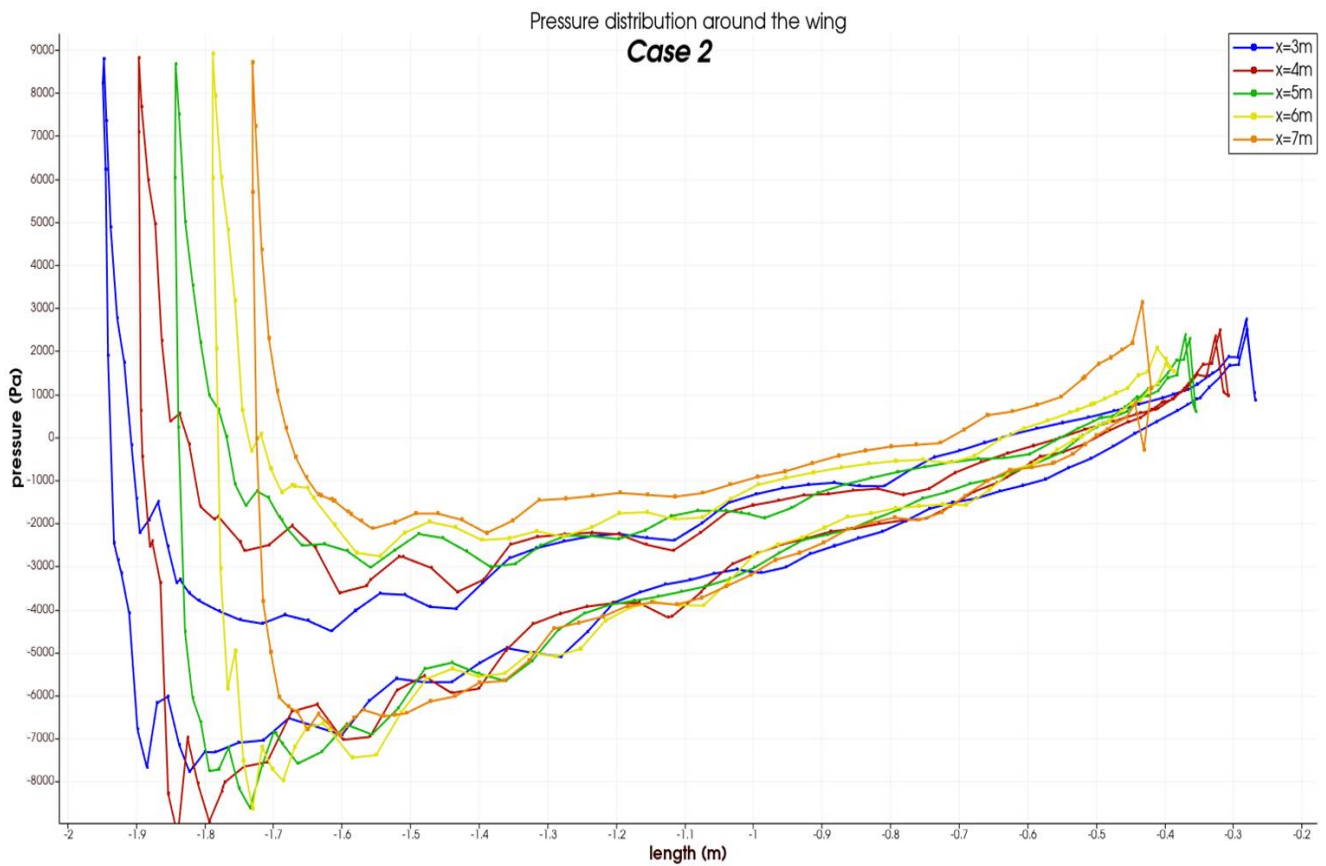


Figure III.15- Pressure distribution around the wing at Case 2.

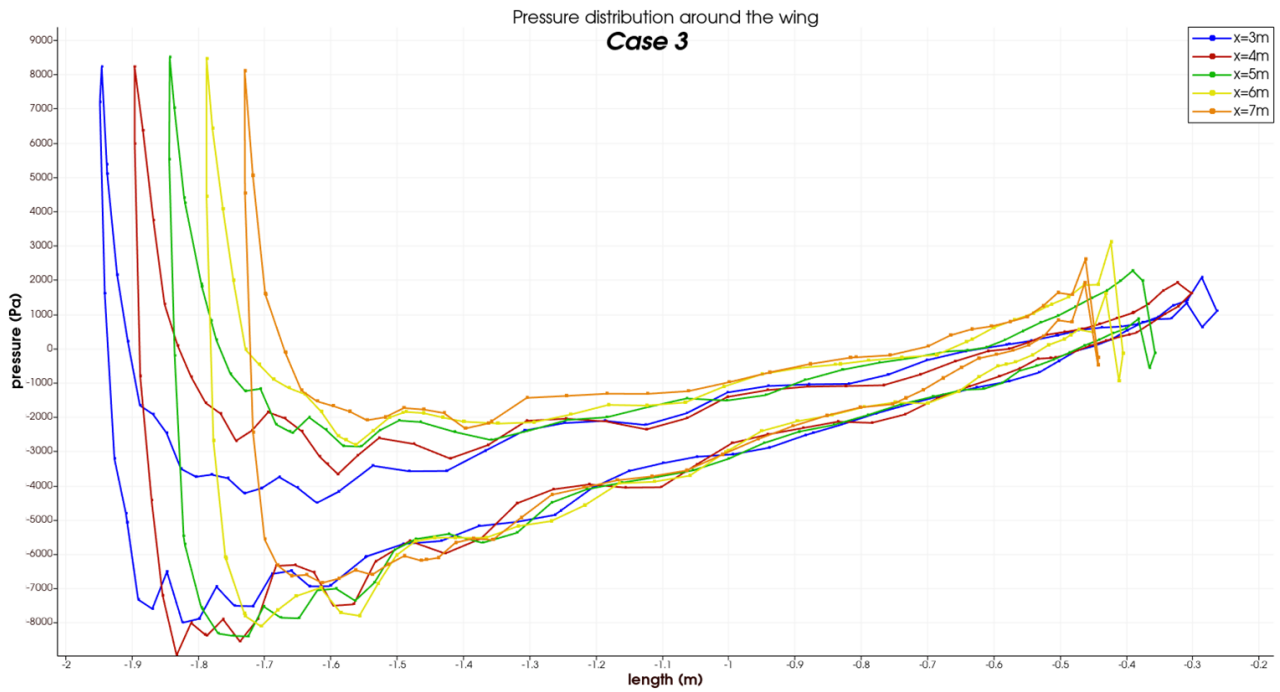


Figure III.16- Pressure distribution around the wing at Case 3.

