



POLITECNICO
MILANO 1863

SCUOLA DI INGEGNERIA INDUSTRIALE
E DELL'INFORMAZIONE

Distributed Electric Propulsion Applied to Micro Aerial Vehicles

TESI DI LAUREA MAGISTRALE IN
AERONAUTICAL ENGINEERING - INGEGNERIA AERONAUTICA

Author: **Davide La Ferla**

Student ID: 248601

Advisor: Prof. Alex Zanotti

Co-advisors: Roberto Albertani, Associate Professor (Ret.)

Academic Year: 2024-25

Abstract

Micro Aerial Vehicles were originally defined as a category of aircraft with the maximum dimension of 150 mm, a maximum mass of 50 g and operating at speeds around 10 m/s. At their typical Reynolds numbers of 50,000 or lower, conventional airfoils experience a significant degradation in aerodynamic performance. Zero-thickness airfoils that are employed to mitigate the drop of performance, decrease the vehicle's payload capabilities. Recently developed Distributed Electric Propulsion (DEP) was applied on conventional aircraft to improve their aerodynamic characteristics at lower flight velocities. This work focused primarily on the development of an original capability to investigate a novel application of DEP to MAVs aimed at improving the performance of conventional airfoils at low and very-low Re numbers. Wind tunnel testing was performed at the Politecnico di Milano using a six-axis micro-sting balance for force and moments estimates. The model, with a rectangular wing of 200 mm wingspan and an Eppler E231 airfoil, was powered by six DC micro-motors and six micro-propellers evenly distributed along the wingspan and blowing on the wing. The numerical investigation was carried out using DUST, a particle-panel solver, augmented by Xfoil, NeuralFoil, and SU2 for detailed airfoil analyses. The good agreement between numerical and experimental results validated the application of numerical models to the subject configurations. Experimental results at 0° angle of attack with the wing in the propeller's wakes showed an increase of the C_L by 83.8% at $Re=26471$ and by 162% at $Re=13235$ with reference to the wing without propellers. At a angle of attack of 7.5° and a propellers' speed of 60000 RPM, numerical results showed that lift is primarily wing generated, with wing's C_L increasing by 112.7% at $Re=26471$ and 303% at $Re=13235$.

Keywords: Low Reynolds number, DUST, Micro Aerial Vehicles, Distributed Electric Propulsion, Wind Tunnel, Micro Propeller, Wing Blowing

Abstract in lingua italiana

I Micro Aerial Vehicles (MAV) sono stati originariamente definiti come una categoria di velivoli con dimensioni massime di 150 mm, una massa pari o inferiore a 50 g e in grado di operare a velocità di 10 m/s. Con numeri di Reynolds operativi dei MAV pari o inferiori a 50.000, i profili alari convenzionali subiscono un significativo degrado delle prestazioni aerodinamiche. I profili alari a spessore nullo, impiegati per mitigare il calo delle prestazioni, riducono significativamente la capacità di carico utile del veicolo. La propulsione elettrica distribuita (DEP) di recente sviluppo è stata applicata ai velivoli convenzionali per migliorarne le caratteristiche aerodinamiche a velocità di volo inferiori. Questo lavoro si è concentrato principalmente sullo sviluppo di un nuovo metodo per studiare l'applicazione della DEP ai MAV volta a migliorare le prestazioni dei profili alari convenzionali a numeri di Reynolds bassi. I test in galleria del vento sono stati eseguiti presso il Politecnico di Milano utilizzando una micro-bilancia a sei assi per la misura di forze e momenti. Il modello, con un'ala rettangolare di 200 mm di apertura alare e un profilo alare Eppler E231, era alimentato da sei micro-motori a corrente continua e sei micro-eliche distribuite uniformemente lungo l'apertura alare. L'indagine numerica è stata condotta utilizzando DUST, un risolutore a particelle-pannelli, supportato da Xfoil, NeuralFoil e SU2 per analisi dettagliate del profilo alare. La buona concordanza tra risultati numerici e sperimentali ha convalidato la fattibilità dell'applicazione dei modelli numerici alle configurazioni in oggetto. I risultati sperimentali a 0° hanno mostrato che le eliche aumentavano il C_L dell'ala dell'83,8% a $Re=26471$ e del 162% a $Re=13235$. A $7,5^\circ$ e 60000 giri al minuto, i risultati numerici hanno mostrato che la portanza è ancora generata principalmente dall'ala, con il C_L di quest'ultima che aumenta del 112,7% a $Re=26471$ e del 303% a $Re=13235$.

Parole chiave: Bassi Numeri di Reynolds, DUST, Micro Velivoli, Propulsione Elettrica Distribuita, Galleria del Vento, Micro Eliche, Soffiaggio su Ala

Contents

Abstract	i
Abstract in lingua italiana	iii
Contents	v
1 Introduction	1
2 Experimental Setup	9
2.1 MAV Model Definition	9
2.1.1 Model's Design and Manufacturing	12
2.2 Wind Tunnel Setup	14
2.2.1 Sting Balance and Calibration	25
2.3 Data Post Processing	32
2.3.1 Aerodynamic Coefficients	32
2.3.2 Experimental Uncertainty	35
2.3.3 Propulsion Parameters	35
3 Numerical Analysis	37
3.1 Software	37
3.2 Airfoil analysis with SU2	41
3.3 Wing's Model and Panel Distribution	47
3.4 Propeller's Model and Panel Distribution	51
3.5 Numerical Full Model	55
4 Results and Discussion	57
4.1 Wing Aerodynamics	57
4.2 Propulsion's Performance	65
4.3 Complete model results	70
4.4 Physical Insights	75

5	Conclusions and Future Work	89
	Bibliography	93
	List of Figures	99
	List of Tables	103
	List of Symbols	105
	Acknowledgments	107

1 | Introduction

Near the end of last century, the USA Defense Advanced Research Projects Agency (DARPA) established standards for Micro Aerial Vehicles (MAV) a category of aircraft with the maximum dimension of 150 mm, a mass of less than 50 g and capable of operating at speeds of 10 m/s or less [1]. Over the years, these definitions evolved and depending on the sources, MAVs could weight almost 200 g and span around 20 cm.

Such vehicles had the primary mission of reconnaissance and surveillance while being operated by a single soldier. However, variable payload options for a variety of missions were integrated as new projects came from research efforts and a number of them reached operational conditions.

Due to the limited dimensions and low velocities, the flight regime of MAVs is the typical low Reynolds number (Re), with maximum values around 5×10^4 . The behavior of the airflow at those Re is different with respect to usual aeronautical applications due to the greater impact of viscosity, which acts near the surface of the airfoil. In this condition conventional and smooth airfoils don't perform as well at higher Re, since they're engineered for different operating conditions. As summarized by Winslow [2] and illustrated in Fig. 1.1, the conventional airfoils exhibit a dramatic drop at lower Re, the typical flight condition for MAV applications. At lower Re, flat plate airfoils can outperform conventional smooth airfoils.

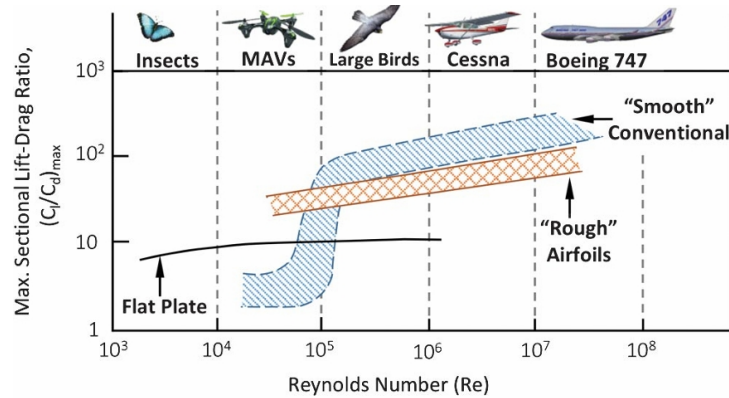


Figure 1.1: Effect of Reynolds number on airfoil maximum sectional lift-to-drag ratio[2].

At low Reynolds number, in the range of 10 to 100×10^3 , the laminar separation point is delayed until close to the trailing edge of the airfoil, even at low angles of attack, because of the increased stability of the boundary layer, which is more resistant to flow transition [3]. The airfoil is effectively in trailing edge stall for most or all of its operational range. This separation increases as Reynolds number is further lowered, reducing even more the aerodynamic efficiency of thicker profiles.

Shimdtz [4] reported in 1967 that below Re of 10^5 thin plate airfoils consistently performed better than conventional ones with the flow more susceptible to separation especially for thicker airfoils. At those Re the aerodynamic coefficients strongly depended on Re for thicker airfoils, while for thin airfoils and flat plates, aerodynamic performance were invariant with Re .

In particular Winslow showed, with CFD simulations, that the ideal airfoil for low Reynolds application should be very thin, and with camber below 6%. NACA 6403 and 4403 were found to be the best profiles analyzed. In particular, the NACA 0012 performed so poorly that the reversed configuration performed better due to the sharp leading edge, leading to a laminar bubble separation, which reattached before the airfoil trailing edge.

A significant body of work on the experimental characterization and design requirements of MAVs can be attributed to T.J. Mueller, G. Torres and to the MAV Laboratory at the University of Florida. In the early 2000s Mueller introduced a novel test rig for wind tunnel tests on MAV wings at low Re , showing the importance of dedicated instrumentation and data analysis applied to flat and cambered thin plates [3]. Torres analyzed different combinations of planform shape, aspect ratio and leading edge geometry for optimal MAV design [5]. It was observed that at such small Reynolds, the most important parameter was the aspect ratio (AR). If the latter was below 1, the coefficient of lift was seen to be non-linear with respect to the angle of attack, while as AR increases, the typical linear relationship was gradually restored. At the same time, the best performing planform shape was found to be the inverse Zimmerman. This shape is obtained by the union of two ellipses, with the same major axis and the minor axis which are one the triple of the other. The smaller ellipse should in the back (trailing edge), from here the 'inverse' classification. Finally, the leading edge shape was examined and it confirmed what already seen in Shimdtz, that a sharp leading edge performs better at low Reynolds numbers.

The first operational practical example of a MAV was developed by AeroVironment Inc. [6, 7]. The vehicle, with a mass of 80 g, designed using Multidisciplinary Design Optimization (MDO) methodology, achieved flights of 30 minutes endurance at a maximum level speed of 13 m/s. Its wing, with a 8.4% maximum thickness conventional airfoil

shape provided, at cruise level flight, a lift coefficient of 0.42 with a lift-to-drag ratio of 6.0. The custom design propellers, with maximum efficiencies of 80% and 68% for geared and direct drive respectively, were coupled to a direct current motor. The total propulsion maximum efficiencies were 40 and 43% in the two cases.

MAVs equipped with thin airfoils (virtually zero-thickness) and inverse Zimmerman wing planforms were extensively studied in a specifically designed wind tunnel [8] with and without propulsion and flight tests at the University of Florida and at Oregon State University. An improvement in lift and lift-to-drag characteristics was achieved by bio-inspired design mimicking bats on a series of wings with zero thickness and highly elastic structures. For the first time, aerodynamic wind tunnel tests were coupled with static and dynamic measurements of the wings' passive shaping using Digital Image Correlation (DIC) system with stereo cameras [9]. Results showed the significant improvement of a full membrane-wing with a selected membrane's pretension versus rigid and flexible carbon-fiber and pliant membranes mixed wing structures with identical shape. The wings exhibited a natural passive variable geometry with dynamic pressure and angle of attack, increasing the camber up to 11% from zero to maximum angle of attack exhibiting a significant increase in lift and pitching stability. An increase of 32% on lift coefficient and a significant improvement in pitching stability, a critical factor in MAV's flight control, was also observed in the full-membrane wings. A significant coupling with propeller speed was also observed due to the effects of propeller slipstream on the elastic wing's membrane shape [10]. The extreme flexibility and pressure shape-changing wings potentially offer a novel flight control methodology. Research at the Politecnico di Milano and Oregon State University have developed a numerical technique for the aerodynamic load estimation on MAVs wings based on the actual strain distribution on the membrane during wing tunnel tests [11]. Based on fluid-structure cosimulation inverse problem the membrane shape and the coefficient of lift could be estimated from measured strain on the membrane, opening to the possibility of a real time solution for flight control.

Propeller-wing interaction represents one of the most emblematic cases of aero-propulsive coupling in aeronautics. Since its early origins, when vortex theories of propellers, based on the classical helical wake model and optimal circulation distributions, provided the first coherent framework to describe induced velocities and wake structure, it became evident that the presence of a lifting surface immersed in an accelerated slipstream could not be treated as a mere superposition of independent effects. Starting from Betz theory of 1919, Okulov et al. [12] proposed a formulation for the interaction between the propeller and its hub, considering a semi-infinite hub.

In its most essential physical representation, a propeller generates an increase in axial ve-

locity (slipstream contraction), a tangential velocity component (swirl), and a rise in total pressure; when this wake impinges on a wing, it locally modifies the dynamic pressure and the effective angle of attack, alters the circulation distribution, and, in nonlinear regimes, can either delay or promote flow separation depending on propeller placement, advance ratio, loading, and wing–propeller spacing. These causes and effects were analyzed in the second half of last century with funding by NASA’s programs [13, 14].

This complex interaction gives rise to three main classes of mechanisms, already formalized in classical wake–interference studies:

1. the increase in local dynamic pressure ($q = \frac{1}{2}\rho U^2$) over wing portions immersed in the slipstream, leading to higher lift coefficients under equal angle of attack and freestream velocity and enabling propulsive high-lift concepts;
2. modifications of induced incidence and downwash, with consequent changes in lift curve slope and induced drag characteristics;
3. three-dimensional and unsteady effects associated with swirl, wake deformation or instability, and boundary layer interaction, which may introduce load non-uniformities, profile drag penalties, and vibro-acoustic issues [15].

From an application standpoint, the traditional inboard tractor configuration has historically been exploited to improve takeoff and landing performance through increases in $C_{L,\max}$ and aircraft control margins, but it also entails critical drawbacks such as non-uniform pressure distributions, asymmetric spanwise loading, sensitivity to partial separation between wing regions inside and outside the slipstream, and potential reductions in overall aero-propulsive efficiency when residual swirl or adverse wake–downwash interference is present. Miley and von Lavante in 1985 [16] conducted a literature survey to identify and review technical material applicable to aerodynamic interference of propeller-wing problems.

Recent experimental and numerical investigations, driven by the renewed interest in UAV and eVTOL aircrafts, such as those carried out by the Department of Aerospace Science and Technology of Politecnico di Milano [17], demonstrate that, depending on operating conditions and installation geometry, such interactions can significantly enhance overall efficiency, particularly at low speeds and high angles of attack, while simultaneously introducing load gradients and flow distortions that require careful design of spacing, inclination, synchronization, and control strategies.

When studying the effects of a number of effectors in a complex physical phenomenon, it is convenient and efficient to introduce dimensionless parameters. One example of such

parameter of great popularity is the Reynolds number. In the case of propulsive effects on aerodynamic characteristics of aircraft, several parameters were applied such as the advanced ratio J in propellers and the reduced frequency k in flapping wings [18]. In the specific case of propellers' effects on fixed wing aerodynamics also the non dimensional fan speed was introduced [19].

In 1957 Kuhn et al. [20] investigated the effect of propeller slipstream on aerodynamic characteristics of a wing equipped with a 50% chord sliding flap and a 30% chord slotted flap, and in doing so he defined C_L'' as the lift coefficient computed considering the slipstream dynamic pressure (q'') instead of the freestream dynamic pressure (q). The new dynamic pressure is defined by adding the two terms, the freestream dynamic pressure and a term which represents the increase in dynamic pressure due to the propeller as the ratio between the propeller thrust and the area of the rotor disk.

In 1938 Stüper [21] investigated the effect of propeller slipstream on wing and tail aerodynamics, using a series of wind tunnel tests. Three types of slipstreams were considered: the first one was a jet without rotation aligned with the freestream, the second one was a propeller slipstream with its axis aligned with the freestream and the third one the propeller slipstream function included the angle with respect to the wing mean chord. To account for the contributions of the increased velocity due to the jet and slipstream, the velocity (U) was modeled as the sum of freestream velocity (U_∞) and relative increase velocity (s) to give $U = (1 + s)U_\infty$. It was found that the jet increased the wing's lift slope and maximum lift delaying the angle at which separation occurred. In particular, separation started outside the jet region, which is of fundamental importance in geometry design, as a banking of the wing due to sudden loss of performance needs to be avoided at all costs. Further testing with a propeller showed that the aerodynamic improvement effects were similar in nature and that the total lift can be split in different components, decoupling the influence of the propeller and the wing. The lift was considered as the sum of the lift generated by the wing itself plus the lift generated by the wing due to the increase in velocity due to the slipstream of the propeller.

Ananda et al. [22] investigated the effects of propeller induced flow on wings at low Reynolds numbers. In particular, a Wortmann wing with moderate aspect ratio was used, at two Reynolds numbers. For $Re = 60000$, a large increase in $C_{L,max}$ was observed as propeller advance ratio decreased. In contrast, an increase in $C_{L,max}$ was not observed for $Re = 80,000$, as the propeller operated in brake state (negative C_T). However, for both cases, a reduction in advance ratio resulted in an increase in the stall angle of attack, accompanied by a decrease in the zero-lift angle of attack. It was also observed that the propeller advance ratio influenced the position and shift of the laminar separation bubble.

The propeller induced flow generated a region of increased turbulent flow on the upper surface of the wing, an effect observed at all angles of attack contributing to the flow reattachment over the downstream portion of the wing.

Within the aeronautical world, distributed propulsion (DP) can be defined (even if formal definition doesn't exist) as a propulsion system where the vehicle thrust is produced from an array of propulsors located across the air vehicle, generally along the aerodynamic surfaces. Moreover, a DP system should operate an improvement of the system-level efficiency, performance and robustness of the air vehicle. Otherwise, any aircraft with two or more propulsors could be classified as such. Within this concept, it is important to make a further distinction between distributed electric propulsion (DEP) and others DP methods. In DEP the thrusting producing devices are linked to the power generating system of the aircraft only by electrical means. In such technology propulsion devices can be specified with greater flexibility with the added benefits from synergistic aero-propulsion coupling [23].

The general idea behind the implementation of DEP, is the use of multiple electrically driven propellers along the wing leading edge to enhance lift during low speed operations, effectively increasing the maximum lift coefficient through boundary layer energization and delayed flow separation. Additional benefits could arise from wingtip-mounted propulsors, which can partially recover induced losses by weakening wingtip vortices, leading to improved aerodynamic efficiency in cruise flight. From a systems perspective, DEP also enables novel control strategies, such as differential thrust for trim control, potentially reducing the reliance on conventional control surfaces.

One prime example of DEP is the X-57 Maxwell aircraft developed by NASA Scalable Convergent Electric Propulsion Technology Operations Research (SCEPTOR) [24]. This experimental aircraft derived from the Tecnam P2006T. A Luigi Pascale's design with a wingspan of 11.4 m and a MTOW of 1,290 kg, was modified as a demonstrator for distributed electric propulsion (DEP) and was equipped with 18 leading edge electric motors with folding propellers with the goal of improving low speed lift. Two wingtip mounted motors are employed in cruise flight. Numerical simulations showed that DEP increases the maximum lift coefficient at takeoff from 3.4 to 5.2 and raises the cruise lift-to-drag ratio from 11 to 20 [25]. Full scale tests confirmed these predictions within 10% [26]. Aeroelastic analyses conducted by Hoover et al. [27] identified whirl flutter instabilities, which were eliminated by stiffening the pylon mount connecting the tip propellers to the wing, ensuring stability up to 500 knots (far beyond the flight envelope). Overall, the X-57 validated the potential of DEP to improve aerodynamic efficiency and stability in future electric aircraft.

The improvement of aerodynamic performances on MAVs at low Re, especially at high AOA, was studied with the application of fixed Gurney flaps on membrane-wings, showing a maximum increase in maximum lift coefficient and lift-to-drag ratio of 26.5% and 22.4%, respectively, at Re numbers of 73,300 and 95,200. Another method that was investigated in wind tunnel tests was the application of a passively deployable flap on the suction side of the wing, close to the trailing edge. The measured average increase in maximum lift coefficient varied from 5% to 15% for different condition of membrane pre-tension and Re number [28].

In recent years, aerodynamic numerical analysis became a fundamental aspect of aeronautical engineering, and its importance shows no signs of diminishing. Regarding MAVs, many different numerical studies have been carried out using CFD tools and simulations. Seminal research was conducted at the University of Florida with the development of original CFD applications on fixed and flapping wings MAV's [18]. More recently, Schroeder and Baeder [29] analyzed the flow around thin, highly cambered airfoil in two and three dimensional flows, using the OVERFLOW and TURNS flow solvers. A low-Mach preconditioner was applied to accelerate convergence and increase accuracy, achieving reasonably good agreement with experimental results. Another example of CFD simulations applied to MAVs was developed by Barroso et al. [30], where bird-inspired tails were analyzed for low Reynolds number applications.

Propeller-wing interactions, represent another common subject of CFD investigations, encompassing a wide range of research topics. In particular, for low Reynolds number applications, Choi et al. [31] modeled the single propeller of a MAV as an actuator disk to study the effects of propeller induced blowing, showing that flow separation was delayed and reattachment was promoted. Following the same idea, Chen et al. [32] used a sliding mesh approach to model the propeller's effects on a Zimmerman wing-fuselage model analyzing its effect on the wing aerodynamic performance.

In this study, a methodology for the improvement of the aerodynamic performance of a conventional "thick" airfoil wing for Micro Aerial Vehicles (MAVs) through the application of Distributed Electric Propulsion (DEP) at low Reynolds numbers is investigated. While zero-thickness wings typically exhibit higher aerodynamic efficiency in low Reynolds number regimes, a thick airfoil configuration offers significant practical advantages for operational MAVs. In particular, it provides internal volume within the wing for the integration of energy storage systems and payload components, a feature that is not achievable with ultra-thin wing designs.

The analysis is carried out through both numerical and experimental approaches. On the

numerical side, particular attention is devoted to assessing the prediction accuracy of the solver adopted for the study, namely DUST. The investigation focuses on evaluating the limits of applicability of the Vortex Particle Method (VPM) implemented in DUST, which has previously been employed for full scale eVTOL aircraft and rotorcraft. In this work, its performance is examined in the context of small-scale MAV configurations operating at low Reynolds numbers. The limits of the method are identified through systematic comparison with experimental measurements. Then, once the validation is terminated, DUST would be used to investigate flow phenomena at this scale and compare them to what happens in full scale models, in order to see difference and similarity of a wing acting inside a propeller stream at different Reynolds regimes.

The experimental campaign is conducted at the small wind tunnel of the Politecnico di Milano. The work begins from scratch with the design and manufacturing of a dedicated test model equipped with a distributed propulsion system. A custom test rig is developed to accommodate the model and ensure proper alignment and repeatability during testing. Load measurements, which are extremely small and on the order of grams, are performed using a newly acquired micro sting balance. A dedicated data processing methodology is implemented to ensure reliable acquisition and interpretation of the measured forces and moments.

One of the primary objectives of the thesis is therefore the creation of a complete experimental setup for investigating distributed electric propulsion applied to small scale vehicles operating in low Reynolds number regimes, such as MAVs. This includes the integration of the propulsion system into the model, the validation of the measurement chain, and the assessment of the feasibility and accuracy of the load measurements enabled by the micro balance.

Furthermore, the work aims to transfer and adapt the expertise developed within the Aerospace Science and Technology Department of the Politecnico di Milano in the field of eVTOL aircraft and rotorcraft to smaller scale vehicles. This translation of knowledge allows established methodologies and numerical tools to be evaluated and refined for MAV applications.

To enable a clearer interpretation of the combined effects of MAV aerodynamics and distributed propulsion, specific simplifications are introduced in the project, which are later described in detail. Overall, the study seeks to provide both a validated experimental framework and a critical numerical assessment, contributing to a deeper understanding of DEP integration on thick airfoil MAV wings at low Reynolds numbers.

2 | Experimental Setup

2.1. MAV Model Definition

Due to the originality of this work, including the lack of any existing reference, the general shape of the model was selected as a rectangular wing planform, a recurrent choice for preliminary work in the aerodynamics of MAV's.

An Eppler E231 airfoil was selected so its thickness of 12% would permit a better payload and systems integration in operational vehicles, while offering better lift-to-drag ratio at low Re numbers with respect to the more documented NACA0012 [33].

The wing has a span of 200 mm with a chord of 40 mm. These measures were chosen to have the maximum aspect ratio, while respecting the constraint of the maximum dimension of MAVs, and still having enough space inside the wing to have the motors and respective wiring. The model has an aspect ratio of 5.

Additionally, an elliptical fuselage was added in the center of the wing as the joint for the sting balance.

Along the span, the nacelles for the six motors, and relative propellers, are present, one every 33 mm in order to leave enough space between one propeller and the next. This meant that the entire wing would be under the influence of the propellers' slipstream, as DP systems would require. Fig. 2.1 reports the dimensions of the model.

For the model under consideration, the aerodynamic chord, the balance axis, and the motor-propeller axis are mutually aligned and coincide with the 0° angle of attack reference of the model.

In this project the distributed electric propulsion consisted in a series of six commercially available DC motors with their propellers obtained from a Cheerson CX-10 commercial toy drone [34], which is represented in Fig. 2.2. Two drones were necessary to have all the motors needed, with spares in case of failure.

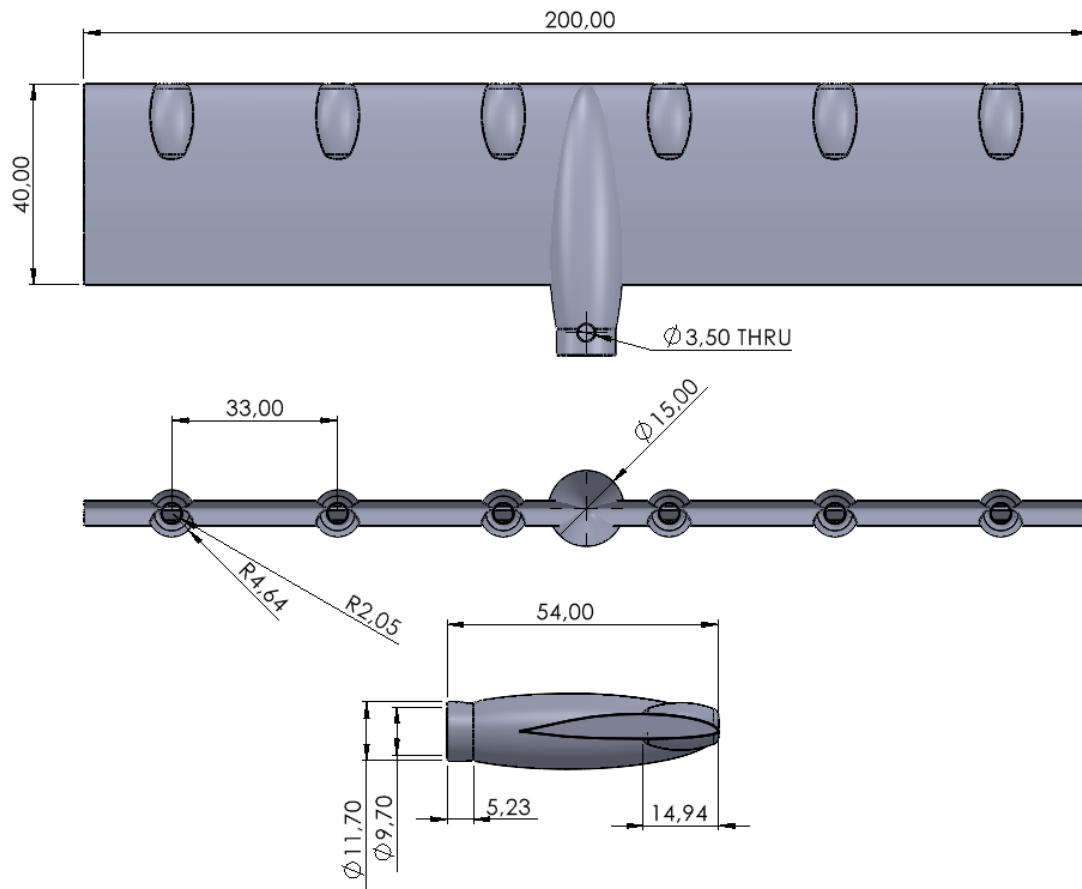
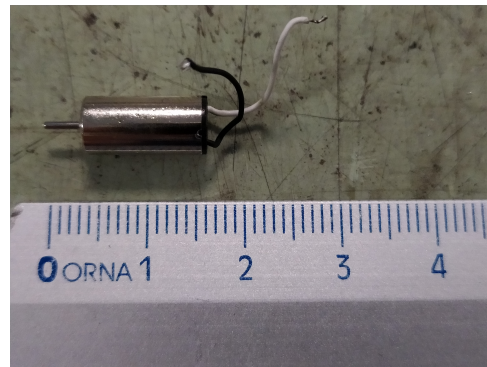


Figure 2.1: Top view, front view and right view of the model with dimensions in millimeters.



(a) Cheerson CX-10 drone.

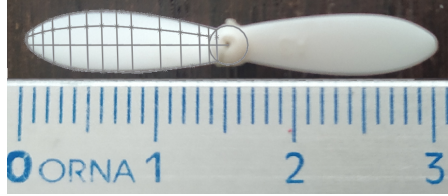


(b) DC motor with measure.

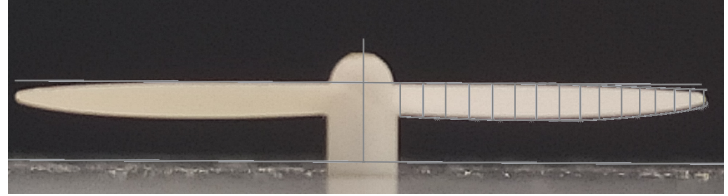
Figure 2.2: Cheerson CX-10 drone used with detail on the DC motor.

One of the motors, which measured 12 mm in length and 6 mm in diameter, is shown in Fig. 2.2b. The motor features a shaft length of 6 mm, onto which the propeller is mounted using an interference-fit joint.

The propellers taken from the commercial drone had a diameter of 30.4 mm and the space between each propeller's blade tip of 2.6 mm. The planform and the front view of the propeller are reported in Fig. 2.3. The propeller airfoil was very thin, consistent with the requirement for thin profiles at low Reynolds numbers. Another adimensional parameter useful to characterize the model is the distance between propeller tips, expressed as a fraction of the radius of these latter. For the model in question such ratio is 0.17.



(a) Planform shape of propeller blade.



(b) Twist profile of propeller blade.

Figure 2.3: Propeller blade planform shape and twist profile from top and lateral view respectively.

The distance between propellers' tips is maintained from the blade tip of the most external propeller to the wing tips, maximizing the surface of the wing interested by the propellers slip stream. The rotational direction of the six propellers of the full model is presented in Fig. 2.4.

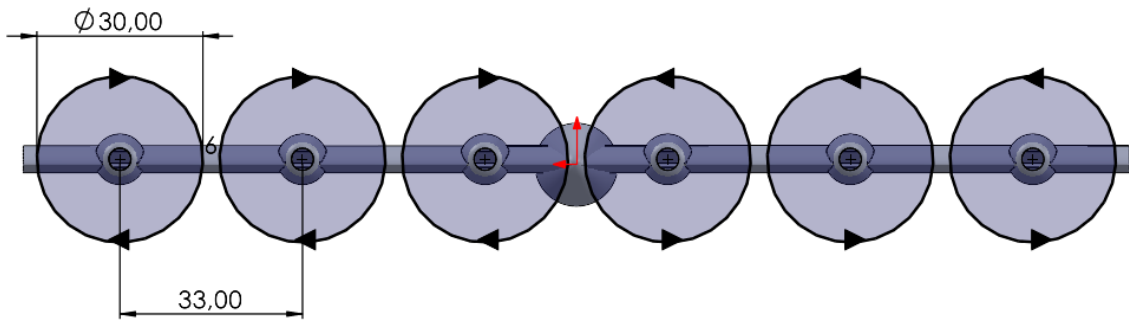


Figure 2.4: Front view of the full model with rotation direction of the propellers.

This configuration was selected for two main reasons. First, the propellers within each semi-span were selected to rotate in the same direction to create a simple reference base-line. Second, the outboard rotors were selected to rotate opposite to the wing tip vortices

to counteract the associated downwash effects. Further studies on the effects of propellers' rotation and their effects on the wing's performance could be run in the future.

2.1.1. Model's Design and Manufacturing

The limited dimensions of the wind tunnel model and low levels of dynamic pressure during testing allowed the fabrication of 3D printed structures, facilitating the realization process once the design was completed.

To ease the assembly process, in particular for motors installation inside the wing, since they are thicker than most of the airfoil, the model was divided into 5 parts, as illustrated in Fig. 2.5. All components were designed in Solidworks and then bonded together once the wiring of the motors was done.

