U.PORTO

FEUP FACULDADE DE ENGENHARIA
UNIVERSIDADE DO PORTO

Aerodynamics and biomimetics analyses of ash seed aiming wind turbines blades application

FELIPE WESTERMANN LOPES CASTILHO

Submission of proposal for master's dissertation

Master's in Computational Mechanics - Fluids

Tutor: Behzad Vasheghani Farahani

Co-tutor: Marco Parente

Coordinator: Jose Machado

Porto, July of 2024

Master's Degree in Computational Mechanics 2023/2024

MECHANICAL ENGINEERING DEPARTMENT

Phone: +351 22 0413474

Edited by

FACULTY OF ENGINEERING OF THE UNIVERSITY OF PORTO

Rua Dr. Roberto Frias 4200-465 PORTO

Portugal

Tel. +351-22-508 1400

Fax +351-22-508 1440

feup@fe.up.pt

http://www.fe.up.pt

ACKNOWLEDGMENT

I am grateful to all those who have supported me throughout the journey of completing this master's thesis.

First and foremost, I extend my appreciation to my parents, Elvira and Sergio, who’s supported and provided me with the resources that were fundamental in my life and academic path. I am also thankful to my sister, Tayane, for her example as an Engineer and for her straightforward advices.

I am grateful to Dr. Behzad Farahani and Marco Parente, my advisors, for his kindness, patience and support, always available when I needed any assistance. And special thanks also to Jose Machado, the coordinator and Professor Paulo Valente for their time and for being open to discuss with me more details about the thesis.

Finally, I am thankful to all my friends and colleagues who have offered their encouragement and assistance along the way. Your support has been invaluable, and I am deeply grateful for your contributions to my academic journey.

To all those mentioned and to those who may not be named here, please accept my gratitude for your support and encouragement throughout this journey.

ABSTRACT

This thesis seeks to investigate the aerodynamics and biomechanics of ash seeds and their potential application in small-scale wind turbines. Maple seeds naturally exhibit a spinning motion during free fall, likely as an evolutionary adaptation for effective dispersal, what could be indicated as a natural selection as proposed by Charles Darwin's theory of evolution. This inherent characteristic, influenced by the seed's shape and weight, underscores the significance of studying natural design and properties for technological innovation, was studied in this thesis using concepts known as biomimicry. The objective of this research is to draw insights from nature to develop a more efficient and sustainable wind blade design.

Given the intricate nature of the computations required for this investigation, the utilization of computational fluid dynamics (CFD) through Ansys Fluent software is proposed to achieve a precise and comprehensive analysis. Flow types, as well as the Newton and Navier-Stokes equations, have been examined. Additionally, detailed discussions on mesh design and its implications for computational simulations have been conducted.

Keywords: Aerodynamics, ash seeds, biomimicry, wind turbines, efficient and CFD analyses.

GENERAL INDEX

Contents

INDEX OF FIGURES

1. INTRODUCTION

1.1 GENERAL ASPECTS

Allied to the development of new forms, shapes and designs related to wind blades and consequently to energy production, with a focus on more efficient systems and with better architected devices, biomimetic engineering is a field that can help engineers and designers in this process of studying and manufacturer new tools and potentially search new energy production forms.

The diversity of solutions in this project is visible. Due to the different types of fly seeds that exist and considering that each of these plants has adapted to a specific region (local environment), it's possible to expect divergent solutions and consequently new perspectives to embroider the theme. Subject to design constraints associated with atmospheric conditions (force of the wind, humidity, and others whether conditions), there are several types of devices that can be designed to achieve a solution that covers all constraints.

In general, the main interest is observing the aerodynamics and mass distribution of ash seeds. Their format, which allows a better dispersion (moving away from the mother tree) draws attention for its simplicity and efficiency. Associate this natural tendency of rotation when they are falling from the trees with vertical wind turbines or other applications could be considered a challenge.

Studying the efficiency and applicability of many aspects related to the biomechanics of these seeds and then associating this information with a possible application for a series of tools for converting movement into energy, with a focus on wind blades, demands attention for a series of details that could be separated in 3 parts: ash seeds design, environmental conditions, and wind blades.

It should be noted that in recent years, the vibrations in wind turbines are a subject that have an important impact, when active for significant periods, trigger a dynamic response in the structure of the generators, promoting greater fatigue in the structure. Therefore, this project also aims at a study of vibrations, so that the entire system can be understood, and possible losses of efficiency caused by vibrations considered.

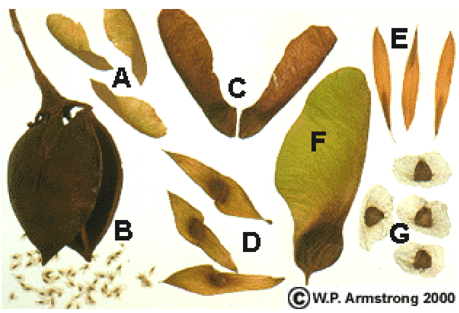


Figure 1. Different species of ash seeds (https://www2.palomar.edu/users/warmstrong/plfeb99.htm)

Helicopters seeds indicated in the Figure 1: A. Box Elder (Acer negundo, Aceraceae); C. Big-Leaf Maple (Acer macrophyllum, Aceraceae); E: Evergreen Ash (Fraxinus uhdei, Oleaceae); F. Tipu Tree (Tipuana tipu, Fabaceae). Flutterer/Spinners: B. Empress Tree (Paulownia tomentosa, Scrophulariaceae); D. Tree Of Heaven (Ailanthus altissima, Simaroubaceae); G. Jacaranda (Jacaranda mimosifolia, Bignoniaceae).

Taking into account the actual necessity to develop new sustainable and more efficient ways to produce energy, this thesis have as objective reply one question, how does it is possible to think creatively, associating with biomimetic concepts, to active this goal of sustainability in the fild of wind energy production?

1.2 WORK STRUCTURE

In engineering, the risks of a final design failure can be something too costly, so this work aims a systematic analysis involving case study, simulations of a computational model, a small-scale prototype design to validate results.

Regarding the case study, a bibliographic review addressing the main concepts of this work is what we have in mind. Analyzing the scientific publications associated with the topic and making possible parentheses, in addition to discussing what the professional environment demands is what we have in mind.

During the process of simulation analysis this project intend to apply a systematic verification and validation (V&V) to check if the process meets with required specifications and if it fulfills its intended purpose. The verification processes involve equating the fluid dynamic analyses (regarding a mathematical representation of the reality); validation process involves to checking whether the specification captures the project's requirements.

It had been created diagram with arrows, as indicated in the Figure 2, that helps to describe the processes of a fluent analysis simulation. Basically, involves a systematic verification of the simulations and how it could be optimized (refine the mesh as well as small adjustments in the simulation conditions).

Giving an overview about the diagram, it was first necessary to establish the geometry that would be implemented in the project and that would come closer to the reality of a natural design, therefore the choice of geometry associated with what was in mind to establish the boundary condition parameters was fundamental.

Later, considering the need to treat the 3D model, regardless of simplification and adaptation to the Ansys software, so that the simulations could take place, the mesh was implemented and readapted. Parameters such as speed, pressure and type of regime (laminar, turbulent, etc.) were also constantly adjusted in order to obtain results that better match reality.

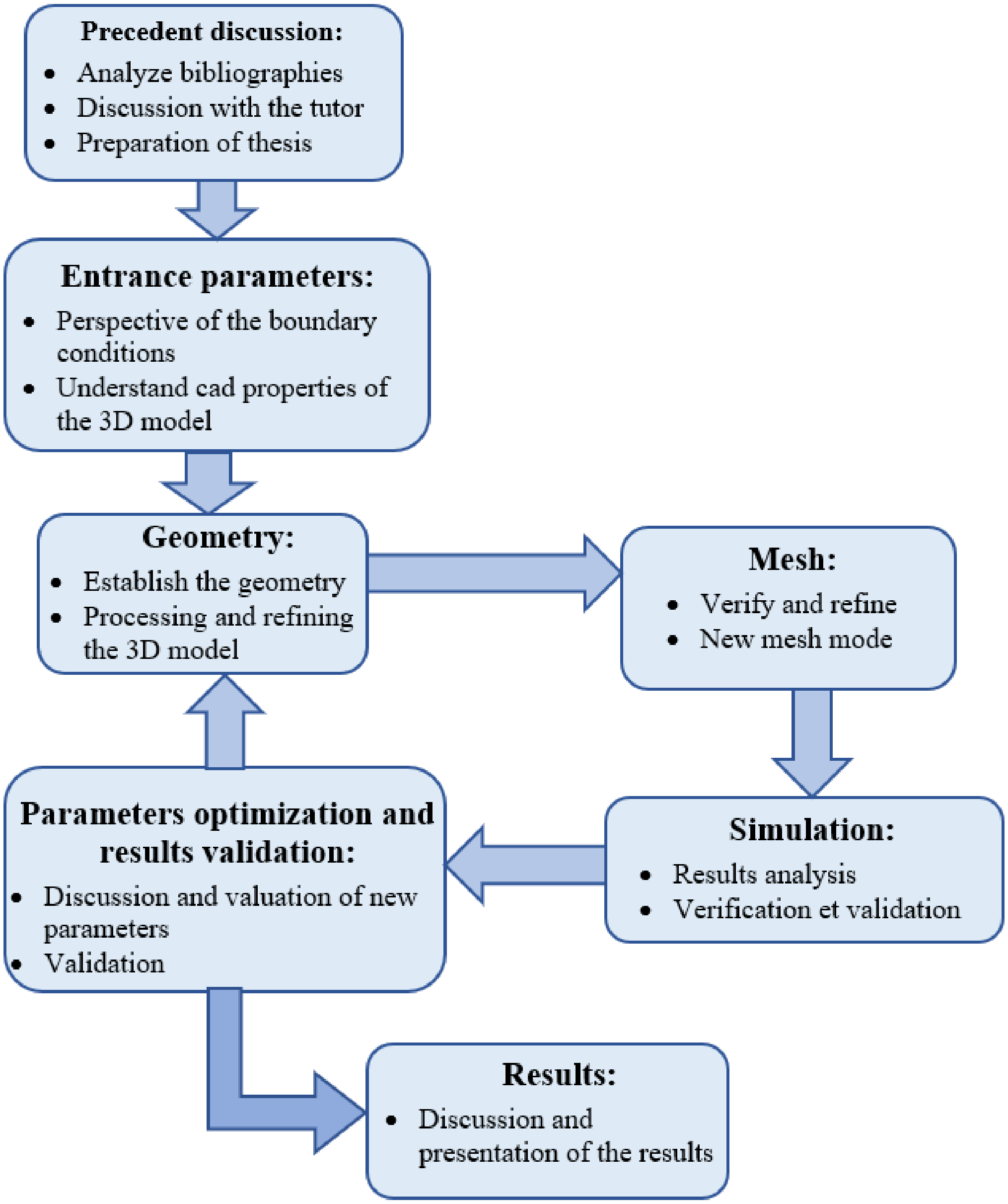


Figure 2. Arrow diagram used as reference for the fluid dynamic simulation in ANSYS software

2 FLUID ANALYSIS, BIOMIMETIC ENGINEERING AND WIND TURBINES

2.1 FLUID ANALYSIS

To perform simulations related to the aerodynamic behavior of winged seeds, this thesis aims to use the principles of fluid dynamic analysis to study the case. CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, hypersonics, weather simulation, natural science and environmental engineering, industrial system design and analysis, biological engineering etc.

The project aims to analyze and discretize the biological structure of the ash-seed and therefore realize simulations using the software Ansys. ANSYS Fluent is a Fluid Simulation software that uses the principles of Computational Fluid Dynamics to solve problems or to simulate mathematical models using the chemical engineering principles like: fluid mechanics, thermodynamics, heat transfer, mass transfer etc.

For all flows, ANSYS FLUENT solves conservation equations for mass and momentum. For flows involving heat transfer or compressibility, an additional equation for energy conservation is solved. ANSYS FLUENT is very popular and widespread software in the field of Computational Fluid Dynamics (CFD). The finite-volume code is primarily purposed for numerical modeling of fluid flow, i.e. solving the Navier-Stokes equations and continuity equation.

Continuity equation for incompressible flow, mass conservation is:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \tag{1}$$

The momentum equations in the x, y, and z directions are:

X-momentum equation:

$$\rho\left(\frac{\partial \mathrm{u}}{\partial t} + u\frac{\partial \mathrm{u}}{\partial x} + v\frac{\partial \mathrm{u}}{\partial y} + w\frac{\partial \mathrm{u}}{\partial z}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) + Fx \tag{2}$$

Y-momentum equation:

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) + Fy \tag{3}$$

Z-momentum equation:

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) + Fz \tag{4}$$

Where:
ρ is the fluid density.
u, v, w are the velocity components in the x, y, and z directions, respectively.
t is time.
p is the pressure.
μ is the dynamic viscosity.

Fx, Fy and Fz are the body force components per unit volume in the x, y, and z directions, respectively (e.g., gravitational forces).

The left-hand side of each momentum equation represents the change in momentum due to unsteady effects and convection. The first term on the right-hand side is the pressure gradient, which drives the fluid flow. The second term on the right-hand side represents the viscous diffusion, accounting for internal friction within the fluid. The last term includes any external body forces acting on the fluid, such as gravity.

The momentum conservation equations presented are directly derived from the Navier-Stokes equations. The Navier-Stokes equations are the fundamental equations governing fluid motion and encapsulate the principles of momentum conservation in fluid dynamics

The Navier-Stokes equations in their general form for incompressible flow are:

$$\rho\left(\frac{\partial \mathrm{u}}{\partial t} + \mathrm{u} \cdot \nabla \mathrm{u}\right) = -\nabla \mathrm{p} + \mu \nabla^2 \mathrm{u} + f \quad (5)$$

Where:
∇ is the gradient operator.
∇^2 is the Laplace operator.
f represents body forces per unit volume, such as gravity.

For most practical applications, especially at high Reynolds numbers, the flow is turbulent, and modeling turbulence is essential. Fluent provides several turbulence models, such as: k-ε Model that is one of the most used models for general-purpose CFD simulations; k-ω Model, used for near-wall treatment and when more accuracy is required in the boundary layer.

The k-ε model is a two-equation model that provides a robust and economical approach for simulating turbulent flows. It includes two transport equations: one for the turbulence kinetic energy (k) and one for the turbulence dissipation rate (ε).

Turbulence Kinetic Energy (k) Equation:

$$\frac{\partial(\rho \mathrm{k})}{\partial t} + \nabla \cdot (\rho \mathrm{k} \mathrm{u}) = \nabla \cdot (\alpha \mathrm{k} \nabla \mathrm{k}) + \mathrm{Gk} + \mathrm{Gb} - \rho\epsilon - \mathrm{YM} + \mathrm{Sk} \quad (6)$$

Turbulence Dissipation Rate (ε) Equation:

$$\frac{\partial(\rho\epsilon)}{\partial t} + \nabla \cdot (\rho\epsilon \mathrm{u}) = \nabla \cdot (\alpha\varepsilon\nabla\varepsilon) + \mathrm{C1}\epsilon\frac{\epsilon}{k}(\mathrm{Gk} + \mathrm{C3}\epsilon\mathrm{Gb}) - \mathrm{C2}\epsilon\rho\frac{\epsilon^2}{k} + \mathrm{S}\epsilon \quad (7)$$

Terms and Coefficients
Gk: Generation of turbulence kinetic energy due to mean velocity gradients.
Gb: Generation of turbulence kinetic energy due to buoyancy.
YM: Contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate.
αk, $\alpha\epsilon$: Effective diffusivity of k and ε, respectively.
C1ϵ, C2ϵ, C3ϵ: Empirical constants.
Sk, Sϵ: Source terms.

The k-ω model is another popular two-equation turbulence model. It is known for its improved accuracy in the near-wall region compared to the k-ε model. This makes it suitable for boundary layer flows under adverse pressure gradients and for flows with separation.

Turbulence Kinetic Energy (k) Equation:

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho k u) = \nabla \cdot (\alpha k \nabla k) + Gk - Yk + Sk \quad (8)$$

Specific Dissipation Rate (ω) Equation:

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho w u) = \nabla \cdot (\alpha w \nabla w) + Gw - Yw + Dw + Sw \quad (9)$$

Dw: Cross-diffusion term.
Gω: Generation of ω.
Yk: Dissipation of turbulence kinetic energy.
Yω: Dissipation of ω.
αk**,** αω: Effective diffusivity of k and ω, respectively.
Sk**,** SωS_\omegaSω: Source terms.

Both the k-ε and ok-ω models are broadly used in CFD simulations due to their relative simplicity and effectiveness in shooting the crucial features of turbulent flows. The preference between them frequently depends on the particular flow traits and the want for accuracy specially areas, which include close to-wall regions wherein the k-ω version usually plays higher. In exercise, CFD engineers may additionally test a couple of turbulence models to identify which presents the first-rate balance of accuracy and computational efficiency for his or her specific utility.