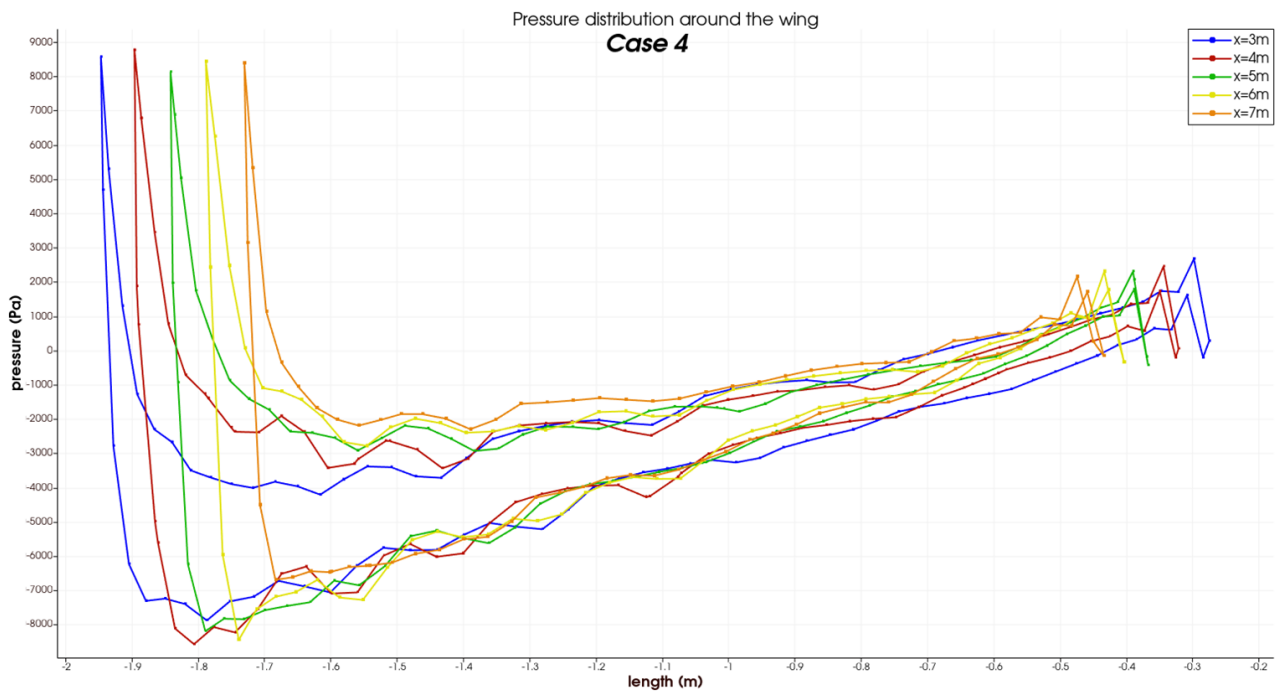
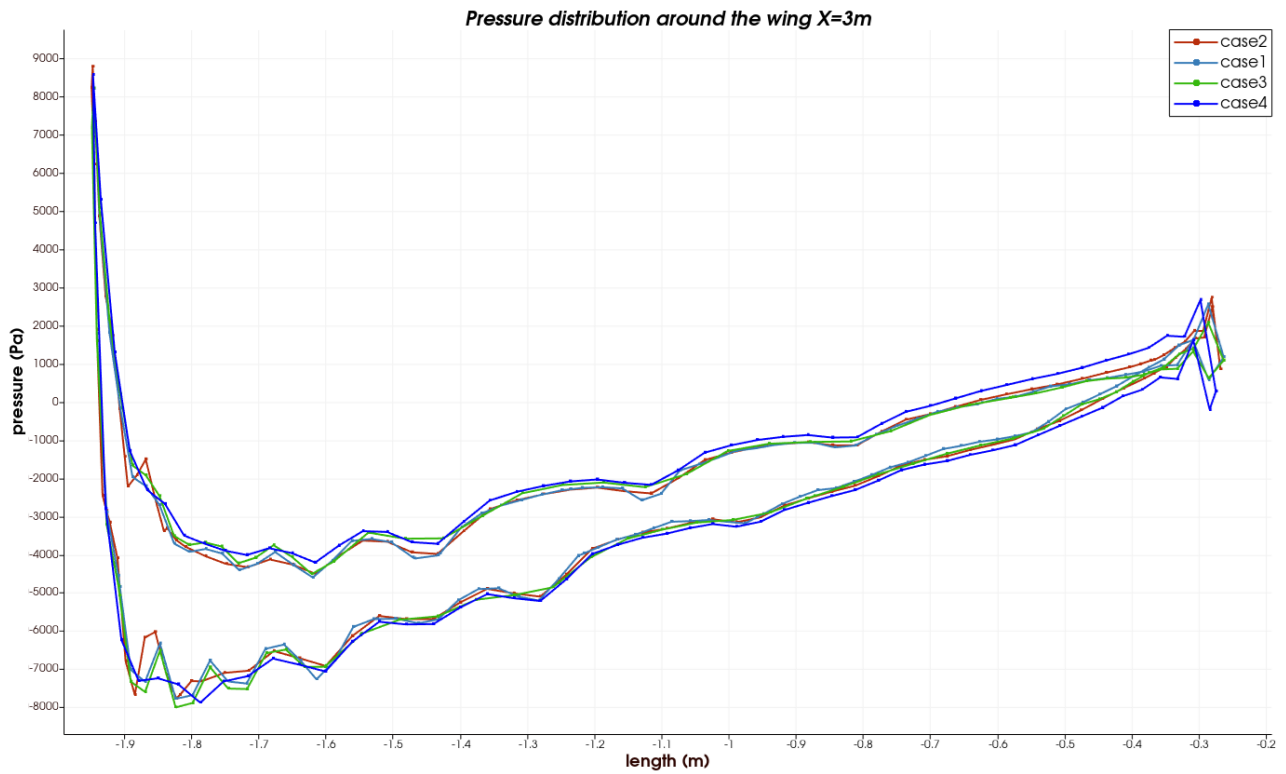


Figure III.17- Pressure distribution around the wing at Case 4



***Figure III.18-** Image to distribute press the four ways on the x axis = 3m.*

III.6- Streamlines

Current lines express curves in space that describe the movement of a flowing liquid and are characterized by the fact that at any moment they contain a tangent parallel to the velocity of the liquid particles at each point.