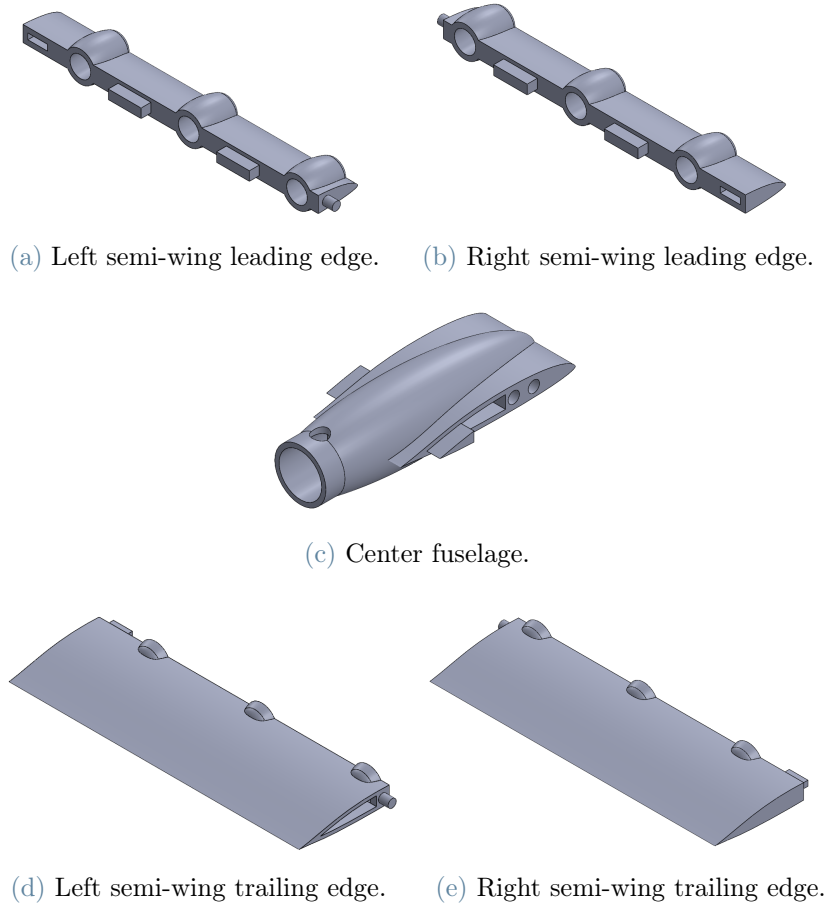


Figure 2.5: Five components of the model for production.

The leading edge components (Fig. 2.5a and Fig. 2.5b) incorporate the majority of the nacelles and include protrusions that serve as supports for the trailing edge section, along

with a single alignment pin on the fuselage side to ensure proper positioning. The main body of the leading edge was printed solid in terms of filler material. The nacelles were designed optimizing the shape for best airflow, with their minimum thickness dictated by the 3D printer. The minimum dimensions of the nacelles internal geometry were restricted by the DC motors, which are reported above.

The trailing edge components of both half wings (Fig. 2.5d and Fig. 2.5e) were hollow, to allow the wires to be run inside the model to the sting balance's fixture point. In addition two alignment pins were designed, one for the leading edge in the tip section of the component and the other for the pseudo-fuselage. The trailing edge also included the end section of the nacelles, making the motors a structural element of the wing structure.

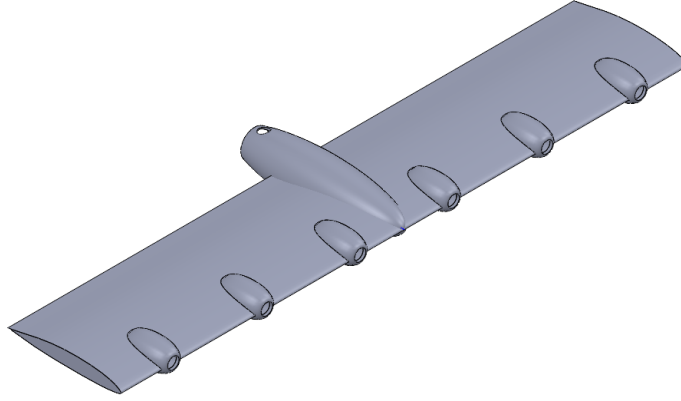
In the center of the wing, an ellipsoidal fuselage was added, Fig. 2.5c, with the primary function of streamlined connection and housing for the sting balance. To fully couple the model and the sting balance, a circular cylinder extrusion and a 3.5 mm diameter hole for the dowel pin were introduced in the ellipsoid beyond the trailing edge. The fuselage was the most challenging component to design due to the precision required by the numerous functions it had to fulfill. Its primary role was to accommodate the sting balance, but it also acted as the junction between the five separate components of the model. Furthermore, most of its internal volume needed to remain hollow to allow routing of the six DC motors wiring, while ensuring that the wire exit path did not mechanically interfere with the balance mounting and with the balance's measurements.

The model was manufactured using the Creality K1 Max 3D printer and modeled by the Creality Print slicing software. By the application of this methodology, the production time was limited to about three hours, and the total cost of the model in terms of material did not reach € 1.

The wiring of the motors on the wing proved to be the most complex task in the manufacturing process, as a total of 12 wires had to be routed within a very limited available volume. Each motor was originally supplied with two leads of approximately 20 mm in length, as they were salvaged from the small Cheerson drone in Fig. 2.2. Consequently, the first step consisted in soldering extension wires to each motor in order to obtain manageable cable lengths.

Throughout the entire process, strict monitoring of color coding was of paramount importance to ensure correct polarity and avoid connection errors. The length of the cable extensions was such that the wires could be routed along the full support structure (3D printed plus metal parts) and reach the exit of the wind tunnel.

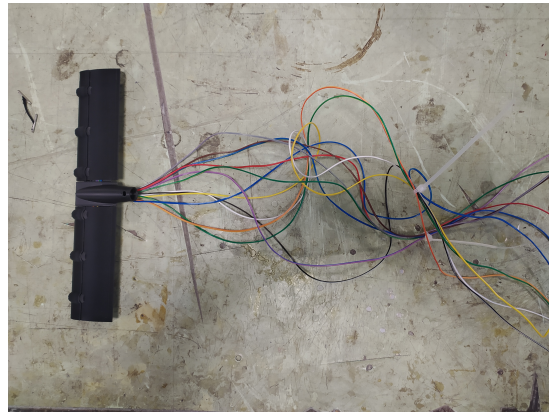
The final model is represented in Fig. 2.6, as both the CAD model and the final product.



(a) CAD model of the wing with the nacelles and the pseudo-fuselage.



(b) Printed model, without the motors and wires.



(c) Full model complete of motors and wires, after bonding.

Figure 2.6: Complete model of the wing with pseudo-fuselage and nacelles.

2.2. Wind Tunnel Setup

The experiments for this project were run at the academic wind tunnel facility at the Politecnico di Milano, also known as "*Galleria Verde*". In the following table (Table 2.1) the main parameters of the facility are reported.

The closed-loop wind tunnel is equipped with three pressure transducers which measure respectively the static pressure across the convergent section (1 and 2 in Fig. 2.7) and the total pressure at the beginning of the test chamber (*Pitot* in Fig. 2.7)[35].

Wind Tunnel Parameters

Parameter	Value
Test Chamber Width	300 mm
Test Chamber Height	450 mm
Test Chamber Length	1000 mm
Maximum Velocity	36.5 m/s
Mean Turbulence Intensity	0.3 %

Table 2.1: Parameters of the Green Wind Tunnel.

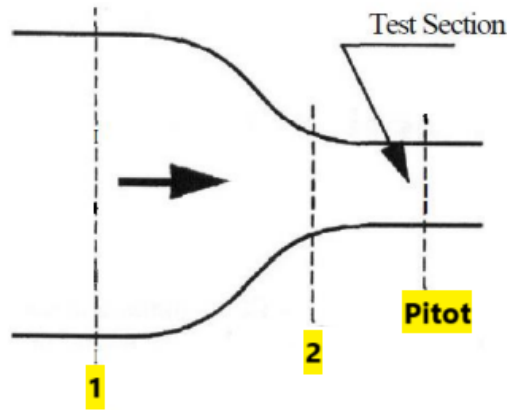


Figure 2.7: Lateral view sketch of convergent section and test chamber, with pressure transducer position.

The pressure transducers give the differential pressure across the convergent section, providing the necessary data for the estimation of the velocity in the test chamber.

The outputs of the wind tunnel transducers are acquired through a National Instruments NI-9205 module, which provides 32 input channels and is installed on a National Instruments Compact DAQ-9174 chassis. Each channel receives an analog ± 10 VDC signal, which is converted into a 16-bit digital representation over the same voltage range. Data visualization and monitoring are handled via a dedicated LabVIEW Virtual Instrument (VI) program.

Data acquisition is carried out by sampling the signals from all transducers at a frequency of 1000 Hz over a duration of 1 s. Consequently, each pressure measurement corresponds to the average of 1000 samples.

For the proper installation of the model mechanically connected with the sting balance with their centerline aligned with the centerline of the wind tunnel, a custom metal support was designed and manufactured. This support was in the shape shown in Fig. 2.8 and was realized starting from a single 10 mm raw iron bar, which was cut and welded in the final shape. The major dimension was such that once inclined of $\pm 30^\circ$, the back part did not touch the floor (or ceiling) of the test chamber. The main length resulted in about 400 mm. The shape of the metal support was chosen in order to maintain a symmetry during the experiment, thus the second half of it. Further more the free end of the metal was equipped with a through hole that made it possible to join it to the other wall of the wind tunnel. Moreover, the shape of the support was such that the model will stay at the same spatial position when changing the AOA.



Figure 2.8: Metal support.

Finally, another 3D custom designed and manufactured component was connecting the metal support to the balance. This piece, shown in Fig. 2.9, features an arch on one end designed to fit around the metal support with enough clutch power to ensure the coupling. To remove the degree of freedom remained, the 3D printed component was also bonded to the metal support. At the other end of this support, the balance tip will be fitted through a hole. A small groove was added on the side of the hole to allow the wires coming out from the balance. The upper part of the support, has a flat surface to place the digital inclinometer between wind tunnel tests for AOA measurements calibration and validation. This surface was used as the reference surface with respect to which every angle of attack was computed.

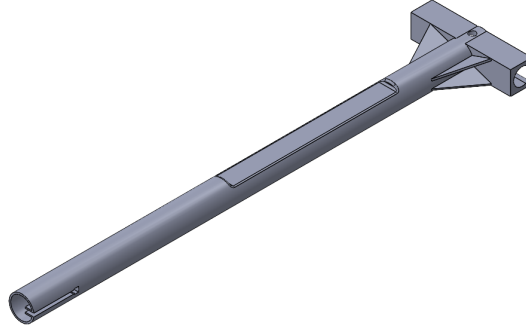
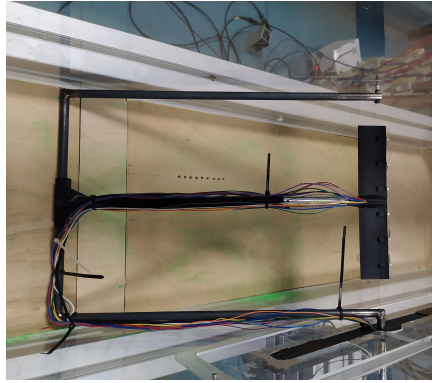
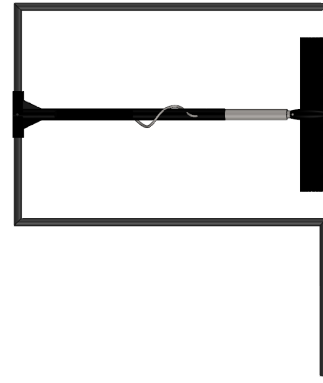


Figure 2.9: 3D printed support for the balance.

The assembled model, with sting balance and supporting arms is illustrated in Fig. 2.10.



(a) Model and supports mounted.



(b) Model and supports CAD.

Figure 2.10: Model, sting balance and supports, mounted inside the wind tunnel and in its CAD model.

The metal support served as the mounting interface between the test rig and the external structure of the wind tunnel. The rightmost end of this support (shown in Fig. 2.8) was connected via an elastic joint to a Parker BE231DQ-NPSN high power-density servo motor (see Fig. 2.11). The elastic joint was designed to transmit only rotational loads between the electric motor and the test rig, isolating other mechanical stresses.

The Parker BE231DQ-NPSN is a brushless AC servo motor engineered for high torque density and performance in a compact package. This motor employ high-performance neodymium magnets and is capable of delivering high continuous stall torque. Additionally, it includes a 2000-line incremental encoder for precise position feedback and control.

In the current test rig, the servo motor enabled high-accuracy and high-repeatability control of the wing's angle of attack, ensuring consistent positioning loading during the

wind tunnel experiments.

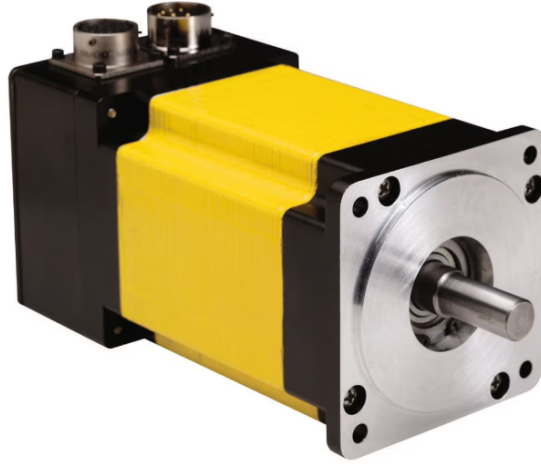


Figure 2.11: Parker BE231DQ-NPSN high power density servo motor.

The test rig's control of angular motion for the model was facilitated by a Parker PX60-040-S2-MU60-033 inline helical planetary gearmotor, shown in Fig. 2.12, mounted between the servo motor and the structural support. This unit integrates a PX60 series mid-precision inline helical planetary gearhead with a matched servo motor interface designed to improve torque delivery, motion control precision, and mechanical stiffness.

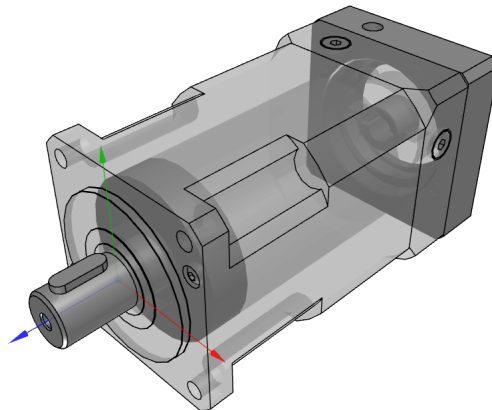


Figure 2.12: Parker PX60-040-S2-MU60-033 gearhead.

The PX series gearheads are engineered to provide a balance of high torque, low backlash, and compact size, making them suitable for servo-driven positioning tasks where smooth, repeatable motion is critical. The gearhead design uses a helical planetary gearset, which

enhances torque transmission efficiency and reduces operational noise compared to conventional gear types. These gearheads are configured in an inline arrangement that simplifies integration with brushless servo motors and facilitate direct coupling to shafts or couplings in motion control assemblies, like the one employed for this setup.

The “040” ratio designation implies a 40:1 reduction within the gearhead, amplifying the motor’s torque at the output shaft while allowing the commanded rotational motion to be finely resolved at the test rig interface.

Collectively, the PX60-040-S2-MU60-033 and the BE231DQ-NPSN assembly enables precise and repeatable angular positioning of the aerodynamic model in the wind tunnel, with a mechanical transmission capable of sustaining load variations without introducing significant backlash or control artifacts.

The full assembly of servo motor, gearhead and flexible joint is presented in Fig. 2.13 mounted on a bosh truss structure adjacent to the wind tunnel.

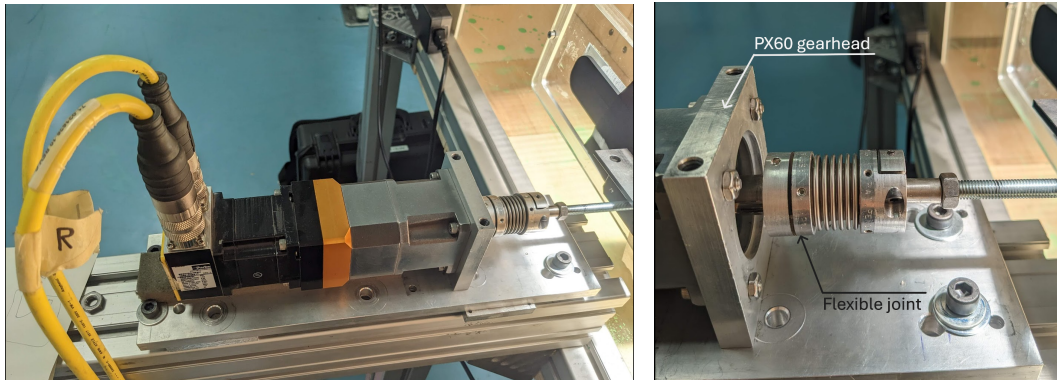


Figure 2.13: Parker BE231DQ-NPSN servo motor and PX60-040-S2-MU60-033 gearhead mounted together and particular with the flexible joint.

The technical data of the gearhead are reported in Table 2.2, together with the exact dimensions. This information derive directly from the manufacturer website [36].

The Parker motor was controlled through its proprietary software, in which it was possible to set the desired angle rotation and monitor in real time the angle of attack of the wing. The zero degree angle of attack was set at the beginning of each day, ahead of any experimental activity, using the reference surface of the 3D printed support. Afterwards, the desired angles of attack were imposed through the control software by applying manual increments of 1° until the target value was reached. This procedure ensured accurate and repeatable setting of the angle of attack for each test run, while minimizing the risk

Technical Data	
Part No.	PX60-040-S2+MU60-033
Series	PS/PX Series
Gearhead Series	PX
Frame Size	60
Ratio	040
Backlash	S2 - Standard
Mounting Kit	MU60-033 - BE23, SM231, SM232, SM233
Square Flange	62 mm (2.441 in)
Bolt Hole	5.5 mm (0.217 in)
Bolt Circle	70 mm (2.756 in)
Pilot Diameter	50 mm (1.969 in)
Output Shaft Diameter	16 mm (0.630 in)
Output Shaft Length	25 mm (0.984 in)
Pilot Thickness	2.5 mm (0.098 in)
Flange Thickness	8 mm (0.315 in)
Housing Diameter	82 mm (3.228 in)
Housing Recess	6.0 mm (0.236 in)
Length	105.3 mm (4.146 in)

Table 2.2: Technical data of Parker PX60 gear head.

of accidental collisions between the model and the wind tunnel floor or ceiling due to potential input errors in the software interface.

Outside of the wind tunnel, a terminal block was used to further extend the wiring using a single 12-pole cable. This terminal block was intended as the point at which the polarity of the wiring could be inverted whenever a change in the motor's direction of rotation was required, even if in the end this didn't occurred. This cable was routed to the box housing the PWM controllers, which was subsequently connected directly to the power supply.

Inside the box, the input positive line was first routed through a master switch controlling the entire propulsion system. The positive and negative supply lines were then split into six individual pairs using another set of terminal blocks, yielding 12 wires that served as inputs to the PWM units. Each PWM output generated an additional pair of wires, which were finally connected to the external 12-pole cable through a second terminal block, completing the wiring architecture.

Special care was dedicated to the potential mechanical and aerodynamic detrimental effects of the motor power wires to the measures of the sting balance.

To avoid, or at least minimizing this problem, two possible configuration were thought. In the first one, the wires were put into a 'U' shape below the balance-model fixture point. In this case the tension of the wires was null and the only contribution regarded the aerodynamic resistance they posed once the wind tunnel was turned on. The second configuration was based on the opposite principle. Supposing the support-balance-model assembly was three springs in series, the best way to minimize interaction was to increase the stiffness of the components. This was done tensioning the wires before linking them to the support. In this configuration the aerodynamic hindrance was minimum, while the structural force passed was unchanged with respect to the other configuration, thus translating into a decreased strain of the components. During the test with the first one it was noted that the zero of the balance was lost after the measurement. This was attributed to the fact that the 'U' configuration, when exposed to the airflow, displaced the wires in such manner that they did not return to the original position after the wind was turned off. This then reflected into a less reliable measure. In the end the second option was chosen because this problem did not occur with such configuration.

To ensure correct and safe routing of the electrical wiring from the model to the power supply, a custom window for the wind tunnel side panel was designed. The component, intended to be 3D printed, consists of two separate parts that fit tightly within the existing opening in the tunnel panel. The first part functions as a fixture, providing structural support and alignment for the assembly. The second part incorporates two openings: a circular hole designed to accommodate the metal support of the model, and an elongated slot that allows the electrical wires to pass through freely, regardless of the angle of attack at which the model is positioned.

The original drone battery provided a nominal output of 3.7 V. However, in order to achieve higher rotational speeds and considering that the DC motors were capable of operating at higher voltages without issues, the supply voltage was increased to 7 V. The motors were controlled by a KERT KAT10VD [37] power supply, represented in Fig. 2.14 provided a constant voltage up to 15 V output and 6 A which results in a maximum power output of 90 W, exceeding the project's requirements.



Figure 2.14: KERT KAT10VD power supply.

For fine tuning during the experiments, each motor was connected in series with a PWM, enabling more precise adjustment of the operating conditions. The PWMs allowed to translate the analog input of the knob into a variable electric current for the motor.

The motors' tuning was set on the RPM of each motor and the total power required was only monitored through the power supply. To measure the rotational velocity of the propellers, a laser tachometer was used.

The instrument used was a Monarch Pocket Laser Tech 200 (PLT200) laser tachometer (in Fig. 2.15a) whose characteristics are reported in Table 2.3.

Specification	PLT200
Pulse duration	Continuous
Laser Wavelength	650 nm
Beam Diameter	4 mm x 7 mm typical 2 meters
RPM Range	5-200,000
RPS Range	0.084-3,333.3
Resolution - Fixed	1 (10 above 99,999)
Resolution - Autoranging	0.001 to 1.0 (10 above 99,999)
Accuracy	$\pm 0.01\%$ of reading or resolution limit
Operating Range	Up to 25 ft. [7.62 m] or up to 70 degrees off perpendicular to target

Table 2.3: Tachometer main characteristics.

The PLT200 ensured high accuracy in measuring the rotational speed of the propeller throughout the entire experiment campaign. The measurement of the RPM of the propellers while the wind tunnel was in function, a special support, represented in Fig. 2.15b, was used to hold the PLT200 with a block of wood used as counterweight. The tachometer was used inside the wind tunnel for propeller speed calibration only, since its interference with the air flow was not acceptable.



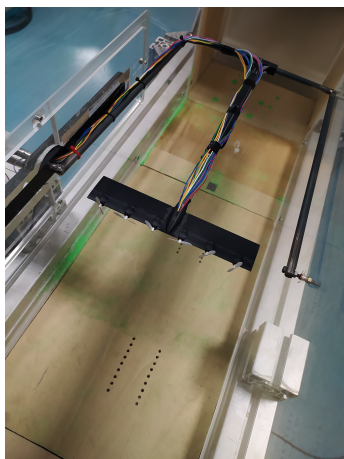
(a) PLT200 Tachometer.



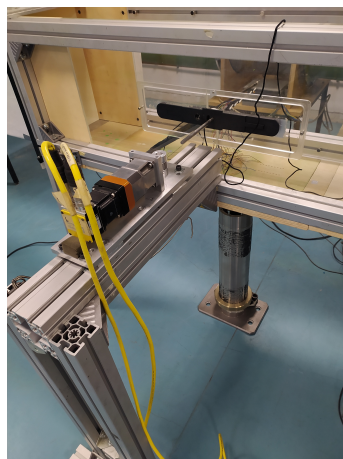
(b) Tachometer mounted on its support.

Figure 2.15: Laser tachometer and its support for wind tunnel operation.

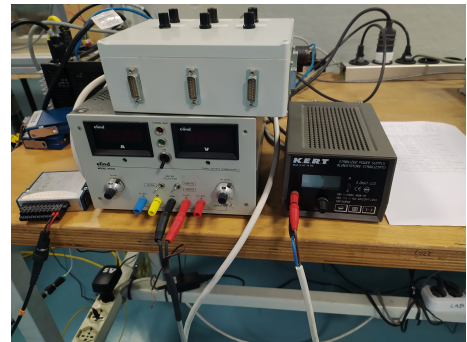
The full wind tunnel model, inclusive of all the details described in this section, is shown in Fig. 2.16.



(a) Inside test chamber.



(b) External motors.



(c) Power supply and control box.

Figure 2.16: Complete experimental setup, with wing, balance, supports, motor, controller and wiring.

In Fig. 2.17 the reference system used in this thesis are reported, with the relative positive direction.

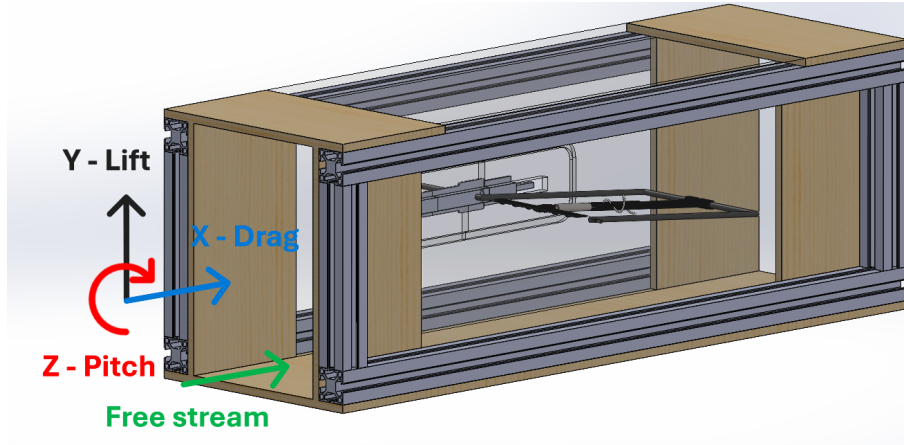


Figure 2.17: Reference system for the wind tunnel results. The signs of the moments follow the right-hand rule.

Finally, the experimental procedure followed for every wind tunnel run consisted of the following:

1. tuning of the propellers to the desired RPM regime, using the tachometer and then turning them off;
2. using the inclinometer to check 0° alignment of the model arm and setting the zero of the external servo-motor;
3. turning on the wind tunnel and setting the correct speed for the test using the relative VI, then turning it off;
4. generate in Matlab a random order array containing the different AOA for the test run;
5. moving the model to the first AOA of the random array using the external servo-motor;
6. setting the zero on the balance VI;
7. turning on the wind tunnel;
8. turning on the propellers;
9. taking the measure for 4 seconds;
10. turning off the propellers and the wind tunnel;

11. Repeat steps 5 to 9 for the number of repetitions of the measure, generally five times;
12. moving the model to the next AOA of the random array using the external servo-motor;
13. Repeat steps 5 to 10 for the total number of measure points.

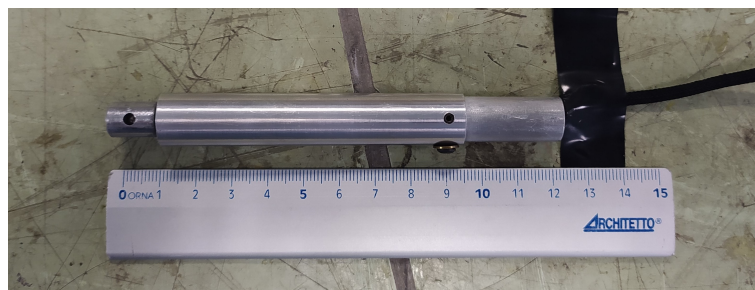
The most critical aspect of the procedure was the randomization of the measurement sequence, implemented to avoid the risk of systematic errors influencing the results. It should also be noted that steps 1, 8, and 10 were performed exclusively during the testing of the Full model configuration.

2.2.1. Sting Balance and Calibration

To measure the loads generated by the wind tunnel model, the Modern Machine & Tool CO., INC. *MC.-10-.375 Balance*, a 0.4 inch diameter & 3.750 inch length wind tunnel sting balance, was used. This particular balance, is a 6 components (axial force, normal force, pitch moment, roll moment, yaw moment and side force) sting balance with high accuracy in all the range of measure and it's represented in Fig. 2.18.



(a) Sting balance without the metal cover.



(b) Sting balance with reference measure in cm.

Figure 2.18: OSU-001 sting balance.

The balance measured structural deformations through six Wheatstone bridges, each associated with one force or moment component. In order to operate correctly, each bridge requires an excitation voltage to establish a reference electrical potential; the resulting voltage variation across the bridge is directly proportional to the strain experienced by the corresponding sensing element. The actual reference tension received by the balance was read by a National Instruments NI-9215 module, while the electrical outputs from the six Wheatstone bridges were acquired using two National Instruments NI-9237 modules, featuring 16 channels each and high resolution analog-to-digital conversion. The different modules were mounted on a National Instrument compact DAQ-9178 chassis; each channel receives an analog signal, which is converted to a 16-bit digital output of the same voltage.

The specifications published by the manufacturer are reported in Table 2.4.

Full Load [Lb or $Lb \cdot In$] (N or $N \cdot m$)			
NF	3 (13.34)	RM	3 (0.339)
AF	2 (8.896)	YM	3 (0.339)
PM	5 (0.5649)	SF	2 (8.896)
Sensitivity (lb/mV/5V or in-lb/mV/5V)			
NF	5.073139e-04	RM	6.406601e-04
AF	4.233831e-04	YM	3.842097e-04
PM	4.545049e-04	SF	4.323792e-04
Specifications			
Excitation Voltage		5 VDC	
Min. Safety Factor		1.5	
Temperature range		0-200 °F	
Thermal zero shift tolerance		0.005 mV/V per 100°F	

Table 2.4: Sting Balance Specifications.

The sting balance was powered by a ELIND 6TD20 power supply, which could output a continuous current voltage of up to 20 V for a maximum current of 1 A, or 6 V at 5 A on the other channel. The motors however, could not be linked through the same power supply, because they required higher power levels thus a dedicated power supply was used as explained in Sec. 2.2.

Data acquisition and processing is based on a LabVIEW VI suite. The sampling is

performed by extracting the average signal of the transducers with a 25 kHz frequency and for a 4 seconds sampling time, therefore each pressure measurement is the result of an average of 100000 samples.

The sampling frequency was chosen such that from its temporal history it was possible to obtain the rotational speed of the propeller in the single propeller test of Sec. 4.2. From a test with the tachometer it resulted a speed of up to 70000 RPM which equals to 1166 Hz, thus the sample frequency was chosen to avoid aliasing and other phenomena.

The calibration matrix given by the sting balance factory resulted erroneous once it was tested with known loads applied. This discrepancy could be dependent on the fact that the last known calibration was done in 2012, more than 10 years prior to the present work. This made it necessary to repeat the calibration procedure *in situ*. Given the complexity of a full calibration on all six axes, just the three axes of interest in this study were re-calibrated, namely axial force (AF), normal force (NF) and pitching moment (PM).

Regarding the calibration procedure, a special support, shown in Fig. 2.19, was created and 3D printed to facilitate the application of the known loads. This support was joined with M3 screws to the calibration block included with the balance package. Different indications of length were drawn on the support to indicate the position of application of the load. To evaluate the pitch moment and its cross correlation with the axial force, an additional metal bar was used, orthogonally to the 3D printed one. The full setup is shown in Fig. 2.19.

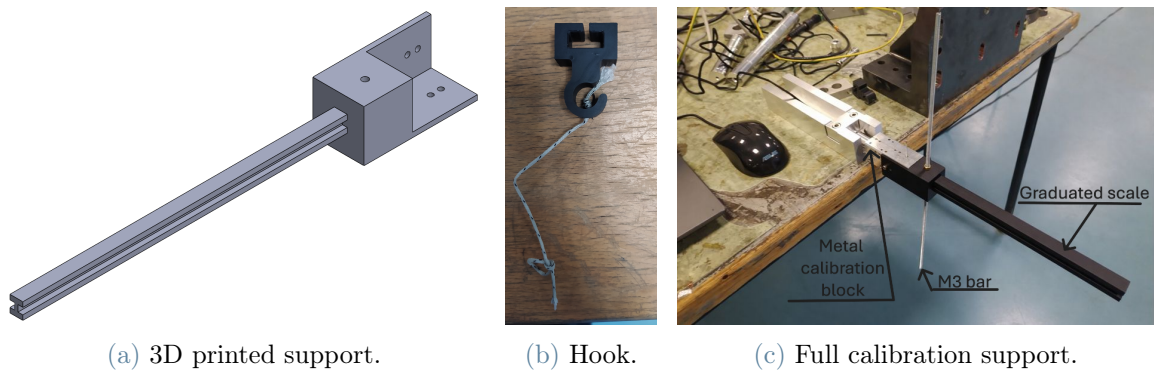


Figure 2.19: Full calibration support with details regarding the model of the 3D printed part and the hook used to position the weights.

Three different weights were used, 50 g , 100 g and 200 g. In order to stimulate each axis individually, the weights were applied in different configurations, as shown by Fig. 2.20. The 3D printed support was used for all configurations except positive axial force, thus

for the configuration at -90° the support was later mounted to evaluate the pitch moment and its cross correlation with the axial force in this configuration.