Newton's Second Law states that the force performing on an object is equal to the mass of the object elevated via its acceleration. This fundamental principle is the premise for the equations governing fluid movement, inclusive of the Navier-Stokes equations and the turbulence fashions utilized in CFD simulations.

$$F = ma \quad (10)$$

Where:
F is the force vector.
m is the mass of the object.
a is the acceleration vector.

In resume, Newton's Second Law is fundamental to fluid dynamics, imparting the underlying standards from which the Navier-Stokes equations and turbulence models like k-ε and ok-ω are derived. These models allow the accurate simulation of complicated fluid flows in CFD software program like ANSYS Fluent

2.2 BIOMIMETIC ENGINEERING

The humanity has always copied nature, in simple and sophisticated things, like airplanes are, in some aspects, the copy of aerodynamic of birds; Velcro that had been based in seeds that has physical structures that allows the seeds to etched in the some surfaces and can be spread; bats and their sonars. Another application of biomimetics was the development of more aerodynamic boat hulls from the study of dolphins. The analysis of how wasps pierce wood with minimal effort inspired the development of a new medical probe.

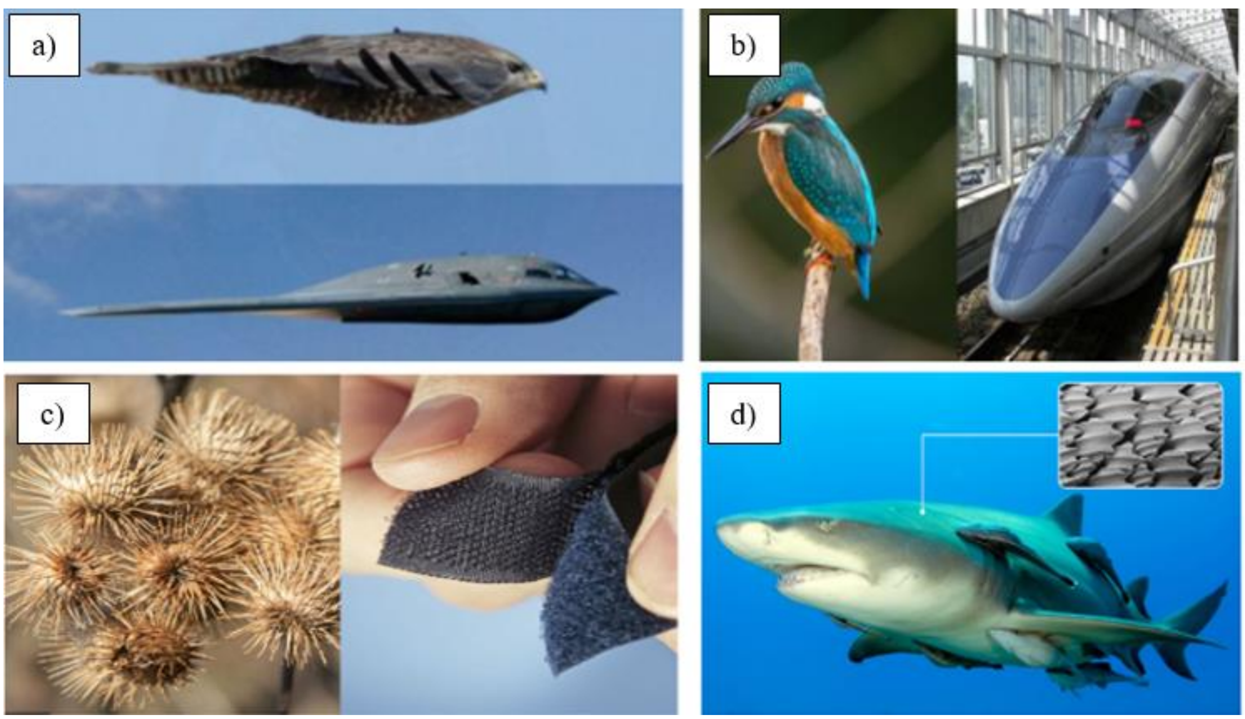


Figure 3 Biomimicry engineering exemples: a) Falcon - Stealth B2 Bomber (USA);b) Kingfisher – SHINKANSEN Hight speed train (Japan); c) Bardane – The Velcro; d) Shark skin – Antibacterial (Sharklet tech). (Source: https://mibellebiochemistry.com/fr/node/207)

Therefore, biomimicry is the science that studies the models of nature. It is considered that the various changes along the Earth's evolution allowed the development of advanced systems. These should be used as inspiration or even imitated to solve current human problems.

Janine Benyus, an American biologist and founder of the Biomimicry Institute, focused on spreading the techniques of biomimetics, was responsible for popularizing this concept in 1997. The novelty of Janine was to propose this inspiration as a working method, a systematic approach to solving problems. In this way, biomimicry encourages nature-based innovation and, as a result, becomes an important ally in creating a more sustainable future. So, it is a way of seeing, valuing and respecting nature and also presenting solutions that combine raw material and energy savings.

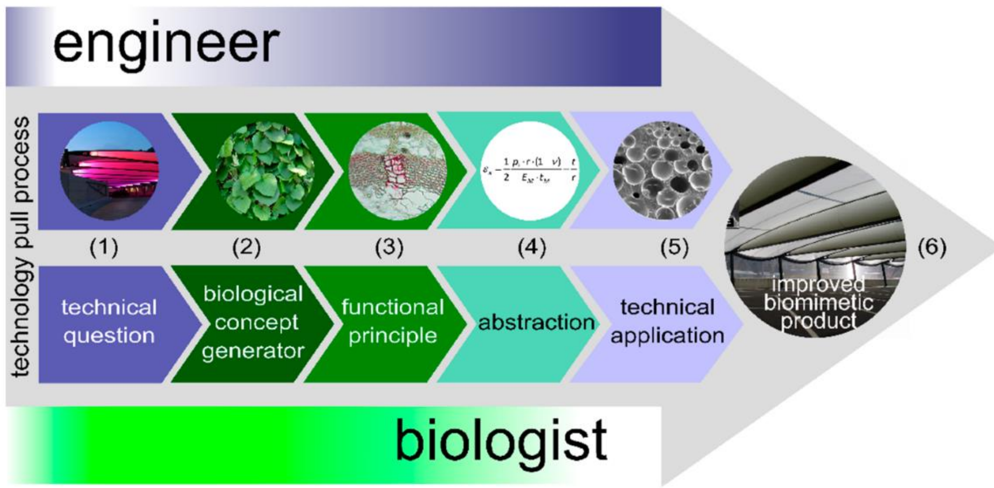


Figure 4. Process Sequences In Biomimetic Research (Speck, O.; Speck, T. An Overview of Bioinspired and Biomimetic Self-Repairing Materials. Biomimetics 2019, 4, 26. https://doi.org/10.3390/biomimetics4010026)

2.3 WIND TURBINES

The wind causes the blades to spin and releases kinetic energy. This causes the shaft in the nacelle to turn as well, and a generator inside the nacelle transforms the kinetic energy of the rotating blades into electrical energy.

Magnets spinning inside a coil of conducting wire can provide electrical energy. How to accomplish such rotation is the crucial question.

When fossil fuels like coal, gasoline, and oil are burned to warmth water, they devise high strain steam which could energy a turbine, which in turn powers an electrical generator. Unfortunately, this additionally emits negative emissions like carbon dioxide and relies on confined sources that have to constantly be dug up from the earth and carried to the floor.

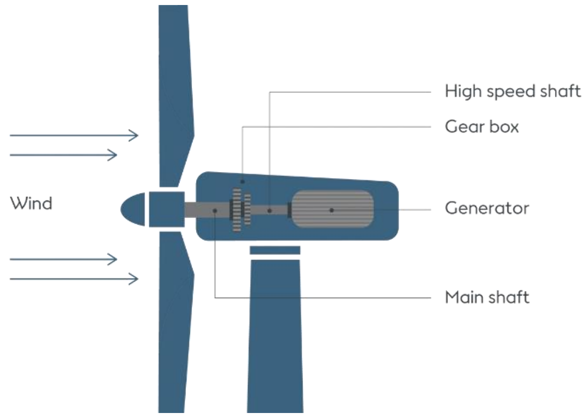


Figure 5. How does wind turbine work (Source: https://us.orsted.com/renewable-energy-solutions/offshore-wind/what-is-offshore-wind-power/how-do-offshore-wind-turbines-work)

2.3.1 WIND BLADES AND AIR FLOW

In essence, drag is caused by air rubbing up against the blade surface. In the same direction as the air flow across the blade surface, drag is perpendicular to lift. However, by bending or twisting the blade and also tapering it along its length, we may lessen this drag-force and create the most effective wind turbine blade design.

The angle between the direction of the oncoming wind and the pitch of the blade with respect to the oncoming wind is called the "angle of attack". As this angle of attack becomes larger, more lift is created but as the angle becomes even larger, greater than about 20^o, the blade will begin to decrease lift.

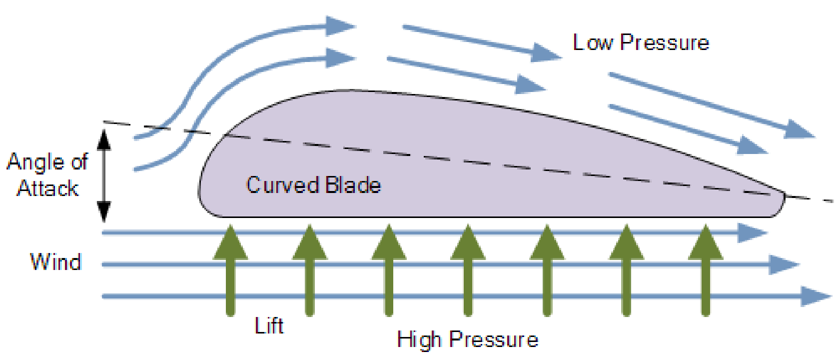


Figure 6. Curved Blade Air Flow (Source: https://www.alternative-energy-tutorials.com/wind-energy/wind-turbine-blade-design.html, October 2022)

In summary, the length of a wind turbine's rotor blades dictates how much wind power can be harvested as they revolve around a central hub, and the performance of wind turbine blades in terms of aerodynamics varies greatly depending on whether they are flat or curved. Flat blades are inexpensive and simple to build, but because of their high drag, they are slow and ineffective.

2.3.2 SUMMARY OF OBJECTIVES

The main objective of this work lies in the interpretation regarding the influence of wind and pressure on the shape of the ash seed and present the concepts that involve the biomimicry engineering regarding to it.

The Figure 6 represents a perspective of what this thesis aims to study. Analise the vertical and horizontal path and study the forces under de seed that's promotes their rotation.

The flow structures show that both lead and edge vortices contribute to the elevator generation and longitudinal rotation. As a result, the aerodynamics increase in lift force, implying that ash seeds with high aspect ratio wing may evolve differently in using fluid mechanisms to facilitate dispersion.

3 METHODOLOGY

The commercial finite element analysis (FEA) program ANSYS Workbench was used to run the simulations. A user-friendly interface for modeling, analyzing, and optimizing complex systems in a variety of fields, such as structural mechanics, fluid dynamics, heat transport, and electromagnetics, is offered by ANSYS Workbench, a comprehensive engineering simulation platform. The software is a good fit for carrying out the suggested simulations, including Computational Fluid Dynamics (CFD) studies using ANSYS Fluent, due to its strong capabilities and extensive library of physics-based solvers.

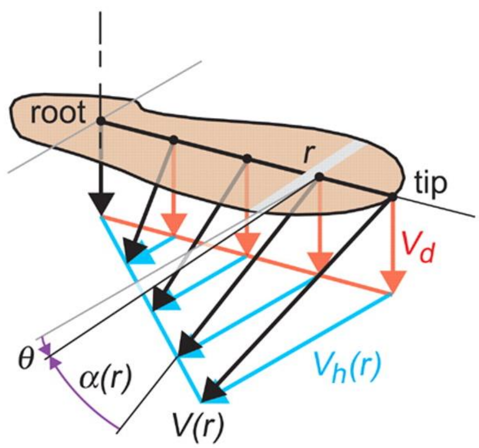


Figure 7. Kinematics and morphology of autorotating seeds (Source:https://mappingignorance.org/2013/05/15/autorotating-seeds-to-fly-or-to-die/)

ANSYS Workbench offers an clean integration of CAD geometry, meshing, boundary situation placing, material traits, and solver choice. This integrated approach reduces the opportunity of mistakes at some point of information switch among numerous software programs and streamlines the simulation technique. Furthermore, ANSYS Workbench offers an adaptable and configurable platform that lets in the addition of cloth models, optimization strategies, and person-defined subroutines as wanted.

Complex geometries can be discretized into high-quality finite elements using the software's sophisticated meshing methods, which guarantee correct results while preserving computational efficiency. Tetrahedral, hexahedral, and hybrid elements are just a few of the many element types that ANSYS Workbench provides. These components could be customized to meet the needs of the given situation. Moreover, the program has adaptive meshing capabilities that enable mesh refining automatically in areas of significant gradients or interest, enhancing simulation accuracy.

For Computational Fluid Dynamics (CFD) analyses, ANSYS Fluent, a aspect of the ANSYS suite, turned into applied in the ANSYS Workbench surroundings. ANSYS Fluent is a CFD solver that allows the simulation of fluid waft, warmth transfer, and related phenomena in complicated geometries. The software program carries superior turbulence models, multiphase glide abilities, and combustion modeling, making it suitable for a huge range of fluid dynamics simulations.

ANSYS Workbench, in conjunction with ANSYS Fluent, provided a comprehensive set of tools for setting up the CFD simulations. By leveraging the combined capabilities of ANSYS Workbench and ANSYS Fluent, the simulations could be conducted with a degree of accuracy and reliability, enabling detailed analysis and optimization of fluid flow phenomena in the system under investigation.

3.1 GEOMETRY PREPARATION

The geometry preparation phase was a critical step in computational fluid dynamics (CFD) simulations. After an analyze of different 3D cad modes of seeds that could better represent the real world, one had been chosen (website where the 3D cad mode had been imported: https://cults3d.com/en/3d-model/various/maple-seed), cleaning, and simplifying geometric models for subsequent meshing and analysis. Initially had been imported the 3D model of ash seeds had been imported for the simulation using ANSYS software.

These model represented the morphology and surface characteristics of ash seed (maple seed), capturing the diversity observed in nature and a possible adaptation for future real turbine model. The models were acquired in standard CAD file formats and were imported into the ANSYS environment for further processing.

Upon importing the 3D models, preliminary checks were conducted to assess the quality and integrity of the geometry. Geometrical inconsistencies, such as gaps, overlaps, or self-intersections, were identified and addressed using Autodesk Meshmixer, software for working with triangle meshes, and ANSYS Design Modeler's geometry repair tools. This step ensured that the imported geometry was suitable for subsequent meshing operations without introducing artifacts or errors into the simulation domain.

Two of the imported 3D models were highly detailed and contained unnecessary features that could complicate mesh generation and increase computational overhead. To streamline the geometry and facilitate meshing, simplification techniques were applied to remove redundant details while preserving the overall shape and morphology of the ash seeds. Additionally, segmentation methods were employed to partition the geometry into distinct regions based on functional requirements or flow characteristics.

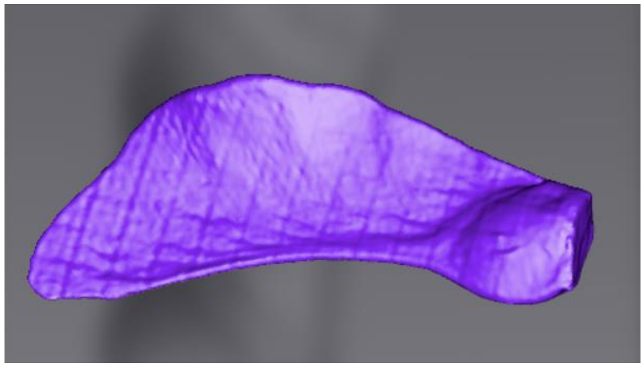


Figure 8. Ash seed 3D model 2 (Source: https://cults3d.com/en/3d-model/various/maple-seed)

3.1.1 MESHMIXER

Meshmixer software was employed to optimize the mesh geometries of the CAD models of ash seeds. This optimization process involved refining the triangular mesh to enhance the accuracy and efficiency of subsequent computational simulations.