The figures below (Figure) illustrate the streamlines around the airplane
($x=0.27$; $y=-23.5$; $z=0.75$)

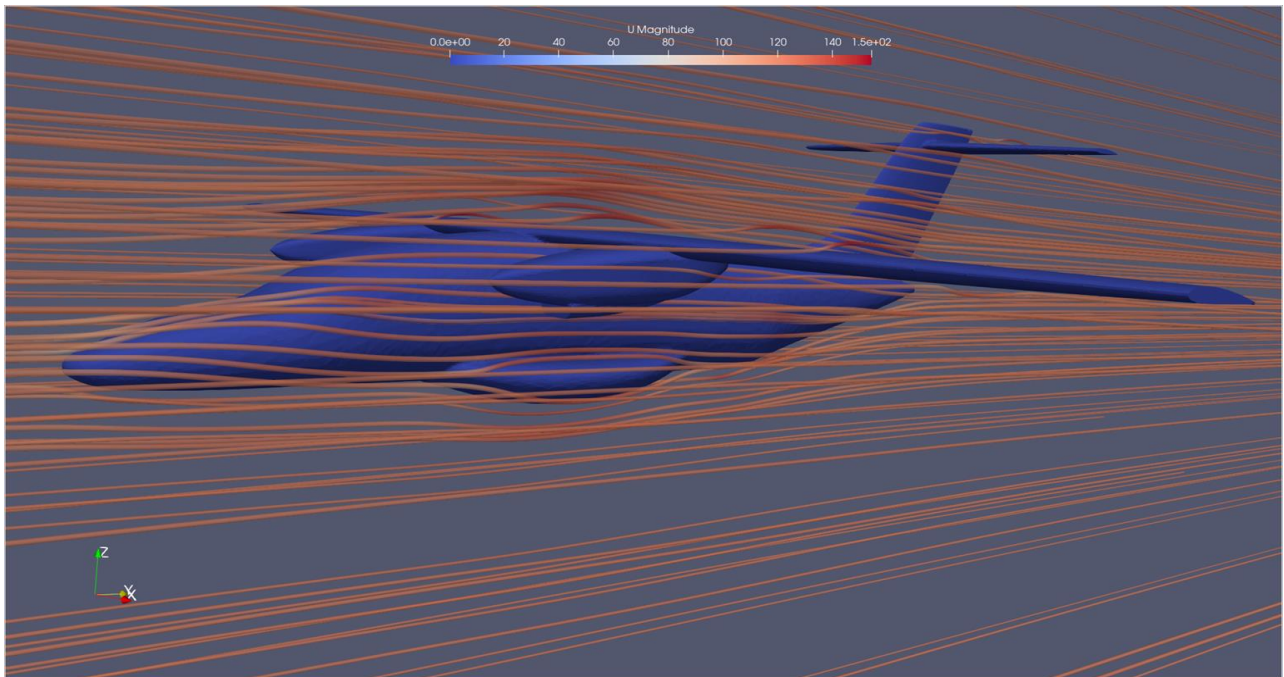


Figure III.19- courantlines around the aircraft.

Dans la figure suivante (Figure), les lignes de courant autour de l'aile sont représentées le long de l'axe x ($x=8.1$ / $y=-23.5$ / $z=0.95$).

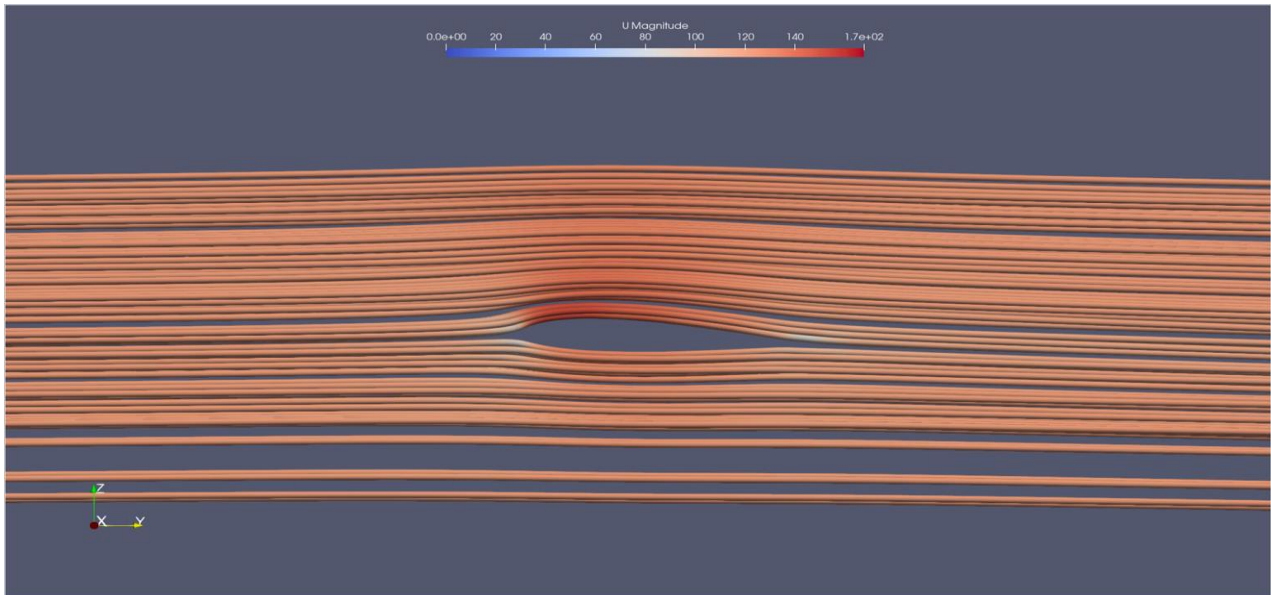


Figure III.20- Streamlines around the wing.