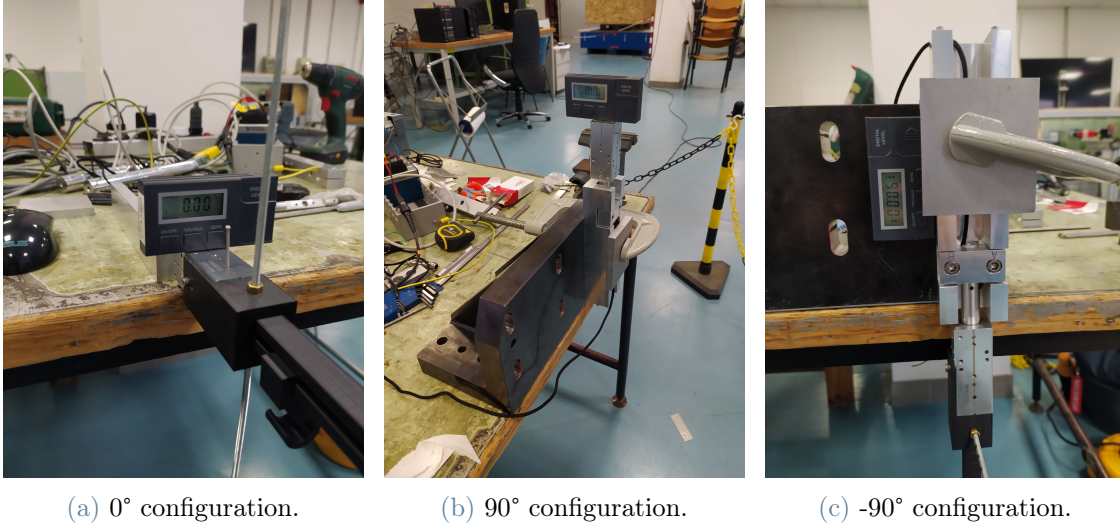


Figure 2.20: Configuration used during the first calibration, with angles of 0° , 90° and -90° with respect to the horizontal.

Between the different configurations, the balance was rotated carefully and was first vertically with the tip upward, then horizontally (with the positive normal force upward and then downward) and finally vertically upside-down, with the metric tip pointing the floor. The correct alignment of the balance assembly was measured with the digital inclinometer also shown in the figures.

For the configurations where gravity itself was not sufficient to maintain the balance at equilibrium, the latter was clamped to a support metal block, as show in Fig. 2.20c.

The different weights were applied in random order to validate the absence of any systematic errors, in accordance with the previously described experimental procedure, and every ten to fifteen measures, an acquisition with zero load was also done. In total there has been 152 measurement points for the computation of the calibration matrix.

The convention for signs of forces and moments, depicted in Fig. 2.22, respected the indication of positive normal force upwards described in the balance technical drawings, while the axial force was chosen positive inward, to signify drag. Finally the pitching moment was chosen as positive for the pitch-up direction and the point with respect to which the moments were computed was the center of the base of the 3D printed support.

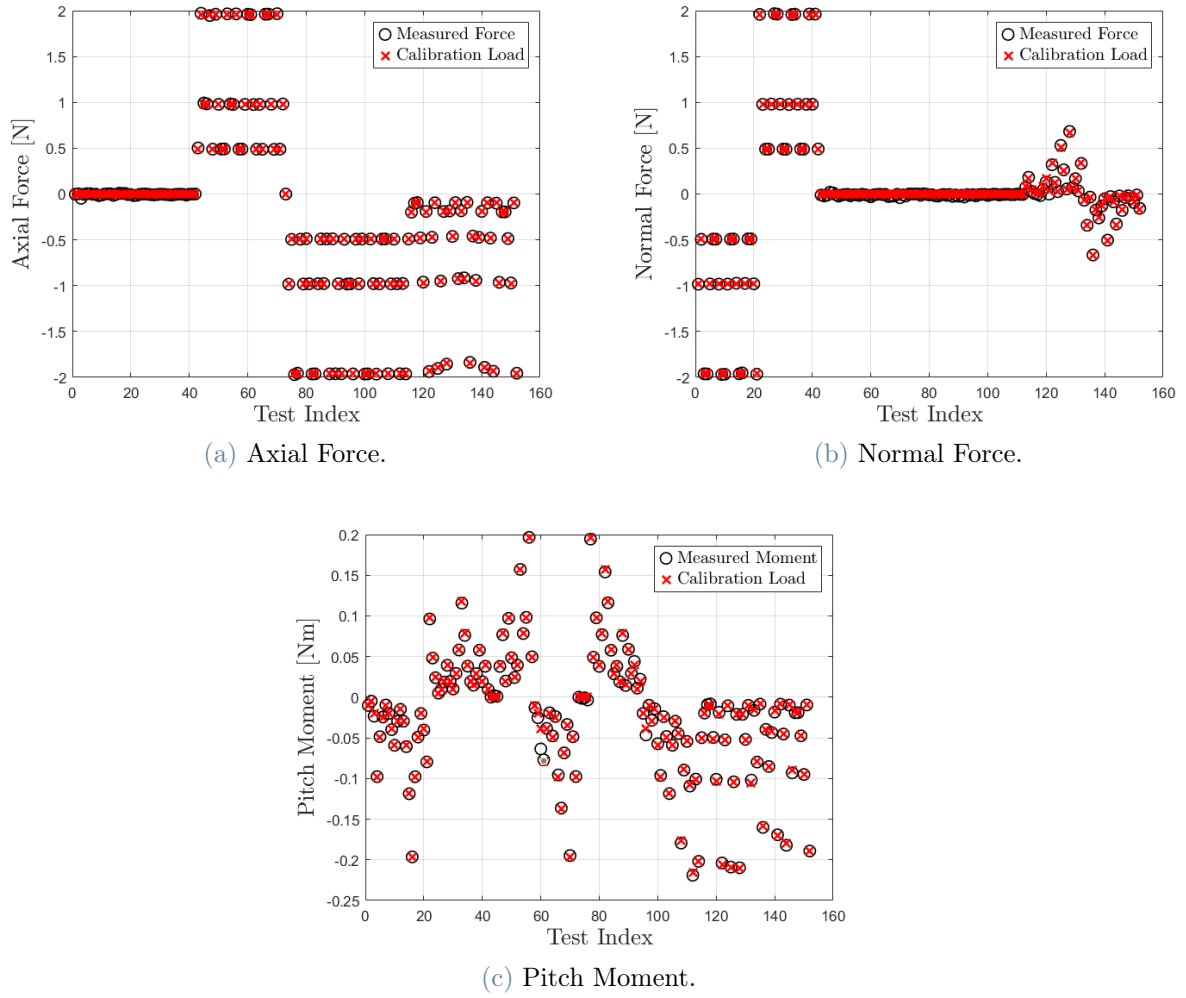


Figure 2.21: Calibration loads and respective measured loads for the three axes considered.

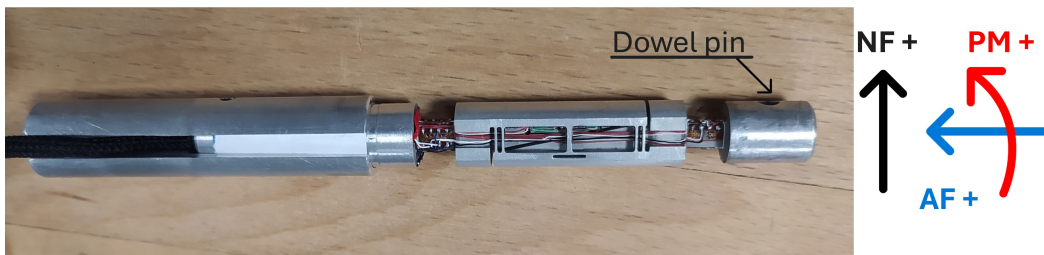


Figure 2.22: Forces and moments reference axes with respect to the sting balance.

This point was 44.35 mm from the dowel pin hole of the balance, while the aerodynamic center of the wing, which is the point at a quarter chord from the leading edge, was at 38.5 mm from the dowel pin hole. This was taken into account during the calibration as the moments needed to be computed from the second point, by using the moment transport

equation expressed in Eq. 2.1

$$\vec{M}_B = \vec{M}_A + \vec{F} \times (\vec{r}_B - \vec{r}_A) \quad (2.1)$$

In the equation above A and B are two arbitrary points, $\vec{r}_B - \vec{r}_A$ is their distance from each other, which for the case analyzed measures 5.85 mm and $\vec{F}$ is the force normal to the distance. In the problem investigated B indicates the aerodynamic center of the wing, while A is the moments' pole used during calibration measures.

Preliminary tests showed an error in the calibration emerged at very low axial forces, exhibiting a wrong negative total drag. To correct this behavior a further set of calibration tests were carried out in the wind tunnel, to mimic the actual working conditions as closely as possible. In the second set of calibration measures, the weight applied were of 5, 10 and 20 g in addition to the ones used previously, and the 200 g weight was not used since the problems of the first calibration arose specifically near zero-load conditions.

In the second calibration, reported in Fig. 2.23, the discrepancy between the measured and applied loads may appear larger than in the first calibration; however, this difference must be interpreted in light of the load scale. In the second set of calibration tests, the applied loads were generally smaller, and therefore a larger apparent deviation does not necessarily correspond to a greater error in practice. Moreover, a wider range of applied loads can be observed with respect to the first calibration, which had a positive influence on the quality and reliability of the results. The resulting calibration matrix is shown in Table 2.5.

Sensitivity (N/ μ V/V or Nm/ μ V/V)						
	NF	AF	PM	RM	YM	SF
AF	-6025.5696	5301.2997	4280.6071	-1072.3112	-36773.381	61047.123
NF	13749.797	-1433.8800	-1557.7068	8599.5697	3889.2511	43657.700
PM	-679.0601	63.67139	207.84948	-570.6694	33.02028	-2438.9177

Table 2.5: Calibration matrix of the sting balance.

Comparing the new and original calibration matrices, it was clear that the diagonal terms were similar while some cross correlation terms changed sign. This implied that in the actual working condition, the non-calibrated components were significantly affecting the outputs of the measures.

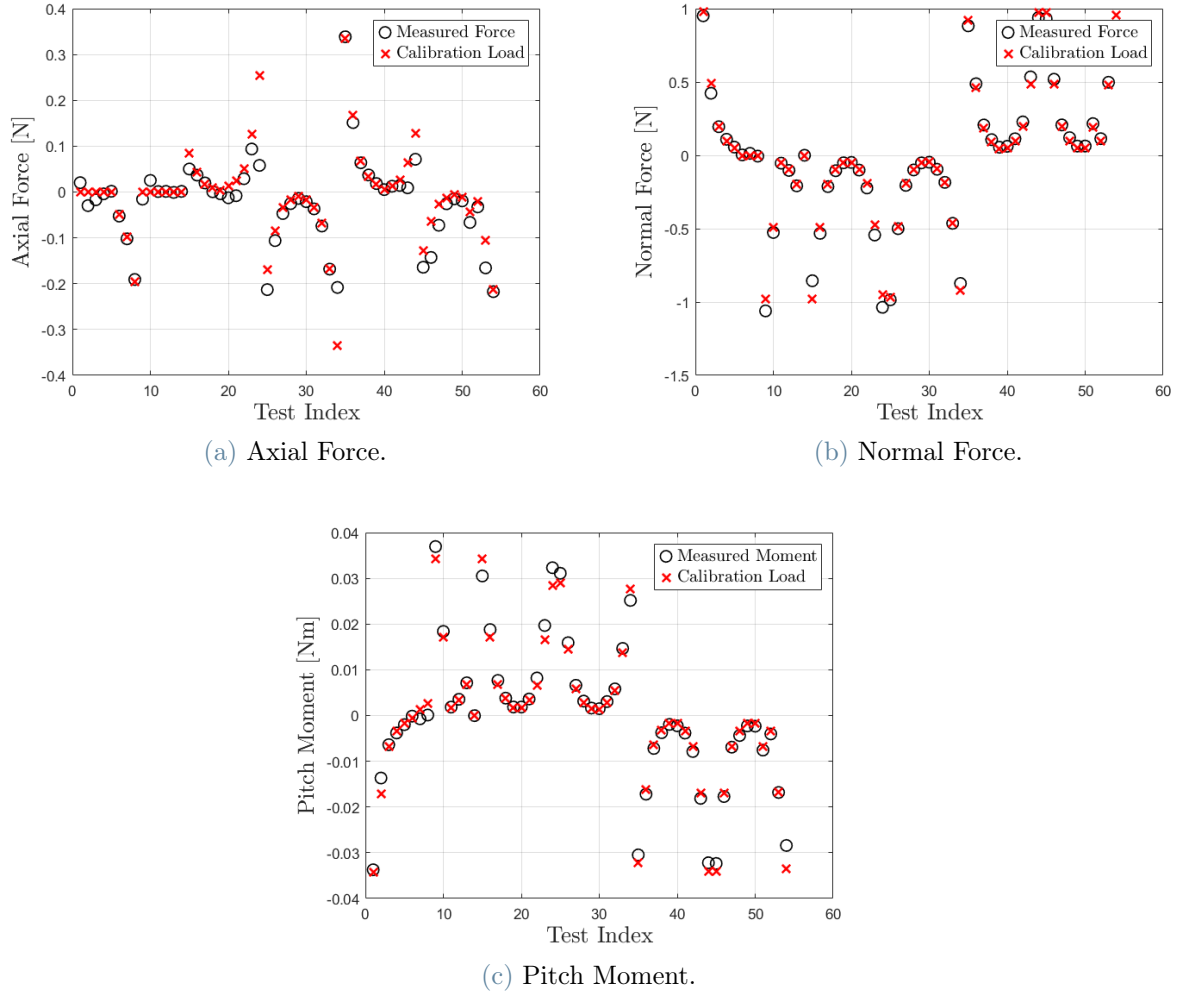


Figure 2.23: Calibration loads and respective measured loads for the three axes considered.

A specific test was conducted to validate the balance's calibration, and to further check the possible mechanical interference of the electric motors wires with the measurement. An angle of attack sweep was performed with the full model placed inside the wind tunnel with a weight of 50 g placed on the fuselage. The results from the balance were then compared to the ideal distribution of the weight due to its vector decomposition in body axis. The results are presented in Fig. 2.24.

The test was conducted while the ceiling of the wind tunnel was open, thus angles of attack around $\pm 180^\circ$ could be reached. This implies simply that the model was put upside down in order to measure the signal from the normal axis in the opposite direction, since gravity always points downward, and rotating the balance was the only option.

The difference in load among the different angles regarding the normal axis is very low, since it scales as $1 - \cos(\alpha)$, thus what seemed to be error of measure due to calibration

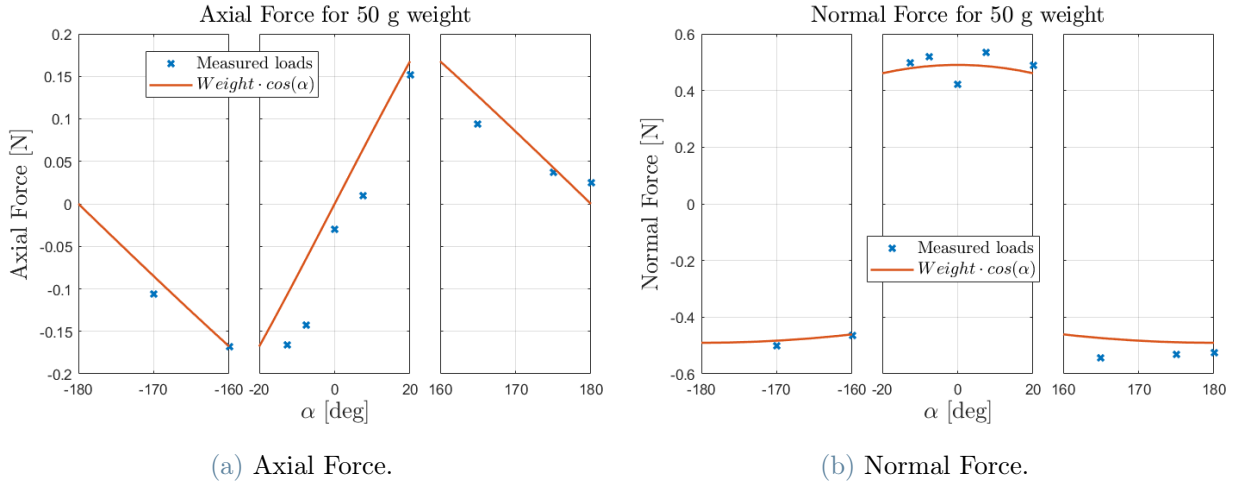


Figure 2.24: Validation of balance's calibration and propulsion power wires installation for mechanical interference.

is only a repeatability issue between measures. For the axial force, this did not occur because the dependency from the angle is linear near the 0° angle of attack configuration. It can be observed that there is good agreement between the measured and expected loads, indicating that the calibration was performed correctly and that the selected wiring configuration did not significantly influence the measurements.

2.3. Data Post Processing

2.3.1. Aerodynamic Coefficients

The aerodynamic coefficients were computed applying a specialized function developed in Matlab. The first operation was to convert the forces measured in body axes into the aerodynamic forces in wind reference axes. After this rotation, the function implemented wind tunnel correction to the measurements and finally the precision interval at probability %P of the measurements was computed.

The aerodynamic coefficients are defined such that C_L and C_D represent the net normal and tangential force coefficients to the freestream direction, respectively [19]. This definition therefore includes the contribution of propeller thrust within the force coefficients. In the case of an isolated wing, these coefficients reduce to the classical definition of lift and drag coefficients.

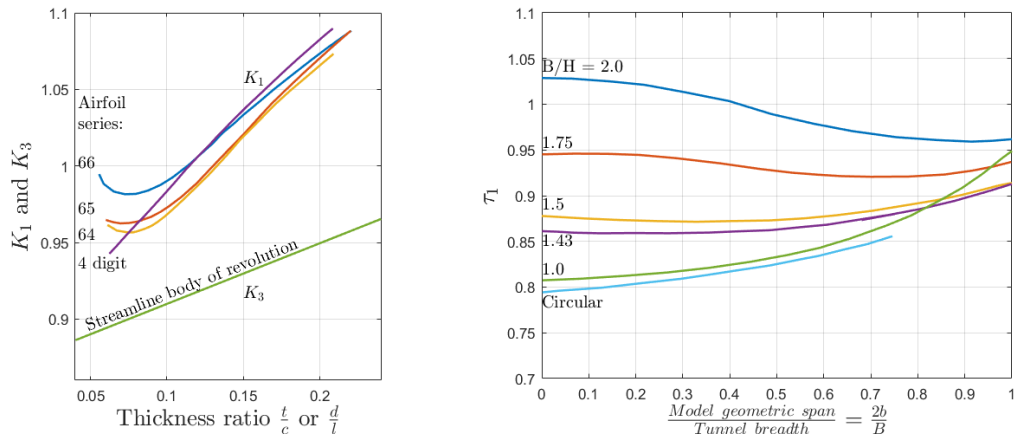
For the wind tunnel corrections there are notably three errors to compute: solid blockage, wake blockage and propeller blockage. These effects have been known since the beginning

of wind tunnel testing and were studied in depth by Barlow et al. [38] and Pankhurst et al. [39].

The solid blockage effect occurs when a body is put inside a wind tunnel because the presence of the model causes a reduction of the wind tunnel stream cross-sectional area thus producing an increase of the velocity (or dynamic pressure) around the model in comparison to free-air condition. The increase of velocity due to solid blockage with respect to velocity measured in the wind tunnel can be evaluated after the measure has been taken with Eq. 2.2, which is valid for blockage values below 10%, which the model in question respects.

$$\epsilon_{sb} = \frac{\Delta U}{U_\infty} = \frac{k_{1-3}\tau_1 V_{body}}{C^{3/2}} \quad (2.2)$$

In the above equation, k_1 or k_3 is the body shape factor different for airfoil (wing) or bodies of revolution (blunt) and can be interpolated through the graph in Fig. 2.25a. τ_1 is a factor depending on the test section shape and on the model span-to-tunnel width ratio; it can be interpolated from Fig. 2.25b considering a $B/H = 1.5$ for the rectangular wind tunnel test section in question. And finally, C is the wind tunnel cross sectional area. The following curves of correction factors were reproduced in Matlab for reference.



(a) Body shape factor with respect to thickness ratio.

(b) Wind tunnel factor with respect to model span-to-tunnel width ratio.

Figure 2.25: Body shape factor and wind tunnel factor for computation of solid and wake blockage [38].

Interference effects are also generated by the wake of the model inside the wind tunnel. However the wake blockage produces two different effects which have to be taken into

account separately. The increase in velocity is computed with Eq. 2.3, while Eq. 2.4 describes an increase of pressure drag of the model due to a pressure reduction on the wake and a decrease of the model base pressure with respect to free-air condition.

$$\epsilon_{wb} = \frac{\Delta U}{U_\infty} = \frac{S}{4C} C_{d_u} \quad (2.3)$$

$$\Delta C_{Dwb_{fus}} = \frac{k_{1-3} \tau_1 V_{body}}{C^{3/2}} C_{D_u} \quad (2.4)$$

In the equations above, S is the wetted surface of the model, which was obtained directly from Solidworks, while C_{D_u} is the measured drag coefficient.

Both solid and wake blockage effects are to be computed for every different parts of the model before summing it all together, and for the case under investigation, the contributions arose from the model, the metal support, the 3D printed one and the sting balance.

The presence of one or more propellers inside the wind tunnel test section also requires corrections. The propeller slipstream accelerates and maintains a velocity higher than the undisturbed wind tunnel velocity. Due to continuity, the same volume of air must pass ahead of the propeller as passes just afterward of the latter. Therefore, the velocity outside the slipstream will be less than U_∞ . Applying momentum theory to the case of a propeller rotating in a wind tunnel, Glauert derived Eq. 2.5 [40].

$$\frac{U_{corr}}{U_\infty} = 1 - \frac{\tau_4 \alpha_1}{2\sqrt{1 + 2\tau_4}} \quad (2.5a)$$

$$with \quad \tau_4 = \frac{T}{\rho A U_\infty^2} \quad (2.5b)$$

$$\epsilon_{prop} = \frac{\Delta U}{U_\infty} = \frac{U_{corr}}{U_\infty} - 1 \quad (2.5c)$$

In the above equations A is the cross-sectional area of the propellers, α_1 is the ratio between A and the cross-sectional area of the wind tunnel and T is the propeller thrust expressed in Newtons.

Finally the corrected speed to compute the aerodynamic coefficients resulted in

$$U_{corr} = U_\infty \cdot (1 + \epsilon_{sb} + \epsilon_{wb} + \epsilon_{prop}). \quad (2.6)$$

where the last term is present only in case of active propellers in the wind tunnel.

2.3.2. Experimental Uncertainty

The experimental data were screened for outliers prior to performing any further analysis. Outlier identification was based on the dataset variance, with acceptable values defined within an interval expressed as a multiple of the standard deviation. In this work, the three-sigma criterion [41] was adopted, whereby data points falling outside the 99.8% probability interval were classified as outliers.

Sample statistics and probability theory were used for the estimation of the precision interval of the results with a 95% confidence. Five repetitions of each experiment were used to compute the sample mean value and the sample standard deviation. The true mean and the true variance were obtained by weighting the sample results with the student distribution t , where ν is the degree of freedoms, which is five in the case of five experiments' repetitions. Thus, for the selected confidence with probability ($P\%$) of 95% and normal distribution, the experimental data points and their precision, are given by

$$x_i = \bar{x} \pm t_{\nu,P} S_x \quad (P\%) \quad (2.7)$$

with t given by the Table 2.6 as a function of the number of data points which are not outliers [41, 42].

ν	1	2	3	4	5
t_{95}	12.706	4.303	3.182	2.77	2.571

Table 2.6: Values of t for 95% confidence interval as a function of the number of degrees of freedom.

2.3.3. Propulsion Parameters

Propellers speed in revolutions per minute (RPM) was obtained during wind tunnel tests directly from the sting balance by estimating the auto-correlation of the time history of the signal from normal force channel.

The result of the operation is a waveform graph with a peak at a specific frequency. The tachometer was applied to verify that the peak frequency exhibited the rotational frequency of the propeller rather than the blade-passing frequency. In the latter case, the two blades per propeller would have required the peak frequency to be divided by two. The information from the sting balance allowed an efficient RPM control during the tests, opening the possibility of a future automatic feed-back control strategy.

For a more efficient correlation between aerodynamics effects and propulsion, dimensionless parameters are typically applied. Those parameters usually couple the propulsion main characteristics with fluid features. An example in flapping wing vehicles studies, or biological fliers, is the reduced frequency k [18, 43]

$$k = \frac{\omega c}{2U_{ref}} \quad (2.8)$$

Where ω , c and U_{ref} are angular velocity of a flapping wing, the wing's reference chord and the reference velocity, respectively. The parameter k represents the ratio between the forward velocity and the flapping velocity, which is related to the flapping frequency.

In a similar way, the parameter $\bar{\Omega}$ was introduced for the case of propellers [19]

$$\bar{\Omega} = \frac{\Omega R}{U_{ref}} \quad (2.9)$$

where ω , R and U_{ref} are angular velocity of the propeller in rad/s, the propeller's radius and the reference velocity, respectively. The parameter $\bar{\Omega}$ represents the ratio between the propeller tip velocity and the reference velocity. It was used as a reference for future works as only one rotational speed was implemented in this thesis.

For a propeller like the one under investigation, another important set of parameters are the efficiency (η) and the thrust (C_T) and torque (C_Q) coefficient.

These parameter are obtained in eqs. (2.10) to (2.12):

$$C_T = \frac{T}{\rho \text{ RPM}^2 D^4}, \quad (2.10)$$

$$C_Q = \frac{Q}{\rho \text{ RPM}^2 D^5}, \quad (2.11)$$

$$\eta = \frac{1}{2\pi} J \frac{C_T}{C_Q}, \quad (2.12a)$$

$$\text{with } J = \frac{U_\infty}{\text{RPM} \cdot D} \quad (2.12b)$$

where T and Q are thrust and torque respectively, D is the propeller diameter and J is the advance ratio of the propeller. In particular J is another representation of the ratio between the propeller rotational speed and the advancing velocity of the vehicle.

3 | Numerical Analysis

3.1. Software

The analysis of distributed propulsion on aircraft of such low dimension is an academically new topic, thus an estimation of the aerodynamic forces acting on such wings have not been reported yet. To solve this problem a series of numerical simulations have been carried out, using primarily the DUST software, with the aid of Xfoil, the NeuralFoil python library and SU2 to validate the results.

DUST is an open-source software developed by Politecnico di Milano and released under MIT license that solves different aerodynamic problems with flexible and reliable approach. The code is written using the latest objected-oriented paradigms of FORTRAN and integrates different aerodynamic models for solid bodies that allows for different model fidelity levels [44].

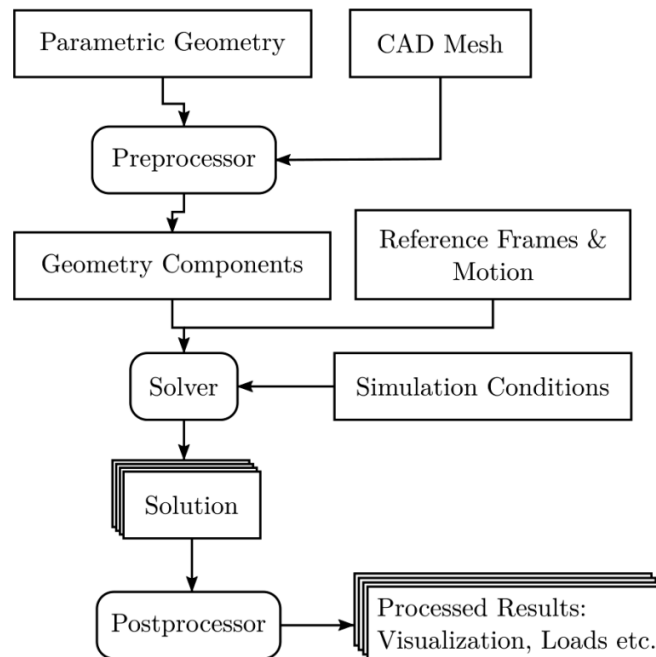


Figure 3.1: DUST workflow [44].

The program is organized into three main stages: pre-processing, solving, and post-processing. Fig. 3.1 illustrates the typical workflow. During pre-processing, components can either be imported from existing mesh files or generated parametrically by defining the geometry in the input files. The solver formulation is restricted to rigid bodies; flexible components cannot be simulated. Solid bodies are represented through surface distributions of singularities, and DUST provides several aerodynamic modeling approaches, such as surface panel, vortex lattice (VLM) and lifting line. The singularity strengths are obtained by solving the boundary value problem for the potential component of the velocity field, enforcing the non-penetration boundary condition. Components are positioned and moved through a hierarchy of reference frames.

Solver inputs include component geometry and mesh, reference frames, simulation settings, and numerical parameters. The solution is advanced in time for a prescribed number of steps, and results are written to the screen or output files as specified. The Kutta condition is enforced by shedding wake elements from the detected trailing edges of lifting bodies. The formulation assumes incompressible potential flow; however, compressibility corrections for steady loads can be introduced via the Prandtl–Glauert correction for panels and vortex lattice elements, or through Mach-dependent tabulated data for lifting lines.

The aerodynamic formulation relies on Helmholtz decomposition of the velocity field and a Lagrangian description of vorticity, based on the vorticity-velocity formulation of the problem [45]. The velocity field $\vec{u}(\vec{r}, t)$ is expressed as the sum of an irrotational (potential) component $\vec{u}_\phi = \nabla\phi$, associated with potential-based elements, and a solenoidal (rotational) component $\vec{u}_\vec{\psi} = \nabla\vec{\psi}$, associated with wake particles.

$$\vec{u}(\vec{r}, t) = \vec{u}_\phi(\vec{r}, t) + \vec{u}_\vec{\psi}(\vec{r}, t) \quad (3.1)$$

Enforcing incompressibility leads to two coupled problems for the scalar potential ϕ and the vector potential $\vec{\psi}$. The time-marching procedure alternates the solution of a three-dimensional boundary element method for the potential part and the Lagrangian evolution of the rotational field. DUST requires only the surface mesh of the model that is used to build a piecewise-uniform distribution of doublets and sources associated with surface panels (as done in Morino formulation[46]).

The wake is shed from a prescribed point of the lifting bodies and is initially modeled as a panel (shares same characteristic of the surface panel of the bodies). As the simulation evolves in time, based on a time step algorithm, the wake develops and at a certain

point (defined by the input given by the user) the wake panels are converted into vortex particles. This method is called Vortex Particle Method (VPM) and allows a more robust wake formulation that can well represent the interactional aerodynamics between different bodies. The code computes aerodynamic loads using different approaches and simplifications based on the aerodynamic model used to describe the different bodies.

It's worth noting that for both surface panel and vortex lattice elements the viscous effect on the aerodynamic load isn't considered (current DUST version 0.8.2-b). Since this project is characterized by low Reynolds in the range of 10 to 50 thousands, the viscous effect is of paramount importance and can't be neglected, consequently the modeling of aerodynamic surfaces was carried out avoiding the linear VLM method and using the lifting line with imposed airfoil performances.

In particular, for the lifting line approach employed generally for high-aspect-ratio lifting bodies such as rotor blades, DUST employs one-dimensional lifting line elements incorporating viscous effects through tabulated two-dimensional airfoil data (.c81 files). Each element consists of a vortex ring and its trailing vortices, with the last vortex filament released into the wake in the spanwise direction. In accordance with Helmholtz's theorem, vortex filaments cannot terminate in the fluid, forming a horseshoe vortex configuration. A single horseshoe vortex yields constant spanwise circulation and is therefore insufficient to represent realistic distributions; consequently, multiple horseshoe vortices are aligned along a common line to capture non-uniform circulation, known as *Lifting Line*. The discrete formulation subdivides the lifting body into uniform-circulation lifting line elements, each modeled as a vortex ring with intensity Γ_{il} , equivalent to a uniform surface doublet panel $\mu_{il} = \Gamma_{il}$. The collection of these elements defines the lifting surface S_l [44].

The circulation intensity (Γ) is determined through a nonlinear fixed-point algorithm linking lifting line circulation to tabulated sectional aerodynamic coefficients. DUST implements two approaches: the loosely coupled Γ -method, based on equating the semi-empirical sectional lift to the Kutta–Joukowski expression, and the α -method, where the local incidence angle, computed from the induced velocity of all wake and lifting elements, is used to retrieve aerodynamic coefficients from tabulated data. The nonlinear system is solved iteratively, accounting for mutual interference among elements and computing circulation from lift through the Kutta–Joukowski theorem.