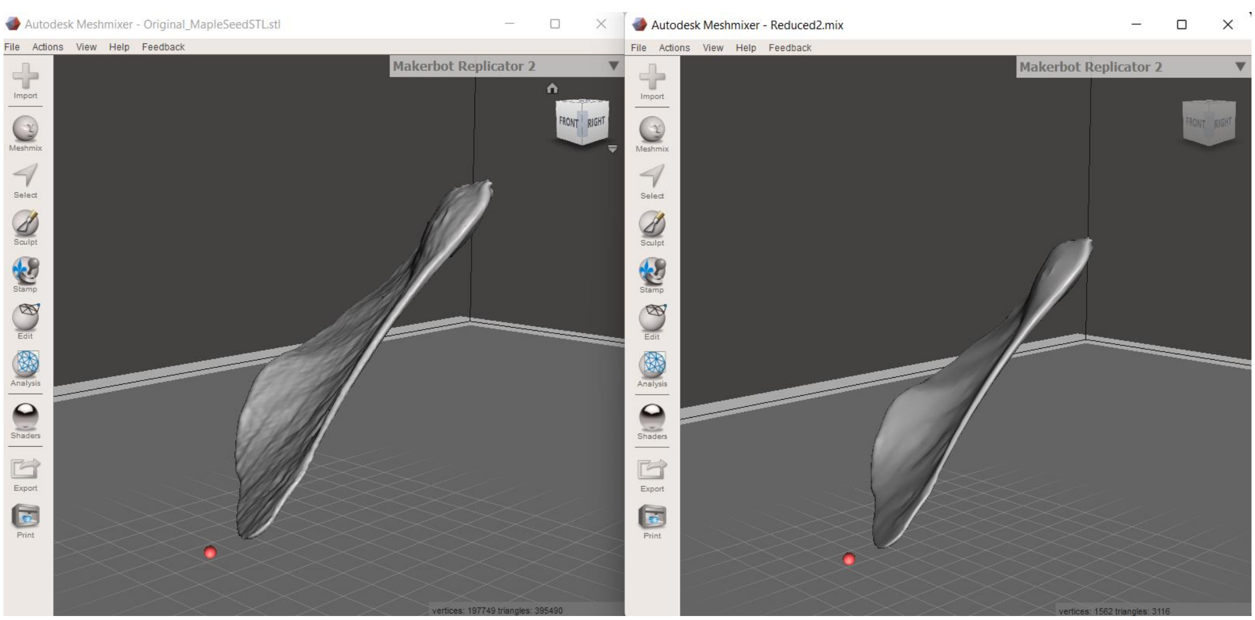


Figure 9. Before and after treating the 3D cade mode superficial mesh using Meshmixer

The optimization process involved selecting the entire 3D CAD model of the ash seed and subsequently reducing the mesh complexity. This was achieved by employing Meshmixer's mesh reduction tools what allows to maintain the model's essential geometric features while decreasing the number of triangles.

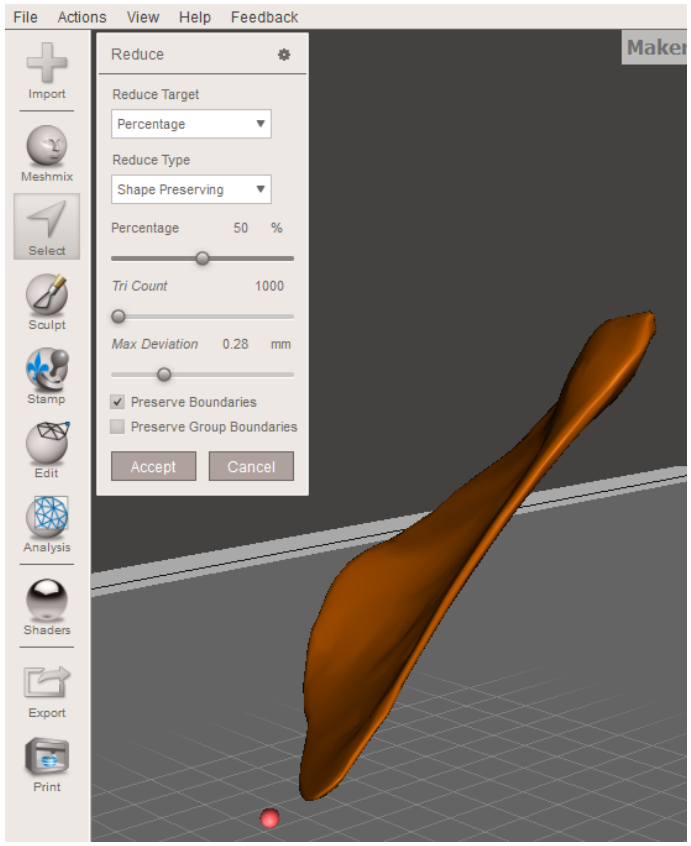


Figure 10. Editing the 3D cad mode applying the reduce feature.

3.1.2 EDIT GEOMETRY IN DESIGNMODELER

Following the mesh optimization, the 3D CAD model was further processed using ANSYS DesignModeler. This step ensured that any remaining gaps or defects were addressed and that the model was transformed into a solid geometry, suitable for subsequent computational fluid dynamics analysis.

To facilitate the computational fluid dynamics analysis, an enclosure was created around the 3D CAD model of the ash seed. This process involved the following steps:

Creation of the Enclosure: An outer boundary was defined around the ash seed model to simulate the fluid domain. This enclosure represents the region through which the fluid will flow around the model. Positioning the Model: The ash seed model was positioned within the center of the enclosure focusing to ensure uniform fluid interaction on all sides. Defining the Fluid Domain:

The space within the enclosure was designated as the fluid domain, where the fluid dynamics simulations would be conducted. This setup allows for accurate analysis of fluid flow behavior around the ash seed model.

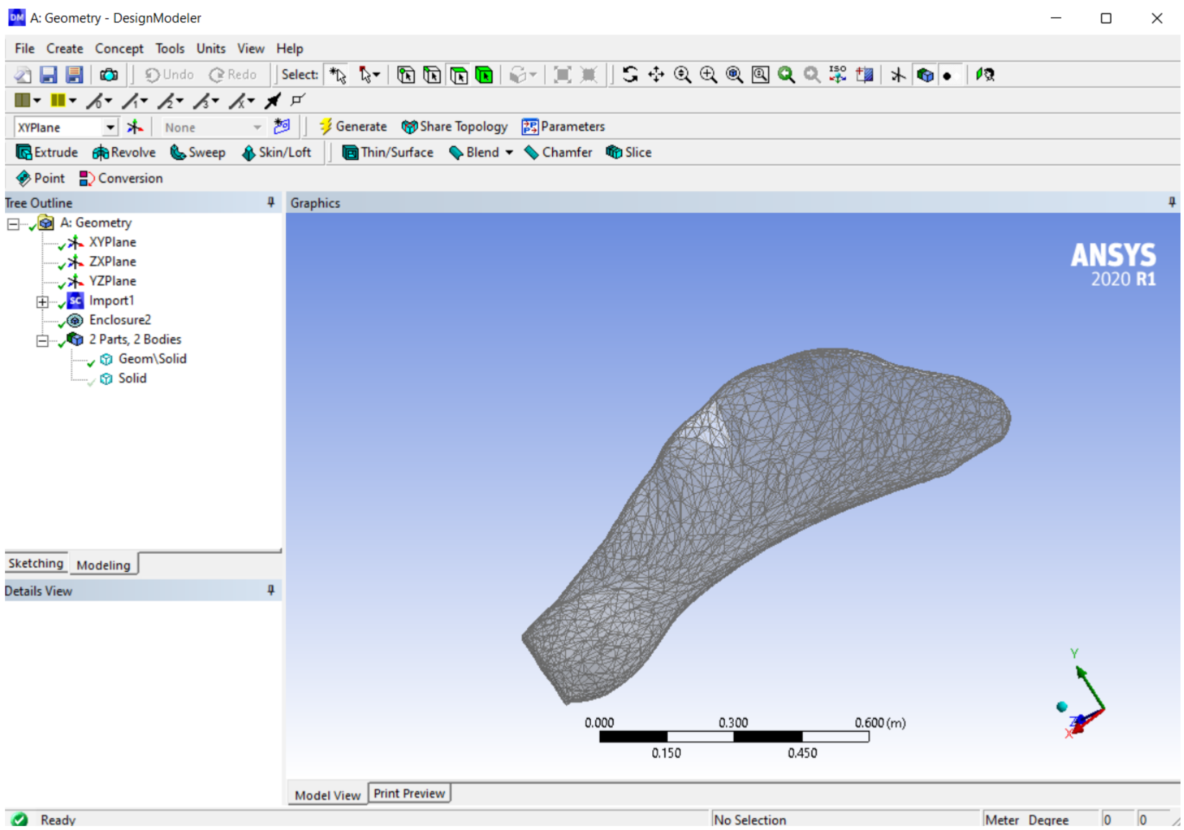


Figure 11. DesignModler from ANSYS

In the Figure 11, the Tree Outline indicate the characteristics that have been incorporated to facilitate the next simulations.

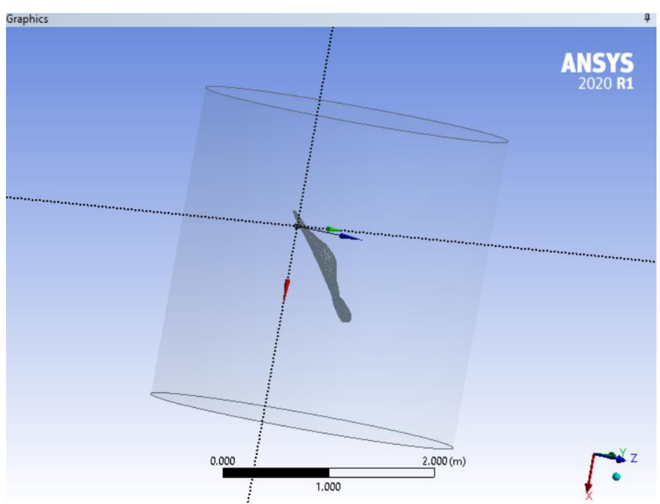


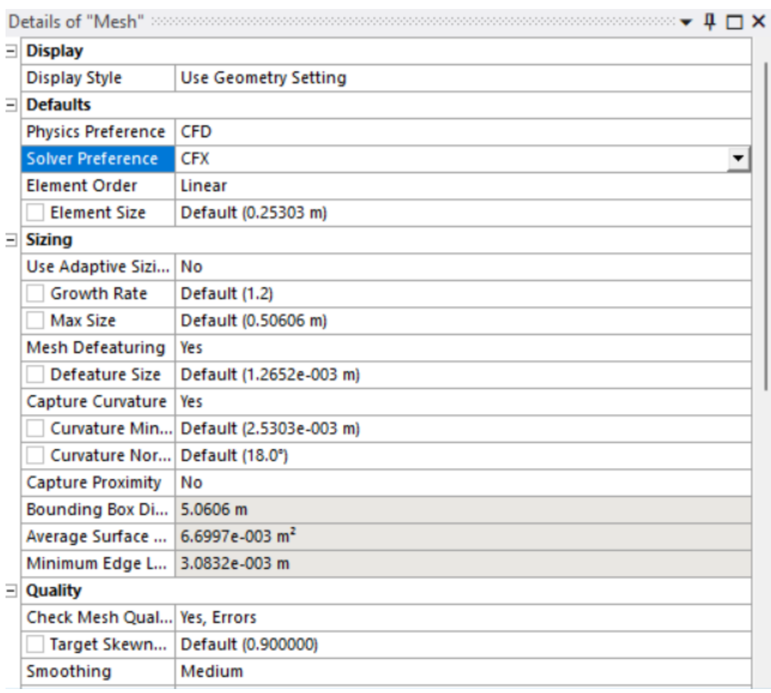
Figure 12. Enclosure applied in DesignModeler

3.2 MESH GENERATION

Mesh generation is a crucial step in computational fluid dynamics (CFD) simulations that discretize the geometry into a small number of components for numerical analysis. The mesh creation process used in ANSYS to simulate fluid flow around ash seed is explained in this section.

The complexity of the geometry, available computing power, and the goals of the simulation all influence the meshing technique selection. To evaluate how much mesh resolution affected the simulation findings, grid independence research was carried out. To find the point of convergence and guarantee accurate simulation results, mesh refinement was gradually increased while changes in important flow variables (such as drag force and velocity profiles) were tracked.

In the first simulation, just to obtain an initial result and see the availability and perspectives of the boundary conditions, it had been used an linear element order and a default size of 0.253m. This means a rough simulation to check if there are any initial errors that could affect the development of the next simulations.



Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	CFX
Element Order	Linear
Element Size	Default (0.25303 m)
Sizing	
Use Adaptive Sizi...	No
Growth Rate	Default (1.2)
Max Size	Default (0.50606 m)
Mesh Defeaturing	Yes
Defeature Size	Default (1.2652e-003 m)
Capture Curvature	Yes
Curvature Min...	Default (2.5303e-003 m)
Curvature Nor...	Default (18.0°)
Capture Proximity	No
Bounding Box Di...	5.0606 m
Average Surface ...	6.6997e-003 m^2
Minimum Edge L...	3.0832e-003 m
Quality	
Check Mesh Qual...	Yes, Errors
Target Skewn...	Default (0.900000)
Smoothing	Medium

Figure 13. Mesh parameters of the first simulation

One of the queries that can be raised is why to choose a linear elements order. First you need it's important to consider that the element order, as used in finite element analysis (FEA) or computational fluid dynamics (CFD), is the degree of the polynomial used to approximate the solution within each element. A first-degree polynomial is used in linear elements.

A linear shape function is a characteristic of a linear element, often known as a lower order element. The mesh region's displacements between nodes change linearly in relation to their separation. Bending is not captured by linear components.

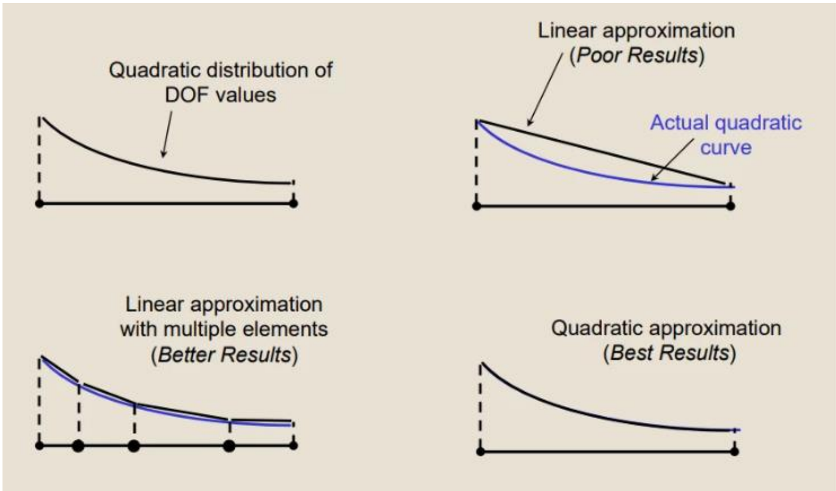


Figure 14. Element Order Comparison (Source: https://featips.com/2019/03/29/linear-vs-quadratic-fe-elements/)

The image bellow can provide a better perspective about the difference between a Firt-Order and a Second-Order element. Whether elements contain only corner nodes (first-order) or if they contain corner nodes and mid-side nodes (second-order). More technically, though, it describes the behavior of the polynomial functions:

Linear (First-Order): $P(x) = a*x + b$
Quadratic (Second-Order): $P(x) = a*x^2 + b*x + c$

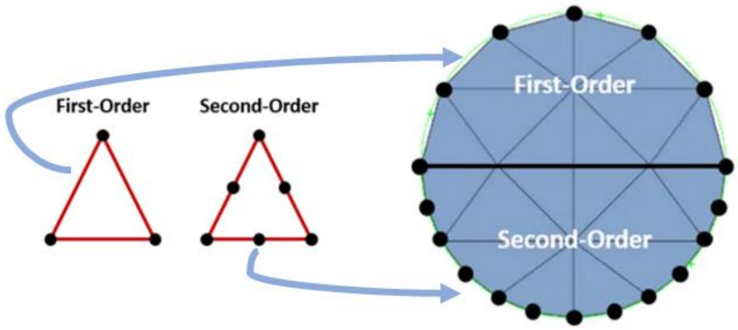


Figure 15. Visual perspective of different order element (Source: https://www.fidelisfea.com/post/first-order-vs-second-order-elements-in-fea)

The Figure 16 shows the initial mesh that was generated to perform the simulations, utilizing a linear first-order mesh design. This mesh was then used to execute fluid flow simulations using ANSYS.

Following the preliminary geometry treatment and after adaptation to ANSYS, the geometry turned into converted right into a solid model to facilitate the enclosure (cylinder geometry around the ash seed). Five key simulations have been carried out with valid results, presenting a reasonable perspective of the methodology. Additional simulations have been completed to understand the information of the software and its fundamentals. The mesh was varied by altering the element size and by switching between linear and quadratic elements. Furthermore, the viscous model settings were adjusted to simulate both laminar and turbulent flow situations.