III.7- Aerodynamic forces

Our main goal is to determine the lift and drag resulting from the airflow around the aircraft. To achieve this, we employed the calculation filter in ParaView to compute these two aerodynamic forces.

In the next step, we will analyze the overhead area of the aircraft and the corresponding area and total space of the aircraft. After that, we will calculate and compare the corresponding forces for withdrawal and lift (Results shown in annexes 1)

To calculate aerodynamic transactions we have to do the next steps

Below:

1. The plane was imported.
2. Press the reflection filter, plans (x).
3. Press the cell selection filter, the entire plane surface
4. Press the extract selection filter.
5. Use the integrated variable filter to calculate the area.
6. Press the surface of the extract.
7. Support the natural surface generation.
8. Press the calculator to calculate strength.
9. Support integrated variables.
10. Press the calculator to calculate the coefficient.
11. Follow the same principle to calculate withdrawal

III.7.1- Lift

The figure shows how to calculate the lift force using ParaView

$$\text{Lift} = \text{normal } z * p * \rho$$

ρ : Fluid Density

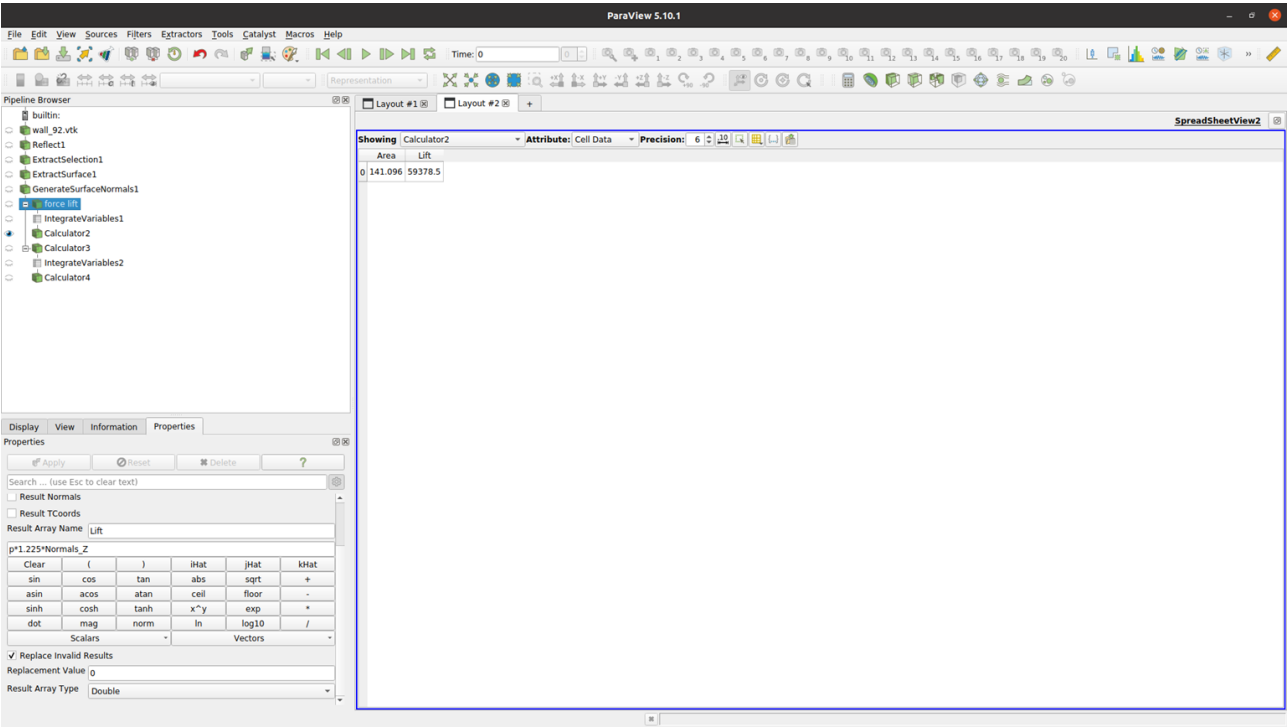
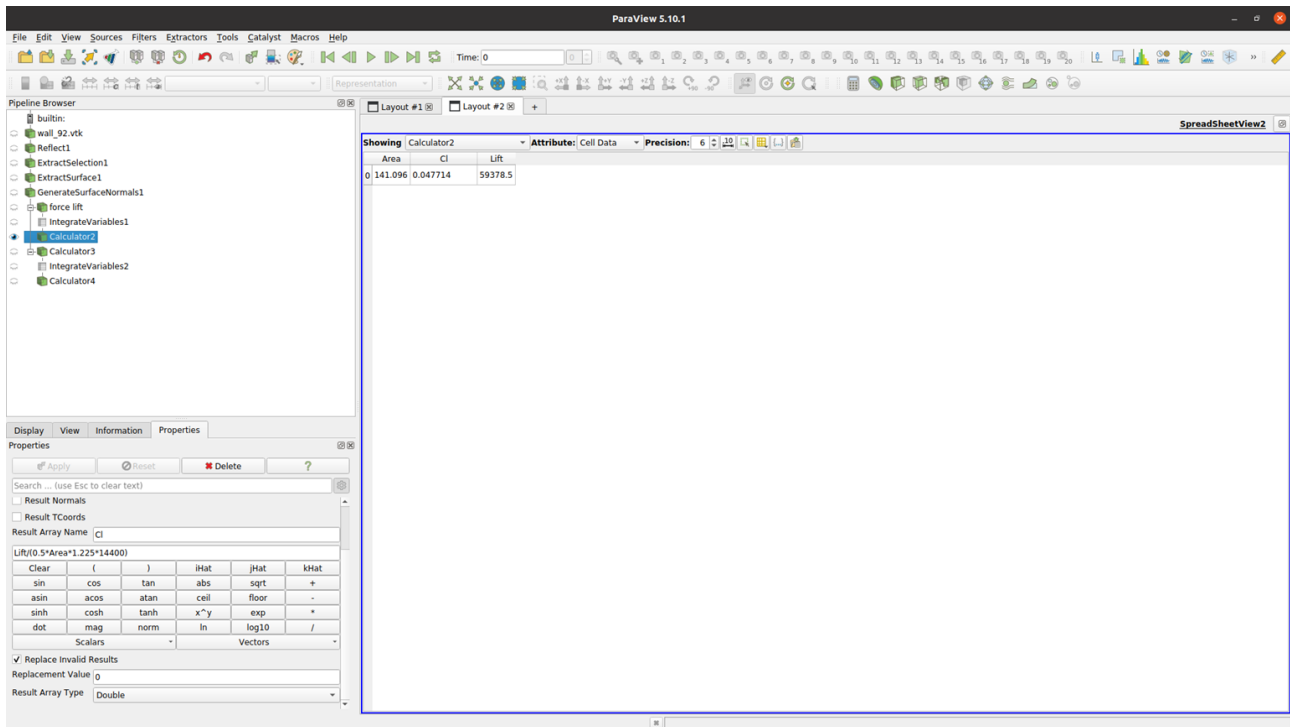