Regarding the airfoil analysis the software of choice was SU2 [47]. SU2 is an open-source computational fluid dynamics (CFD) suite developed for multiphysics analysis and design optimization using unstructured mesh topologies. The core of the suite is a Reynolds-averaged Navier–Stokes (RANS) solver, around which others type of solvers were

implemented for multiphysics analysis. This solver is capable of simulating compressible and incompressible flows, as well as turbulent ones, thanks to the implementation of the Spalart–Allmaras (S-A) and Menter shear-stress transport (SST) turbulence models. SU2 was developed in standard C++ and relies solely on widely accepted open-source software. SU2 also implements numerical algorithms for convergence acceleration which solved the performance issues arose by the use of C++ structures.

Spatial discretization in SU2 is performed using a finite-volume method with a standard edge-based formulation on a dual grid, where control volumes are constructed according to a median-dual, vertex-based scheme. In this approach, each control volume is formed by connecting the centroids of the surrounding cells with the midpoints of the corresponding faces and edges associated with a given node. To combat low frequency errors that affect the whole domain (typical of iterative relaxation schemes), SU2 implements a multigrid approach. SU2 automatically generates a hierarchy of coarser grids from the provided fine grid to accelerate convergence of the numerical solution. This multigrid approach computes corrections to the fine grid solution on coarser levels and applies this logic recursively. SU2 also offers the capability of unsteady simulations with the implicit and explicit time integrations schemes implemented.

For the investigation of the E231 airfoil, only a steady analysis was performed, thus the time integration capabilities were not used. Furthermore, the full multiphysics capabilities of SU2 were not employed, as the problem under consideration was limited to the external aerodynamic flow around an airfoil.

For the preliminary investigation of airfoil performance, Xfoil and NeuralFoil were used. Xfoil is a numerical analysis tool for two-dimensional subsonic airfoils, developed by Mark Drela at the Massachusetts Institute of Technology in 1986 [48]. It combines a linear-vorticity panel method for the inviscid outer flow solution with an integral boundary-layer formulation to account for viscous effects. The viscous–inviscid interaction is achieved through a coupled iterative procedure, allowing the solver to predict lift, drag, and moment coefficients while also capturing boundary-layer development, transition, and separation phenomena.

NeuralFoil [49, 50] is a python library which was developed with machine learning to extrapolate results from the training data based on Xfoil. Due to this approach, the python script is faster than Xfoil computation, requires less input from the user, does not have convergence problem and the data manipulation to create the c81 file necessary for DUST can be automated in the python script. Included in Neurafoil there is also the aerosandbox toolkit which allowed to plot and design any airfoil profile, and was used to

obtain the best geometric agreement with the experimental model.

3.2. Airfoil analysis with SU2

The analysis of the E231 airfoil using Xfoil, at such Reynolds number revealed that the stall condition was always present regardless of the angle of attack. This behavior relates closely to the literature [51] as the maximum thickness viable for such low Reynolds number application should be of 6 to 9 % while the E231 has a maximum thickness of 12%.

Nevertheless concerns about Xfoil capabilities at such low Reynolds were raised, thus a series of classical, higher fidelity CFD analyses were carried out using SU2.

Fig. 3.2 shows the boundaries and the regions of interests of the computational domain. It measures 2 m in radius (corresponding to $50c$), where c is the airfoil chord, selected as the reference length ($c = 0.04$ m).

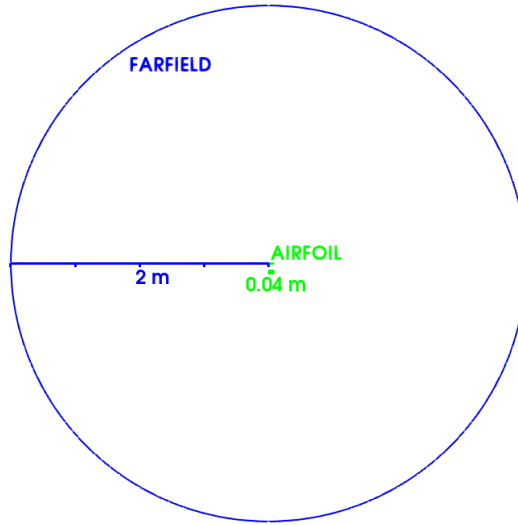


Figure 3.2: Computational domain used in the simulations.

The boundary conditions (BCs) were set as the farfield ones, while the airfoil had an additional no slip BC. The free-stream values that serve as boundary conditions for the farfield were: $Ma = 0.029388$, $Re = 23500$ and $T = 288.15$ K. The free-flow velocity (U_∞) is automatically calculated by the SU2 software with the previous values, yielding a magnitude of 10.0 m/s in the rightward direction.

A hybrid mesh is chosen for this project. In particular, an unstructured mesh is used

for most of the domain to improve computational efficiency, while a structured mesh is generated around the airfoils to better capture the boundary layer behavior. Additionally, a coarser mesh is used for the far field to further increase the computational efficiency, while a higher number of cells is used near the airfoil surfaces and in the wake region to better capture complex flow features.

A focused view on the structured mesh near the airfoil is illustrated in Fig. 3.3.

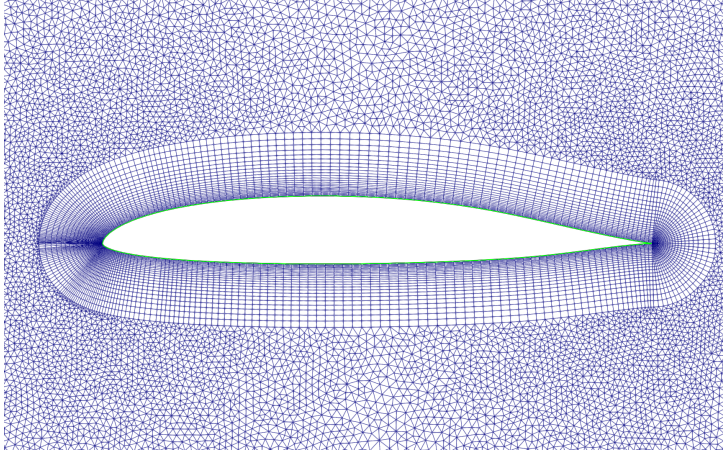


Figure 3.3: Structured mesh near the airfoil.

Moreover, another crucial aspect of the mesh construction is the dimension of the last cells in the structured boundary layer mesh. These elements must have dimensions comparable to those of the first triangular elements of the unstructured mesh outside the boundary layer, in order to improve simulation stability and convergence. In addition, a smooth transition function is applied between the refinement and the coarser regions to avoid abrupt variations in cell size and increase stability. This can be observed in Fig. 3.4.

It should be noted that the refinement around the airfoils has to be strictly controlled during the generation of finer meshes, since very small unstructured elements near the boundary layer can lead to divergence, due to their low numerical dissipation. The mesh displayed in this section has 57965 nodes and is derived from the grid convergence study discussed below. The number of nodes is reported instead of the number of elements because SU2 is a node-centered solver; therefore, the node count is the relevant metric for both grid-convergence assessment and computational cost analysis

The parameters for the boundary layer construction were carefully chosen to achieve a wall y^+ value of $\simeq 1$. The methodology and considerations for selecting the y^+ target value were based on guidelines from *Fluid Mechanics 101* [52].

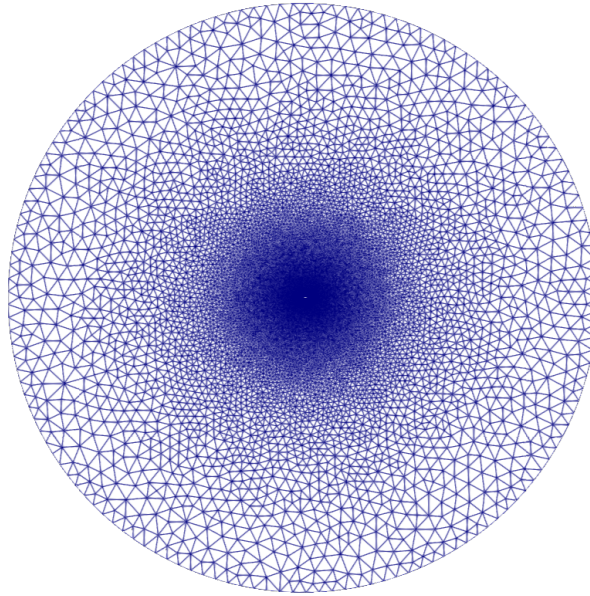


Figure 3.4: Complete mesh structure with refinement near the airfoil.

Finally, the airfoil trailing edge is maintained sharp, as it helps wake development in the numerical computations.

Regarding the numerical setup for this analysis, the RANS (Reynolds-Averaged-Navier-Stokes) solver was used. The numerical methods employed in the simulations are designed to ensure both accuracy and stability. The ROE approximate Riemann solver was the flow numerical method chosen. Roe's method provides an exact resolution of linearized characteristic waves at cell interfaces, ensuring good resolution of convective phenomena. At low Mach and low Reynolds number conditions, such as those relevant to the present analysis, Roe's scheme benefits from its relatively low numerical diffusion compared to more dissipative flux formulations. This method is used with MUSCL (Monotonic Upstream-Centered Scheme for Conservation Laws)[53] reconstruction in order to have a second-order scheme, which is more accurate.

The turbulence model selected for this study is the SST (Shear Stress Transport) $k - \omega$ model, coupled with the LM (Langtry-Menter) transition model. This approach is able to account for laminar-to-turbulent transition driven by separation, including the formation and reattachment of laminar separation bubbles, which are typical features of low Reynolds number flows on airfoils. The SST model is adopted because it provides improved reliability in the presence of adverse pressure gradients and incipient separation, conditions that dominate the aerodynamic behavior of airfoils at the Re in question.

An adaptive CFL (Courant–Friedrichs–Lewy) condition is applied, with a maximum value

set to 100, guaranteeing a good trade off between stability and computational speed. Finally, the Green-Gauss method is used for spatial derivatives. The mesh quality allows to use it instead of the WLS (Weighted Least Squares) method, which has a higher computational cost.

The Cauchy lift-based convergence criterion is selected with a tolerance of 10^{-7} and applied over a window of 100 iterations to ensure high solution accuracy. This choice is due to the central role of aerodynamic lift in this analysis.

A grid convergence study is performed to prove the robustness of the results and to choose the proper mesh to apply in the simulations. Fig. 3.5 shows the trend of the lift and drag coefficients (C_L and C_D) and their respective percentage errors with respect to the number of nodes used to discretize the computational domain.

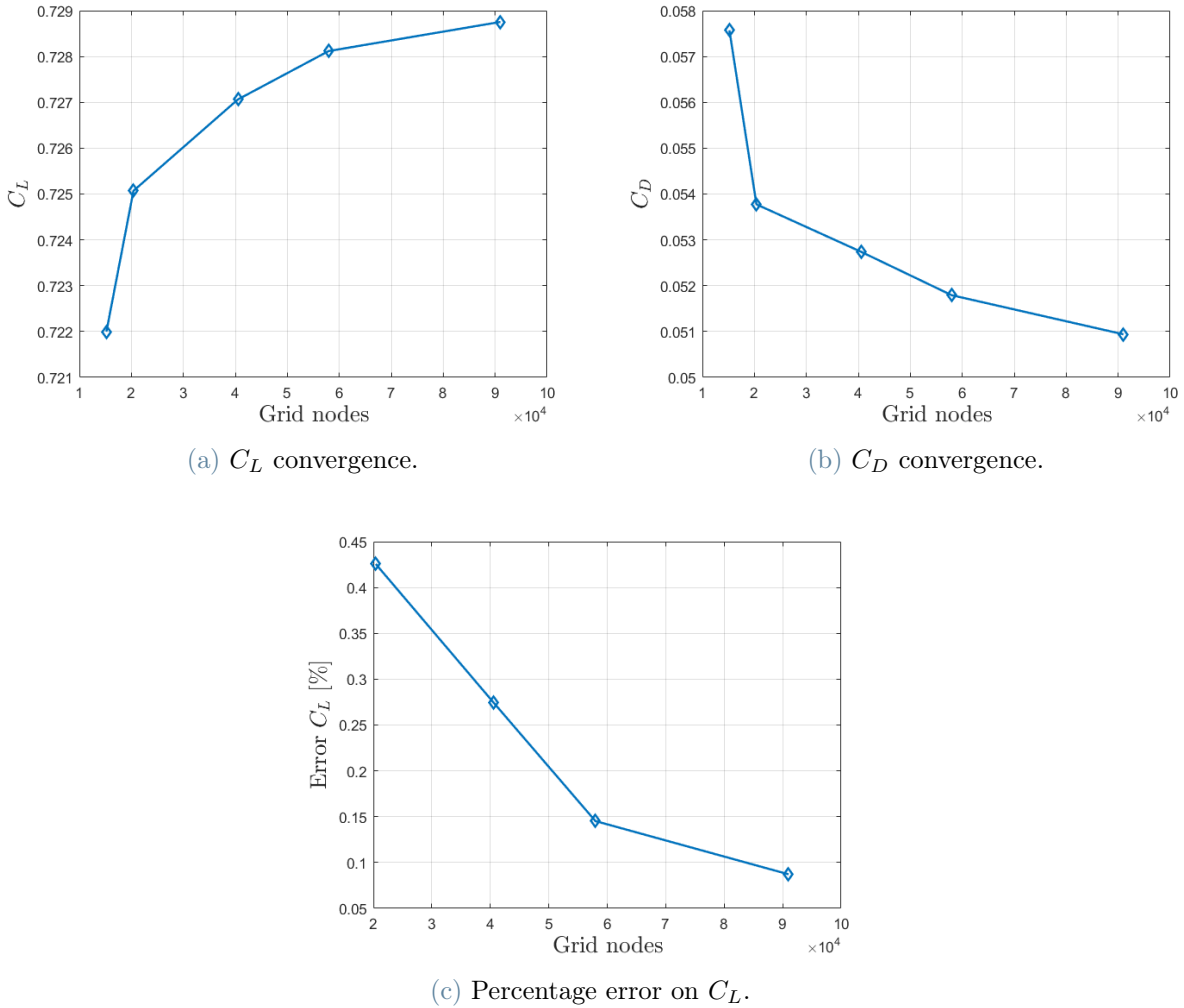


Figure 3.5: C_L and C_D convergence with percentage errors, with the abscissas axis expressed in number of nodes. $Re = 23,500$, $\alpha = 5^\circ$.

In this study, the mesh is parametrized with constant proportions between different parts of the mesh when using different cell numbers. In addition the angle of attack chosen for the grid convergence analysis was 5° , as a lower angles of attack did not show significant trends in the aerodynamic coefficients, even though such effects would be expected for an asymmetric airfoil section. The grid convergence was reached with a 57965 nodes mesh. This mesh was chosen because the trend of C_L and C_D showed a plateau even if the relative error between successive refinements was below 1% for the entire analysis. The validation of the result of this higher fidelity method, was conducted against the experimental data, once it became available. Experimental and numerical result was compared in the same conditions. It's critical to remark that the comparison was done between three-dimensional experimental results (finite wingspan) and a bi-dimensional (infinite wingspan) simulation.

Comparison SU2 vs. Experiment

Alpha	C_L SU2	C_L exp.	$\Delta C_L\%$	C_D	C_D exp.	$\Delta C_L\%$
-5°	-0.2814	-0.2800 ± 0.004	-0.5	0.0499	0.0441 ± 0.0097	13.2
0°	0.1903	0.2060 ± 0.002	-7.6	0.0502	0.0929 ± 0.0018	-46.0
5°	0.7281	0.5694 ± 0.0035	27.8	0.0518	0.1645 ± 0.016	-68.5

Table 3.1: Comparison for lift and drag coefficient, with relative errors, between SU2 simulations and experimental results, for different angles of attack.

In Table 3.1 a comparison between the results from the SU2 simulations and the experimental data is shown. The setup was defined with the objective of minimizing the error in the lift coefficient at 0° angle of attack; consequently, the results at the other angles, may be less representative of the measured C_L . The simulations tend to overpredict the lift coefficient at higher angles of attack, which is consistent with expectations. The behavior of the drag coefficient was not examined in this case, even if reported in Table 3.1, as a discrepancy between the numerical predictions and the experimental measurements was anticipated. This is attributable to the comparison between a two-dimensional airfoil model and a full three-dimensional configuration, in which additional contributions from induced drag and the presence of the nacelles lead to higher measured drag forces.

Once the numerical analysis has been validated, it was useful to analyze the flow field around the airfoil, which is reported in Fig. 3.6, to understand the physics of the phenomenon. The airfoil does not show sign of separation or transition along its chord. The

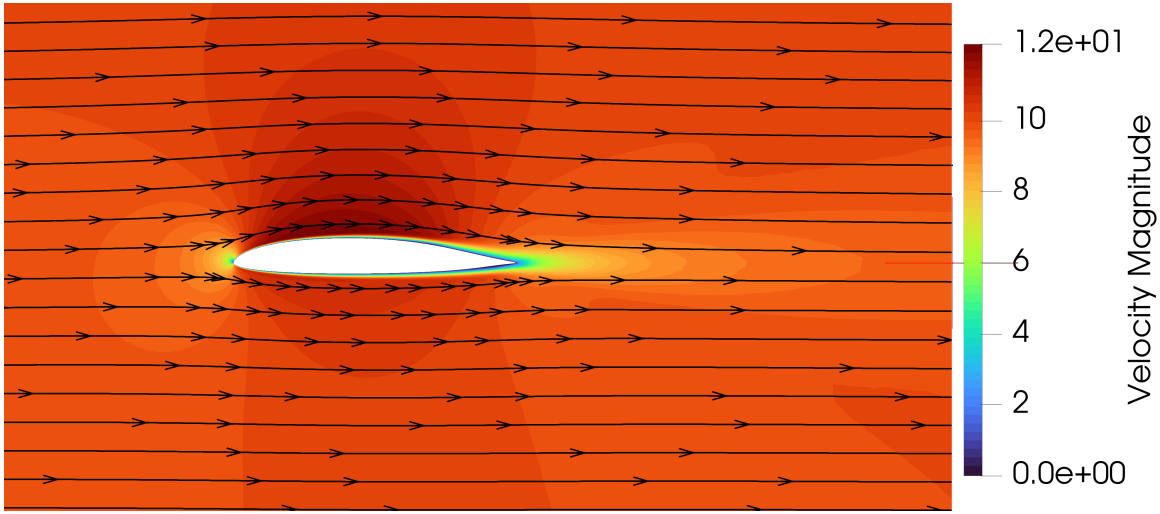


Figure 3.6: Velocity flow field in m/s around the E231 airfoil, with streamlines. Conditions are $Re=23500$, $AOA=0^\circ$.

flow on the upper and lower side of the airfoil rejoin together beyond the trailing edge without causing any particular flow phenomena. The most noticeable behavior, which was to be expected, is the thickness of the boundary layer, which reflects the importance of viscous phenomena at this Reynolds.

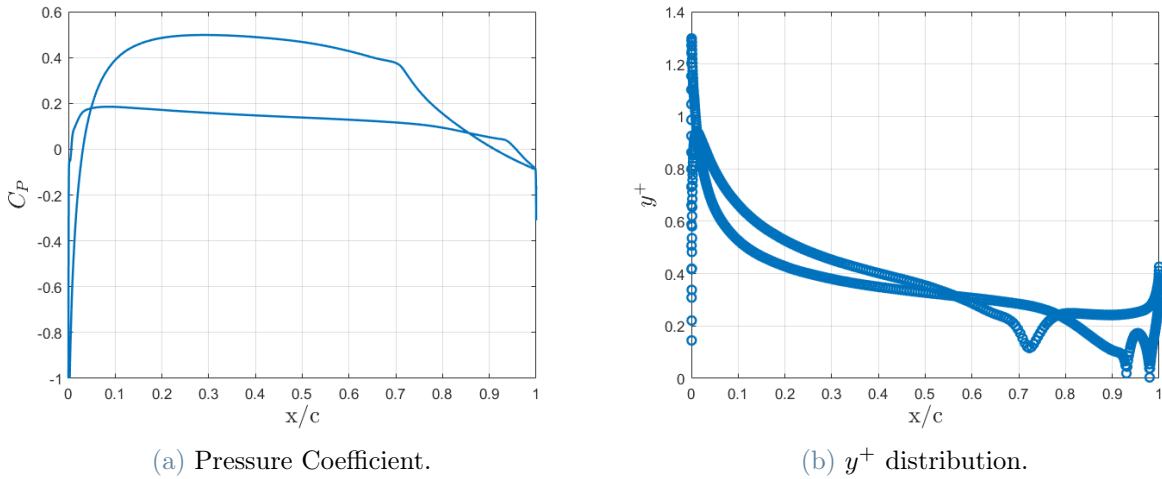


Figure 3.7: Pressure coefficient and y^+ distribution expressed as fraction of the chord.

From the pressure coefficient distribution reported in Fig. 3.7a, the airfoil appears to have separation around 70% of the chord, which confirms theoretical expectation of a separated thick profile at low Re , although exhibiting a very different distribution with respect to the Xfoil one.

Regarding y^+ , the maximum value of one was reached only in a handful of points near the stagnation point of the airfoil, while for the rest of the chord length it remained well under such value.

Neurafoil, and thus Xfoil, on which it is based on, predicted that the E231 airfoil would be stalled even at 0° of angle of attack. This numerical analysis conducted with SU2 clearly shows that this assumption does not hold. Thus it can be concluded that for this airfoil in particular, Xfoil analysis is not to be trusted for the prediction of aerodynamic coefficients at very low Reynolds, which is the case under investigation.

3.3. Wing's Model and Panel Distribution

The most efficient way to model a surface in DUST is to use the built-in parametric definition for surfaces. Thus, both the propeller and the wing were modeled in this way.

Regarding the wing's modeling, DUST implements the ability for the VLM to be corrected with the aerodynamic tables of the c81 file to compute viscous drag. However, this approach was initially thought not to be possible due to the high discrepancies between the values of the linear potential calculations and the values inside the tables obtained with NeuralFoil.

As was explained in further details in Sec. 3.2, the data obtained through NeuralFoil, did not match the one obtained with the experiment. In light of this, a c81 table was created using the experimental data, and doing so, the VLM with airfoil table correction returned to be a viable strategy.

However, lifting-line and linear VLM approaches were also investigated in order to provide a more comprehensive assessment of DUST's capabilities at such low Reynolds number regimes.

The discretization using the lifting line was applied only to the spanwise distribution of panel. The VLM approach, for the linear and the non linear distributions, needed to be decomposed into panels also in the chord-wise direction .

For ease of comparison of results, the same parameter was chosen across different strategies whenever possible. In fact, for all the three cases the spanwise distribution resulted in 50 panels, as the convergence study will explain. Regarding the other discretization direction, the wing was divided into five panels, as using a larger number would have hindered the convergence of the nonlinear solution approach. Consequently, five panels represented the maximum achievable discretization.

To create the tabulated airfoil performance table, the same discretization of angles of attack was maintained with respect to the experiments, with the extensions beyond such values done by Viterna correction, as explained in Sec. 3.4. The selected Reynolds numbers also retraced the experiment ones, described in Table 4.1. The Mach number was considered up to $M = 0.1$ for the table, as the velocity on the main wing did not reach higher values. As expected, the lifting line method considerably reduced the computational time with respect to the linear and non-linear VLM.

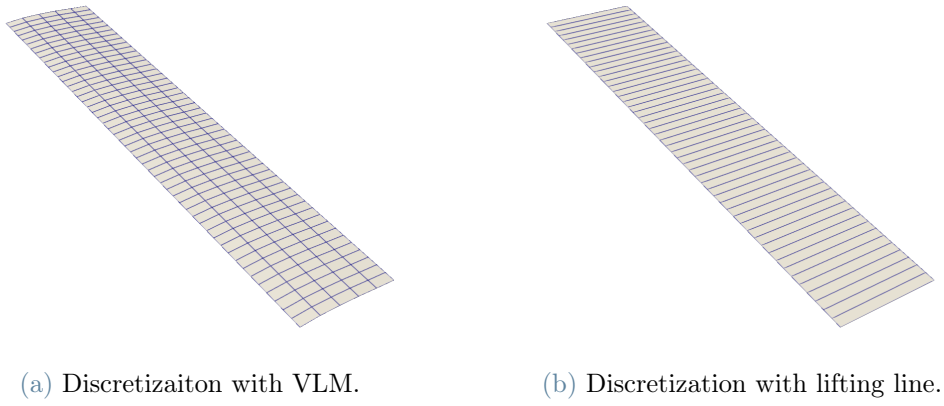


Figure 3.8: Two different discretization models for the wing.

According to the VLM approach, the Eppler E231 was discretized in 150 points and imported as a .dat file for the profile computation inside DUST. Even if the mean line was discretized in five panels only, a highly detailed dat file was useful for the visualization of the results.

To gain confidence in the numerical analysis and potentially run further simulations, an analysis involving the wing's grid convergence without the propellers was carried out. The performance of the wing airfoil was modeled using experimental data obtained from the configuration including the nacelles, without the propellers. Consequently, in the numerical model the nacelles were not explicitly represented, as their aerodynamic influence was already incorporated into the airfoil performance data.

Since DUST is a particle based solver, the convergence based on increasing the elements' number in which the geometry is subdivided needs to be carefully evaluated because an excessive number of panels may result in numerical instability.

In this project the highest possible number of panels needs to be evaluated avoiding to incur in numerical instabilities or exceptionally long computational times, considering that the effect of the six propellers would need to be evaluated in the full model while the

simulations will be run on a laptop. Moreover, the non symmetrical E231 wing's section allowed the convergence study to be carried out at 0° of angle of attack by considering both non-zero lift and drag coefficients.

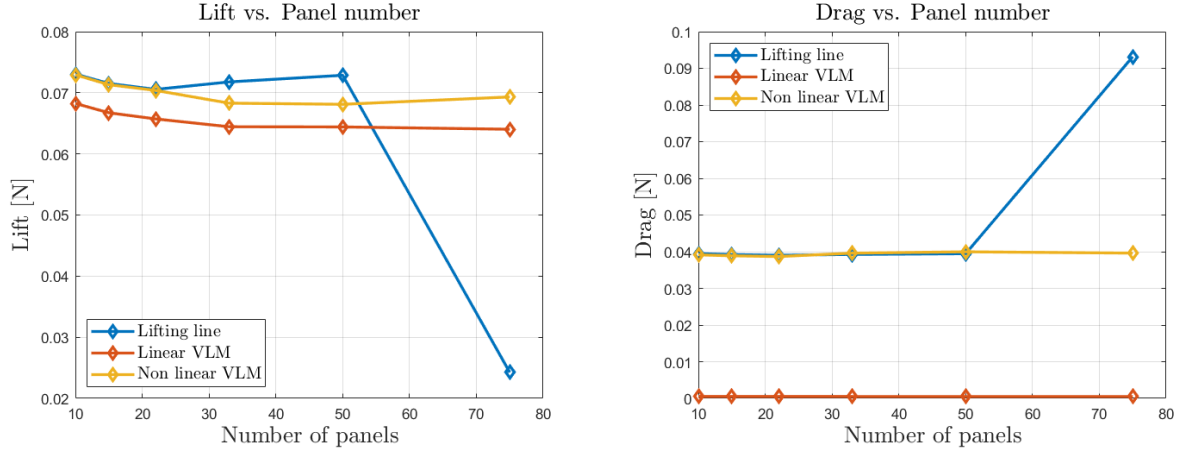


Figure 3.9: Spanwise number of panel convergence for the wing, both for lift and drag.

From Fig. 3.9, it can be observed that the 50 panel discretization provides the most suitable resolution for the intended results. In particular, the lift coefficient predicted by all three methods converges toward similar values as the number of panels increases; however, the lifting-line approach becomes numerically unstable when the surface is discretized with more than 50 panels. A similar behavior is observed for the drag coefficient, for which the lifting-line method again diverges at higher discretization levels. Conversely, the linear VLM exhibits the expected trend, as this formulation is inherently unable to capture the viscous drag contribution of the wing.

During the computation of the aerodynamic coefficients at different airspeed and angles of attack, the time step was chosen based on the propeller time step convergence study, as this value was intended to be consistently adopted in the full model simulations.

Thus, an analysis focused on time step convergence was carried out. Since the wing in the simulation was steady, this investigation primarily aimed to determine how small the time step could be chosen without inducing numerical instabilities, particularly when VLM table corrections were enabled. The analysis showed that for the linear VLM and the lifting line, the time step could be decreased until matching the propeller's time step without generating instabilities. The selected time step was then defined with reference to the propeller's simulation. This meant defining a very close distance between generated particles in the wake causing convergence problems for the non linear VLM, as the sum

of the correction from the non linear system and the closeness of the particles, made it diverge frequently. To overcome this problem, different strategies needed to be applied.

Initially, Aitken relaxation was employed, together with the option of activating the non-linear corrections after a prescribed number of time steps. This strategy made it possible to first obtain a robust solution using the linear VLM formulation and subsequently introduce the airfoil table corrections, accounting for them through a user-defined numerical relaxation factor. Although this approach improved the convergence behavior, the results were still not considered fully satisfactory. The second strategy investigated, to be used in combination with the first, involved the implementation of a sequence of restarts. The underlying idea was to first obtain a converged solution using a relatively large time step and then progressively restart the simulation with decreasing time step values, until the target time step required for the propeller simulations was reached. This combination of parameters allowed for a converged solution to be obtained for the wing with the non linear VLM.

At the velocity of 10 m/s all three wing discretization were used for an evaluation and comparison of the results, as shown in Fig. 3.10. The linear VLM could not capture the viscous drag and the stall angle of attack, as expected. This approach was however capable to estimate successfully the lift distribution around the 0° angle of attack.

Lifting line and non linear vortex lattice methods provide excellence agreement in the total load on the wing, as well as their trends. For the other velocities, only the lifting line approach was analyzed, as the complete model used that wing discretization.

In conclusion, the wing was modeled with 50 spanwise points of a lifting line, each of which had the relative airfoil performance given by the c81 table generated through the experimental results.

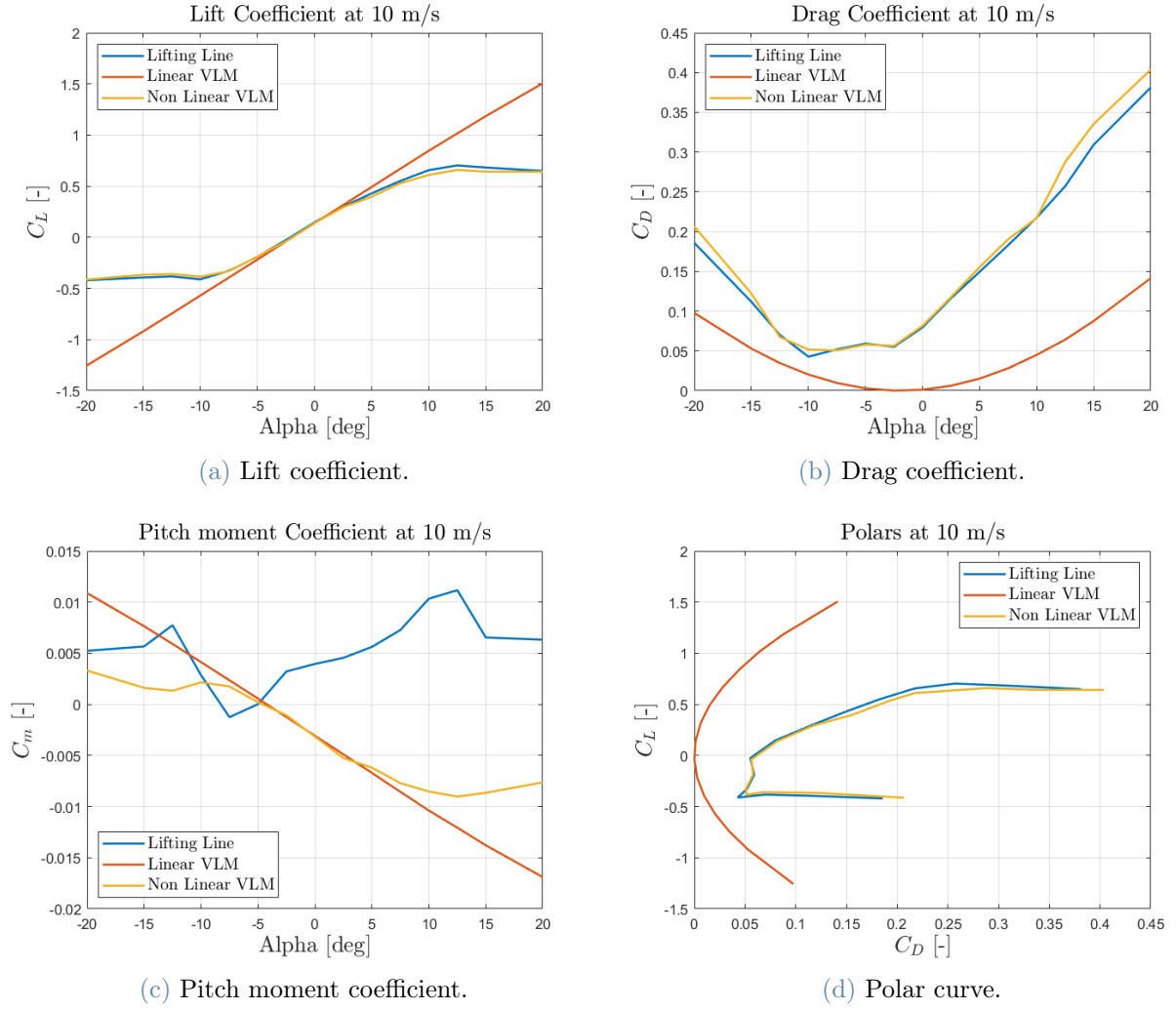


Figure 3.10: Aerodynamic coefficients for the clean wing (without propellers) at 10 m/s with different meshing of the aerodynamic surface, $Re = 26471$.