Table 1. Representation of how the simulation had been conduced

	Mesh		Mode
Simulation	Element size	Oder	Viscous
1	0.253m	Linear	Laminar
2	0.253m	Quadratic	Laminar
3	0.082m	Quadratic	Laminar
4	0.082m	Quadratic	K - epsilon
5	0.082m	Quadratic	K - omega

Figures 16 and 17 illustrate the mesh alterations proposed in this thesis, as indicated in Table 1. There was no change in the number of elements between simulations 1 and 2. therefore the same image was used.

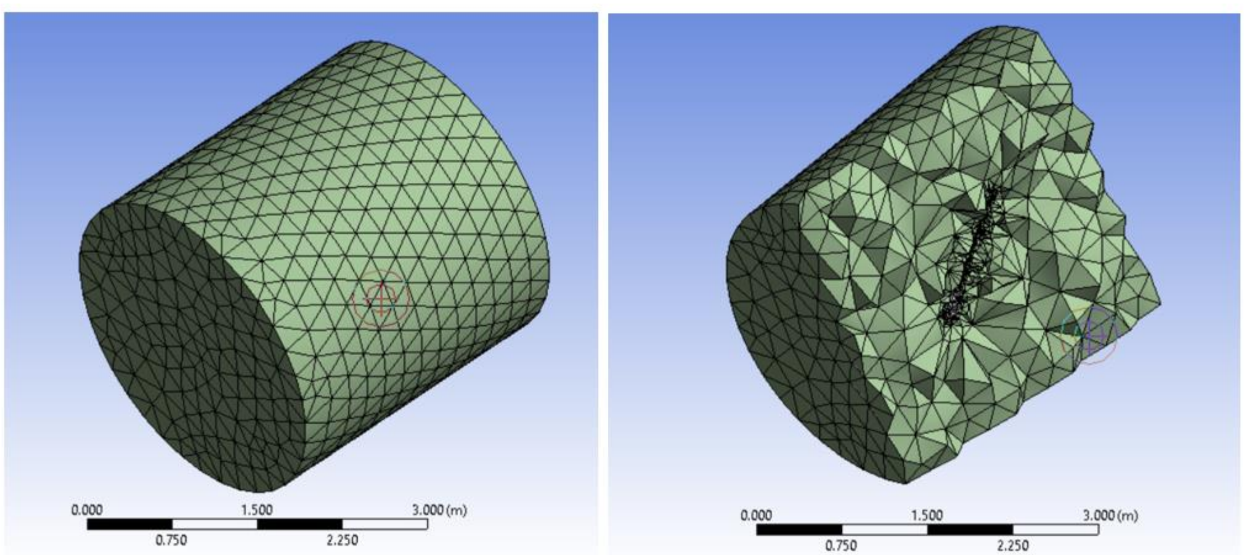


Figure 16. Mesh generated for the first and second simulation

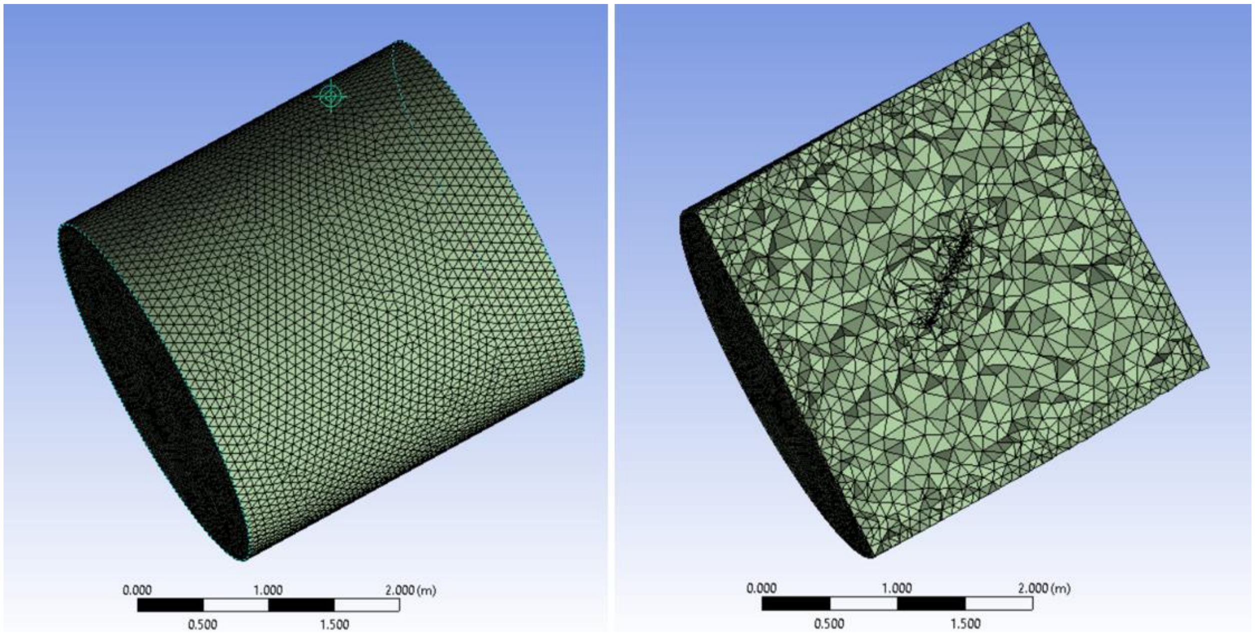


Figure 17. Mesh generated after refining process

3.3 BOUNDARY CONDITIONS

Durind the simulations it had been decided to use a steady-state condition, it means that the flow properties (like velocity, pressure, temperature) are assumed to be constant over time. In this case the simulation aims to find a solution that represents the average or unchanging flow behavior.

This contrasts with a transient simulation, where the flow properties change with time. Transient simulations are used for scenarios where the flow is unsteady, like a valve opening or a pulsating flow what is not the case here, where the fluid, in this case air, will pass surround the seed, as it happens in a wind turbine, taking in considerations ideal conditions.

Another parameter that had been set was the velocity formulation. Initially a simulation using absolute velocity was used because the velocity at each point is calculated relative to a fixed, global reference frame. This is suitable for most simulations where the domain is not rotating significantly. Considering the initial objective is analyze the turbulence around the seed this approach allows the simulation to occur in a more linear way, avoiding deviations and a faster calculation

In the order hand, relative velocity formulation calculates the velocity relative to a designated moving reference frame. This is useful for simulations involving rotating machinery, where a part of the domain (like an impeller) is moving, and the flow needs to be analyzed relative to that motion.

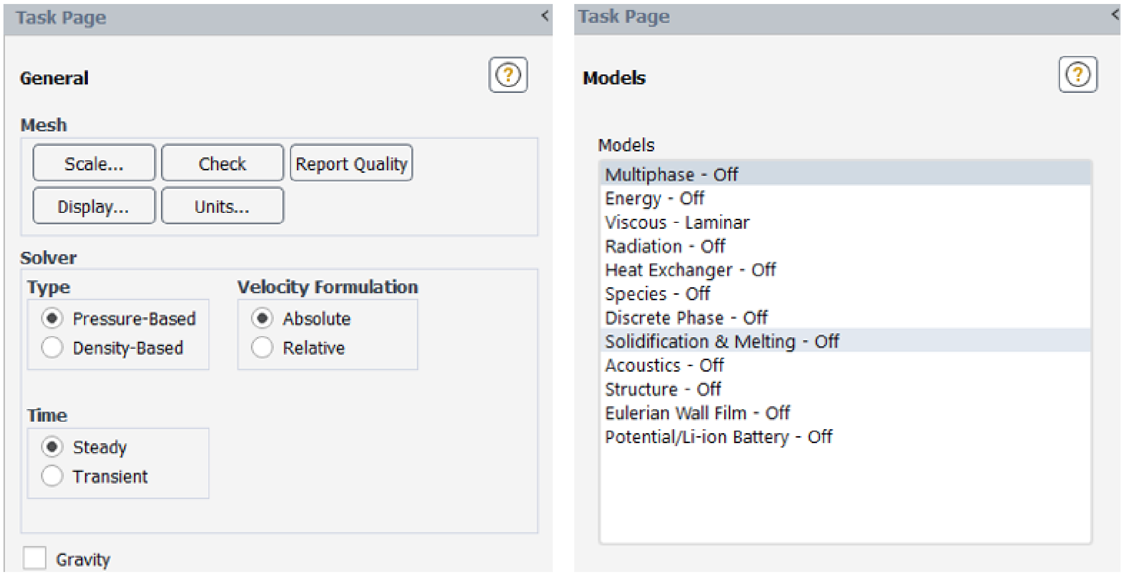


Figure 18. Setup parameters used in ANSYS (alteration had been done according to the Table 1)

The average velocity of the air passing through a wind turbine can vary widely based on several factors, including the location of the turbine, the height of the turbine above ground, and the local wind conditions. However, to provide a general estimate, typical wind speeds for efficient wind turbine operation range from 3 to 25 meters per second (m/s). Information collected in General wind power information, https://renewablesfirst.co.uk/, access in march 2024.

As the wind speed increases, the blades of the wind turbine spins faster. However, wind turbines are designed to stop operating beyond a certain speed in order to prevent damage to the blades. This speed is referred to as the cut-out speed, which is 25 m/s in this case. As the wind speed drops back down below the cut-out speed, the turbine can be allowed to restart and generate electricity. Information collected in information,/towardsdatascience.com/wind-energy, access in march 2024.

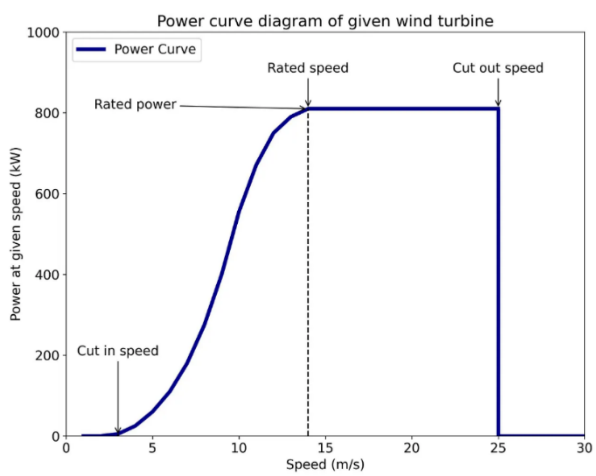


Figure 19. Power curve of the selected wind turbine. Image by Author (Source: https://towardsdatascience.com/wind-energy-physics-and-resource-assessment-with-python-789a0273e697)

The graph present in the Figure 19 is an example of the variation in energy production in relation to the wind speed that passes through the turbine. During the simulations it was decided to apply an initial speed of 25 meters per second. Being aware of it represents, usually, the limit speed (cut out speed) because the objective is to study regions of greater turbulence and air resistance.

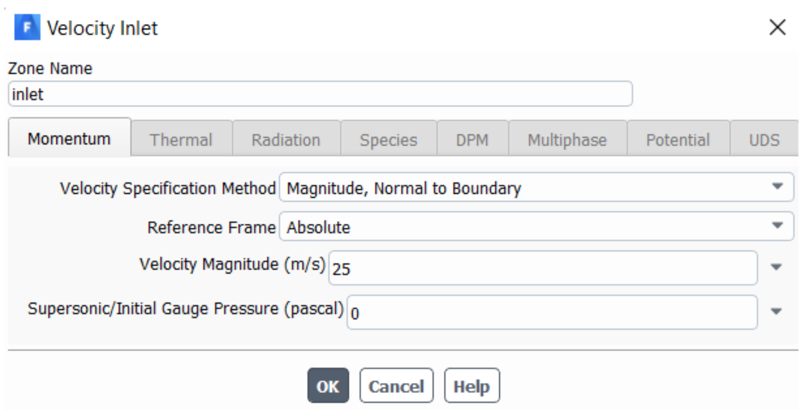


Figure 20. Setup parameter related to inlet velocity used in ANSYS

Now, talking about the outlet parameter, for this simulation it had been applied a pressure of 10 Pascal. Using a gauge pressure of 10 Pascal at the outlet is generally a practical choice to ensure numerical stability and convergence in CFD simulations. While it's not a strict rule, it serves as a useful default value that works well in many aerodynamic applications.

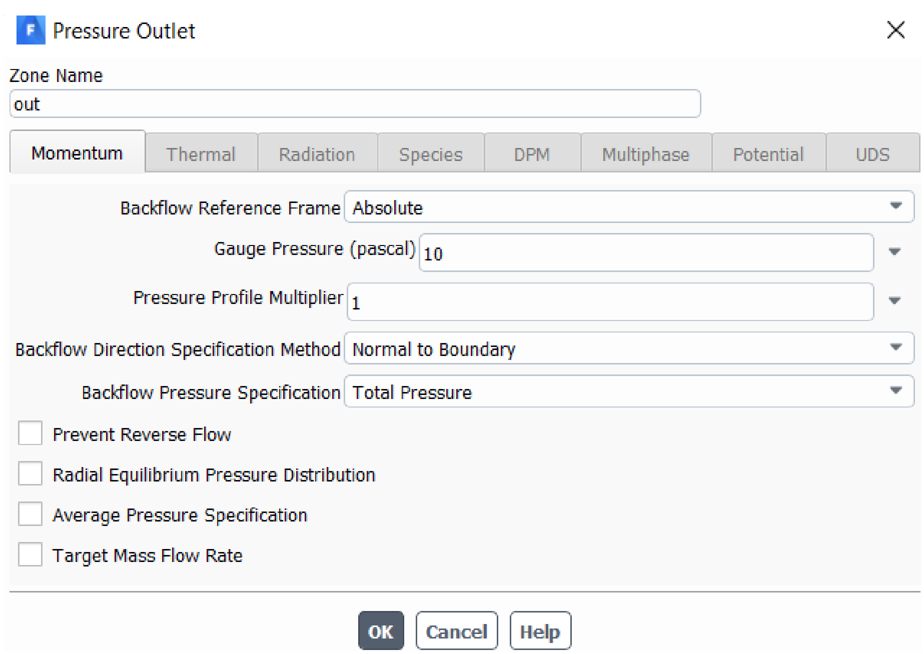


Figure 21. Setup parameter related to outlet pressure used in ANSYS

3.4 MATERIAL PROPERTIES

For the simulations the air, representing the fluid, and the aluminum, representing the seed, had been applied. The Figure 22 indicate the parameters for each material like, density and viscosity.