Figure III.21- Lift calculation in ParaView.

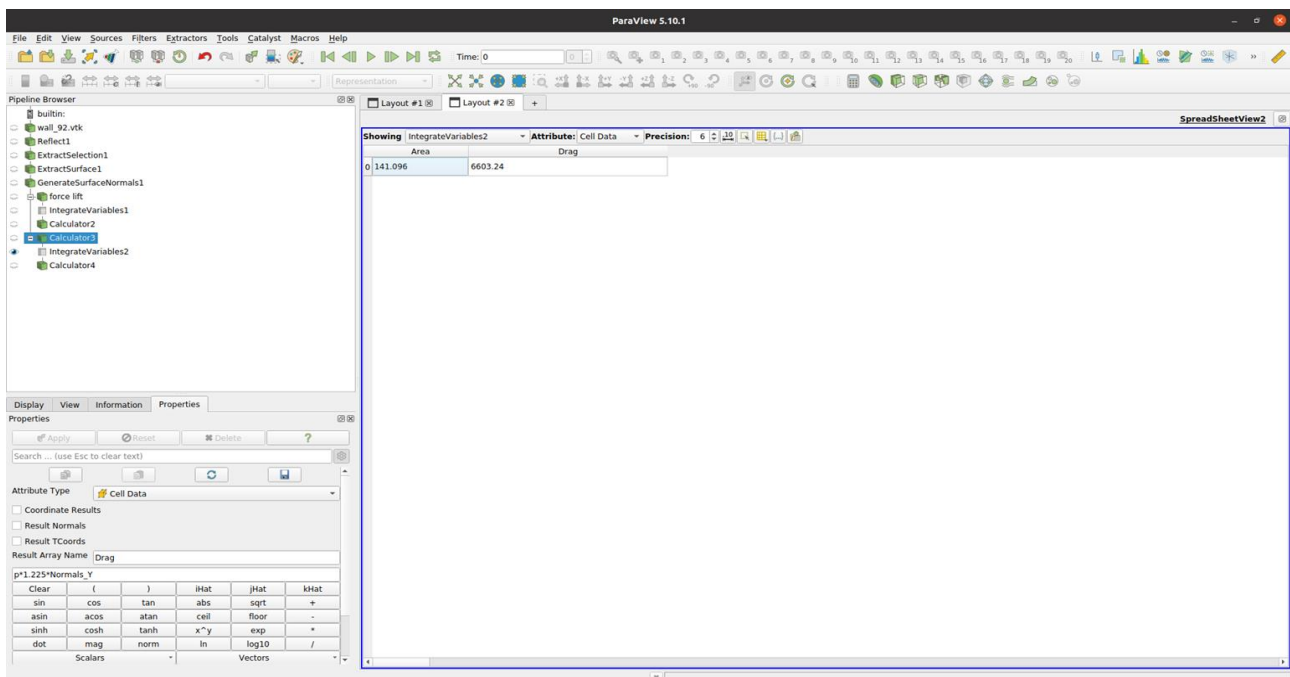
Lift coefficient

The figure shows how to calculate the lift coefficient with ParaView



III.7.2- Drag

Figure illustrates the procedure for calculating drag force using ParaView



Drag coefficient

The figure shows how to calculate the drag coefficient with ParaView.