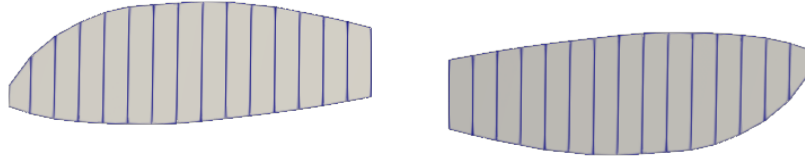
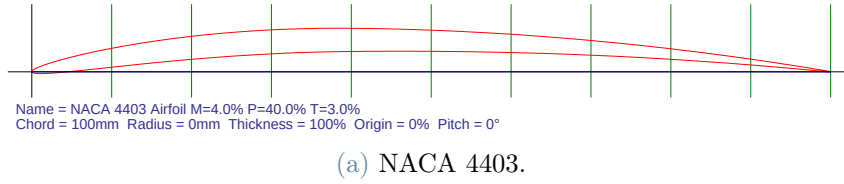
3.4. Propeller's Model and Panel Distribution

To ensure a reasonable computational time for the full model and due to the necessity of accurately capture the viscous phenomena, the propeller was modeled as a lifting line.

The commercial propeller used in this thesis had elliptical shaped blades which were difficult to capture accurately through the built-in parametrization tools. DUST conveniently includes a feature that allows the generation of a cubic spline through the aerodynamic centers (AC), defined by the XYZ coordinates, of several sections. This spline can be subsequently interpolated with a user-defined number of points to achieve the desired geometric resolution.

Each blade's section was defined, in addition to the AC coordinates, through the chord dimension, the twist angle and the airfoil characteristic. The twist and the chord were obtained graphically, from close up photos, reported in Fig. 2.3, of the propeller. Using Solidworks it was possible to compute the length of the different segments and relative angles between them. The aerodynamic characteristics of the airfoil were inputted by .c81 file, which were created using NeuralFoil.

Moreover, by using the aerosandbox toolkit the NACA 4403, represented in Fig. 3.11a, was chosen as the propeller's airfoil.



(b) Propeller blades discretization.

Figure 3.11: The airfoil shape and the discretization using lifting line for the propeller blades.

The constructed tables encompass Reynolds numbers between 10^3 and $2.2 \cdot 10^4$, with Mach numbers defined up to $M = 1$. At the highest rotational speed, the propeller static tip velocity reaches approximately

$$U_{tip} = \omega \cdot r = 2\pi \cdot \frac{RPM}{60} \cdot r = 95 \text{ m/s}, \quad (3.2)$$

yielding $M = 0.316$ and $Re \simeq 20000$ for the maximum propeller rotational speed of 60000 RPM. Nonetheless, in the early stages of the simulations, these quantities may be overpredicted before converging to their actual physical values thus the higher range of definition.

For the angles of attack, a discretization step of 0.5° was applied up to 15° , followed by 5° increments up to 30° , and 15° increments for the remaining range up to 180° . The same distribution was mirrored for the corresponding negative angles. For angles beyond the deep stall configuration (beyond 10° to 15°), NeuralFoil automatically implements the

Viterna [54] corrections shown in Eq. 3.3, in which the airfoil is considered as a thin flat plate and the aerodynamic coefficients are computed for such shape.

$$C_{D_{\max}} = 1.11 + 0.018 \cdot AR \quad (3.3a)$$

$$A_1 = \frac{C_{D_{\max}}}{2} \quad (3.3b)$$

$$A_2 = (C_{L_{\text{stall}}} - C_{D_{\max}} \sin(\alpha_{\text{stall}}) \cos(\alpha_{\text{stall}})) \frac{\sin(\alpha_{\text{stall}})}{\cos^2(\alpha_{\text{stall}})} \quad (3.3c)$$

$$B_1 = C_{D_{\max}} \quad (3.3d)$$

$$B_2 = C_{D_{\text{stall}}} - \frac{C_{D_{\max}} \sin^2(\alpha_{\text{stall}})}{\cos(\alpha_{\text{stall}})} \quad (3.3e)$$

$$C_l = A_1 \sin(2\alpha) + A_2 \frac{\cos^2(\alpha)}{\sin(\alpha)} \quad (3.3f)$$

$$C_d = B_1 \sin^2(\alpha) + B_2 \cos(\alpha) \quad (3.3g)$$

$$C_m = -\frac{\sin(\alpha)}{4} \quad (3.3h)$$

In these equations, AR is the aspect ratio of the wing, while $C_{D_{\max}}$ indicates the drag coefficient at an angle of attack of 90° , and finally, the stall subscript refers to the stall angle of attack of the profile.

A sensitivity analysis was conducted up front for both the spatial and time discretization. The latter was the most important parameter in order to accurately capture the experienced flow phenomena. For the space discretization of the propeller, the only parameter available is the number of elements in which the span is divided, as shown in Fig. 3.11b.

Estimating the propellers performance using a classical CFD approach was considered unsuitable because of the prohibitive computational cost associated with URANS simulations. The solution to the above problem was the direct application of DUST to the propeller model.

To evaluate the best meshing to accurately describe the blade of the propellers, a grid convergence study was carried out. In this study, the blade was discretized initially in 10 panel spanwise and the number of panel was gradually increased until 50, with a ratio of 1.5 between sequential increments. The results, represented in Fig. 3.12, show that for 33 or less panels the thrust computed is almost identical, within a 2% margin, while with 50 panels, the result diverge. This behavior is due to the panel being too small, causing numerical instabilities inside the wake, where particles are being generated too close to each other.

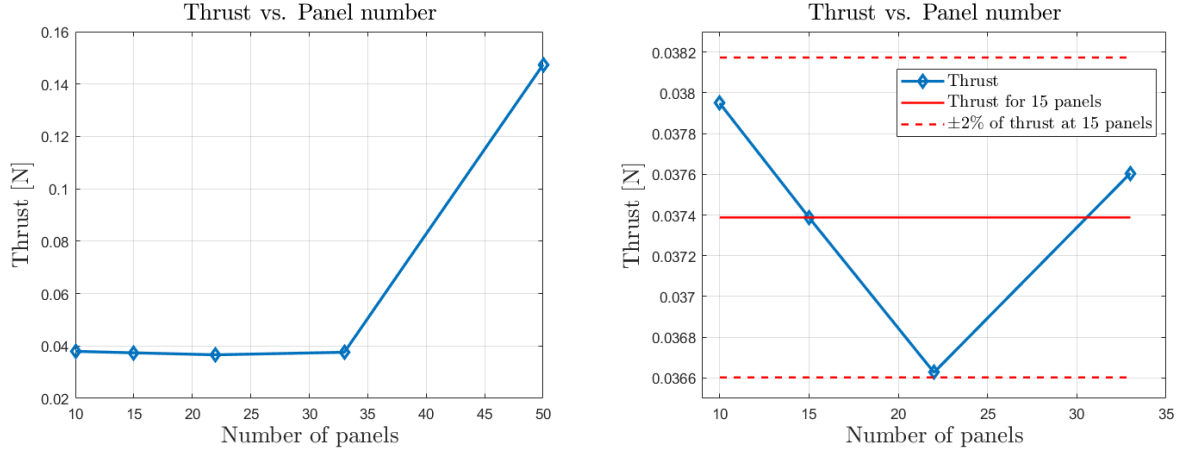


Figure 3.12: Spanwise number of panel convergence for the propeller, with the magnified view on the right, $\text{RPM} = 60000$, $U_\infty = 0 \text{ m/s}$.

According to the above results, the propeller blade was discretized in 15 panels in the spanwise direction, as this resolution provided the best compromise between results accuracy and computational time. A higher number of panels would have yielded a geometry that more closely represented the actual blade shape and reduced the degree of averaging in the final results, but this would have significantly increased computation's time.

A similar consideration applies to the choice of the time step. In this case, the analysis focused on determining the number of time steps per revolution required to adequately represent the underlying physics of the problem. The results of the analysis are presented in Fig. 3.13. While increasing the number of time steps per revolution inevitably leads to higher computational costs, it does not necessarily guarantee a corresponding improvement in accuracy. Conversely, selecting fewer time steps per revolution may compromise the physical fidelity of the simulation, preventing the correct capture of the relevant aerodynamic phenomena.

Finally, a temporal discretization corresponding to 40 time steps per propeller revolution was selected, which resulted in a time step size of $dt = 2.5 \times 10^{-5} \text{ s}$. This time step was selected because, although the percentage error of the thrust between successive discretization decreases below 1% when using 60 time steps per revolution, the thrust values remain within 1% of those obtained with the chosen time step across all the tested temporal discretizations; thus, a faster computational time was preferred, at the expense of a small reduction in accuracy. Such a small time step explains the convergence difficulties encountered with the non linear VLM formulation discussed in Sec. 3.3, as wake particles are generated close to each other and this favors the arise of numerical singularities.

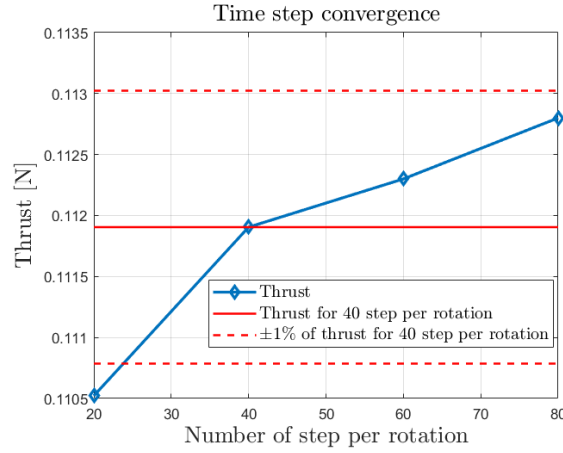


Figure 3.13: Time step convergence for propeller.

3.5. Numerical Full Model

Following the simulations of the individual components, a numerical analysis of the complete configuration was performed. The propellers were positioned upstream of the wing, and the aircraft trim condition was set to an angle of attack of 7.5° at a freestream velocity of 5 and 10 m/s. The above conditions were selected because experimental results suggested that the distributed propulsion aerodynamic effect on the wing was most significant around such angle of attack.

The numerical model was divided in two distinctive components as showed in Fig. 3.14, with the relative name that will be applied throughout the analyses. The name *Full model* will refer to the combination of the effects of both components acting together.

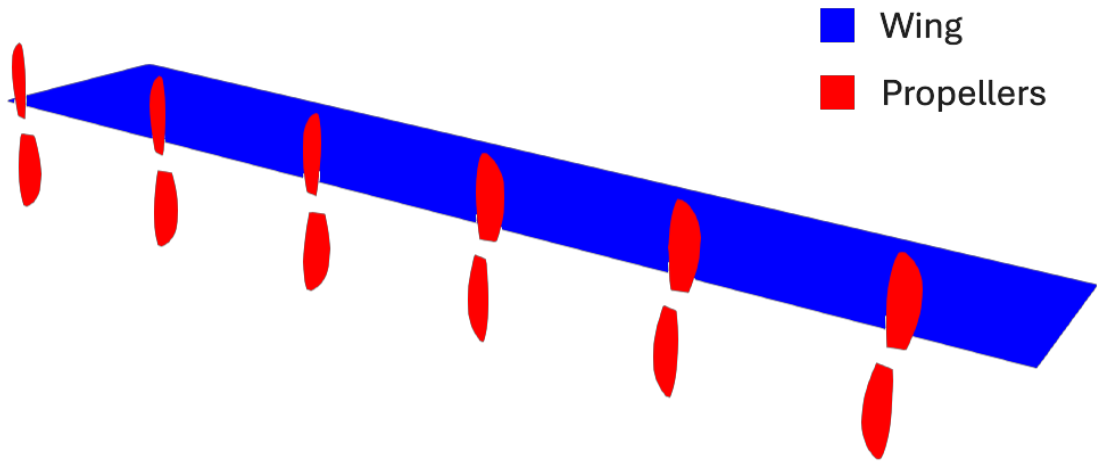


Figure 3.14: Nomenclature of the two components of the numerical model; the combination of both is defined as *Full model*.

The propellers' rotational speed was set at 60000 RPM and their respective $\bar{\Omega}$ are reported in Table 3.2 as a function of the freestream velocity. Only runs at two values of this parameter were analyzed; further investigations on the relevance of $\bar{\Omega}$ will be performed in future work.

RPM	Velocity	J	$\bar{\Omega}$
60000	5 m/s	0.1667	18.85
60000	10 m/s	0.3333	9.42

Table 3.2: $\bar{\Omega}$ and relative advance ratios for the different freestream velocities with constant propeller speed of 60,000 RPM.

For the wing discretization, both lifting line and non linear VLM were analyzed. In the non linear VLM configuration, the convergence problem that previously arose in the clear wing was noticed again. The corrections applied in Sec. 3.3 were not sufficient to guarantee convergence due to the smaller timestep necessary for the correct representation of the propellers wake in respect to the wing wake, generating particles very close to each other, leading to numerical oscillations and instabilities. Considering all the above the lifting line configuration was applied to this analysis for both the wing and the propellers.

Regarding the time discretization, the same used for the propeller analysis was used, and a total of 20 full rotation of the propellers were considered, in order to achieve fully developed flow across the wing.

4 | Results and Discussion

4.1. Wing Aerodynamics

The beginning of the experimental campaign in the wind tunnel was designed to provide the baseline with the clear wing, the wing with the nacelles but without propellers.

The tests of the wing without the propellers were conducted at four different Re , as reported in Table 4.1 with the dynamic pressure. The freestream velocities were selected for doubling the dynamic pressure between successive steps providing a smoother variations in the aerodynamic response, avoiding excessively large increments which could potentially lead to abrupt changes in the flow behavior. The Re number was based on the wing's mean aerodynamic chord of 40 mm.

Airspeed vs. Dynamic Pressure and Reynolds number

Speed	Dynamic Pressure	Re
5 m/s	14.6 Pa	13235
7.5 m/s	33.1 Pa	19853
10 m/s	62.0 Pa	26471
15 m/s	132.6 Pa	39706

Table 4.1: Wing's Re numbers based on the chord and dynamic pressure for the airspeed settings of the wind tunnel.

The angle of attack sweeps were from -20° to 20° with smaller step around zero and larger around the extremes. Between $\pm 10^\circ$ the AOA step was one, while it was 2.5° outside the $\pm 10^\circ$ range. In the following graphs, only the angles higher then -7° are presented, as lower angles of attack are not relevant for usual flight conditions. For each air speed and angle of attack, 5 measurements were operated for statistical analysis, explained in

Sec. 2.3. Finally, the aerodynamic coefficients of each run were estimated from Eq. 4.1 and comparisons from different configurations could be presented.

$$C_L = \frac{L}{\frac{1}{2}\rho_\infty V_\infty^2 S} \quad C_D = \frac{D}{\frac{1}{2}\rho_\infty V_\infty^2 S} \quad C_M = \frac{M}{\frac{1}{2}\rho_\infty V_\infty^2 c S} \quad (4.1)$$

Lift coefficients, at various Re numbers, are presented individually and combined in one figure for comparison. Fig. 4.2 shows the expected behavior with the linear part of the lift curve similar at all Re number but with the non linear portion increasing the lift at the increase of Re. In particular an increase in the stall angle from about 5° to 10° at higher Re numbers can be noticed. Measurements at lower velocities were accompanied by a higher noise, while from 7.5 m/s onward, the confidence interval of the presented value increased drastically.

The drag coefficient, represented in Fig. 4.4, shows less variation at various Re numbers. Also for C_D the higher error bars reappear themselves at the lowest Reynolds regime.

The pitching moment coefficient exhibits a highly non linear behavior, more apparent at lower velocities. At higher velocities the curves at 10 and 15 m/s are very close to each other. The pitching moment coefficient suffers from a higher uncertainty and irregular trend with AOA, with respect to the other coefficients, also because of the difficult procedure for calibrating the pitch axis around a zero value, but primarily because the pitch moment is highly nonlinear at the lower Reynolds regimes.

Figure 4.8 presents the polar curves corresponding to the different free stream velocities investigated. As the velocity increases, the polars exhibit a more conventional aerodynamic behavior, whereas at lower velocities the nonlinearities observed in the other coefficients remain evident.

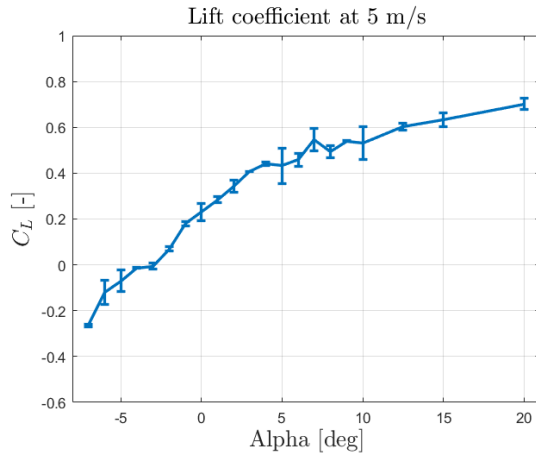
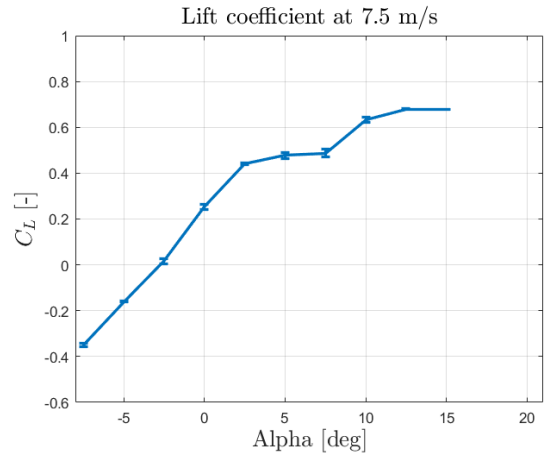
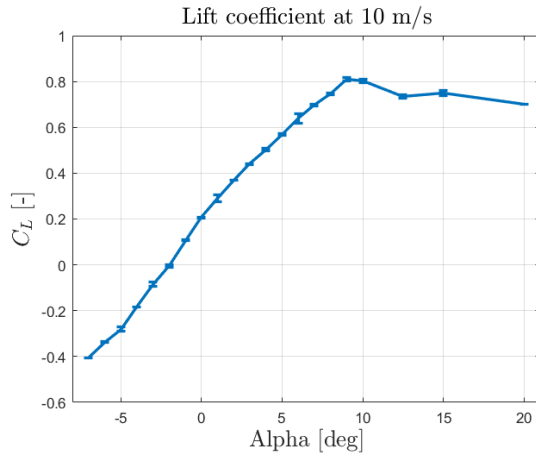
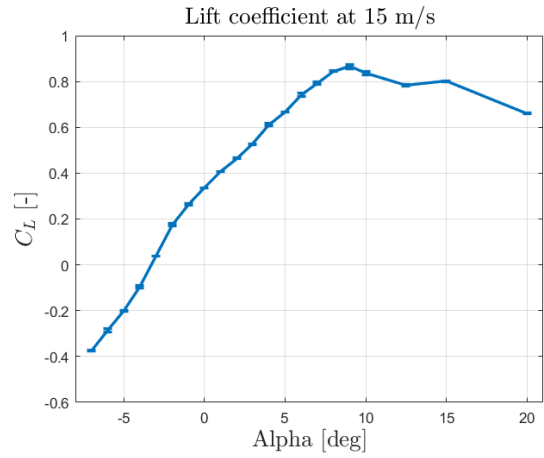
(a) Lift coefficient at 5 m/s, $Re = 13235$.(b) Lift coefficient at 7.5 m/s, $Re = 19853$.(c) Lift coefficient at 10 m/s, $Re = 26471$.(d) Lift coefficient at 15 m/s, $Re = 39706$.

Figure 4.1: Lift coefficient curves against angle of attack at different airspeed for the clear wing.

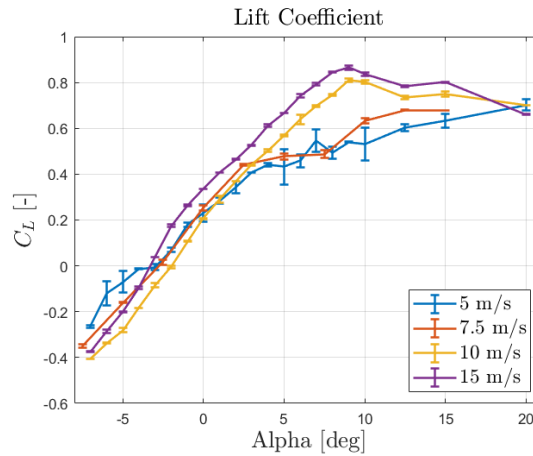


Figure 4.2: Lift coefficient at different freestream velocities.

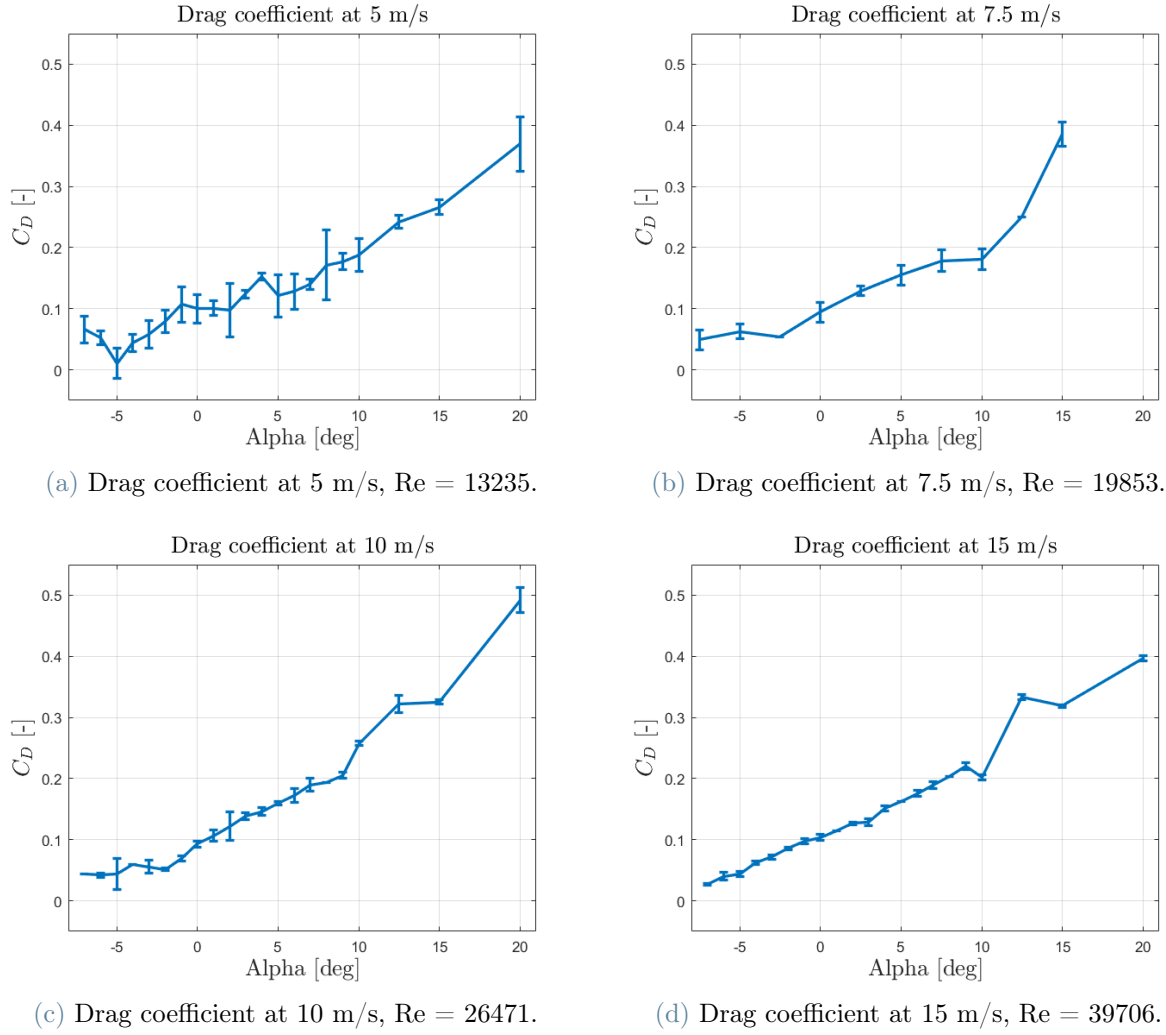


Figure 4.3: Drag coefficient curves against angle of attack at different airspeed for the clear wing.

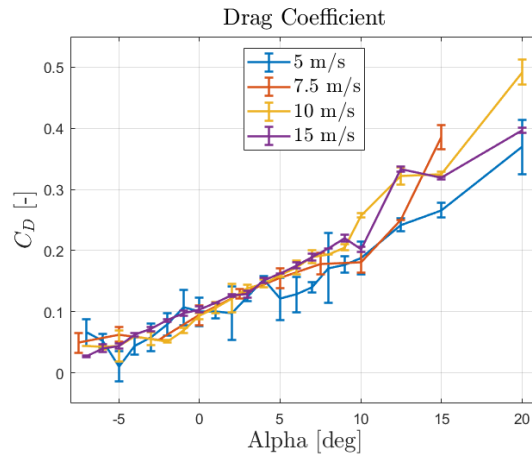
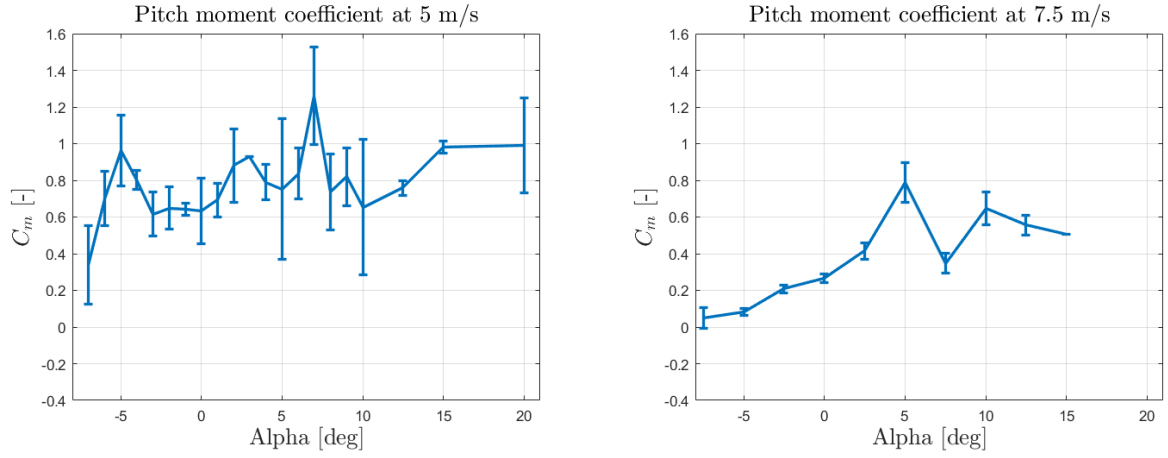
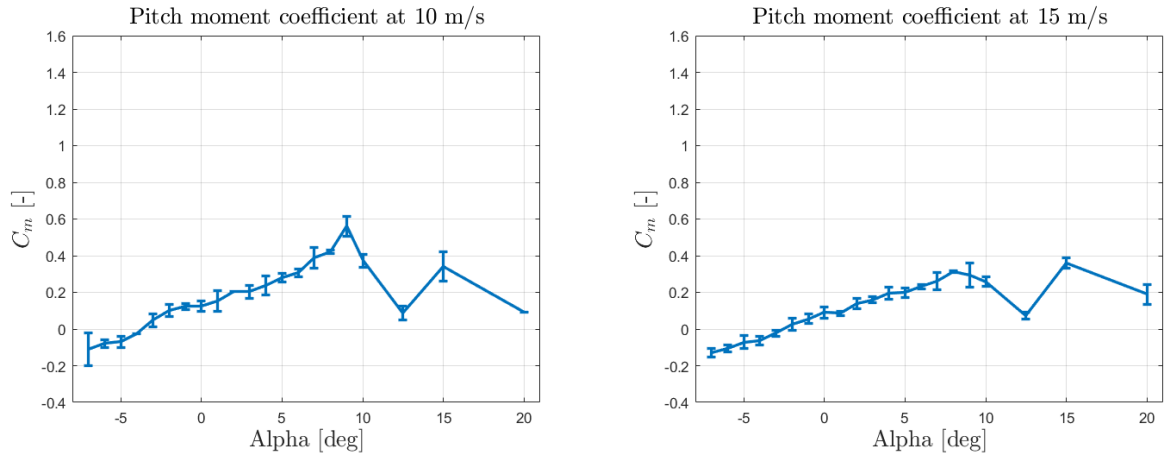


Figure 4.4: Drag coefficient at different freestream velocities.



(a) Pitch moment coefficient at 5 m/s, $Re = 13235$. (b) Pitch moment coefficient at 7.5 m/s, $Re = 19853$.



(c) Pitch moment coefficient at 10 m/s, $Re = 26471$. (d) Pitch moment coefficient at 15 m/s, $Re = 39706$.

Figure 4.5: Pitch moment curves against angle of attack at different airspeed for the clear wing.

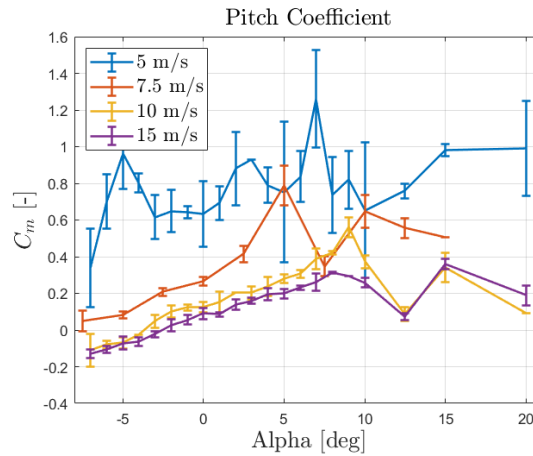


Figure 4.6: Pitch moment coefficient at different freestream velocities.

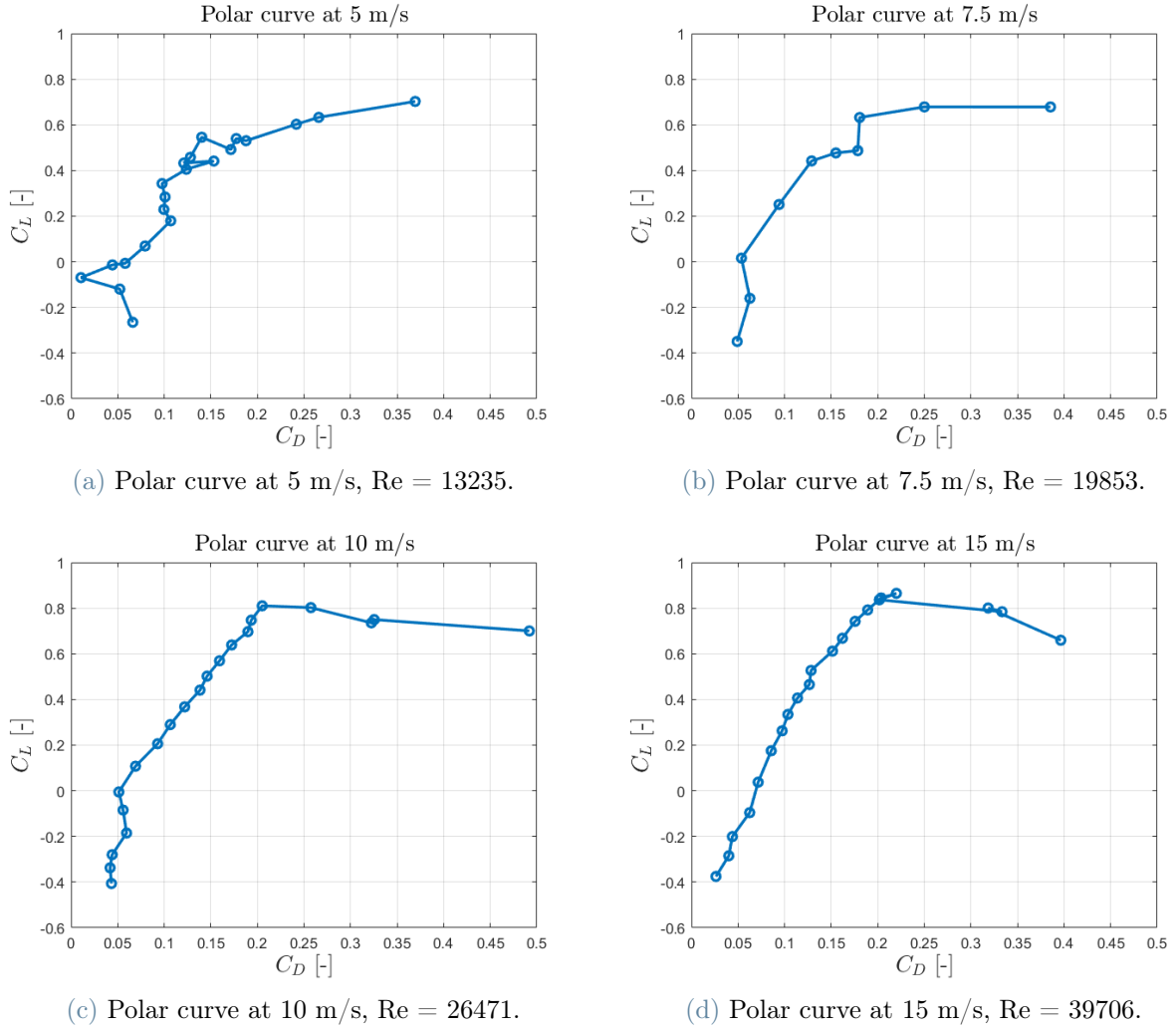


Figure 4.7: Polar curves with coefficients at different airspeed for the clear wing.