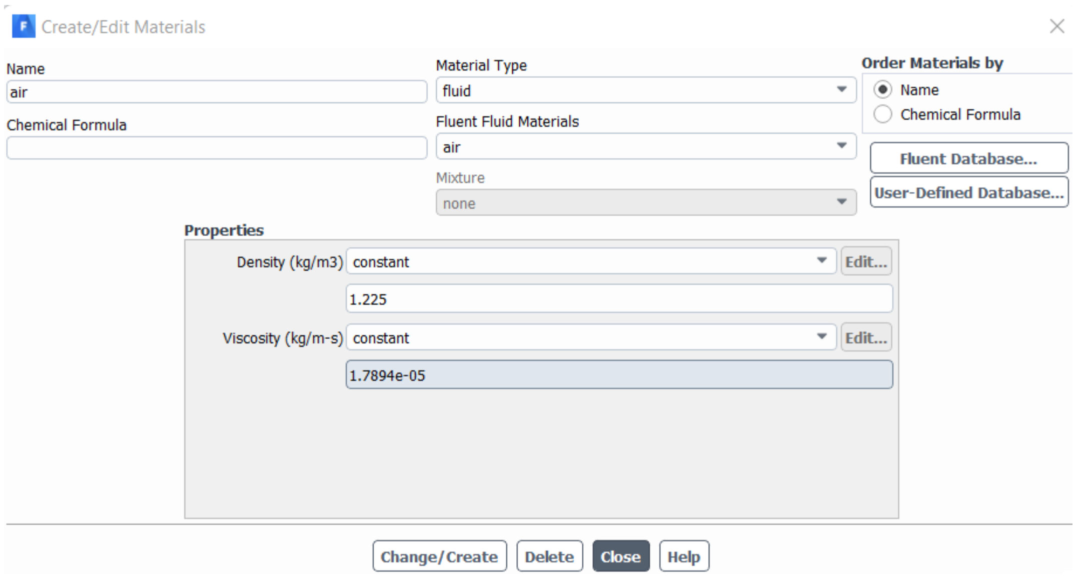


Figure 22. Property of the fluid, air

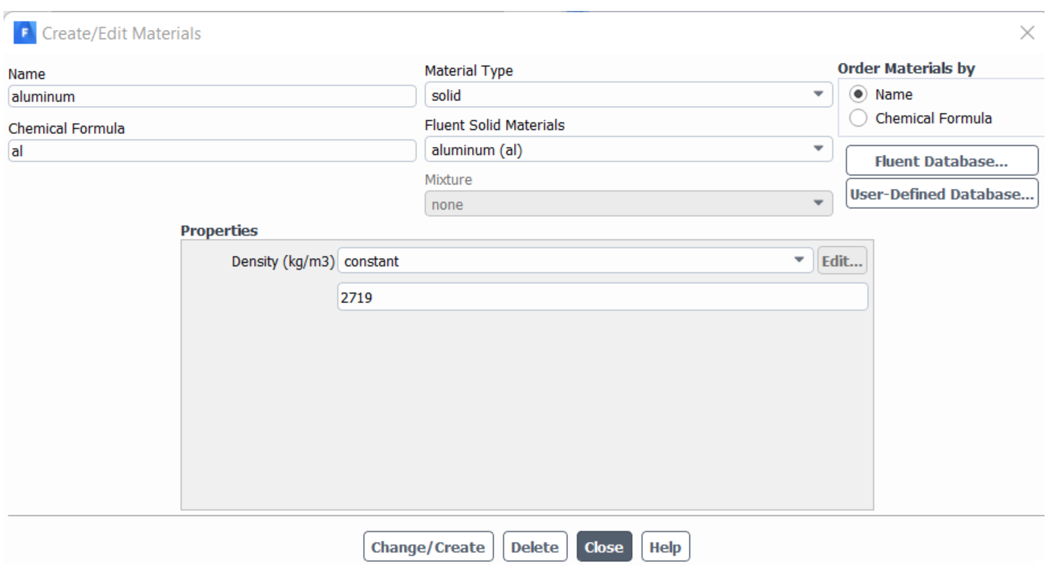


Figure 23. Property of the solid (seed), aluminum

4. CONVERGENCE AND VALIDATION

Convergence in CFD simulations refers to the point where the solution becomes stable and no longer changes significantly with further iterations. It indicates that the numerical solution has approached the true solution of the governing equations. While residuals are a measure of the error in the solution of the equations at each iteration. They are calculated as the difference between the left-hand side and the right-hand side of the discretized equations. As the solution converges, residuals should decrease to very small values.

The number of iterations refers to the number of times the solver processes the entire computational domain to update the solution variables. Each iteration involves solving the discretized equations to refine the solution.

A maximum number of iterations of 300 had been set for the simulation. If the solution converges before reaching this maximum, the solver would stop. In the first scenario the results converged around 100. Was possible to see a tendence of stabilization of the graph about the diagram of velocity, it means the happened a convergence and the calculation could be validate.

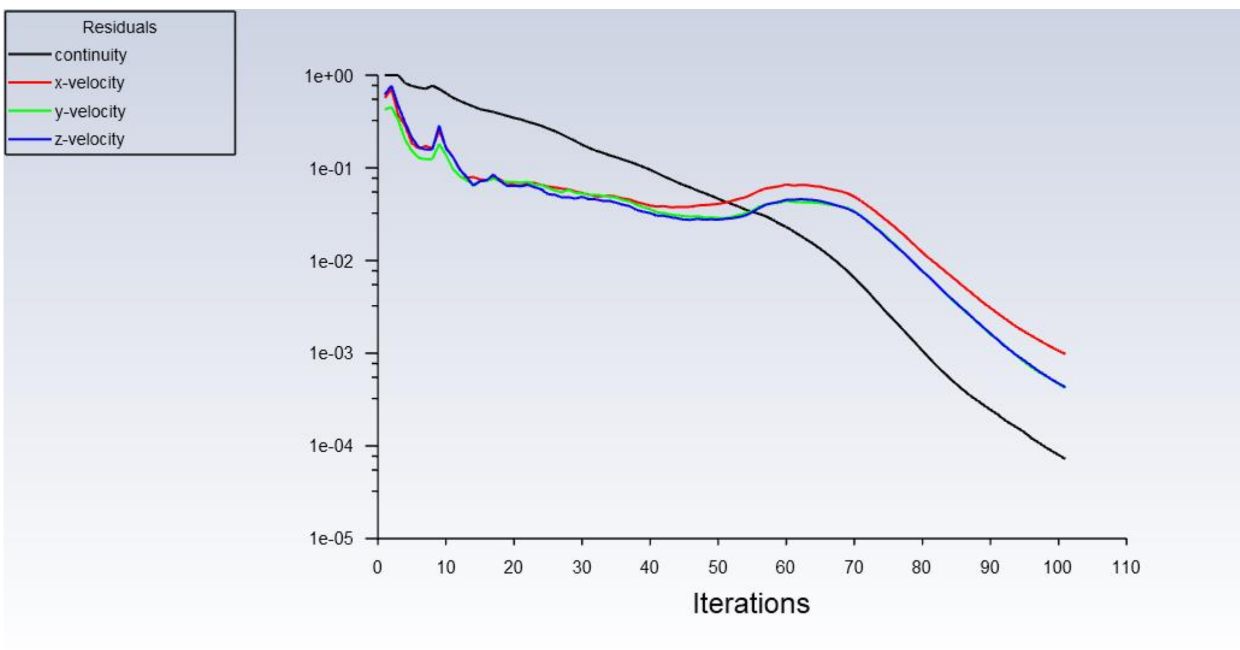


Figure 24. First simulation convergence – Linear element order, not refined mesh

It is important to carefully consider the number of iterations set for a CFD simulation in ANSYS, as it has direct implications for both the computational cost and the accuracy of the results.

Each iteration involves recalculating the flow variables across the entire computational domain, which requires processing power and time. More iterations mean more computational cycles, leading to longer simulation times.

Higher iteration counts demand more CPU time and, in some cases, more memory usage. This can increase the overall computational resource consumption, potentially leading to higher costs, especially if using cloud-based or high-performance computing resources.

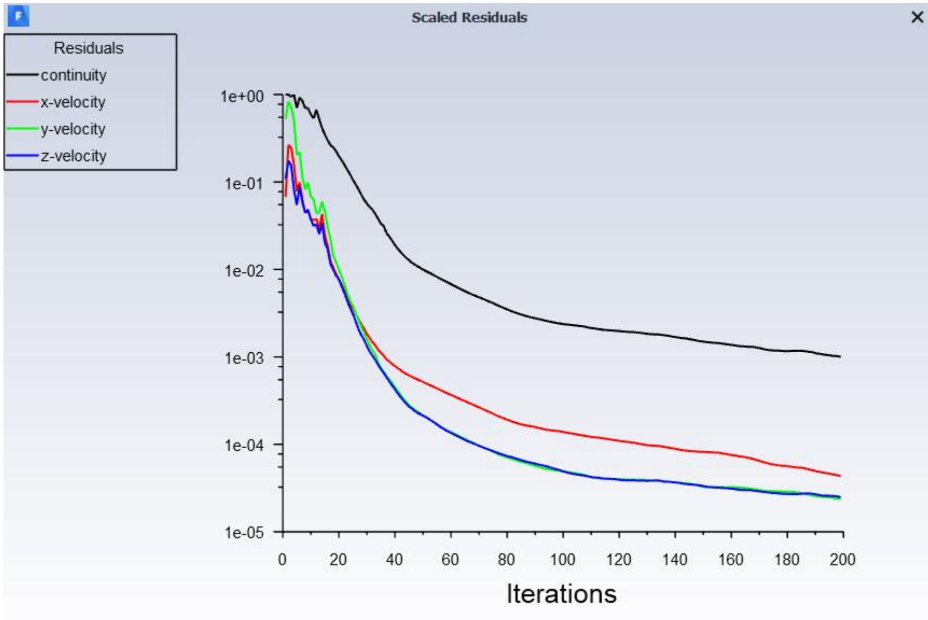


Figure 25. Second simulation convergence - Quadratic element order

In simulation 2, it was possible to notice a greater number of interactions, as indicated in the Figure 25, for the results to converge, which was expected due to the greater complexity in calculating quadratic order elements.

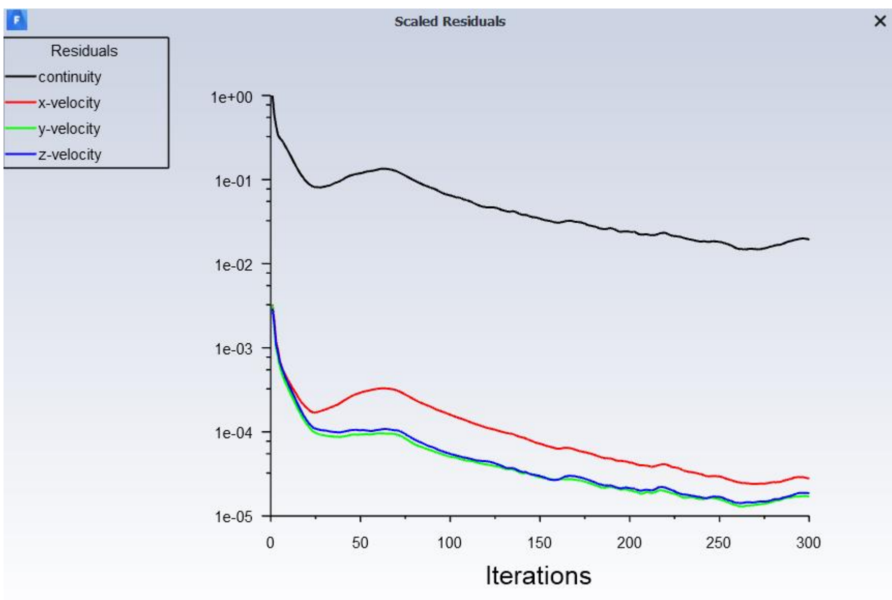


Figure 26. Third simulation convergence - Refined mesh and quadratic element order

As was to be expected, there was a greater number of interactions during simulation 3, while the error started from a smaller number.

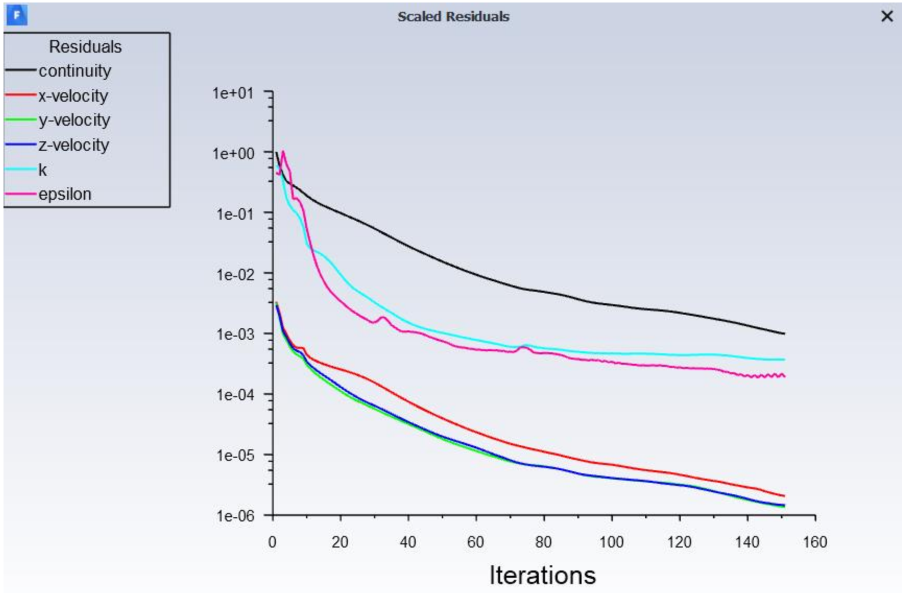


Figure 27. Fourth simulation convergence - Refined mesh and quadratic element order, viscous mode epsilon

One very interesting point that is worth discussing is the way in which the results converged in simulation 4, K-epsilon viscosity model. Smaller errors, with results still close to the laminar regime of simulation 3, imply a prosperous model for this type of simulation.

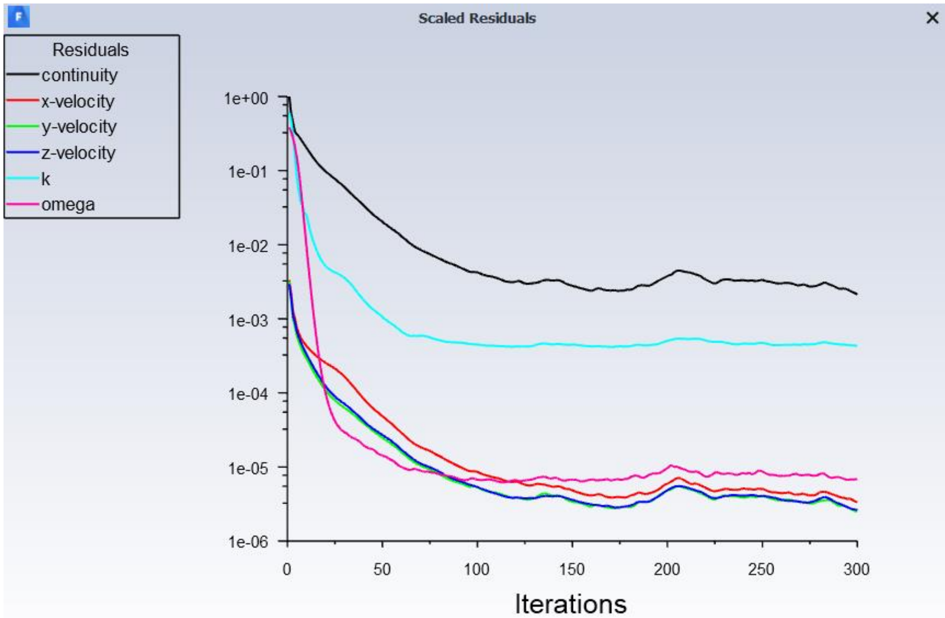


Figure 28. Fifth simulation convergence - Refined mesh and quadratic element order, viscous mode omega

Despite the greater number of interactions required to converge the omega viscosity model, with with quadratic element and refinement parameters, the results were very satisfactory, especially with regard to the convergence of the velocities and the omega. Rate ω is reduced due to viscous and other dissipative effects. It is crucial for maintaining a realistic and balanced turbulence model.

5. RESULTS AND INITAL ANALYSES

For a better visual interpretation of the results and to analyze the fluid passing surrounding the ash seed, results parameters had been applied.