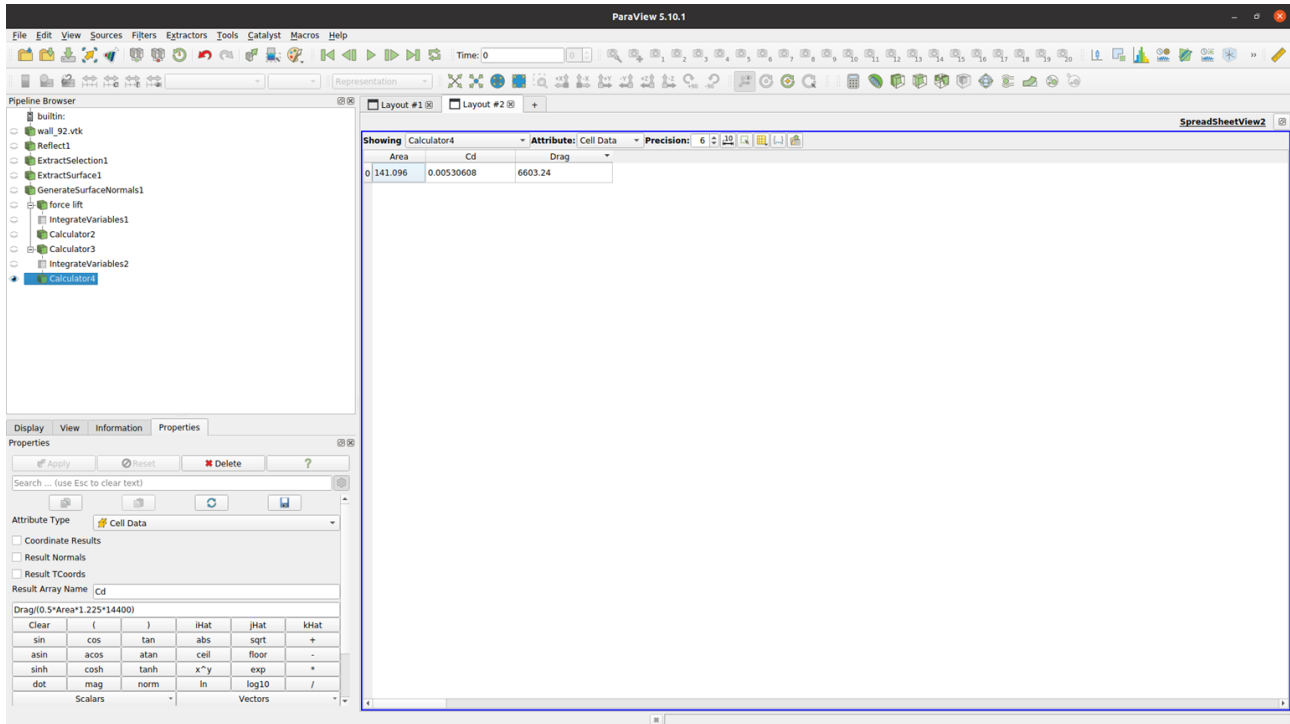


Figure III.24- Calculating the drag coefficient in ParaView.

The results of calculating the lifting and dragging force and their treatment are shown in the following table:

	Area (m)	Force Lift (N)	Force Drag (N)	C_L	C_D
Case1	141.096s	59378.5	6603.24	0.047	0.0053
Case2	141.504	58480.3	5922.82	0.046	0.0047
Case3	140.326	62206.2	7206.19	0.050	0.0058
Case4	140.604	60307.6	8344.68	0.048	0.0067

Note

In short, based on the experience observations of force yield from strikes on different surfaces, the results obtained from a full flat surface were found to closely match the reference forces. This indicates that force measurements on the full level surface were consistent and reliable compared to other surfaces tested. However, more specific information about the experiment may be useful for a comprehensive assessment.

III.6 Conclusion

High-quality images obtained with the free, open-source visualization tool ParaView confirm the accuracy of the results, in line with the physical reality of the phenomenon.

General conclusion and perspectives

During this experiment, we understood the difficulty and importance of numerical simulation stages and their significant role in technological advancement. The aircraft design phase is challenging, complex, and crucial. However, the meshing phase cannot be neglected, as it plays a vital role in the simulation process. The most challenging stage we encountered was adapting the mesh and attempting to develop a method to improve it to suit our complex engineering and limited computer resources. It took us a considerable amount of time and experimentation. A good mesh provides better and more realistic results within a reasonable analysis time.

Through the stages we've been through, I've learned to use the unique freeCAD software with a different workshop, including draft and cfdof workshop that provides a cfmesh interface to me to create a network and openFoam to analyze and learn to use paraview and the wonderful features it offers.

As I am experimenting with the Linux ubuntu operating system, it was great with the gnome desktop, easy to use and customize the user interface, and we also learned to use Linux terminal commands, manage files, and use free open source software.

With the time we have spent studying network technologies and trying to establish a way to improve the network, my future perspectives for this work are as follows:

- Optimize the mesh further for this complex geometry.
- Carrying out a simulation process with the presence of the motor fan.
- Developing and improving the engineering and design of an aircraft, conducting simulations and studying the distribution of aerodynamic forces.
- Application of method No. 6 and No. 7 to improve the mesh of the tourist plane and perform simulation.

References

- [1] « aérodynamique mécanique du vol performances avion », cours théorique BIA, Édition 2013-2014 ,André ,paris.
- [2] H. Nouri and F. Ravelet, « Introduction à la simulation numérique des écoulements.
- [3] Les étapes d'une simulation CFD
- [4] CFD Open. Openfoam user guide.OpenFOAM Foundation.
- [5] FREECAD USER GUIDE site web <https://www.freecadweb.org/wiki/workbenches/fr>.
- [6] DR,FRANJO JURETIC managing direttore and founding partner CFMesh USER GUIDE
- [7] Hybrid mesh for PML - FEniCS Project
- [8] mechanical.quora.com
- [9] SITE WEB <https://github.com/jaheyns/CfdOF>
- [10] LAALAOUI, KENZA (2016), "*Amélioration de l'écoulement autour d'une aile d'avion par l'utilisation de winglets de type trapézoïdale*"Département de mécanique, Université Batna 2.
- [11] DERDOUKH, Meniar (2021), "*Simulation de l'écoulement autour d'un avion touristique avec FreeCAD*".Département de mécanique, Université Batna 2.
- [12] CHERGUI, Aymen & BENHIZIA, Halim (2022), "*Etude de l'écoulement dans un coude circulaire avec OpenFOAM*"Département de mécanique, Université Batna 2.
- [13] LAOUICHI, Sif Alhak (2016), "*Simulation numérique bidimensionnelle autour d'un modèle de véhicule*"Département de mécanique, Université Batna 2.
- [14] ZEKKOUR, Ahmed (2016), "*Influence des différentes formes de winglets sur l'écoulement autour d'une aile d'avion*".Département de mécanique, Université Batna 2.
- [15] «Adaptive Meshing for Finite Element Analysis of Heterogeneous Materials»
Y.H. You, X.Y. Kou*, S.T. Tan Department of Mechanical Engineering, The University of Hong Kong, Pokfulam Road, Hong Kong
- [16] SITE WEB [tps://www.formula1-hdictionary.net/drag.html](https://www.formula1-hdictionary.net/drag.html)
- [17] SITE WEB <https://gmsh.geuz.narkive.com/SligD0Kv/hexahedral-mesh-inside-a-sphere>
- [18] wikipedia https://en.wikipedia.org/wiki/Delaunay_triangulation
- [19] BONGAGNA ONDZET, Mely Grace Delange (2022), "*Caractérisation de l'écoulement à l'entrée d'une conduite avec FreeCAD*".Département de mécanique, Université Batna 2.