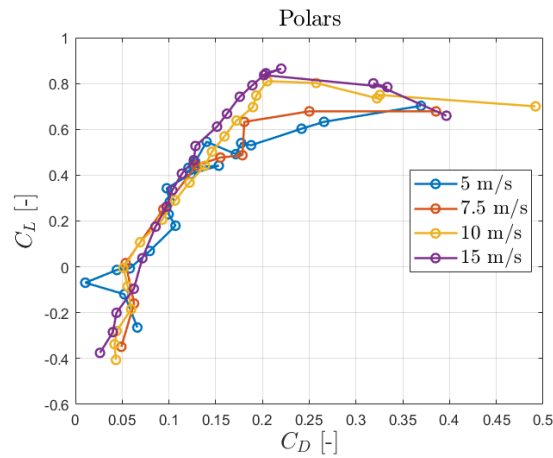


Figure 4.8: Polar curves at different freestream velocities.

Useful metrics to characterize the performance of the wing regard the lift to drag ratio and the pitch moment coefficient as functions of the lift coefficient.

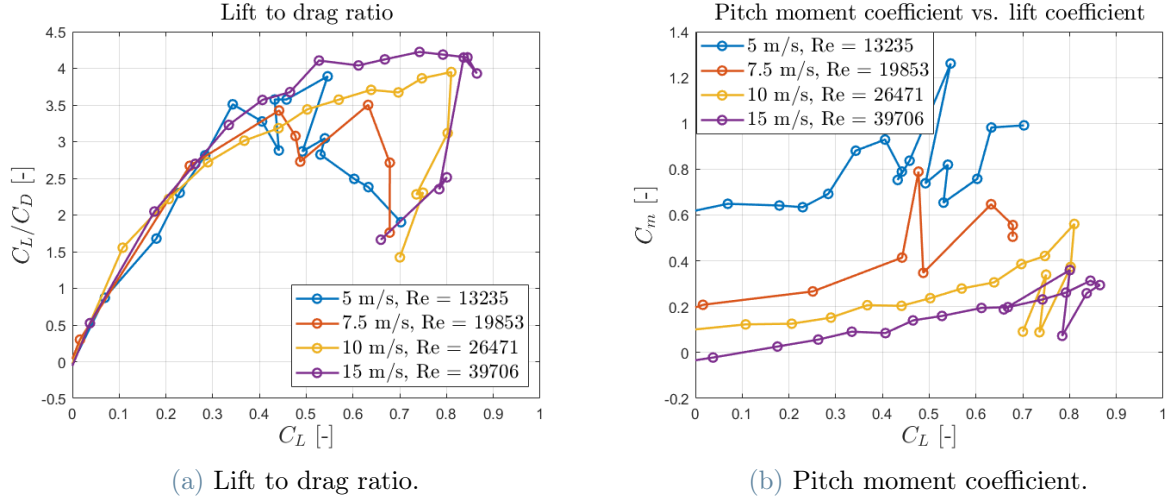


Figure 4.9: Lift to drag ratio and pitch moment coefficient versus C_L , for the different free stream velocities considered, for the clear wing.

Fig. 4.9a shows that the lift to drag ratio remain constant within the linear region of the $C_{L\alpha}$ curve for the different Reynolds considered, The increase in stall angle of attack is reflected in a higher maximum L/D value, which rises from 3.5 at 5 m/s to 4.2 at 15 m/s, and occurs at correspondingly higher values of C_L . The behavior of C_m is reported in Fig. 4.9b and it shows the nonlinearities already observed for the lower free stream velocities, while as the latter increases the more linear behavior is re-established.

The experimental results were employed to validate the numerical simulations. Fig. 4.10 shows the comparison of results on the Clear Wing at the freestream velocity of 10 m/s.

The wing on DUST was modeled using the airfoil table derived from the experimental tests, as explained in Sec. 3.3, thus the simulation closely matches the experimental results as expected. The discrepancy between the two curves was attributed to the use of three dimensional data to model a two dimensional airfoil, which was subsequently extruded into a three dimensional wing. This repeated transition between two and three dimensional representations introduced errors that propagated until this final result.

Once the wing model was finalized, the aerodynamic investigation could be undertaken. Accordingly, polar curves were computed for different velocities, corresponding to those used in the wind tunnel experiments.

Finally, a confrontation of aerodynamic coefficients between different freestream velocities

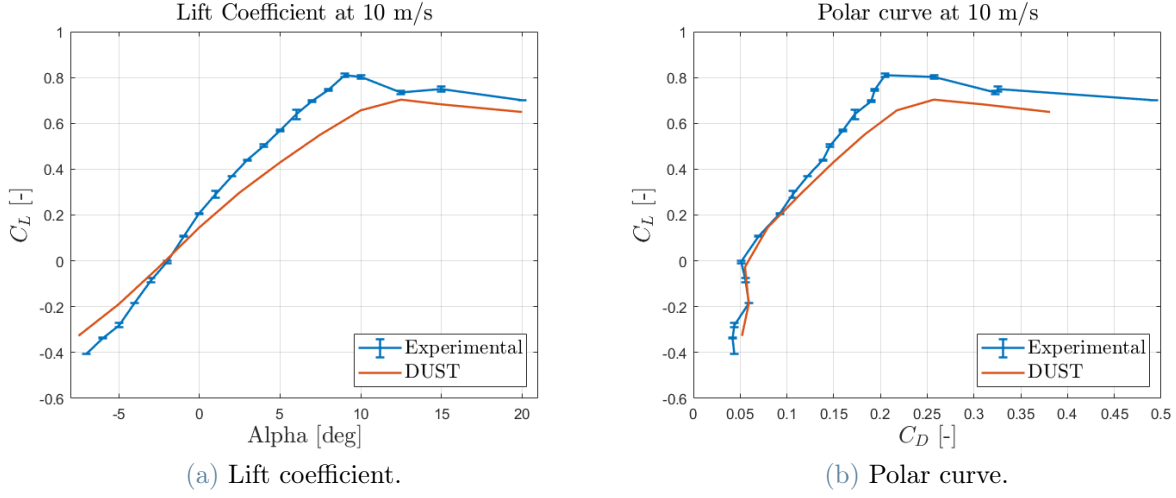
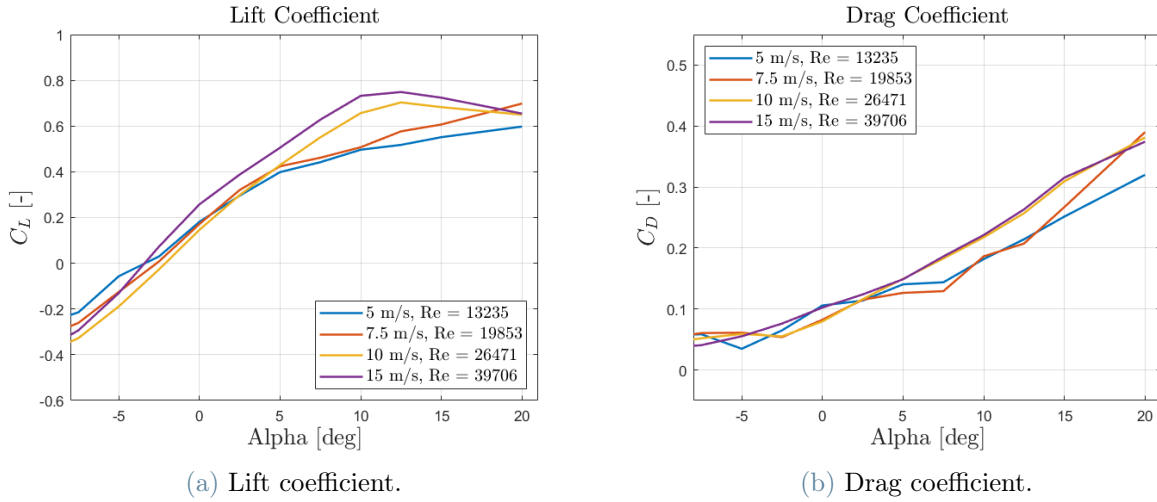


Figure 4.10: Lift and drag coefficients (through polar curve) comparison between numerical simulation and experimental results at 10 m/s ($Re = 26471$) in isolated wing configuration.

was carried out in Fig. 4.10.

From the graphs above, it can be observed that increasing the freestream velocity leads to higher aerodynamic loads, even when considering the adimensional coefficients alone. In particular, the comparison between 7.5 and 10 m/s in Fig. 4.11a highlights an increase in the stall angle of attack, the same as with the experimental results. Among the parameters analyzed, the pitching-moment coefficient exhibits the least linear behavior, although repeatable trend begins to emerge as the velocity increases.



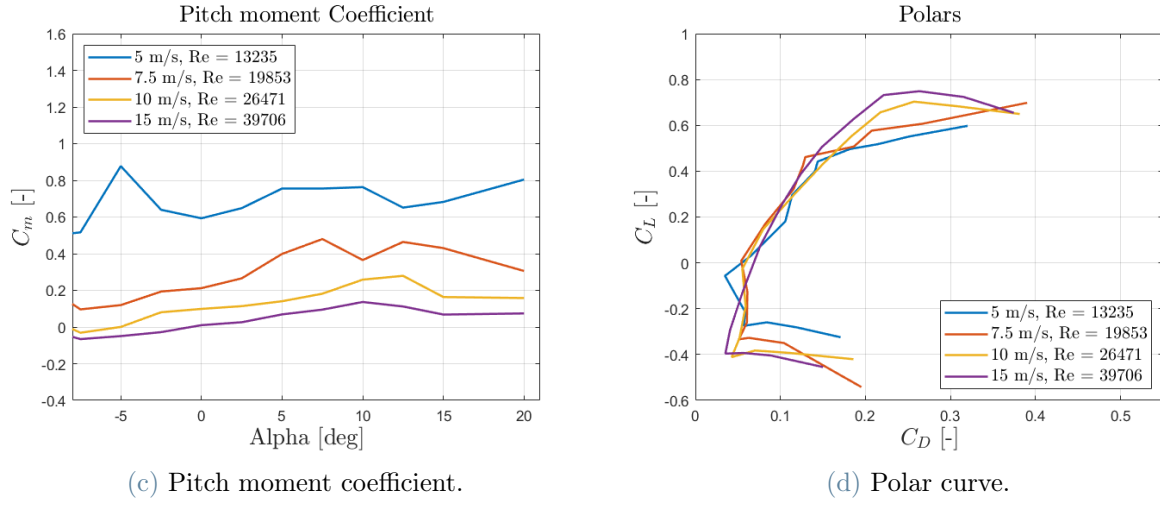


Figure 4.10: Aerodynamic coefficients for the clean wing (without propellers) at different freestream velocities.

4.2. Propulsion's Performance

The experimental characterization of the propulsion's system started with the evaluation of the relationship between the voltage supply and the RPM of the propeller. The RPM are measured by the tachometer described in Sec. 2.2 and the results are presented in Fig. 4.11, showing a linear behavior, as expected.

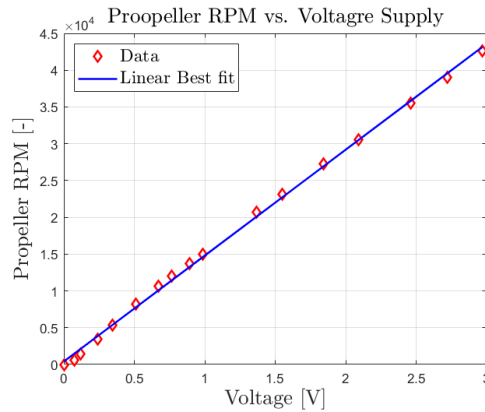


Figure 4.11: Verification of the linear relation between voltage supply and propeller rotational velocity in RPM.

The next test was about static thrust measurements of a single propeller with respect to the RPM. In this instance the velocity of the propeller was measured through the balance, applying the auto-correlation procedure described in Sec. 2.3.3. The results of multiple

runs are shown in Fig. 4.12, where different colors correspond to different runs, and the continuous line represents the best fit of the data obtained using the least-squares method.

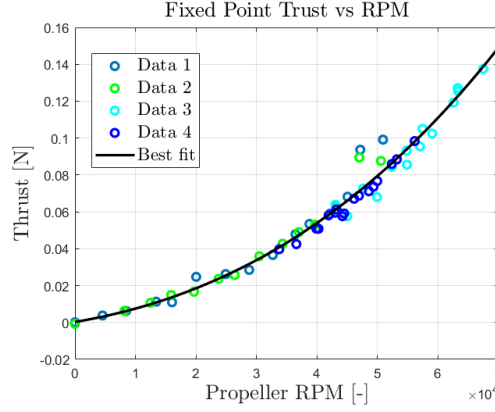


Figure 4.12: Multiple runs measuring propeller's static thrust versus propeller rotational velocity in RPM.

The small propeller exhibits a behavior different from normal full sized one, since the thrust continues to increase as the RPM increases, instead of reaching a maximum. This behavior can be explained by the fact that in conventionally sized propellers, the blade airfoils become increasingly stalled as the velocity rises. In contrast, for the small propeller considered here, the flow initially operates at very low Reynolds numbers; consequently, an increase in velocity shifts the flow toward a more typical aerodynamic regime, leading to improved performance.

A crucial test was the monitoring of the rotational velocity of the propeller with increased free stream flow with respect to static conditions.

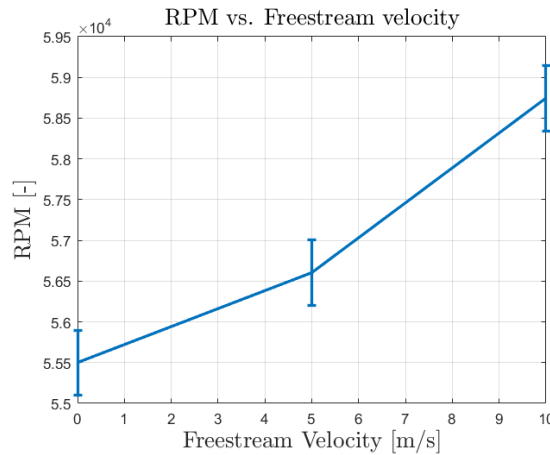


Figure 4.13: Rotational speed of propeller versus wind tunnel freestream velocity at constant input power of 7 V.

The results, illustrated in Fig. 4.13, show the small change in propeller speed with freestream velocity. The main contributor to the statistical experimental error in the measure of the propeller rotational speed was due to the challenge of tuning each propeller at the same speed. The micro motors installed on the wing are sensible to the vibration generated by each other, so the tuning process was operated recursively until a small difference between the propellers speed was found (within 50 RPS).

With the propeller model fully defined, the first numerical run was the estimate of single propeller thrust versus propeller rotational speed in RPM, as illustrated in Fig. 4.14a.

The propeller's pitch angle was set to 13° , since the best agreement with the experimental thrust was sought, and that value provided the closest match, as shown in Fig. 4.14b.

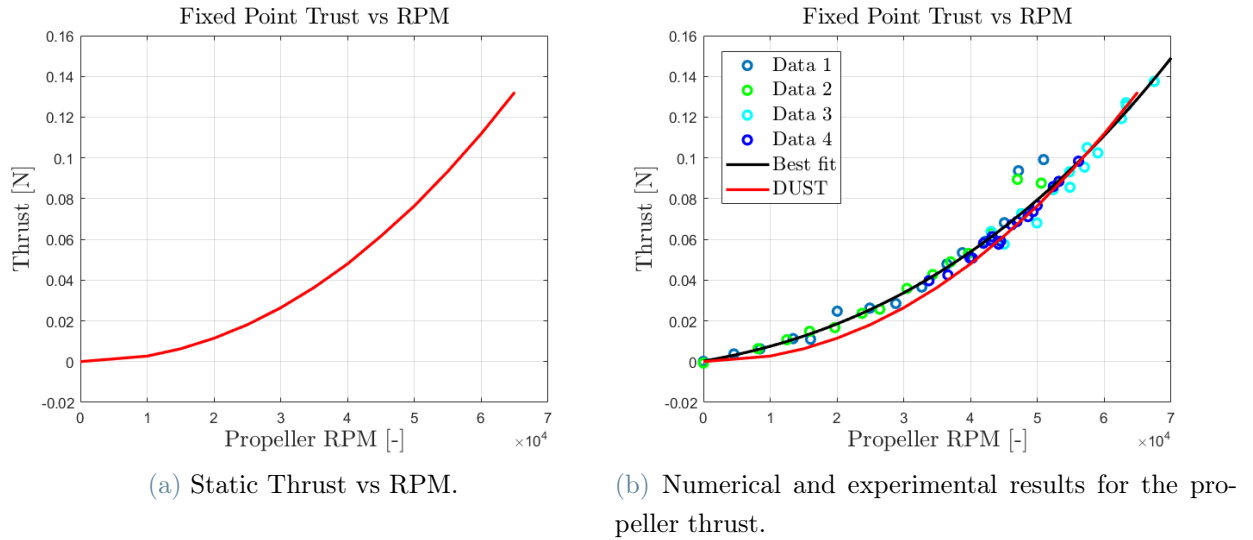


Figure 4.14: Comparison between numerical and experimental results for the propeller thrust. Input voltage 0 to 7 V for experimental values.

As illustrated Fig. 4.14b, the simulation slightly under-predicts the thrust at lower rotational regimes. However, for the rotational speed applied for all wind tunnel tests of 60,000 RPM, simulation and experimental thrust are in perfect agreement. Therefore, the numerical results for the single propeller are representative of the actual physical behavior and the associated flow field could be considered realistic. The propeller was modeled using airfoil performance data generated with NeuralFoil, therefore, the agreement between numerical and experimental results suggests that this Python-based tool provides reliable aerodynamic predictions at low Reynolds numbers for thin airfoils such as the NACA 4403. The reliability of NeuralFoil for thick profiles at low Re, is already discussed in Sec. 3.2.

The simulations included the estimation of the torque of the propeller. However the comparison with the experimental results was not possible due to the non availability of the experimental torque because of the temporary downgrade of the sting balance from six-axes to three-axes.

The parameters typically used for describing propeller performance (efficiency (η), thrust (C_T) and torque (C_Q) coefficient, explained in Sec. 2.3.3) are represented in Fig. 4.15, where a separate plot for C_Q was used due to the difference in value scales.

J	η	Re
0	0	12516
0.0831	0.1340	12526
0.1662	0.2534	12556
0.2493	0.3522	12605
0.3324	0.4207	12673
0.4156	0.4330	12761
0.4987	0.3296	12868
0.5818	-0.0741	12992

Table 4.2: Propeller efficiency as a function of advance ratio and Reynolds number computed at 2/3 of the propeller radius, at 60000 RPM.

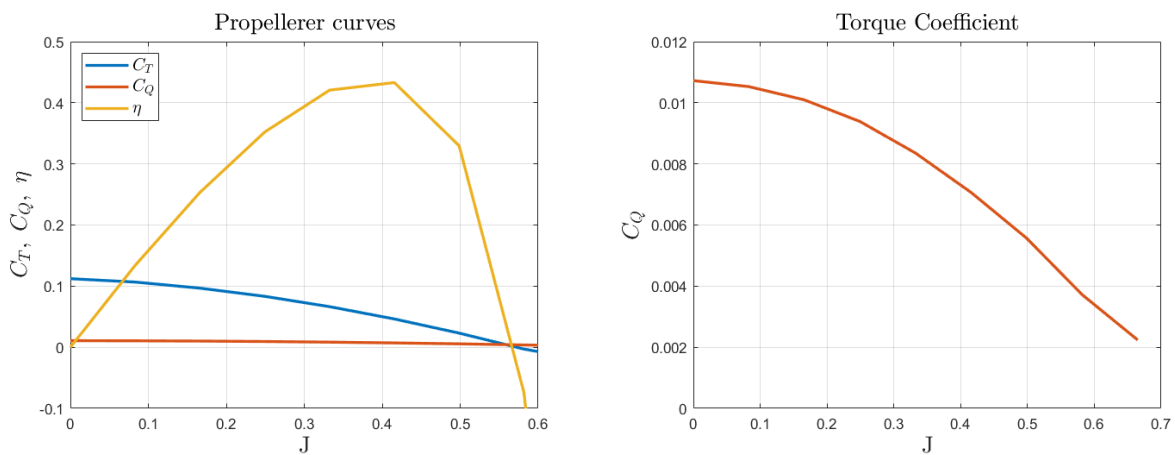


Figure 4.15: Thrust coefficient, torque coefficient, efficiency for the propeller with the magnifying on the torque coefficient.

Although the behavior of the coefficients are typical for propellers of larger dimensions,

the maximum value of efficiency is significantly lower in respect to larger propellers, as expected due to the extremely low Re number, as illustrated in Table 4.2.

Furthermore, the three coefficients intersect the x-axis for a value of $J \simeq 0.56$, corresponding to a freestream velocity of $U \simeq 17$ m/s. The advance ratio J was varied by modifying the freestream velocity while maintaining a constant rotational speed of 60,000 RPM, consequently, the Reynolds number evaluated at two thirds of the propeller radius varied only slightly with increasing freestream velocity, since the dominant contribution to the local velocity is associated with the rotational speed.

A sanity check with simulations of a single propeller at 10 m/s freestream at different angles of attack was run. The results, shown in Fig. 4.16, show a practically constant level of thrust symmetrical with respect to 0° , as expected. The maximum thrust is lower than the static maximum thrust as it should be.

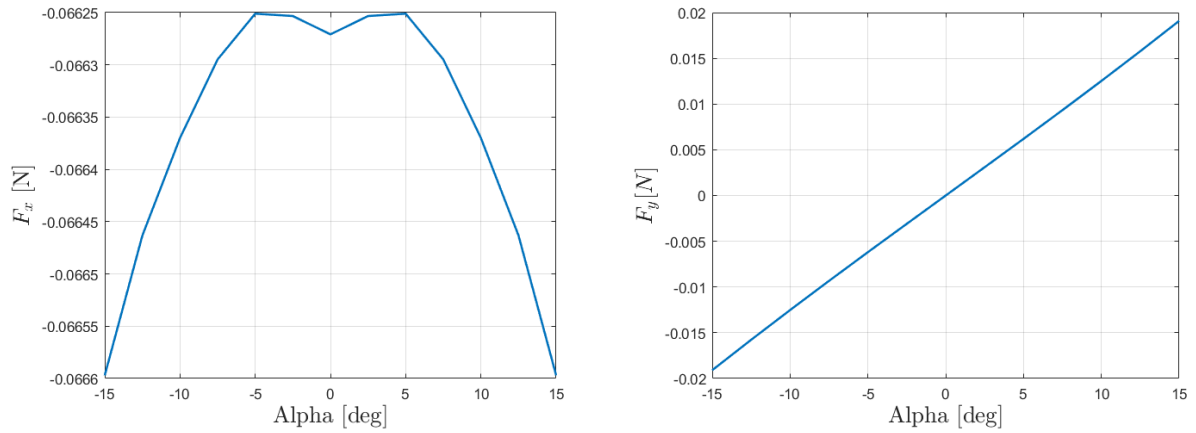


Figure 4.16: Forces respect to body axis of a single propeller at different angles of attack, at 10 m/s and $\text{RPM} = 60000$.

Furthermore, it is noticeable that for the x-axis component of the force, the variation between angles of attack is smaller with respect to the y-axis component. This is simply due to the proportionality with sine and cosine of the two components, which are respectively linear and constant around zero. In general the thrust produced by the propeller increases slightly (in modulus) as the angle of attack increases.

The next runs for the propulsion characterization regarded the interaction of the six propellers, without the wing downstream. Similarly to before, the thrust of the propeller was computed for different angles of attack at a fixed 10 m/s freestream velocity and the results are presented in Fig. 4.17. The symmetry around 0° was lost, however the total

loads generated were similar to the interaction-less case, as F_x differs only about 0.25% between the two cases. For F_y the difference, visible only in a magnified view, is irrelevant.

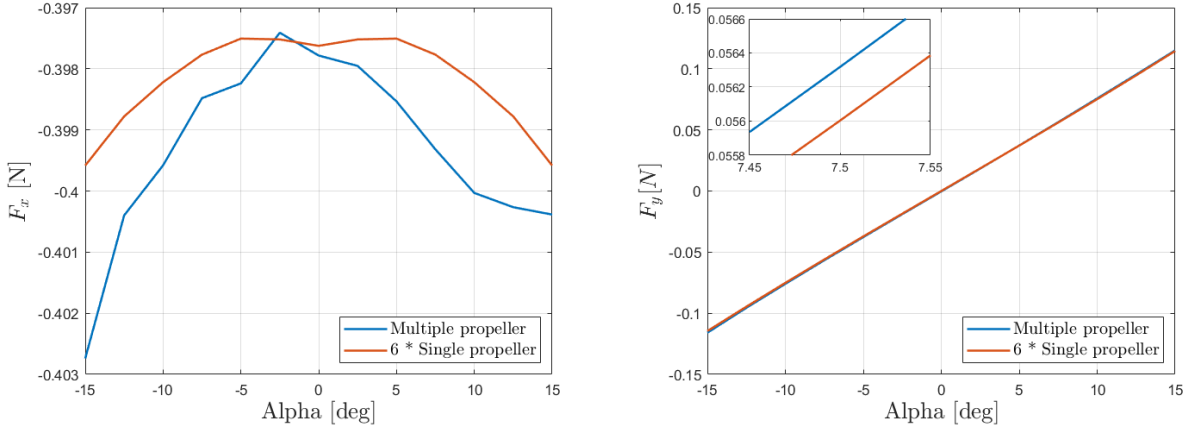


Figure 4.17: Loads along wind axis of multiple propeller at different angles of attack, at 10 m/s, with comparison with single propeller multiplied by 6 and $\text{RPM} = 60000$.

The forces obtained for the angle of attack of 7.5° are reported in Table 4.3 as it is the angle that will be used for the full model in Sec. 4.3. These results show that for a higher freestream velocity, the thrust produced by the array of propellers is lower, as expected.

Velocity	F_x	F_y
5 m/s	-0.5752 N	0.0768 N
10 m/s	-0.3994 N	0.0563 N

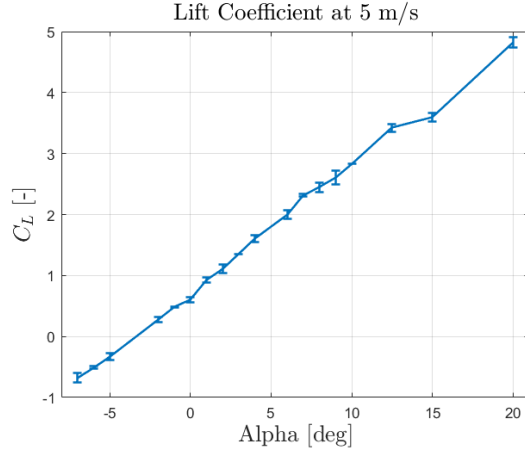
Table 4.3: Force components generated by the propellers for the two velocities tested at 7.5° .

This test was carried out to check that it was feasible the estimate of a meaningful thrust of the propeller on the full model, before starting full model simulations.

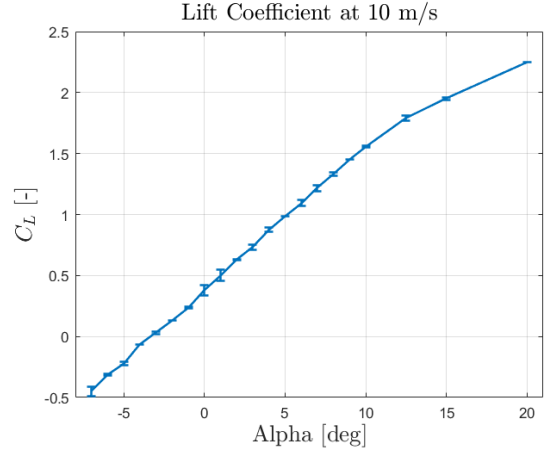
4.3. Complete model results

Once the tests on the wing and the propeller separately were completed, the experimental campaign proceeded with tests on the complete model. These tests focused on the determination of the aerodynamic coefficients of the configuration with all six propellers operating. The tests were conducted for a propeller rotational speed of approximately

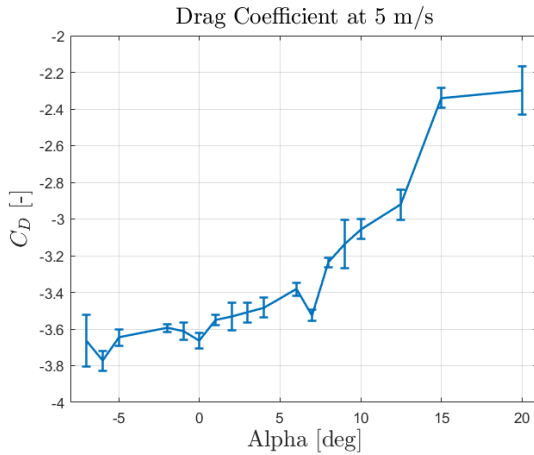
60000 RPM and for two different airspeeds, 5 and 10 m/s. The two speeds were selected because 10 m/s should be the typical operational flight speed of these category of vehicles, while at 5 m/s the effect of the blowing of the propeller was thought to be more relevant.



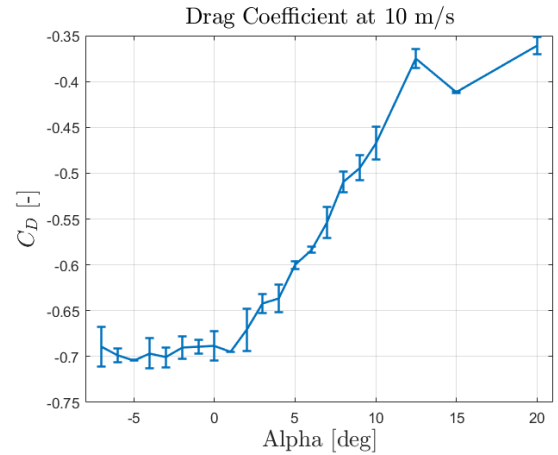
(a) Lift coefficient at 5 m/s, wing's $Re = 13235$.



(b) Lift coefficient at 10 m/s, wing's $Re = 26471$.



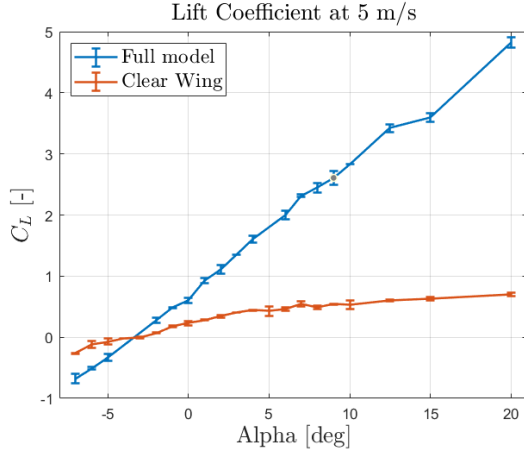
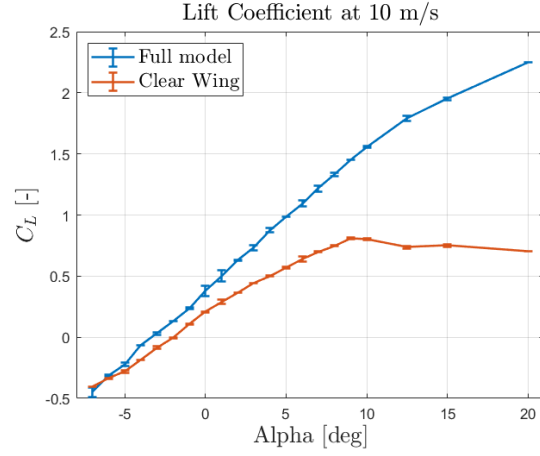
(c) Drag coefficient at 5 m/s, wing's $Re = 13235$.