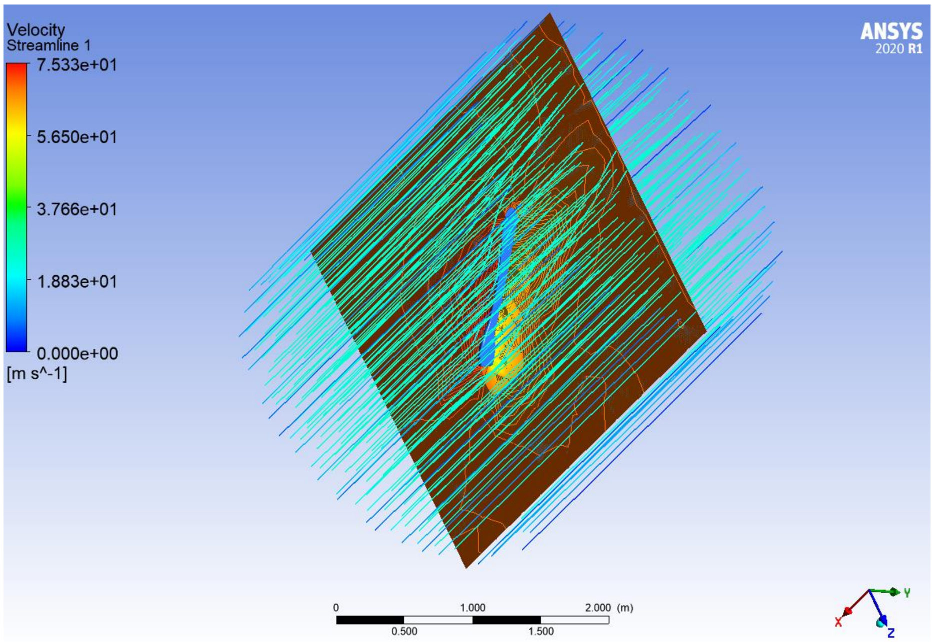


Figure 29. Velocity streamline and pressure plane - Simulation 1

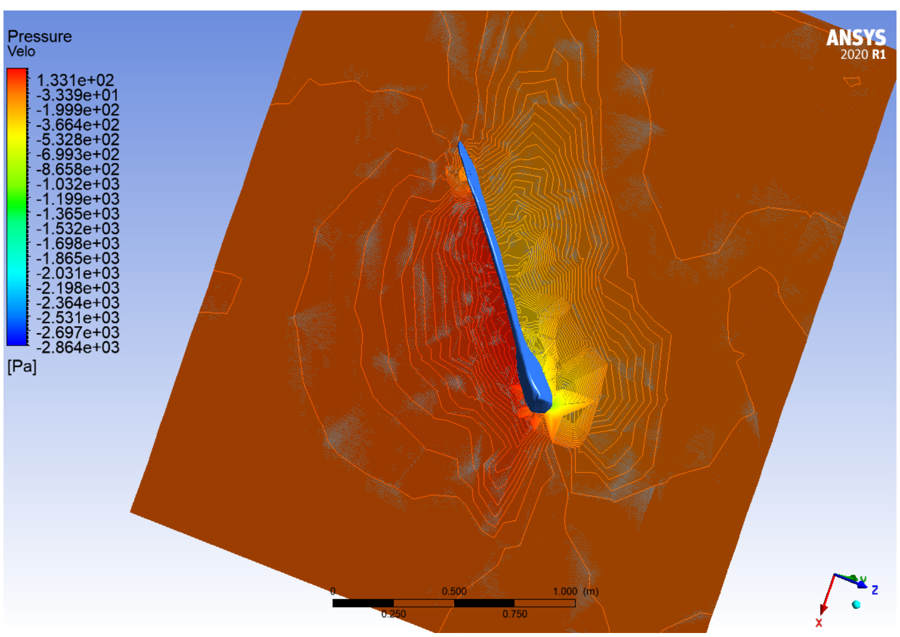


Figure 30. Planning with the pressure value around the ash seed - Simulation 1

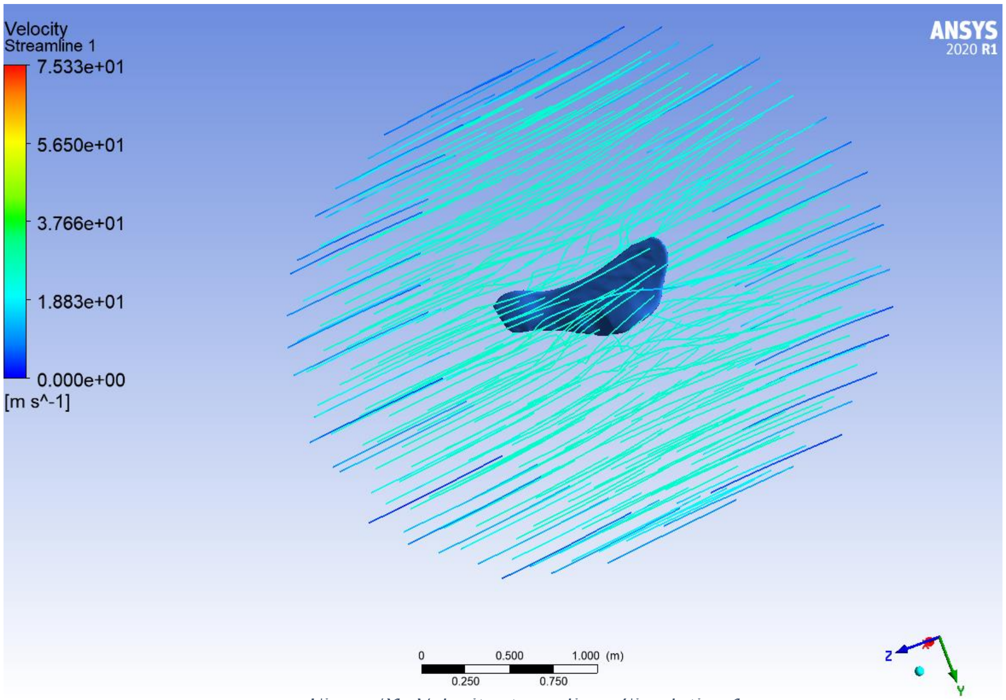


Figure 31. Velocity streamline - Simulation 1

Second simulation, quadratic element:

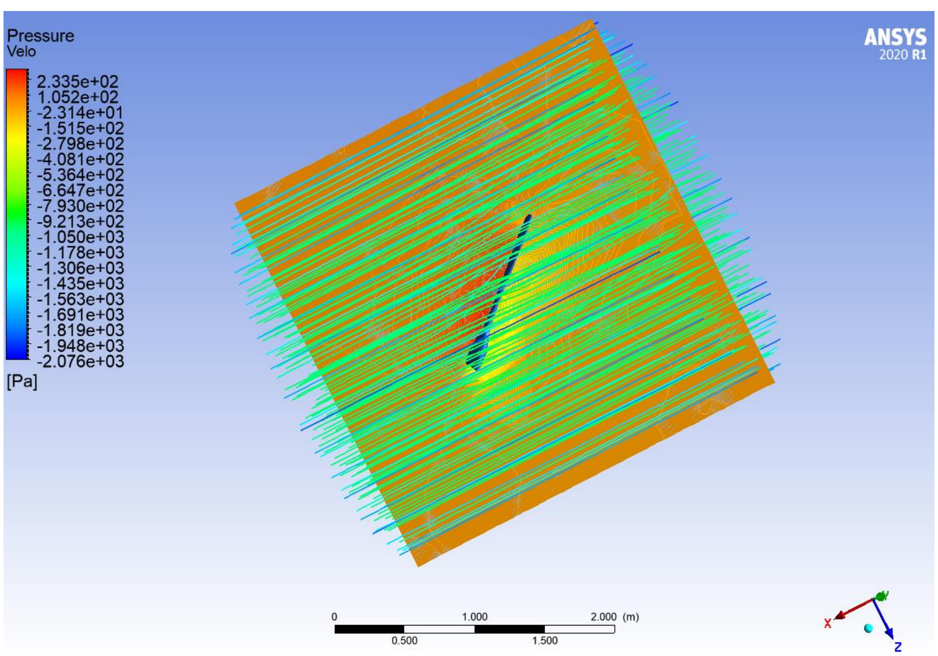


Figure 32. Velocity streamline and pressure plane - Simulation 2

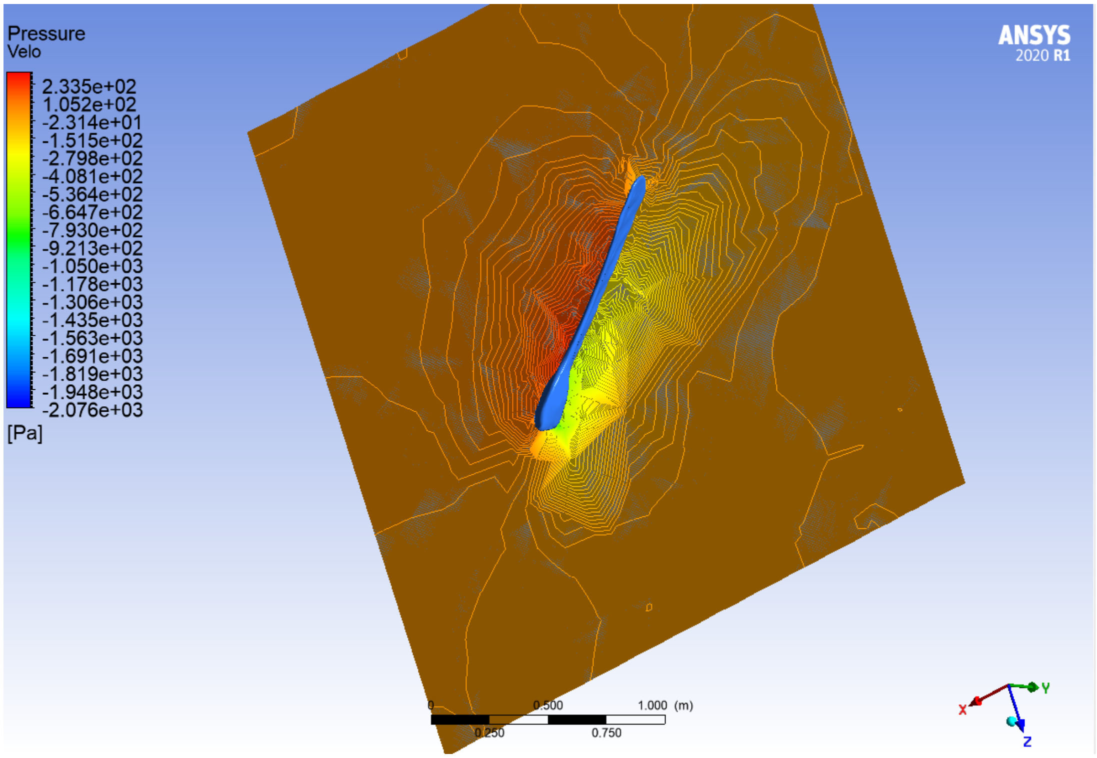


Figure 33. Planning with the pressure value around the ash seed - Simulation 2

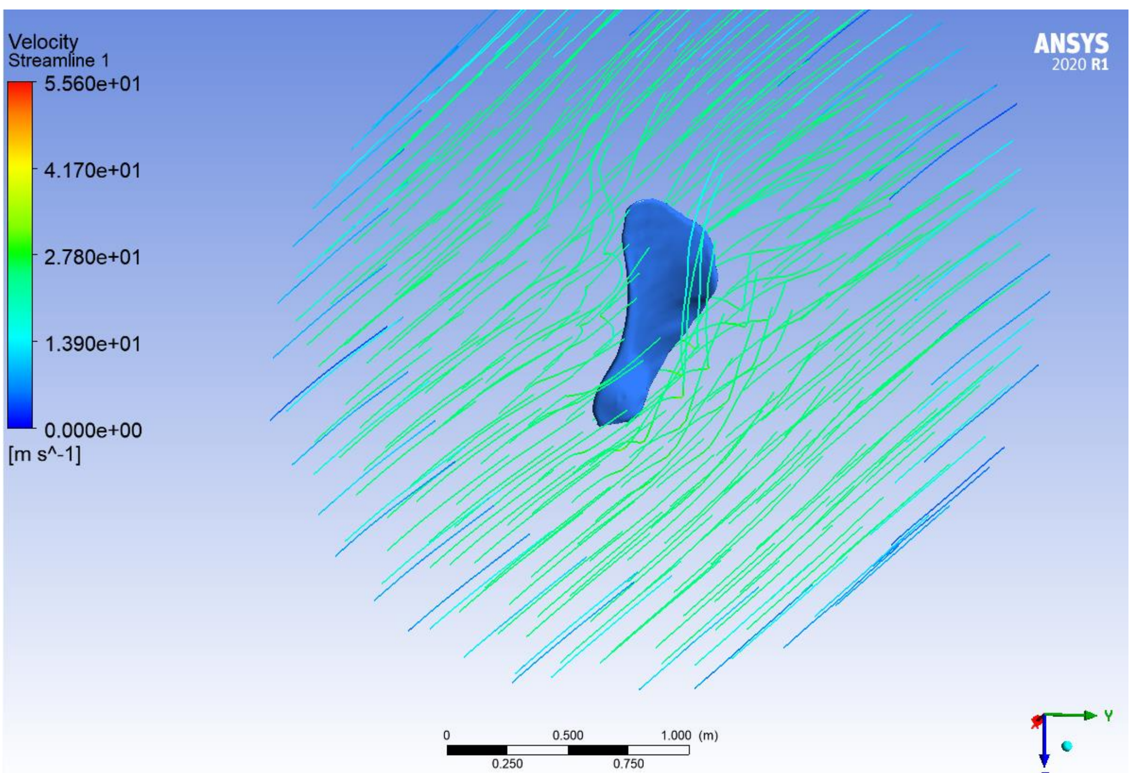


Figure 34. Velocity streamline - Simulation 2

Third simulation, quadratic element with the refinement of the mesh:

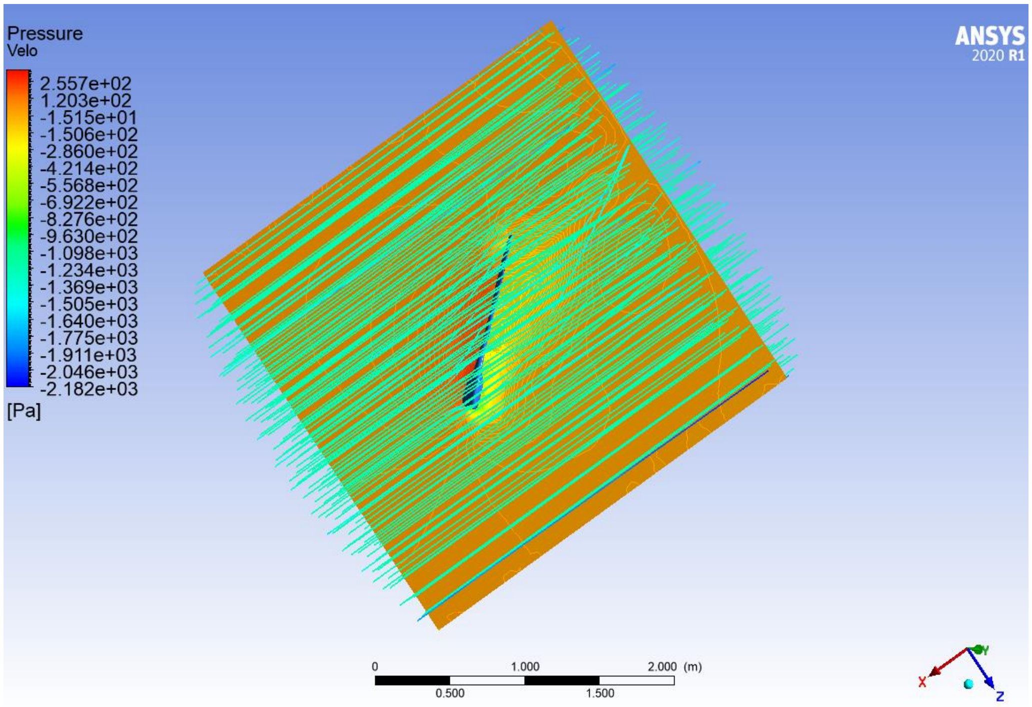


Figure 35. Velocity streamline and pressure plane - Simulation 3

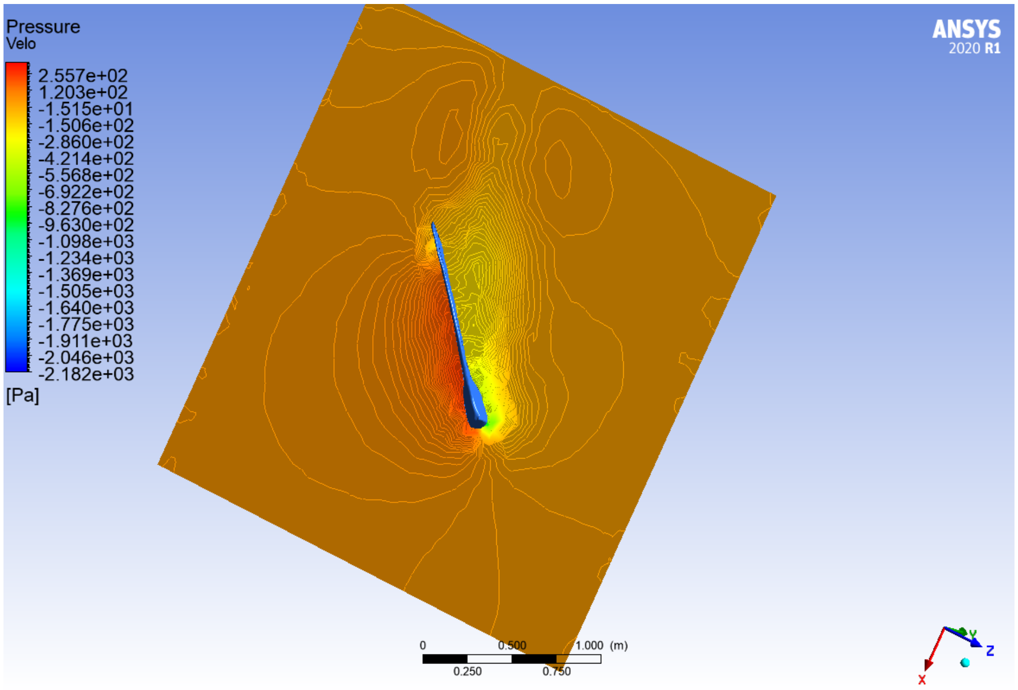


Figure 36. Planning with the pressure value around the ash seed - Simulation 3

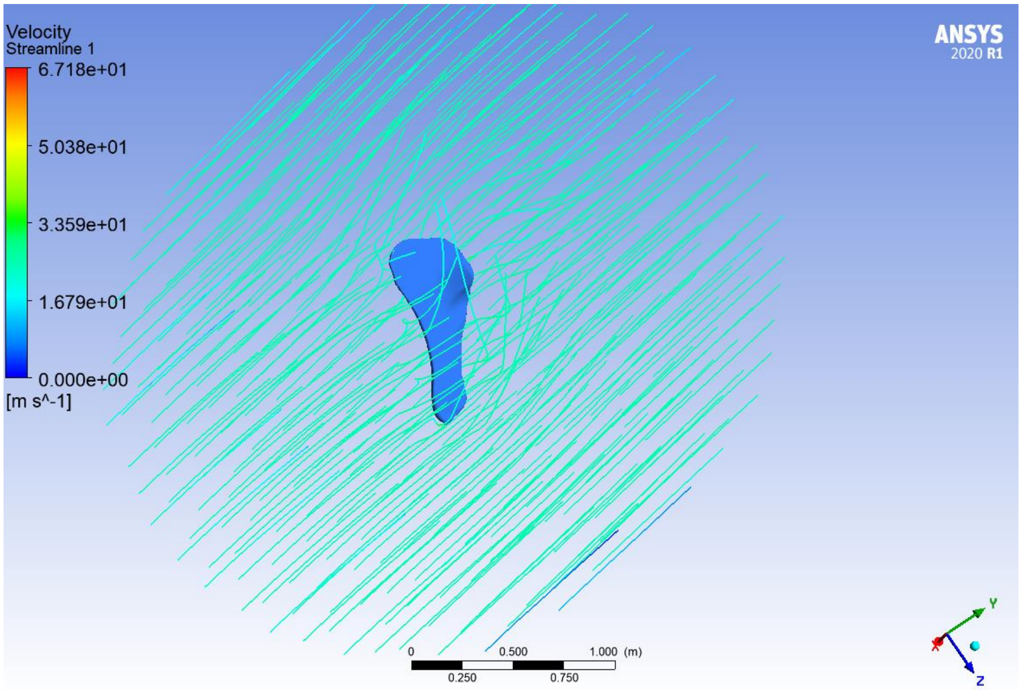


Figure 37. Velocity streamline - Simulation 3

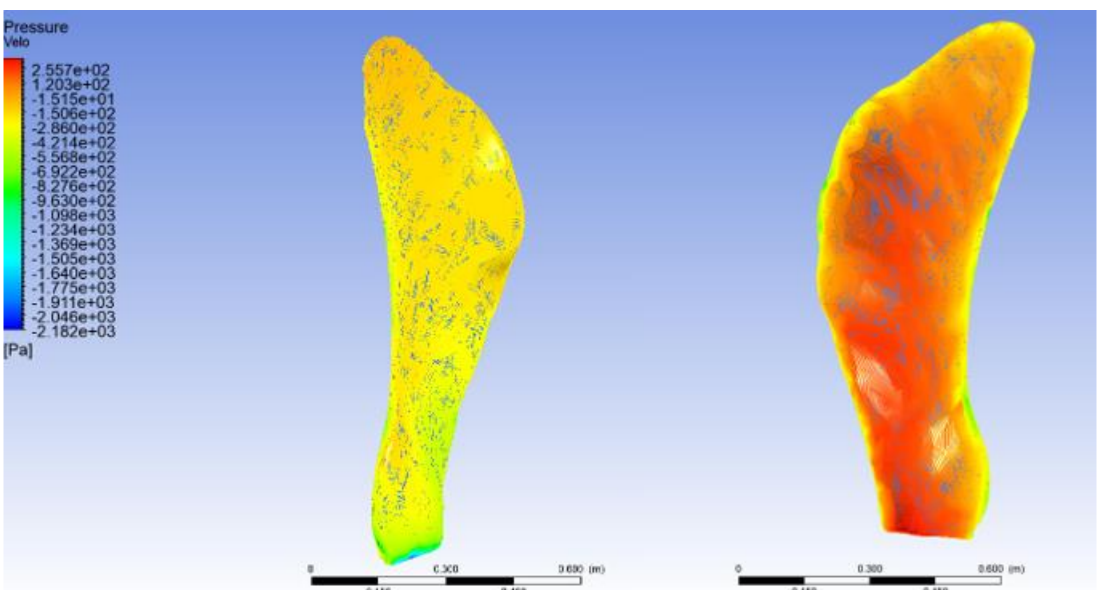


Figure 38. Pressure around the two faces of the seed - Simulation 3

Fourfh simulation, quadratic element with the refinement of the mesh, mode of viscous K-epsilon:

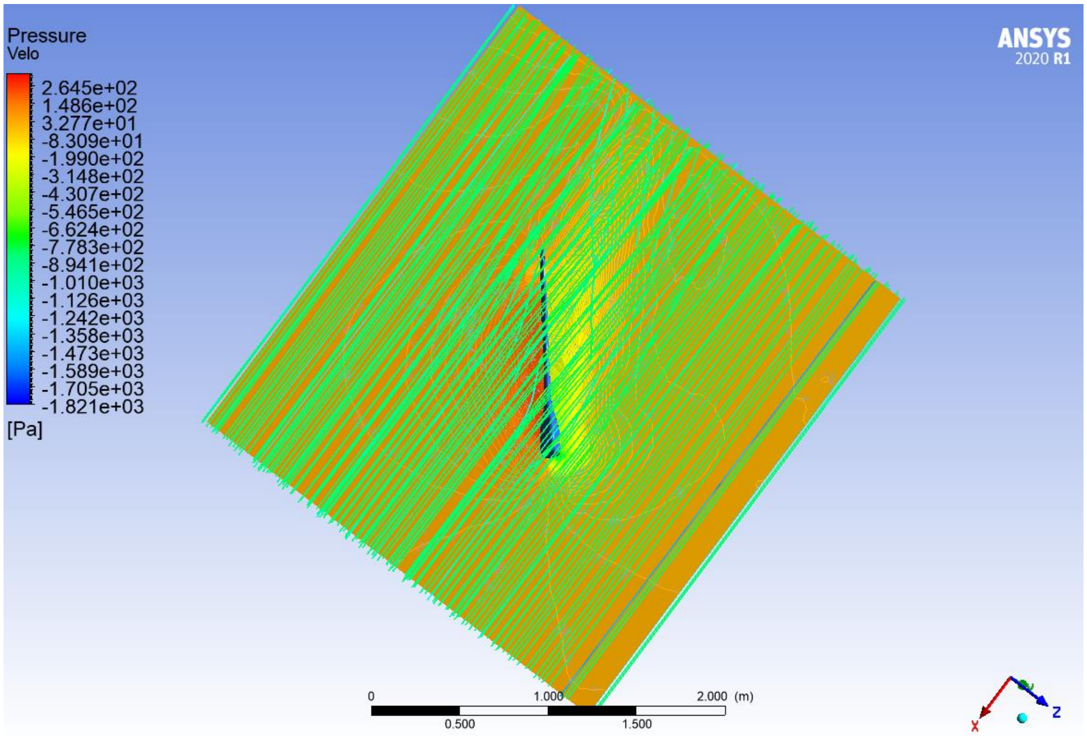


Figure 39. Velocity streamline and pressure plane - Simulation 4

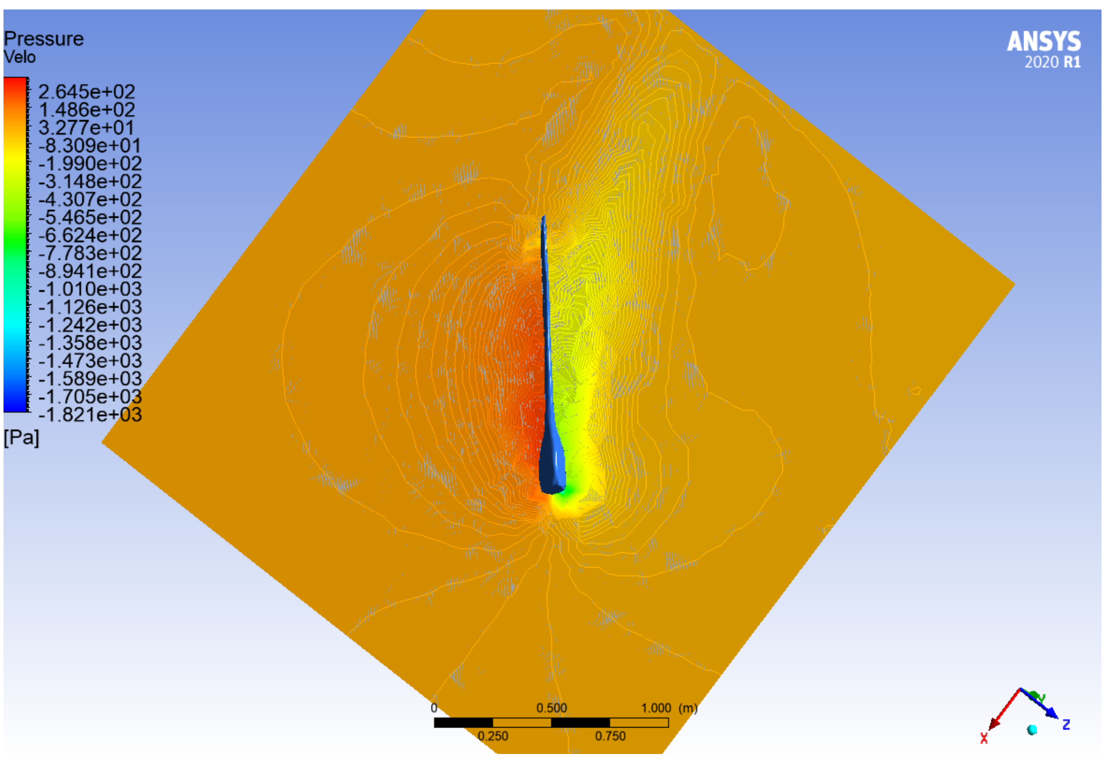


Figure 40. Planning with the pressure value around the ash seed - Simulation 4

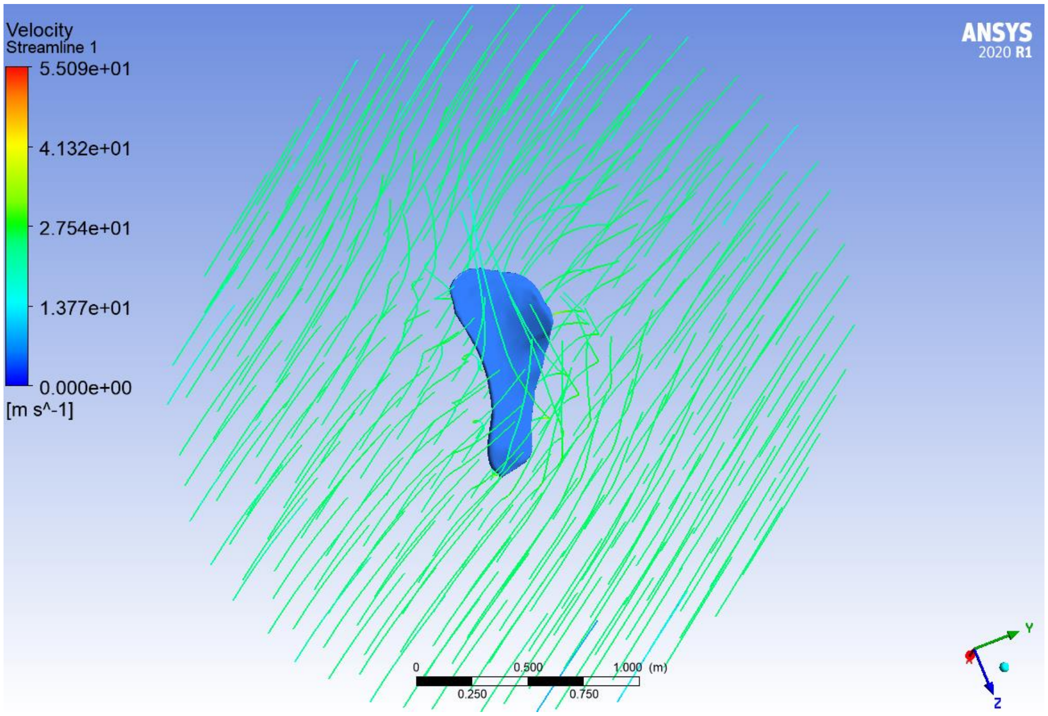


Figure 41. Velocity streamline - Simulation 4

Fifth simulation, quadratic element with the refinement of the mesh, mode of viscous K- epsilon:

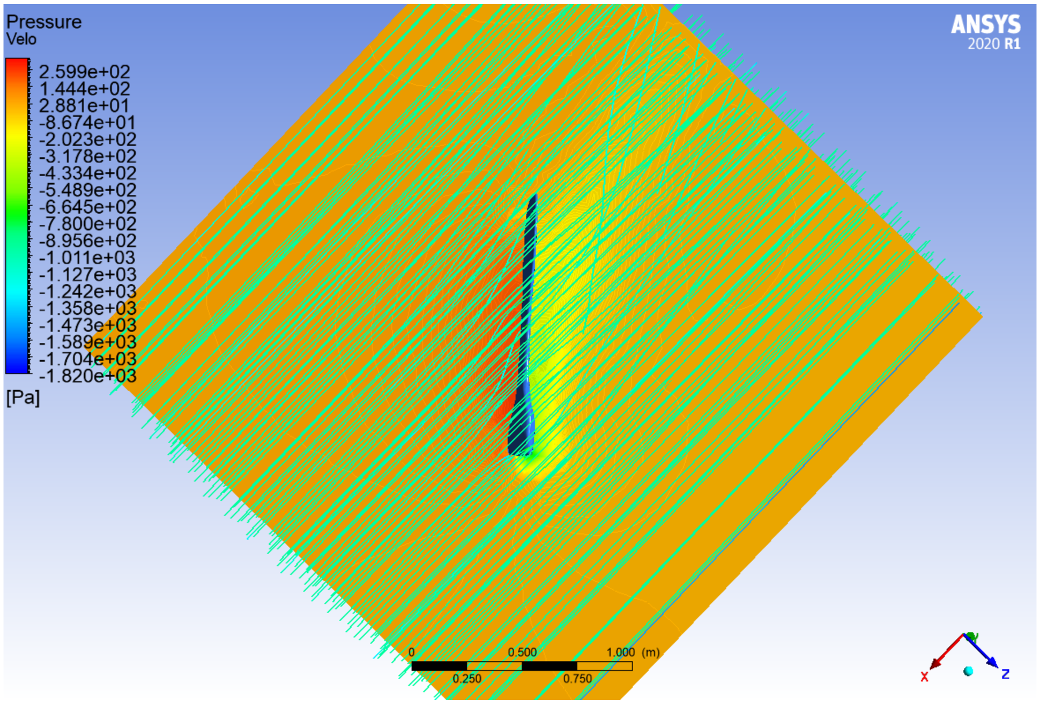


Figure 42. Velocity streamline and pressure plane - Simulation 5

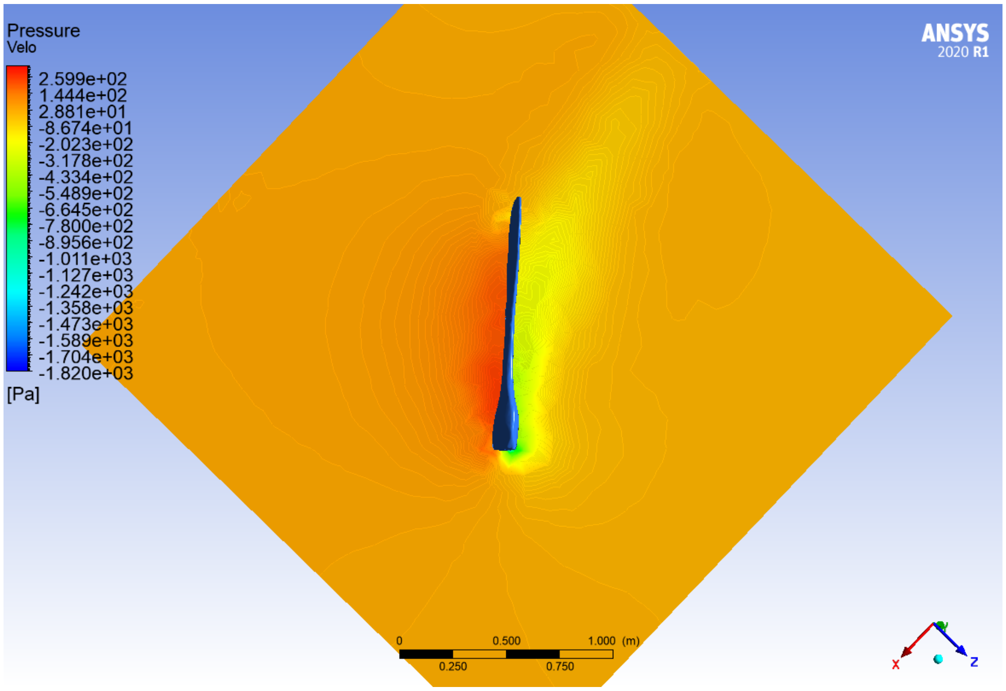


Figure 43. Planning with the pressure value around the ash seed - Simulation 5

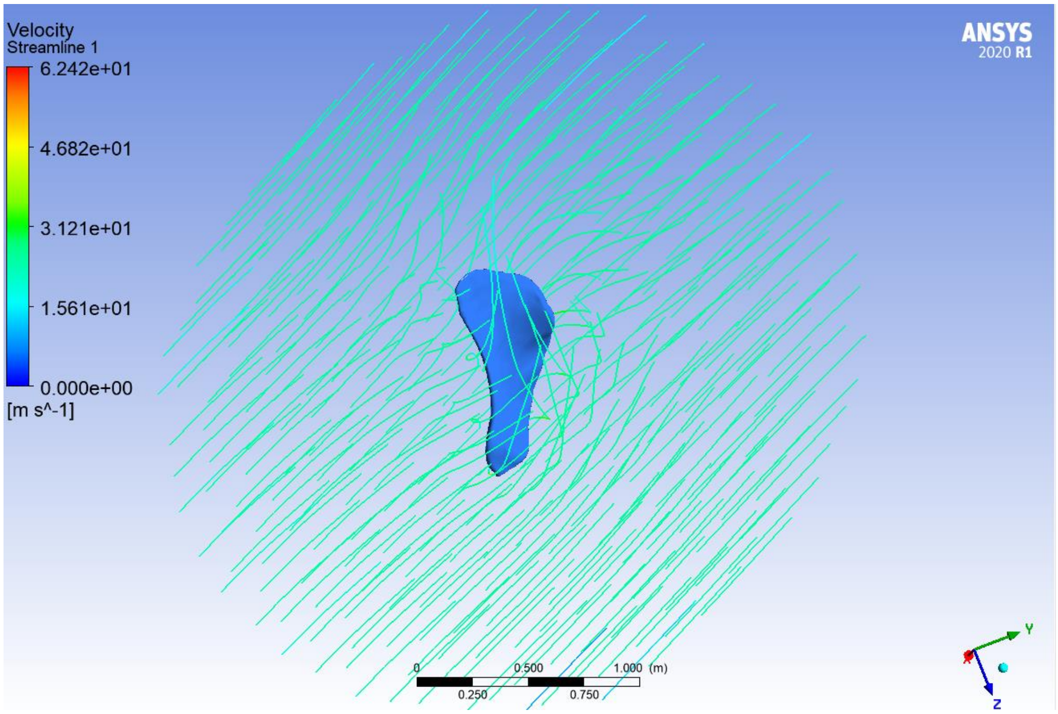


Figure 44. Velocity streamline - Simulation 5

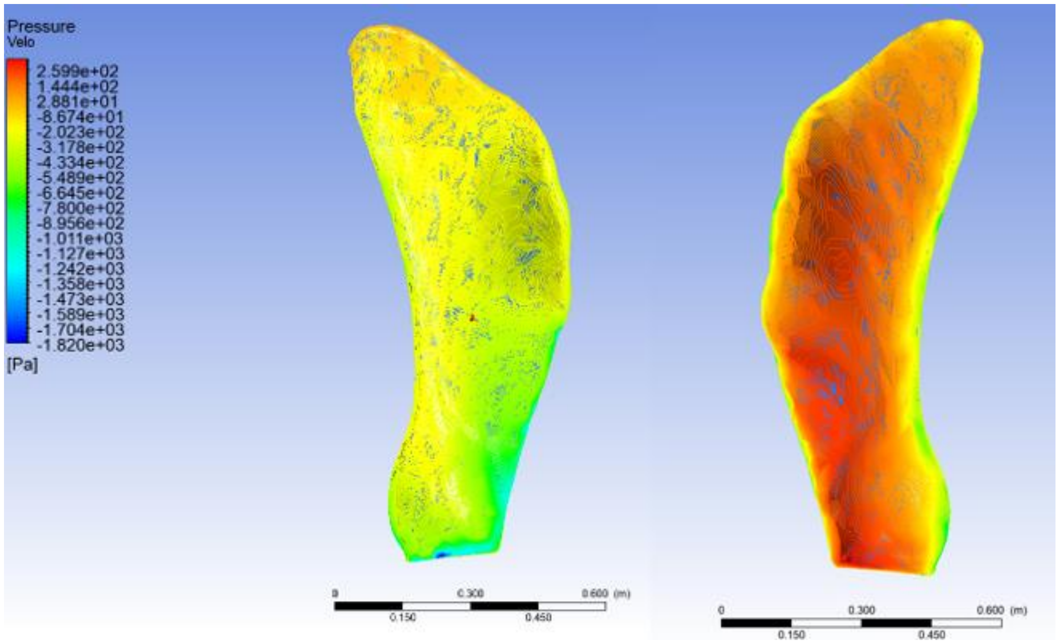


Figure 45. Pressure around the two faces of the seed - Simulation 5

Its important to say, just to clarify the reference, that the pressure values in the simulation are relative to a reference pressure (atmospheric pressure), that's why it possible to see negative pressure as results. Negative pressure regions indicate areas of high flow velocity and low pressure, which are instrumental in the rotation and dispersion mechanisms of the ash seed.

6. PROTOTYPE DEVELOPMENT

To advance the theoretical findings of this research, a practical approach was undertaken to develop a prototype inspired by the aerodynamic properties of ash seeds. This development took place at EirLab, a digital fabrication workshop associated with the ENSEIRB-MATMECA University, part of the Polytechnic Institute of Bordeaux. EirLab provides a comprehensive suite of rapid prototyping tools, including 3D printing, laser cutting and engraving, waterjet cutting, and electronic circuit production. Additionally, the lab offers training in the use of these tools and fosters a multidisciplinary community that aids in transforming ideas into tangible products.

For the creation of the prototype, I utilized a Form 3L 3D printer from Formlabs, which employs Stereolithography (SLA) technology. Unlike Fused Deposition Modeling (FDM), which can struggle with high-precision parts, SLA 3D printing excels in producing detailed and accurate components. This technology uses liquid resin that is selectively cured by a laser to form solid parts, followed by necessary post-processing steps to achieve the desired precision and finish.

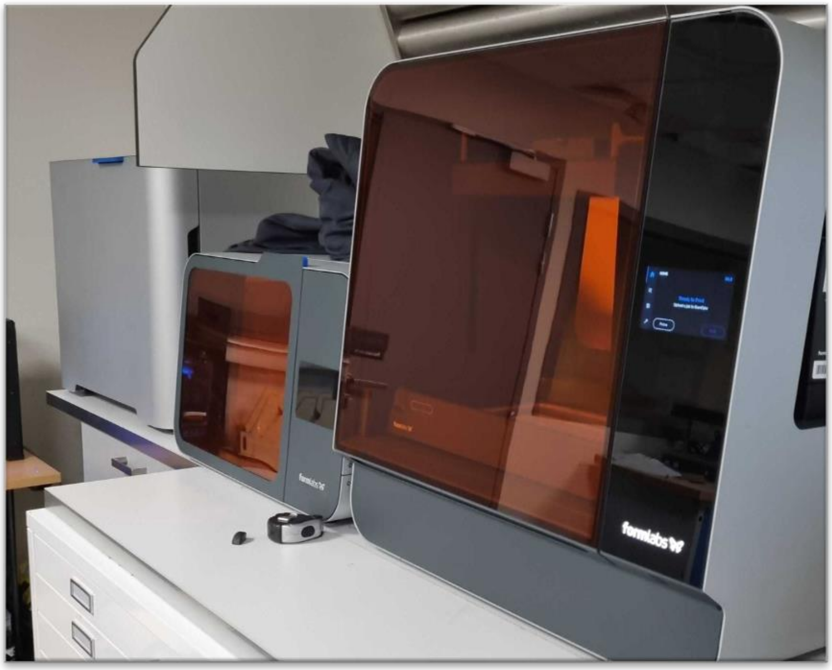


Figure 46. 3D machine used to print the prototype disigns

The choice of the Form 3L SLA printer was driven by its ability to produce high-resolution parts that closely replicate the intricate shapes and fine features of the ash seed-inspired blade design. The advanced capabilities of SLA printing were essential to accurately capture the aerodynamic features critical for the prototype's performance.

Figure 47. 3D models that had been printed in EirLab

During the process of developing the prototype, it became evident that translating the complex 3D cad geometry model features of the ash seed into a physical model was not straightforward. The intricate details captured in the digital model challenging to replicate with the 3D printing technology, necessitating a simplification. This simplification was essential to facilitate the fabrication process while still maintaining the central aerodynamic characteristics of the original design. Despite these challenges, the prototype demonstrated significant potential. This experience underscored the importance of iterative design and prototyping in achieving a balance between theoretical precision and practical manufacturability.

Included in this thesis is a picture of the initial prototype design, featuring a small motor coupled with a representation of a vertical wind turbine. This prototype showcases the potential for a practical and functional model, where the rotational motion of the blade, inspired by the ash seed, can be harnessed to generate mechanical energy. This mechanical energy can subsequently be converted into electrical energy, demonstrating a viable pathway from biomimetic design to real-world energy solutions. The promising results of this initial design underline the feasibility of developing an efficient and sustainable vertical wind turbine, leveraging the natural aerodynamic properties of the ash seed.

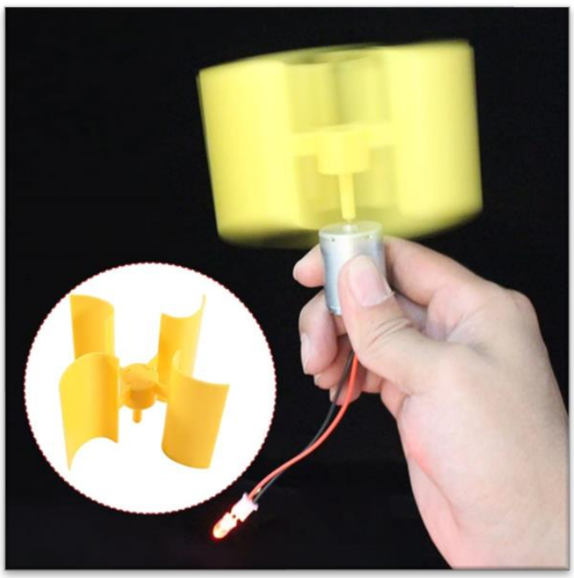


Figure 48. Example of the initial energy generator prototype model

Included in this thesis is a picture of the initial prototype example design, featuring a small motor coupled with a representation of a vertical wind turbine. This prototype showcases the potential for a practical and functional model, where the rotational motion of the blade, inspired by the ash seed, can be harnessed to generate mechanical energy. This mechanical energy can subsequently be converted into electrical energy, demonstrating a viable pathway from biomimetic design to real-world energy solutions. The promising results of this initial design underline the feasibility of developing an efficient and sustainable vertical wind turbine, leveraging the natural aerodynamic properties of the ash seed.

For e better approach and future practical studies it could be implemented the concepts associated with thee Strouhal number (St), a dimensionless value that describes oscillating flow mechanisms. It is commonly used in fluid dynamics to characterize vortex shedding, oscillating flows, and other periodic flow phenomena. The Strouhal number is defined as:

$$St = \frac{fL}{U} \tag{8}$$

Where:
f is the frequency of vortex shedding (or any characteristic frequency of the oscillatory flow) in Hz.
L is a characteristic length (for example, the diameter of a cylinder or the chord length of a blade) in meters.
U is the free-stream velocity of the fluid in meters per second (m/s).

Strouhal number can then be used to compare with empirical data or simulations to understand the vortex shedding and oscillatory behaviors in your turbine design.

7. CONCLUSION

This study has successfully investigated the aerodynamics and biomechanics of ash seeds, exploring their potential applicability in the design of small-scale vertical wind turbines. The research employed a series of computational fluid dynamics (CFD) simulations using ANSYS Fluent software, focusing on the refinement of the mesh and analysis of flow characteristics around the seed-inspired blade design.

Initially, the mesh was refined three times to ensure a high level of accuracy in the simulations. The mesh refinement process included: Utilizing linear elements; Transitioning to quadratic elements; Gradually decreasing the element size to capture finer details of the flow dynamics.

These refinements were crucial to obtaining the best results for the pressure and velocity distributions around the seed-inspired blade. ANSYS Fluent was shown to be a valuable tool, allowing detailed visualization and analysis of the pressure and velocity fields. The convergence of the results further validated the reliability of the simulations, providing a clear view of the flow behavior and informing the next steps in the design process.

In several simulations, regions of negative pressure were observed around the ash seed as air flowed past it. These negative pressure values are relative to atmospheric pressure and do not imply an absolute vacuum. Instead, they highlight areas where the local pressure is significantly lower due to accelerated airflow over the seed's surface.

This phenomenon, driven by the seed's shape and aerodynamic properties, facilitates its natural rotation and dispersal. Understanding these pressure differentials is crucial for designing efficient vertical wind turbine blades. By mimicking the ash seed's ability to create low-pressure zones, we can potentially enhance the aerodynamic performance and rotational efficiency of turbine blades, ultimately contributing to more effective energy generation.

The findings suggest that the ash seed possesses significant potential for vertical wind turbine applications. Its unique shape and natural tendency to rotate during free fall minimize turbulence and associated vibrations, which are critical factors in improving the efficiency and stability of turbine blades. This natural aerodynamic efficiency inspired by the ash seed can be harnessed to develop a more effective and sustainable wind turbine prototype.

This research represents the initial steps toward leveraging biomimicry in wind turbine design. Further studies are recommended to optimize the blade design and conduct experimental validations. Exploring different seed shapes and incorporating more complex fluid-structure interactions could yield even more efficient turbine models. The promising results obtained in this study lay a solid foundation for ongoing and future research in this innovative area.

7.1 FUTURE DEVELOPMENTS

With the results obtained, validated and the start of the prototyping process, it can be said that the next steps may be to complete a real small-scale generator and verify the real possibility of creating a power from a blade design inspired by an ash seed. By continuing to innovate and research in these areas, it is possible to see a great perspective in key role like: offshore wind turbines, smart grids, material and design innovations, etc.

REFERENCES

Mazzolai B, Beccai L and Mattoli V (2014) Plants as model in biomimetics and biorobotics: new perspectives. Front. Bioeng. Biotechnol. 2:2. doi: 10.3389/fbioe.2014.00002

S. Sudo, N. Matsui, K. Tsuyuki, and T. Yano, "Morphological design of dandelion in Proceedings of the XIth International Congress and Exposition" (Society for Experimental Mechanics, 2008).

R. Nathan, F. M. Schurr, and O. Spiegel, "Mechanisms of long-distance seed dispersal trends". Ecol. Evol. 23, 638 (2008). https://doi.org/10.1016/j.tree.2008.08.003, Google ScholarCrossref

H. N. Ridley, "On the dispersal of seeds by wind," Ann. Bot. 19, 351 (1905). https://doi.org/10.1093/oxfordjournals.aob.a089006.

https://us.orsted.com/renewable-energy-solutions/offshore-wind/what-is-offshore-wind-power/how-do-offshore-wind-turbines-work Accessed: October 2022

B. Deori, S. Barman, S. Das, M. Hussain, S. M. Basumatary and K. K. Sharma, "Experimental Study on the Performance of Lenz Vertical Axis Wind Turbine". Journal of Material Science and Mechanical Engineering (JMSME) Print ISSN: 2393-9095; Online ISSN: 2393-9109; Volume 2, Number 9; April-June, 2015.

Greene, D.F. and Johnson, E.A. (1990) The Aerodynamics of Plumed Seeds. Functionas Ecology, 4, 117-125. https://doi.org/10.2307/2389661

Nathan, R. (2006) Long-Distance Dispersal of Plants. Science, 313, 786-788. https://doi.org/10.1126/science.1124975

Patel, Karna S., Saumil B. Patel, Utsav B. Patel, and Prof Ankit P. Ahuja. "CFD Analysis of an Aerofoil." International Journal of Engineering Research 3, no. 3 (March 1, 2014): 154–58. http://dx.doi.org/10.17950/ijer/v3s3/305.

King, Matthew Lee. "A CAD-centric Approach to CFD Analysis With Discrete Features." Diss., CLICK HERE for online access, 2004. http://contentdm.lib.byu.edu/ETD/image/etd570.pdf.

https://simutechgroup.com/ansys-15-cfd-enhancements/

https://www.alternative-energy-tutorials.com/wind-energy/wind-speed.html

https://todayshomeowner.com/eco-friendly/guides/how-much-power-does-a-wind-turbine-produce/