Annexes

Annexe 1

In the next step, we will analyze the overhead area of the aircraft and the corresponding area and total space of the aircraft. After that, we will calculate and compare the corresponding forces for withdrawal and lift

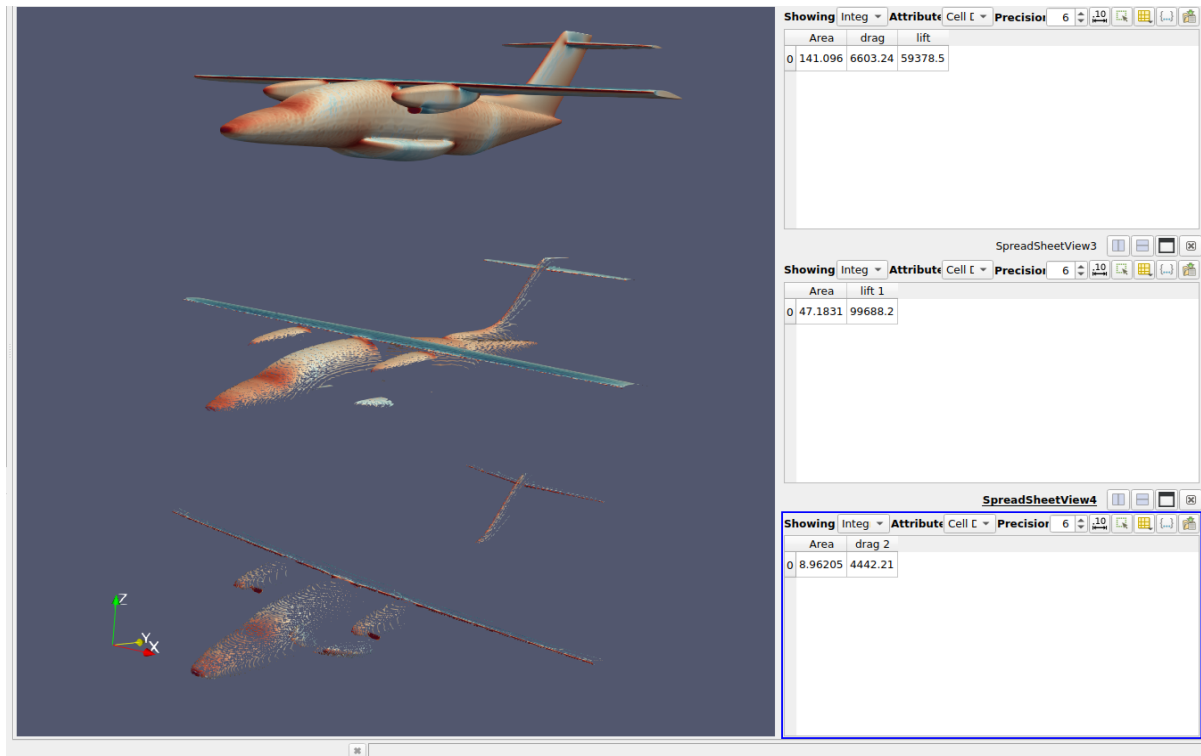


Figure A1.1-An image showing the spaces used in the calculation.

We compare the results of the calculation of forces in the following table:

	upper		frontal	
	Area	lift	Area	drag
R _{reference}	38.2953	72332.8	12.1252	47630.06
Case1	47.1831	99688.2	8.96205	4442.21
Case2	41.8555	30226.6	10.517	697.735
Case3	43.3855	20992.1	6.84449	818.449
Case4	38.3626	15823.8	9.90423	795.355

Annexe 2

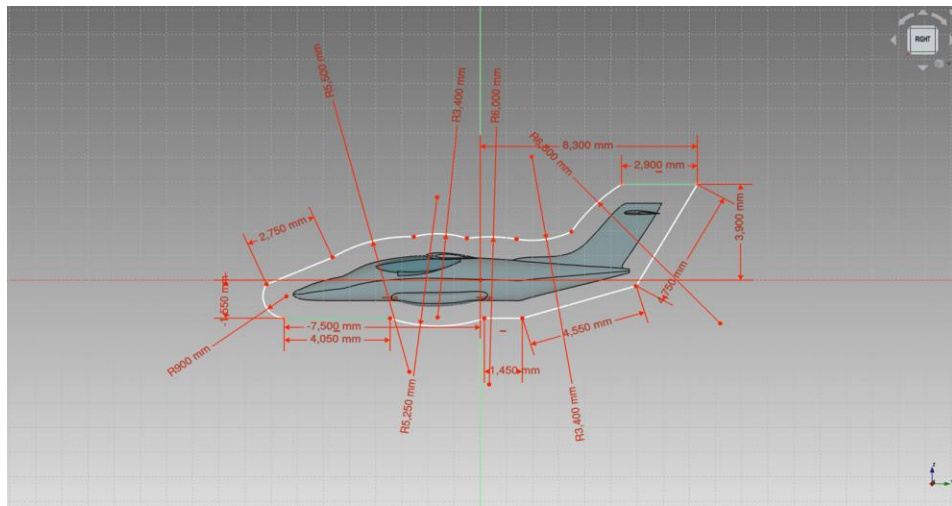


Figure All.1-Sketch about a plane.

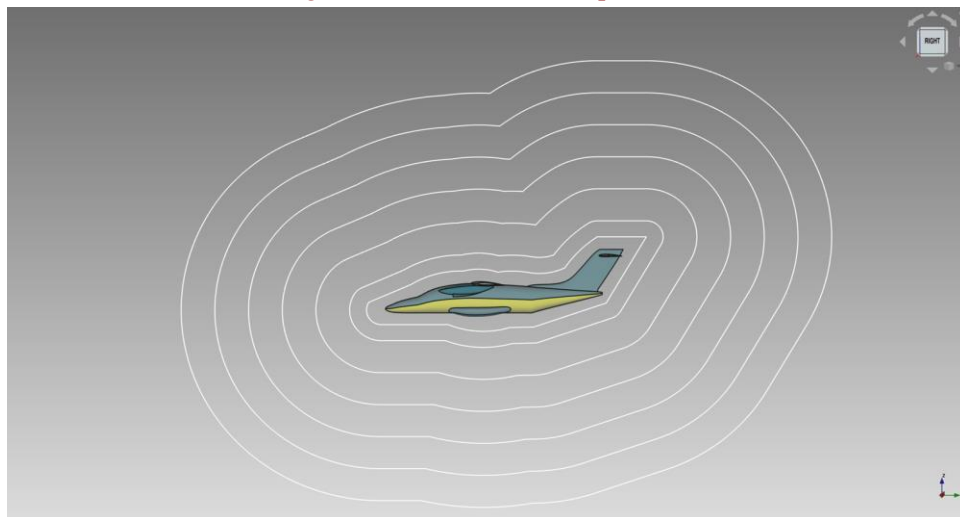


Figure AII.2-Use offset feature on drawing.