(d) Drag coefficient at 10 m/s, wing's $Re = 26471$.

Figure 4.18: Lift and drag coefficients for the full model at both the velocities tested.

In the experimental tests, differently from the numerical simulations, it was not possible, at the present stage of preliminary tests, to decouple the contribution from the vertical component of the propeller's thrust from the lift associated with the enhanced wing performance due to the propellers' blowing effect. Due to this limitation, only considerations on the total loads may be done from the experimental results. Future work will try to shed more light on the experimental wing's lift enhancement. This type of analysis is conducted in Sec. 4.4 with the aid of numerical simulations.

(a) Lift coefficient at 5 m/s, wing's $Re = 13235$.

(b) Lift coefficient at 10 m/s.

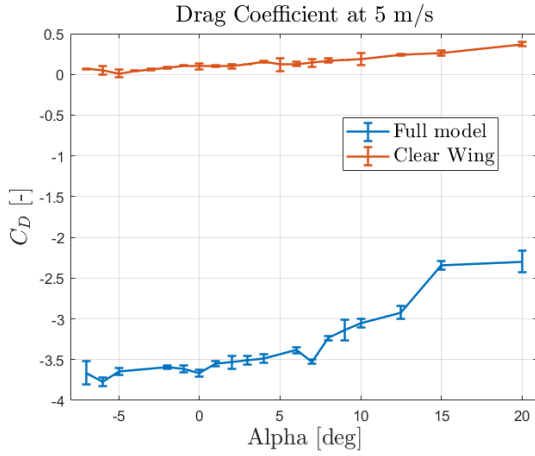
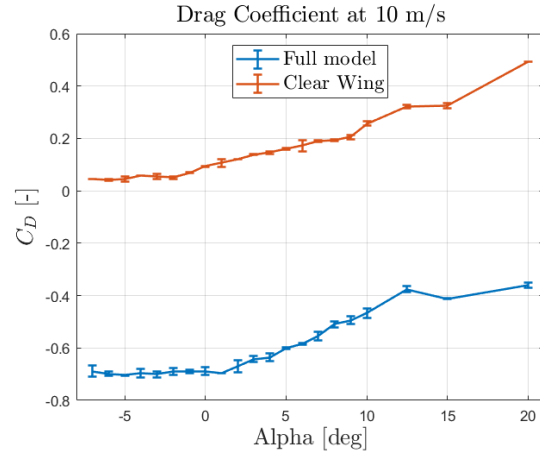
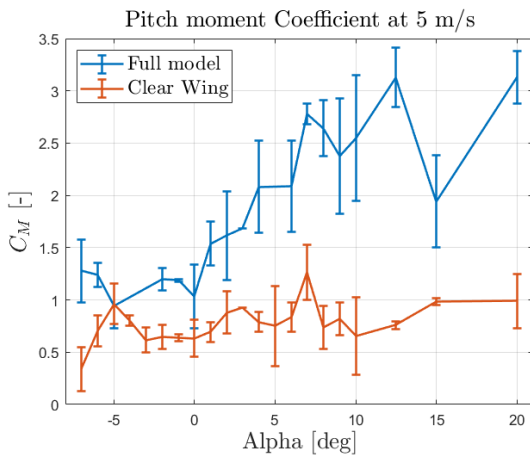
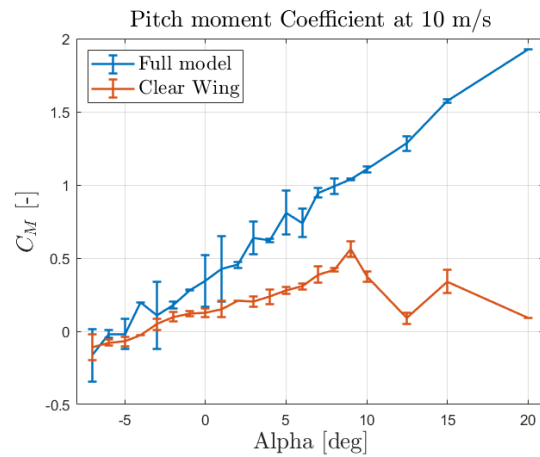
(c) Drag coefficient at 5 m/s, wing's $Re = 13235$.(d) Drag coefficient at 10 m/s, wing's $Re = 26471$.(e) Pitch moment coefficient at 5 m/s, wing's $Re = 13235$.(f) Pitch moment coefficient at 10 m/s, wing's $Re = 26471$.

Figure 4.19: Lift, drag and pitch moment coefficients for the full model at both the velocities tested, against the corresponding alone wing configurations.

The results obtained from the full model test were compared to the ones derived from the clear wing. These comparison are presented in Fig. 4.19.

It is useful to remember that the lift coefficient of Full Model is defined as net stream-normal force coefficient incorporating the effects of wing and propellers. The figures show the expected increase of the net lift coefficient in the Full Model case with respect to Clear Wing, however the increase is smaller in the case of freestream of 10 m/s. The drag coefficient, which is a net streamwise force coefficient, is negative, since the propeller thrust has negative sign. The blowing effect of the propellers, increasing the local velocity of the flow across the wing surface, results in an increase in the maximum angle of attack thus increasing the possible aircraft's attitude, effectively extending the operational envelope. This represents a key advantage of DEP, as it demonstrates the capability to perform maneuvers that would otherwise be unattainable within the original aerodynamic limits of the configuration.

The thrust, representing the dominant contribution to the longitudinal load by the cosine of the angle of attack, exhibits the expected behavior with maximum absolute values around zero angle of attacks and decreasing towards higher AOA. Absolute levels of thrust are lower at higher freestream velocities, as expected. At larger angles of attack, in fact, the thrust-producing capability of the propeller is reduced due to the altered angle at which the flow reached the blades, as predicted by the numerical simulations presented in Sec. 4.2. Consequently, the reduction in the total longitudinal load at high angles of attack can be attributed to the combined effect of increased wing drag and degradation of propeller performance under inclined flow conditions.

Propellers operation led to a more linear trend in the pitching moment, although with a noisy behavior. For a freestream velocity of 5 m/s, the coefficient exhibits higher noise and reduced linearity, similar to the isolated wing case. The primary difference between Clear Wing and Full Model conditions however is the dramatic increase of nose-up moment, introduced by the propellers' effects on the wing.

In the experimental campaign, the only configuration in which the propeller thrust did not affect the lift coefficient was at 0° angle of attack, since the thrust has no vertical component at this condition. This made it possible to isolate the performance improvement attributable solely to the blowing effect, excluding any contribution from the thrust component itself.

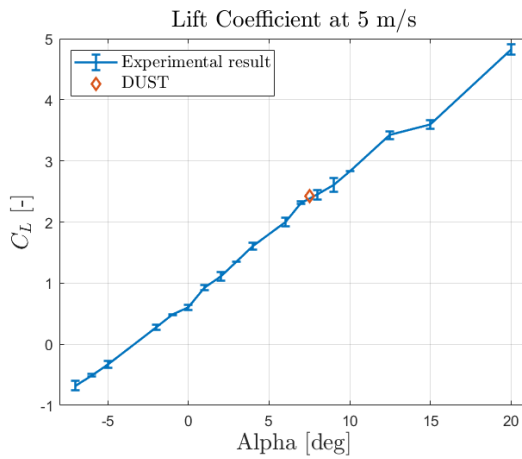
Table 4.4 shows that the blowing effect increases the lift coefficient at both freestream velocities considered. However, the impact is more pronounced at the lower velocity, where the propellers operate more effectively relative to the incoming flow.

Velocity	Configuration	C_L	$\Delta C_L\%$
5 m/s	Clear Wing	0.2294	+162%
	Full model	0.6014	
10 m/s	Clear Wing	0.2060	+83.8%
	Full model	0.3787	

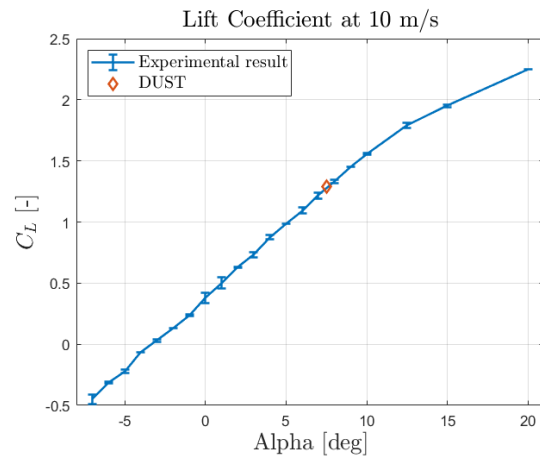
Table 4.4: Comparison of wing performance between the Clear Wing and the Full model, for the two freestream velocities analyzed at 0° AOA.

The original tests matrix included different airspeeds and at different RPM regimes. Unfortunately, during the test at 5 m/s the right-most external motor experienced a fatal failure due to the overheating, exacerbated by the higher than specified voltage supply. The model was supposed to be repairable, as testified by the different parts presented in Sec. 2.1.1, however the bonded components made the process not feasible. One possible solution would have been to remove the portion of the leading edge housing the faulty motor and subsequently reprint that section, however in the event of a successful reassembly, neither the 3D-printing process nor the bonding procedure could guarantee a perfect geometric match between the original and repaired components. In such new conditions it would have been advisable to run all the tests again with the repaired model. The whole process was discarded for time constrains. Furthermore, the main scope of the work, the design, set up and testing of the facility was considered achieved.

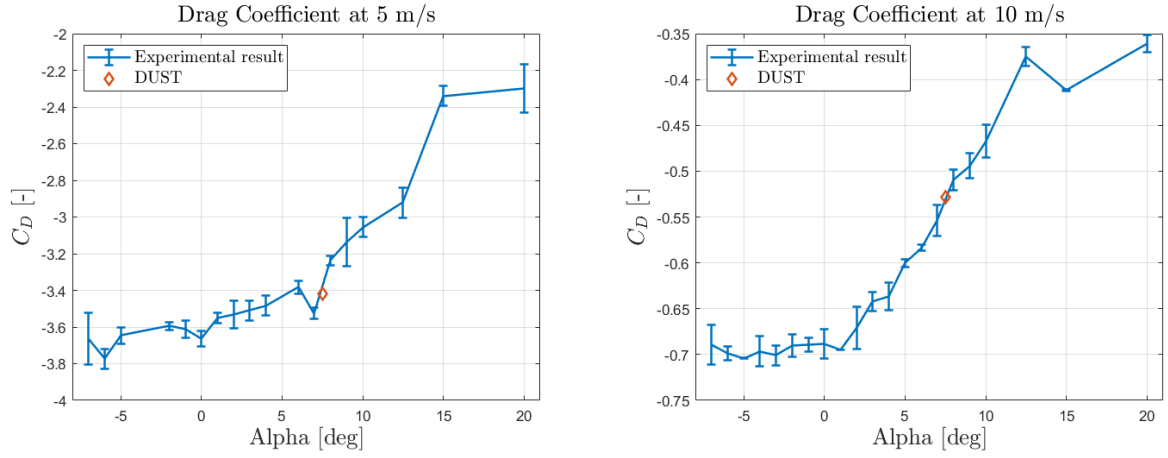
Going back to the numerical approach, simulations were carried out at 7.5° AOA for both the velocities analyzed, and the comparison with experimental results is illustrated below.



(a) Lift coefficient at 5 m/s, wing's $Re = 13235$.



(b) Lift coefficient at 10 m/s, wing's $Re = 26471$.



(c) Drag coefficient at 5 m/s, wing's $Re = 13235$. (d) Drag coefficient at 10 m/s, wing's $Re = 26471$.

Figure 4.19: Lift and drag coefficients comparison for the full model against experimental results. Propeller speed 60,000 RPM, voltage input 7 V.

Fig. 4.19 shows the close match between simulation and experimental results, thus the mid fidelity approach of DUST could be successfully employed for this type of project when the input data for the modeling are based on physical results obtained experimentally. Further implementation in 2D analysis could avoid the use of the experimental data entirely.

4.4. Physical Insights

The numerical approach, once validated, allows for useful insights in the flow physics and phenomena thanks to DUST post-processing capabilities, among which velocity flow field computation stands out.

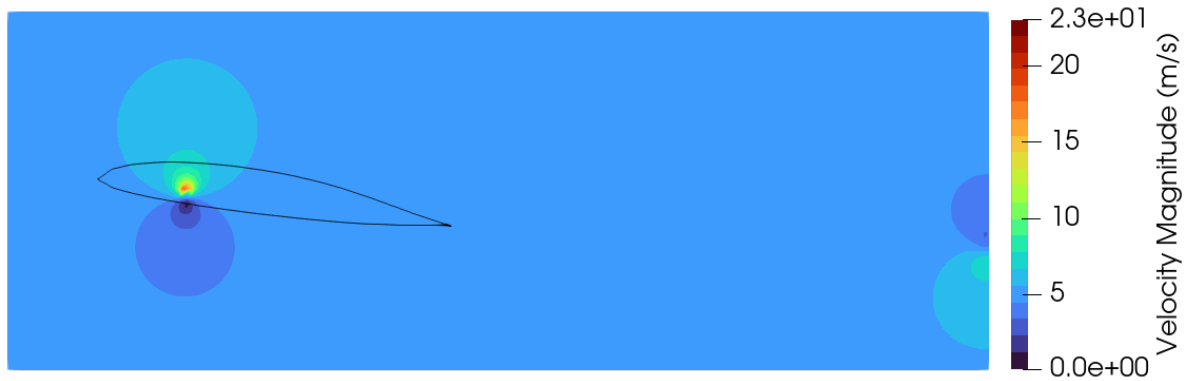


Figure 4.20: Velocity in m/s around the mid-span airfoil at 5 m/s freestream and at 7.5° .

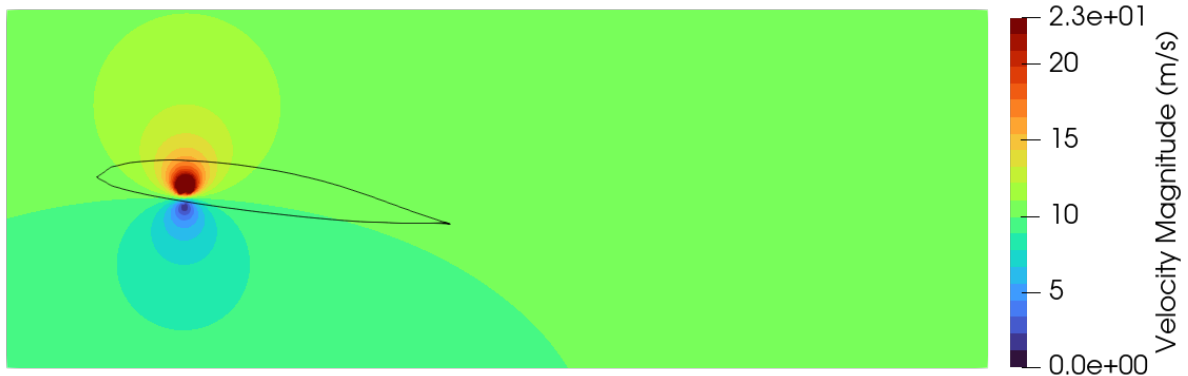


Figure 4.21: Velocity in m/s around the mid-span airfoil at 10 m/s freestream and at 7.5° .

Figs. 4.20 and 4.21 show the flow field around the wing with a splice applied in the center point of the wing. From these slices it is possible to see the doublet DUST used in the lifting line approach, as it is the only component present. Background velocity respects the free stream one. Even if these flow representations may not give many insights in the flow study, they serve as a baseline for the propeller interaction shown below. The disturbance in the otherwise uniform flow in Fig.4.20 is due to the wake of the wing not being outside the domain yet at the same time step of the other representation.

After the clear wing, further investigation of the flow was carried out for the isolated propeller, for both the fixed point thrust and the interaction of the propeller with the free stream flow. Accordingly, Fig. 4.22 reports the velocity flow field generated by a single propeller. The primary parameter that can be extracted from this representation is the magnitude of the induced velocity produced by the propeller. This quantity can be employed as a first-order approximation to estimate the magnitude of the difference between freestream and propeller wake velocities providing an information about the potential for propeller effects on the wing aerodynamics. In the region adjacent to the propeller, corresponding to the expected wing location, the induced velocity is approximately 16 m/s, thus in the operational range for MAVs of 10 to 15 m/s a beneficial effect from the propellers' blowing effect could be expected.

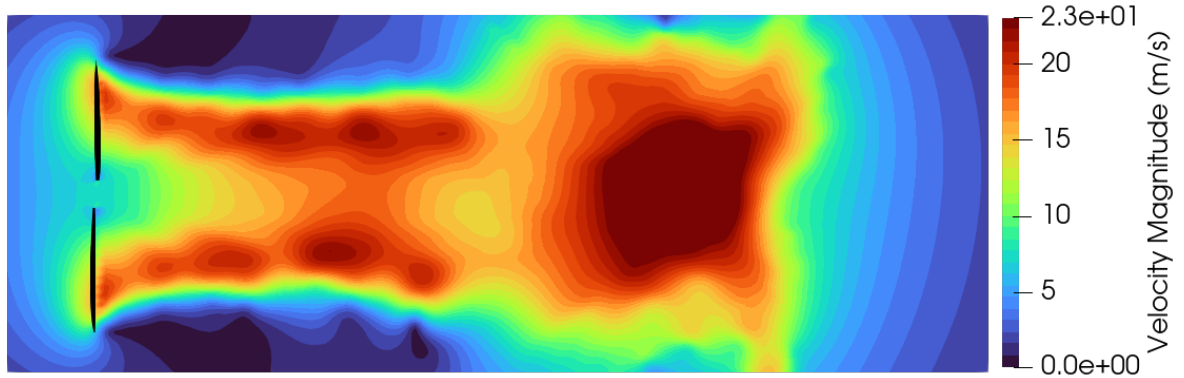
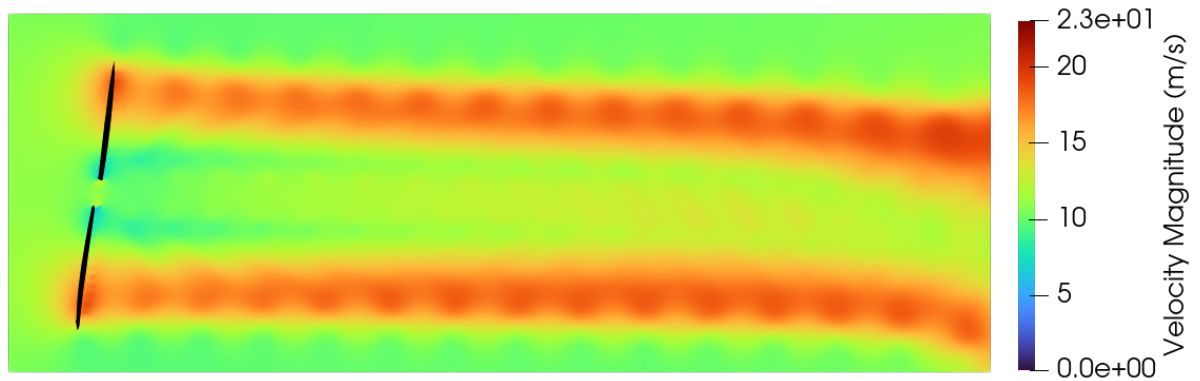
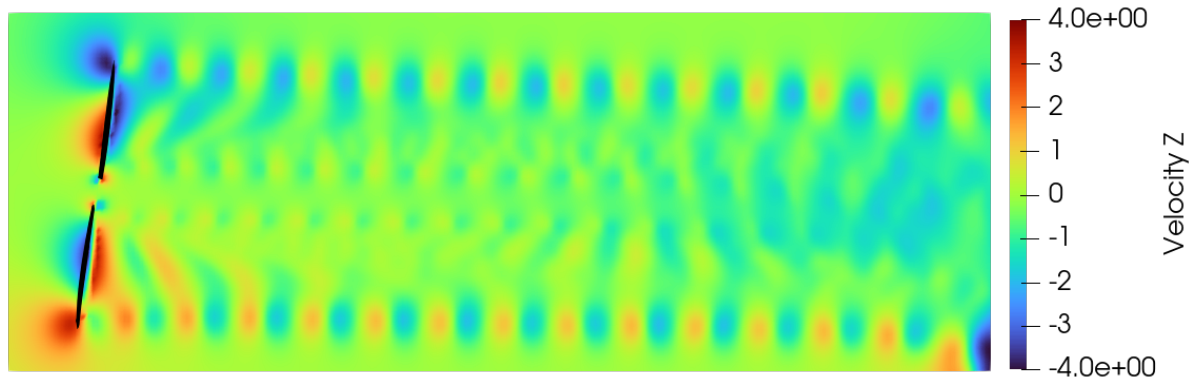


Figure 4.22: Axial velocity in m/s around the propeller in static conditions at 0° .



(a) Velocity magnitude.



(b) Velocity normal to the propeller's plane.

Figure 4.24: Velocity in m/s, total magnitude and normal to the propeller's plane at 10 m/s freestream and at 7.5° and $\text{RPM} = 60000$.

Figs. 4.24a and 4.23a show that the region closest to the propeller axis is only marginally affected by the blowing effect. However, in the present configuration, this region is occupied by the wing itself. Conversely, the region where the induced velocity reaches its

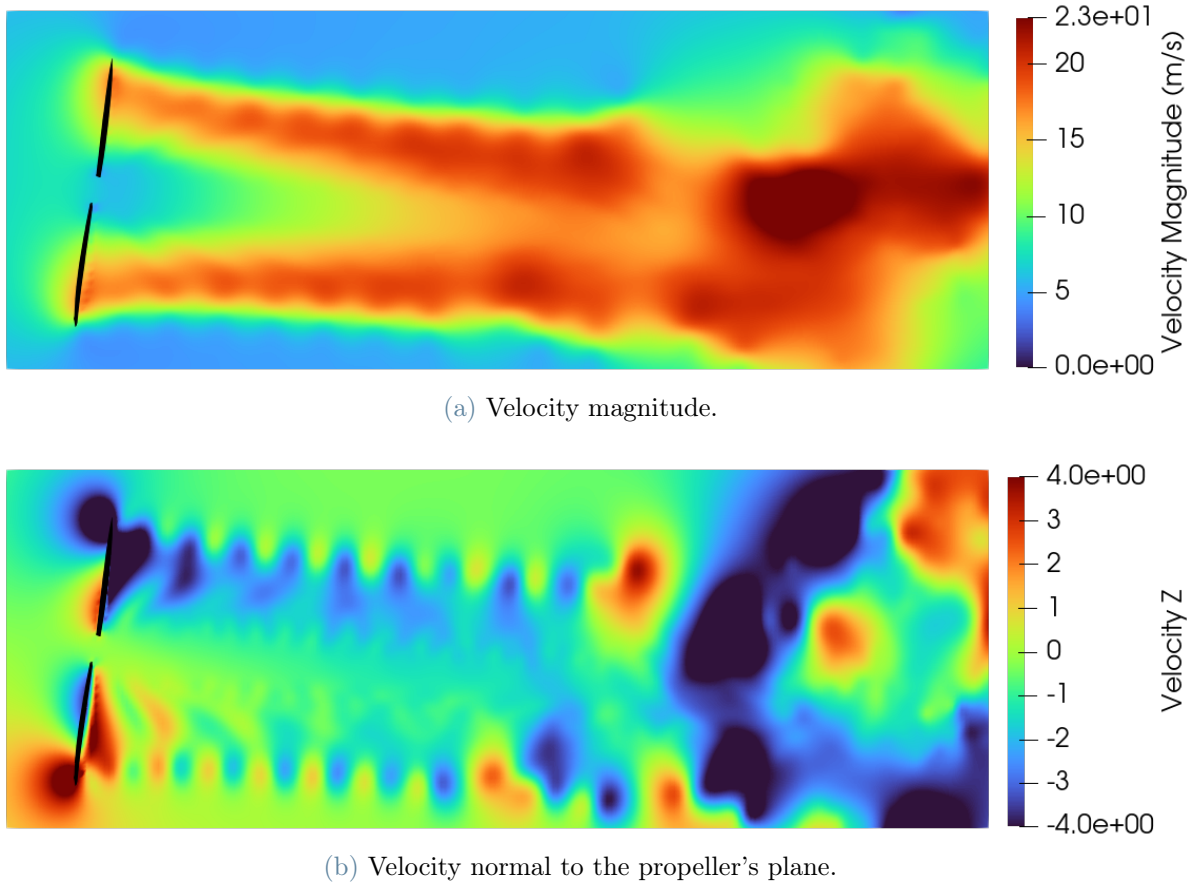


Figure 4.23: Velocity in m/s, total magnitude and normal to the propeller's plane at 5 m/s freestream and at 7.5° and $\text{RPM} = 60000$.

maximum coincides with the wing surface, thereby the blowing effect is expected to occur. Figs. 4.23b and 4.24b show the component of the velocity normal to the plane, and the clear influence of the rotating blade inside the flow, as the upper and lower sides of the wake reach opposite values throughout the field. For both velocity components represented, the effects at 5 m/s free stream are more intense, but extend less downstream with respect to the higher velocity.

The interaction between the propellers was further analyzed, in absence of the wing, by looking at the flow field in the wing plane, presented in Fig. 4.25. The propeller wakes were observed to remain distinct throughout their development; however, mutual interaction effects were evident. In particular, the inward curvature of the wakes generated by the external propellers and the diverging tendency of the central wake indicate aerodynamic coupling between the slipstreams.

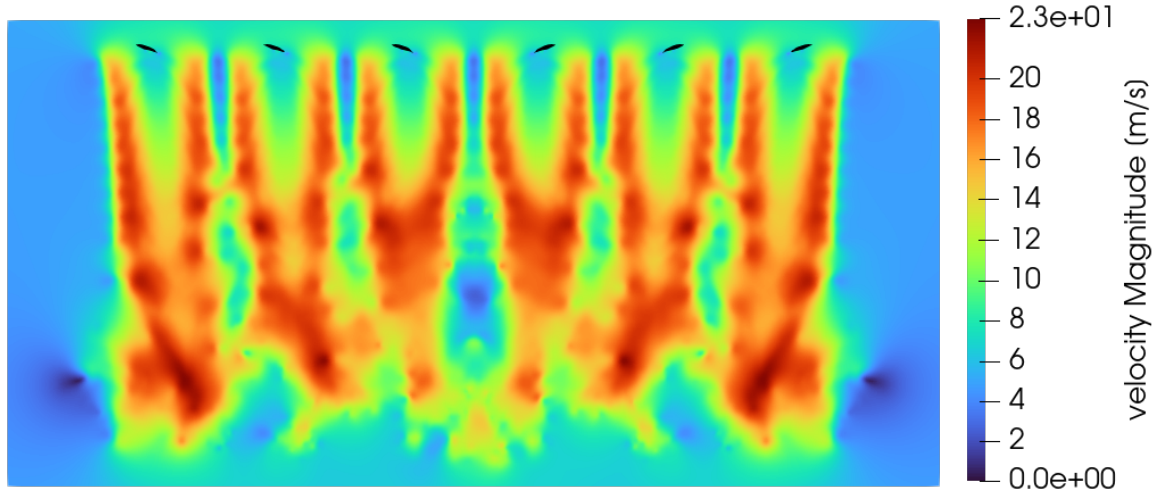


Figure 4.25: Top view of velocity flow field around the six propellers in 5 m/s at 7.5° and $\text{RPM} = 60000$.

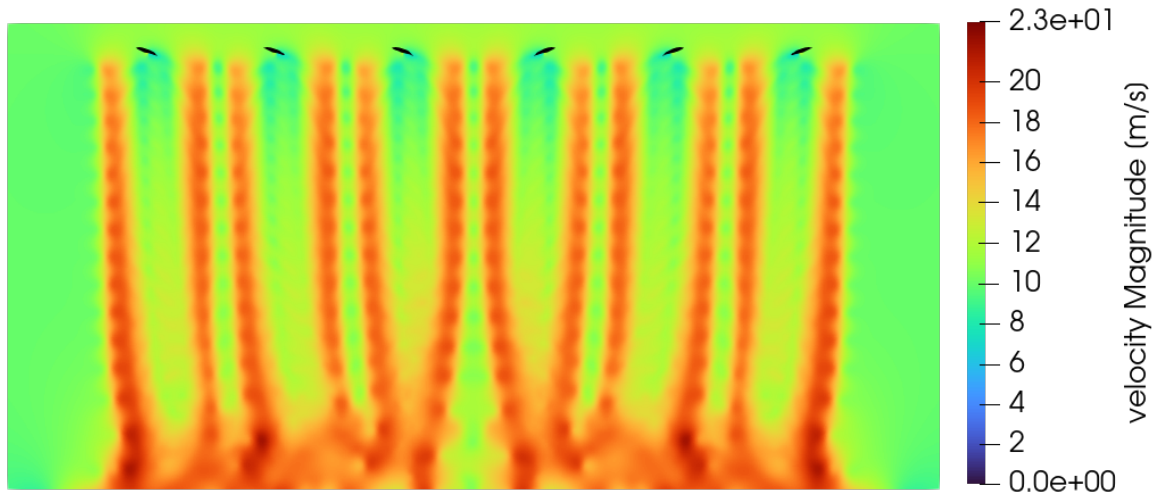
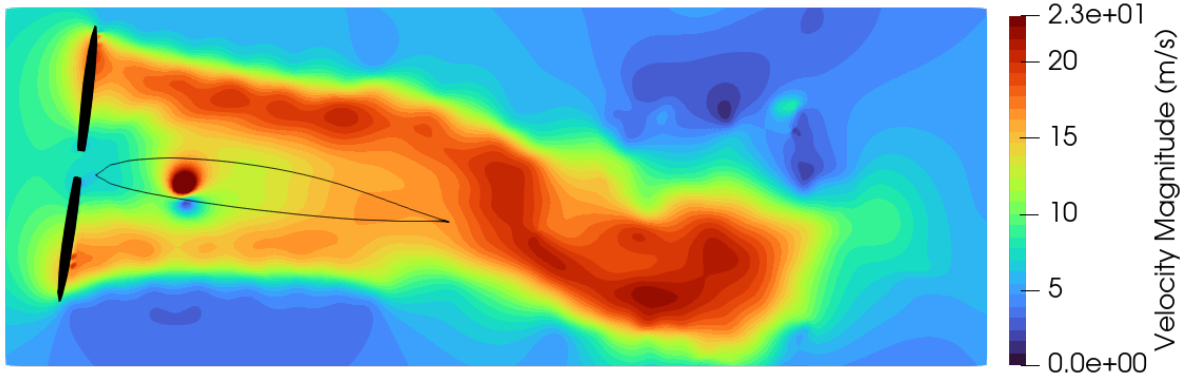


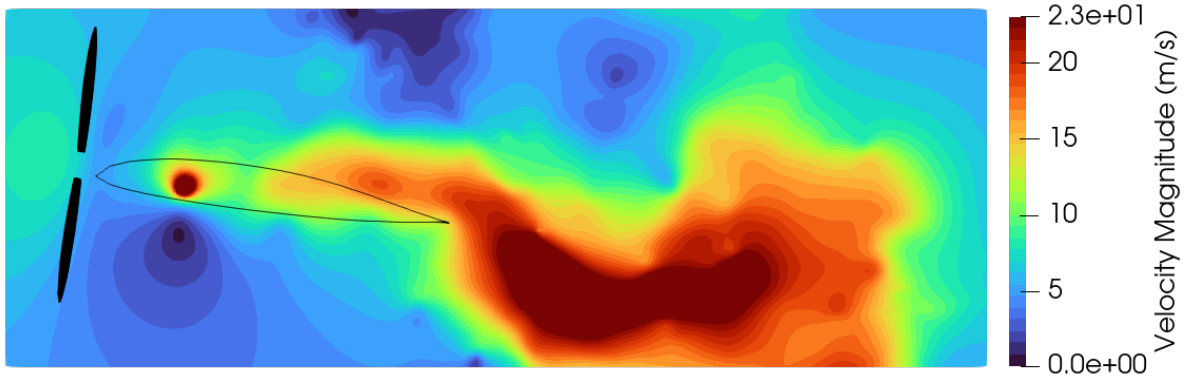
Figure 4.26: Top view of velocity flow field around the six propellers in 10 m/s at 7.5° and $\text{RPM} = 60000$.

At the lowest freestream velocity, the blowing effect generated by the propellers is more intense, resulting in a higher induced velocity in the region corresponding to the wing location. Conversely, at 10 m/s, the blowing effect extends further downstream, although with reduced peak intensity. In both cases, the phenomenon remains relevant to the present study, since the wing chord is smaller than the region influenced by the propeller slipstream. Furthermore, the 5 m/s case shows an increase in non stationary behavior in the region of the wing, which may affect negatively the global performances of the model.

From the Full Model simulation results it was possible to extrapolate the flow fields for both airspeeds considered. Such results are presented in Figs. 4.23 and 4.24.



(a) Propeller axis.



(b) Wing's centerline.

Figure 4.27: Velocity magnitude in m/s, through a propeller axis and at the wing's centerline at 5 m/s freestream and at 7.5° and $\text{RPM} = 60000$.