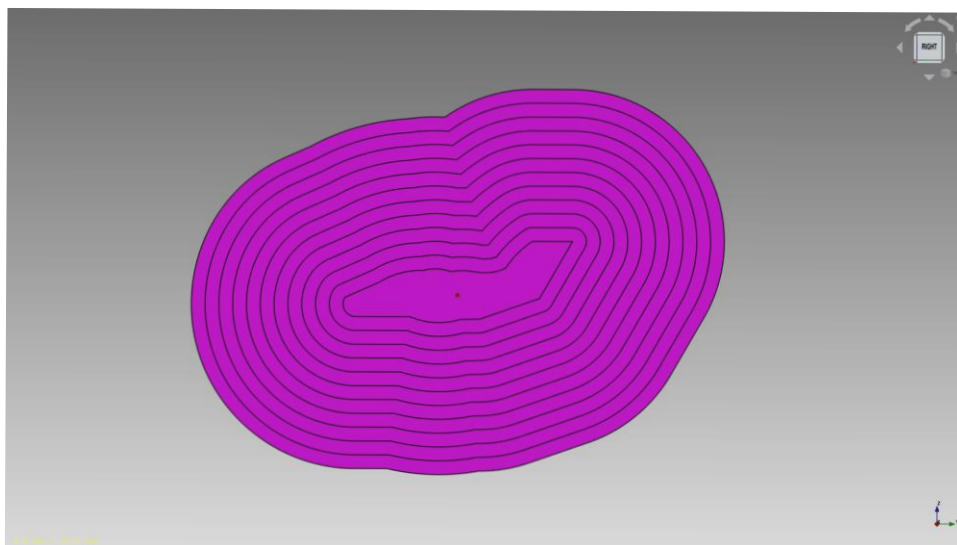


Figure AII.3-Extrusion for graded offset layers.

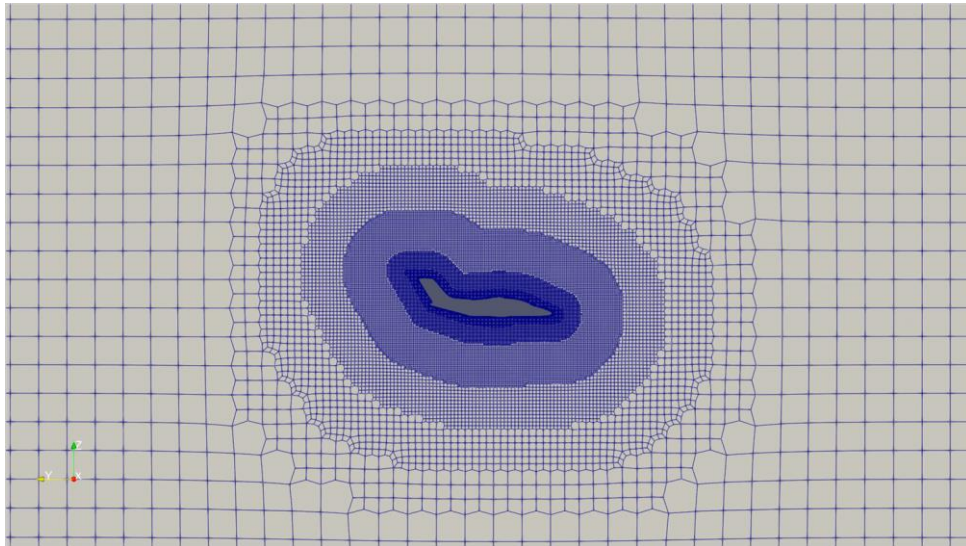


Figure AII.4-Side section of grid results

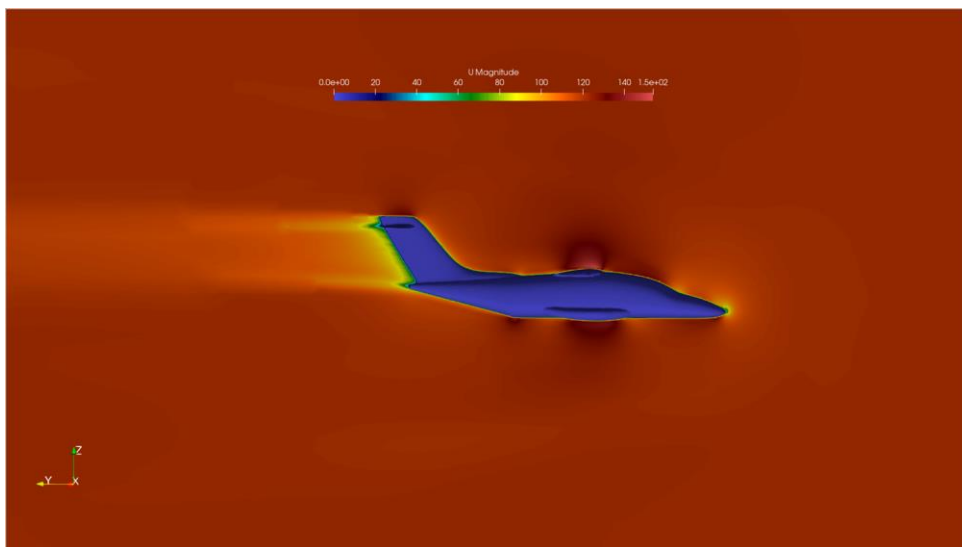


Figure AII.5-Side section of simulation results.