The figures above illustrate the propeller blowing effect in two distinct planes: one passing through the propeller axis and the other through the midspan of the wing. As expected, the blowing effect is significantly more pronounced in the plane aligned with the propeller axis. In contrast, in the plane located at the wing center, the interaction between the propeller slipstream and the wing wake gives rise to a region characterized by non-stationary flow behavior. In particular, at a freestream velocity of 10 m/s, the interaction region is located well downstream of the wing. Conversely, at the lower freestream velocity, this region directly impinges on the trailing edge of the wing, leading to an increase in drag, as will be shown below. Furthermore, the comparison highlights once again the differences in both intensity and spatial extent of the blowing effect across the investigated Reynolds number regimes.

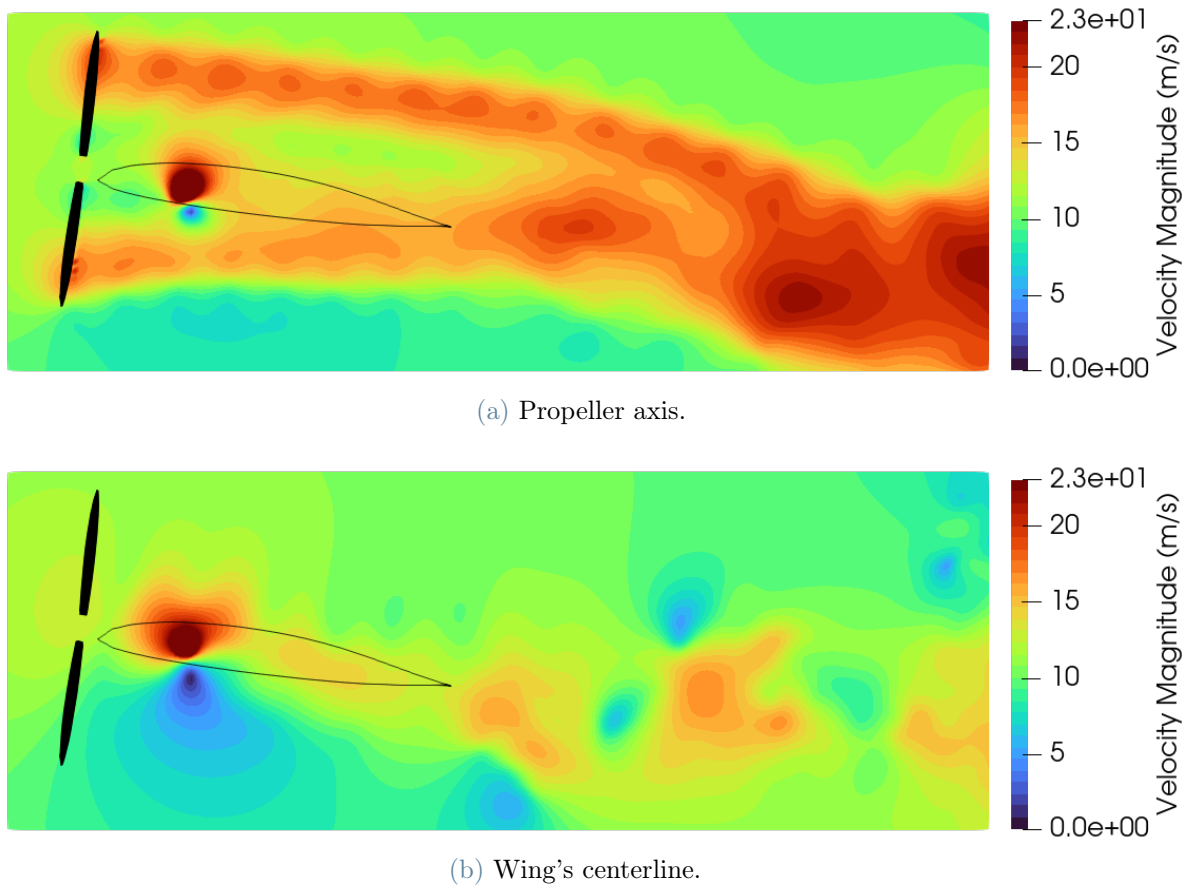
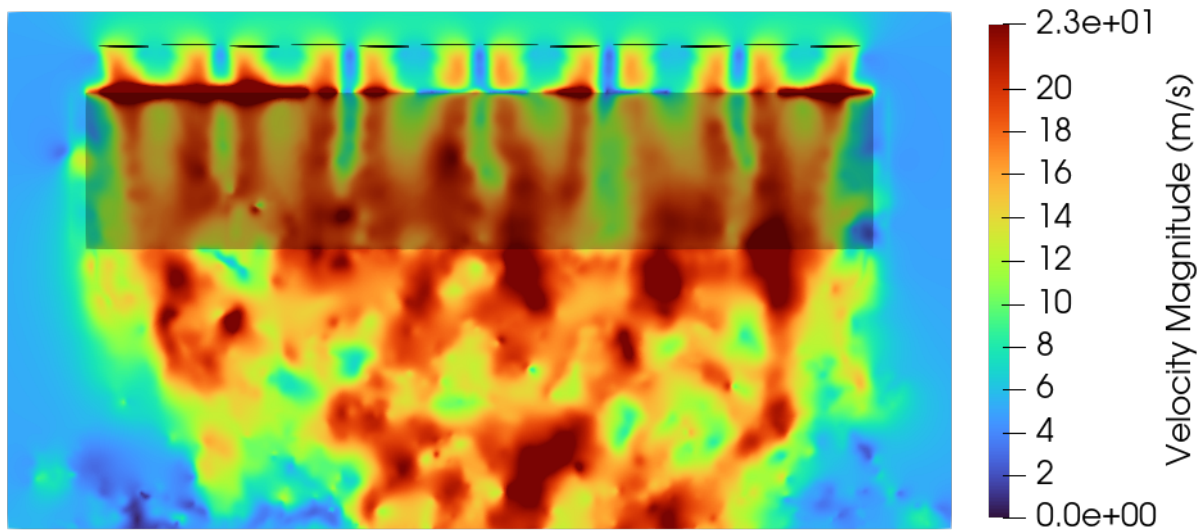
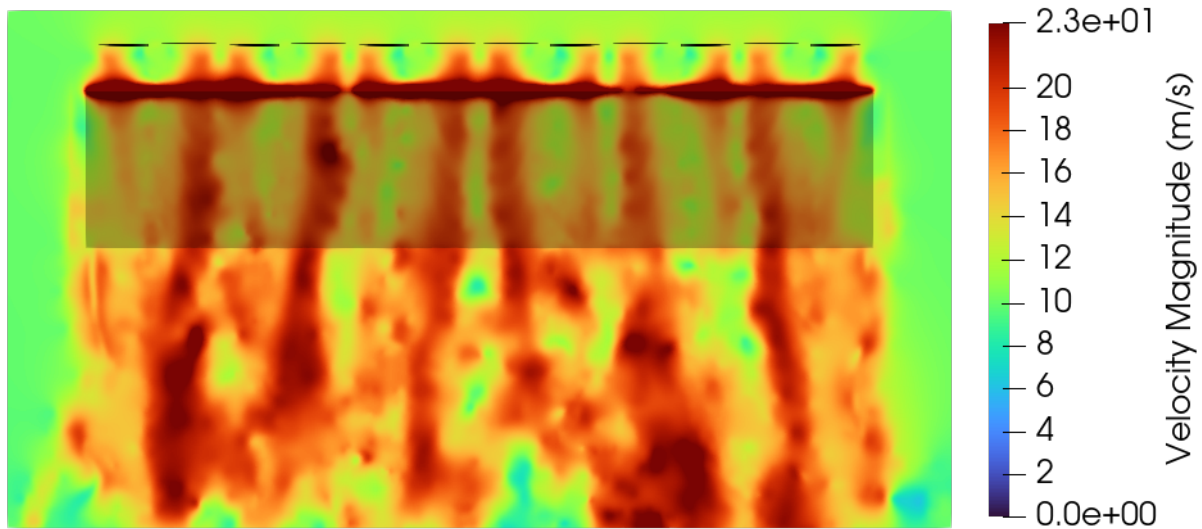


Figure 4.28: Velocity in m/s, through a propeller axis and at the wing's centerline at 10 m/s freestream and at 7.5° and $\text{RPM} = 60000$.

Fig. 4.29 reports the flow field of the full model in the plane of the wing. The effect of the wing downstream of the propellers can clearly be observed. With respect to Figs. 4.25 and 4.26, the propeller slipstream's region of influence remain unchanged in extension, but the presence of the wing alter the flow, causing higher non stationary behavior. However, as expected and consistent with the principles of DP, the influence of the propellers extends over the entire wing span, thereby confirming the physical consistency and relevance of the simulation. The left-to-right asymmetry is attributed to numerical oscillations in the solution, which propagated differently across the two halves of the computational domain. Nevertheless, the averaged total loads are consistent with the corresponding experimental measurements, thereby supporting the validity of the overall flow field predictions.



(a) 5 m/s freestream.



(b) 10 m/s freestream.

Figure 4.29: Velocity flow field normal to propellers' plane at 7.5° and 60000 RPM.

From the three dimensional flow field it was possible to compute the gradients and then the Q-criterion, from which it was possible to infer a representation of the vortical structures down stream the propellers and the wing. Such results are presented in Figs. 4.30 and 4.31. The difference in the intensity and influence area of the vortices generated by the propellers is shown, and in the 5 m/s case the influence is stopped clearly before the end of the wing, contributing to the increase in drag explained below.

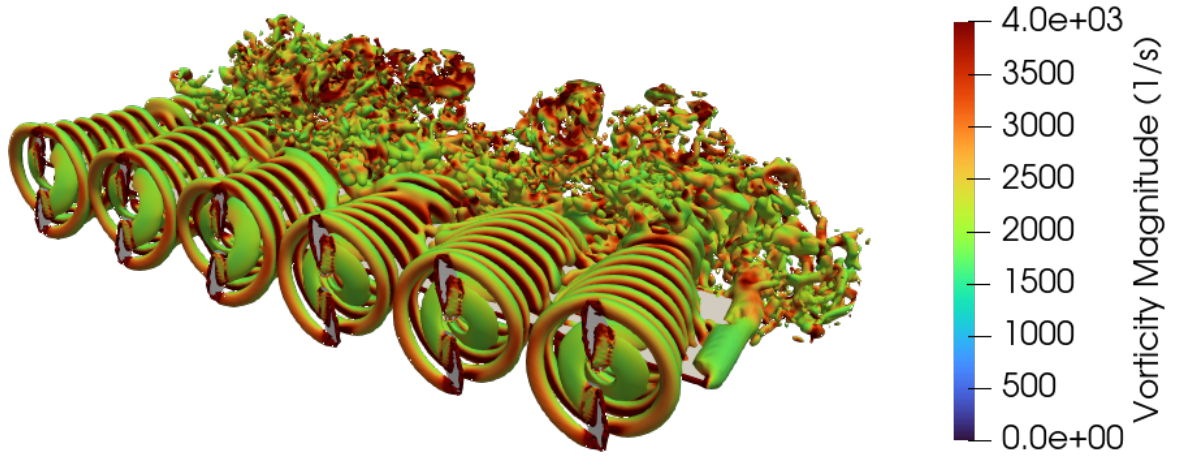


Figure 4.30: Turbulence representation using Q-criterion at 5 m/s freestream velocity, 60000 RPM and 7.5°AOA.

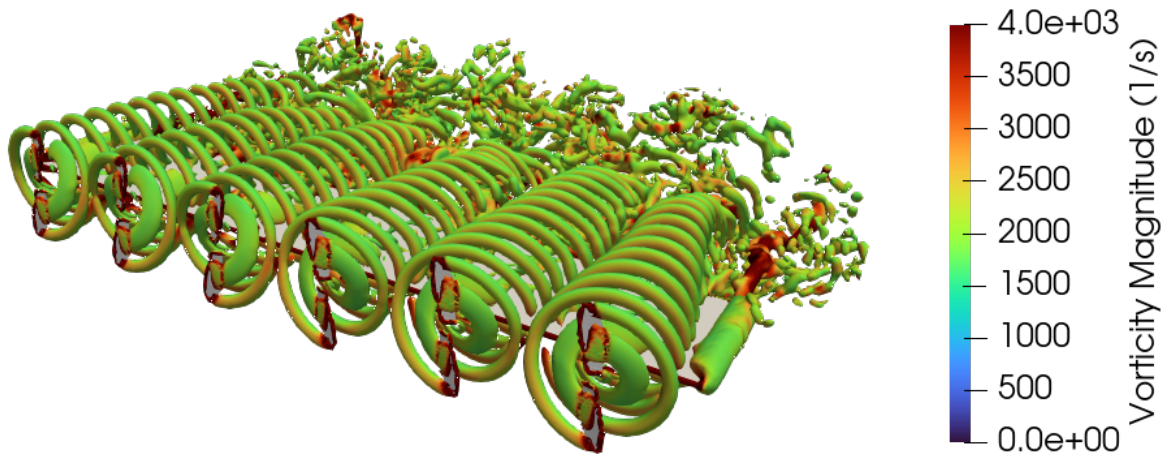


Figure 4.31: Turbulence representation using Q-criterion at 10 m/s freestream velocity, 60000 RPM and 7.5°AOA.

From the flow field analysis carried out in this section, results were obtained that are consistent with observations previously reported for full scale configurations. In particular, Savino et al. [55] showed that at advance ratios $J=0.5$ and $J=0.75$, a similar interaction mechanism between the propeller tip vortices and the wing wake occurs at higher Reynolds numbers. Specifically, the same increase in vorticity intensity was also observed as the free stream velocity decreased, and a greater improvement in wing performance was observed for the lowest advance ratio, corresponding to the lowest free-stream velocity, where the propeller operates more effectively.

It is therefore remarkable that phenomena originally studied on full scale models, op-

erating at higher Reynolds numbers and higher advance ratios, are also observed in the present case. In this work, the Reynolds number is orders of magnitude lower, and the advance ratio is approximately one third of the values previously investigated. Despite these substantial differences in operating conditions and scale, the same underlying interaction between the propeller wake and the wing flow field emerges, confirming the robustness of the physical mechanism.

Another fundamental aspect of the simulations of the full model, was the possibility to separate the different contributions of the different components toward the global loads. The results of the test are presented in Table 4.5, and the reported values were obtained by averaging the computed results over the final iterations, as the inherent unsteadiness of the problem causes the quantities to oscillate about a mean value at each time step.

Velocity	Component	F_y	C_L	%	F_x	C_D	%
5 m/s	Wing	0.2182	1.7812	73.4%	0.1380	1.1265	32.9%
	Propellers	0.0790	0.6449	26.6%	-0.5569	-4.5461	-132.9%
	Full Model	0.2972	2.4261	100%	-0.4189	-3.4196	-100%
10 m/s	Wing	0.5743	1.1720	91%	0.1270	0.2592	49.1%
	Propellers	0.0569	0.1161	9%	-0.3857	-0.7871	-149.1%
	Full Model	0.6312	1.2881	100%	-0.2587	-0.5280	100%

Table 4.5: Force coefficients in wind axes, divided by component, for the two freestream velocities analyzed.

Table 4.5 shows that the direct contribution of the propellers to the global lift was significantly lower than the contribution due to the wing.

At the freestream velocities of 10 m/s and 5 m/s, the blades generate more thrust force than the wing generates drag, thus the total force contribution in the x-axis (wind reference) will result in a negative value. The thrust produced in both cases in the full model is comparable to the one produced by the propeller alone expressed in Table 4.3, thus the effect of the wing on the upstream propeller is minimal.

Reducing the freestream velocity from 10 m/s to 5 m/s resulted in an increase in thrust generated by the propellers, from 0.399 N to 0.562 N. This variation primarily affects the longitudinal force, with only a minor influence on the vertical direction. The observed difference in percentage load distribution can therefore be attributed to this effect: while the propeller contribution varies only slightly, the wing performance changes significantly

between the two velocities, leading to a redistribution of the overall lift budget. The blowing effect is more pronounced in the 5 m/s configuration, as evidenced by the adimensional aerodynamic coefficients, which attain higher values in this case due to the lower dynamic pressure. Furthermore, at a freestream velocity of 5 m/s, the propellers generate a higher induced velocity compared to the 10 m/s case, as shown in Fig. 4.25 and Fig. 4.29. As a result, the wing experiences a higher drag force under the lower-speed condition. The same trend is not observed for the lift force, since the increased non-stationary behaviors and momentum addition produced by the propeller slipstream limit the improvement of the aerodynamic performance.

The most important aspect of this set of simulations, and the main objective of this work, is the comparison between the wing performance with and without the propellers' wake. The numerical approach allowed in the wing-propeller complete model the separation of the contribution of the wing and the propellers, thus the difference in aerodynamic performance with the wing without propeller was found. The results are reported graphically in Fig. 4.32 and the numerical comparison of the wing performance with and without the contribution of the propellers is reported in Table 4.6.

Velocity	Configuration	C_L	$\Delta C_L\%$	C_D	$\Delta C_D\%$
10 m/s	Clear Wing	0.5509	+112.7%	0.1830	+41.6%
	Blown wing	1.1720		0.2592	
5 m/s	Clear Wing	0.4420	+303%	0.1438	+683.4%
	Blown wing	1.7812		1.1265	

Table 4.6: Comparison of wing performance between the Clear Wing and the Full model wing component, for the two freestream velocities analyzed.

It can be clearly observed that DEP provides a substantial improvement in wing lift coefficient with an increase of a factor of approximately four at a freestream of 5 m/s and two at 10 m/s, at the same propeller's rotational speed. The difference can be attributed to the relative influence of the propeller's wake effect which is significantly greater at 5 m/s.

It was shown that the vorticity at high RPM regime at 5 m/s freestream velocity increased more the drag than the lift, with detrimental effect on the lift-to-drag ratio. It is thus of great importance a future study on the effect of propeller rotational speed coupled with freestream velocity for the best lift improvement with the minimum drag penalty, a situation potentially applicable to take off. The conditions with increase in lift and

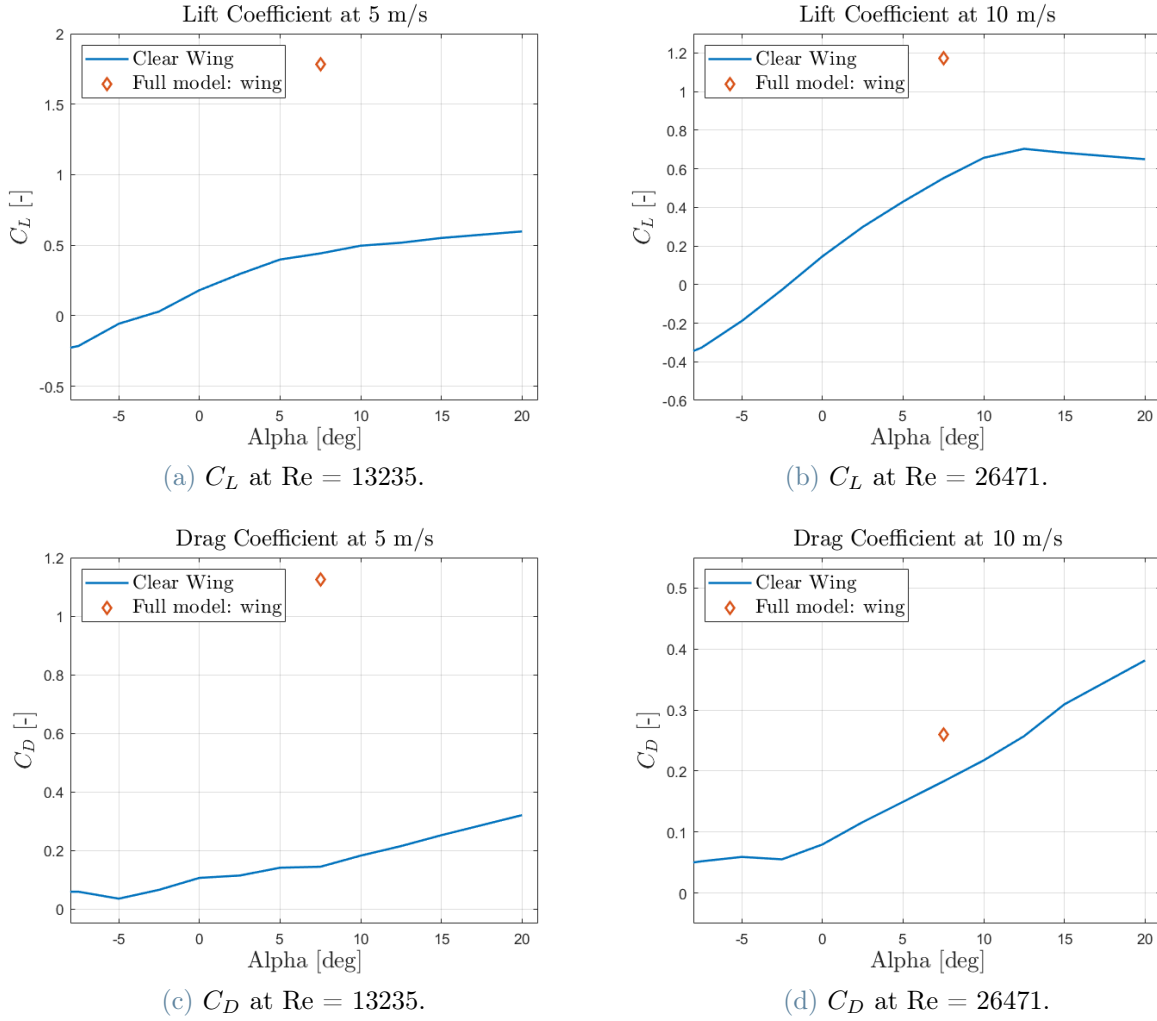


Figure 4.32: Comparison of lift and drag coefficients between the Clear wing and the wing component of the Full model.

significant increase in drag could be relevant for landing performance and potential reduce or elimination of flaps.

Because of the excellent agreement between the numerical and experimental results (showed in Fig. 4.19), it is reasonable to assume that the same percentage distribution of loads between the wing and the propeller was maintained also in the real model, allowing the improvements of the wing performance described. Moreover, since the thrust was found to be only marginally affected by variations in angle of attack and by the presence of the wing downstream, the conclusions presented in this analysis can reasonably be extended to the complete set of polar curves presented in Sec. 4.3.

In conclusion, the application of DUST to low Reynolds number flows, while also requiring very small time steps for propeller resolution, leads to numerical oscillations in the wake,

thereby reducing the reliability of the results. In the absence of experimental data for validation, the credibility of the numerical solution remains uncertain. Therefore, in its current state, DUST should be regarded primarily as an analysis tool capable of providing qualitative insight into flow phenomena and the underlying physics, once it has been validated against experiments. However, it cannot yet be considered a fully predictive tool for this class of problems. Despite this problem, DUST still proved to be able to give useful insights in flow phenomena once its results were validated against experiment. New capabilities are already being developed to allow differentiated time steps between components to mitigate numerical instabilities.

5 | Conclusions and Future Work

In this project, a novel experimental setup for powered Micro Aerial Vehicles, which included a newly acquired micro six-axes sting balance, was developed at the Politecnico di Milano's small academic wind tunnel. Because of time constraints the balance was re-calibrated on only three out of six axes, without detrimental effects on the planned testing campaign. A full scale model of an MAV's wing with six motor-propeller units was designed from scratch and manufactured to study the effects of distributed electric propulsion on the wing. A custom test rig was also designed to allow the testing of the model, with particular attention to wire routing to avoid interference with the measurements. The test rig, completed of the model was validated for accuracy and feasibility of measures at Re between 13 to 40 thousand. The aerodynamic coefficients obtained showed reliable behavior across different Reynolds.

Part of the project consisted in the development of a virtual twin of the wind tunnel model with the objective to validate the numerical simulation analysis. The wing was modeled using the experimental data, while the propeller used data extrapolated by NeuralFoil. The validation was organized in three steps: 1) Clear wing (without propellers), 2) Propeller only and finally 3) Full model, including the six propellers. For all three steps, the numerical analysis showed excellent agreement with experimental testing, proving the capability of DUST to accurately represent the interactional aerodynamics also at such low Reynolds regimes. The latter numerical configuration could allow the evaluation of the wing's lift decoupled from the propeller's thrust.

The combined experimental and numerical investigation demonstrated a significant increase in wing aerodynamic performance due to the application of Distributed Electric Propulsion.

The experimental results showed that, at zero angle of attack and for a propeller rotational speed of 60000 RPM, the blowing effect produced a substantial lift enhancement. In particular, at $Re=26471$, the wing lift coefficient C_L increased by 83.8%, while at $Re=13235$ the increase reached 162%. This lift augmentation was accompanied by an increase in the pitching moment in the nose-up direction.

The extended flow field analysis capabilities provided by DUST allowed a deeper understanding of the load contributions. At an angle of attack of 7.5° , it was observed that the majority of the lift is generated by the wing itself, rather than by the vertical component of the propeller thrust. However, the drag associated with the wing also increases under these conditions. At 7.5° angle of attack and 60000 RPM, the lift coefficient of the wing increased by 112.7% at $Re=26471$, and by 303% at $Re=13235$, confirming the strong effectiveness of the blowing effect at lower Reynolds numbers. This lift increase, however, occurred together with a drag rise: C_D increased by 41.3% at $Re=26471$ and by 680% at $Re=13235$. These results show that while the reduced advance ratio enhances lift through stronger slipstream-wing interaction, it also significantly penalizes the overall aerodynamic efficiency of the wing, particularly at the lowest Reynolds number investigated, an effect that could result in improved short landing capabilities.

Finally, the same behavior observed at higher Reynolds numbers and advance ratios was also identified in the present study, indicating that the fundamental flow mechanisms remain substantially unchanged across scales. Although the intensity of the phenomenon decreases at lower Reynolds numbers, the underlying physical principles governing the interaction are preserved.

This study represents the beginning of a new line of research and included the design and development of a novel testing capability. As such an investigation with a larger test-matrix with all main aspects of the subject was not possible.

Accordingly, future work should expand upon the knowledge developed, starting with a redesign of the wind tunnel model to address the issues identified during testing. In particular, the adoption of an aluminum model with integrated assembly and disassembly features would enable the replacement of damaged components. In addition, the 3D-printed support should be redesigned using a stiffer material; for instance, an iron bar similar to those used in the external support, could significantly improve the stiffness and stability of the setup.

Additional efforts should focus on performing rigorous calibration checks of the sting balance on all six load components, verifying and, if necessary, improving its accuracy. Attention should also be given to the mechanical characteristics of the electric motor wires crossing the balance, ensuring that their stiffness or routing does not influence the measurements.

Future experimental campaigns should include the execution of the full test matrix, comprising angle of attack sweeps, freestream velocity sweeps, and propeller rotational speed sweeps. Given the large number of required test runs, a Modern Design of Experiments

(MDOE) approach should be adopted to efficiently define the experimental matrix and optimize the investigation of the parameter space.

Since the augmentation of lift due to the propeller wake effect on the wing has been proved, future work in wind tunnel model could include a design with DEP for increased lift for take off and landing and two propellers at the wing tips for cruise, in order to study also wing tip vortices interaction at low Reynolds numbers.

Regarding the software limitations, new developments in the DUST solver are already being carried out to implement a differential time step for the wing and the propeller. Such implementation would guarantee that the wake particles generated by the wing were released at a slower rate, diminishing the possibility of numerical instabilities arising in the flow field, and globally increase the reliability of the numerical predictions in such low Re, high RPM test cases.

Bibliography

- [1] Martin Waszak, John Davidson, and Peter Ifju. *Simulation and Flight Control of an Aeroelastic Fixed Wing Micro Aerial Vehicle*. 2002.
- [2] Justin Winslow, Hikaru Otsuka, Bharath Govindarajan, and Inderjit Chopra. Basic understanding of airfoil characteristics at low reynolds numbers (104–105). *Journal of Aircraft*, 55:1–12, 12 2017.
- [3] Thomas J. Mueller and James D. DeLaurier. Aerodynamics of small vehicles. *Annual Review of Fluid Mechanics*, 35(Volume 35, 2003):89–111, 2003.
- [4] F. W. Schmitz. Aerodynamics of the model airfoil part i. Technical Report RSIC-721, Redstone Scientific Information Center, November 1967. Translated from German, Airfoil Measurements.
- [5] Gabriel Torres. Aerodynamics of low aspect ratio wings at low reynolds numbers with applications to micro air vehicle design /. 09 2002.
- [6] Joel Grasmeyer and Matthew Keennon. *Development of the Black Widow Micro Air Vehicle*. 2001.
- [7] Michael A. Dornheim. Tiny drones may be soldier’s new tool. *Aviation Week & Space Technology*, (1):42, 6 1998.
- [8] Roberto Albertani, Parvez Khambatta, Adam Hart University of Florida, Lawrence Ukeiley, Matias Oyarzun, Louis Cattafesta, and Gregg Abate. *Validation of a Low Reynolds Number Aerodynamic Characterization Facility*. 2009.
- [9] R. Albertani, Bret Stanford, James Hubner, and P. Ifju. Aerodynamic coefficients and deformation measurements on flexible micro air vehicle wings. *Experimental Mechanics*, 47:625–635, 10 2007.
- [10] Roberto Albertani, Richard DeLoach, Bret Stanford, James Hubner, and Peter Ifju. Wind tunnel data base development and nonlinear modeling applied to powered micro air vehicles with flexible wing. 08 2006.

- [11] Mattia Alioli, Pierangelo Masarati, Marco Morandini, Trenton Carpenter, N. Brent Osterberg, and Roberto Albertani. Membrane shape and load reconstruction from measurements using inverse finite element analysis. *AIAA Journal*, 55(1):297–308, 2017.
- [12] V L Okulov, J N Sørensen, and W Z Shen. Extension of goldstein’s circulation function for optimal rotors with hub. *Journal of Physics: Conference Series*, 753(2):022018, sep 2016.
- [13] B. Chandrasekaran. *Method for the Prediction of the Installation Aerodynamics of a Propfan at Subsonic Speeds: User Manual*. National Aeronautics and Space Administration, 1986.
- [14] Antony Jameson. The analysis of propeller-wing flow interaction. 02 1970.
- [15] Quentin R. Wald. The aerodynamics of propellers. *Progress in Aerospace Sciences*, 42(2):85–128, 2006.
- [16] S. J. Miley and E. von Lavante. Propeller propulsion system integration: State of technology survey. 03 1985.
- [17] Alex Zanotti, Luca Menini, Alberto Savino, Donato Grassi, and Luca Riccobene. Experimental investigation of wing-propeller aerodynamic interaction in evtol configurations. *Aerospace Science and Technology*, 152:109348, 2024.
- [18] Wei Shyy, Yongsheng Lian, Jian Tang, Dragos Viieru, and Hao Liu. *Aerodynamics of Low Reynolds Number Flyers*. 01 2008.
- [19] Aaron T. Perry, Phillip J. Ansell, and Michael F. Kerho. Aero-propulsive and propulsor cross-coupling effects on a distributed propulsion system. *Journal of Aircraft*, 55(6):2414–2426, 2018.
- [20] Richard E. Kuhn and Jr Hayes, William C. Wind-tunnel investigation of effect of propeller slipstreams on aerodynamic characteristics of a wing equipped with a 50-percent-chord sliding flap and a 30-percent-chord slotted flap. 1957.
- [21] J. Stuper. Effect of propeller slipstream on wing and tail. 1938.
- [22] Gavin Kumar Ananda Krishnan, Robert W. Deters, and Michael S. Selig. *Propeller Induced Flow Effects on Wings at Low Reynolds Numbers*. 2013.
- [23] Hyun D. Kim, Aaron T. Perry, and Phillip J. Ansell. A review of distributed electric propulsion concepts for air vehicle technology. In *2018 AIAA/IEEE Electric Aircraft Technologies Symposium (EATS)*, pages 1–21, 2018.

- [24] Nicholas K. Borer, Michael D. Patterson, Jeffrey K. Viken, Mark D. Moore, JoeBen Bevirt, Alex M. Stoll, and Andrew R. Gibson. *Design and Performance of the NASA SCEPTOR Distributed Electric Propulsion Flight Demonstrator*. 2016.
- [25] Alex M. Stoll, JoeBen Bevirt, Mark D. Moore, William J. Fredericks, and Nicholas K. Borer. *Drag Reduction Through Distributed Electric Propulsion*. 2014.
- [26] Alex M. Stoll. *Comparison of CFD and Experimental Results of the LEAPTech Distributed Electric Propulsion Blown Wing*. 2015.
- [27] Christian B. Hoover, Jinwei Shen, Andrew R. Kreshock, Bret Stanford, David J. Piatak, and Jennifer Heeg. *Whirl Flutter Stability and Its Influence on the Design of the Distributed Electric Propeller Aircraft X-57*, pages 1–14. 2017.
- [28] N. Osterberg and R. Albertani. Investigation of self-deploying high-lift effectors applied to membrane wings. *The Aeronautical Journal*, 121(1239):660–679, 2017.
- [29] Eric Schroeder and James Baeder. *Using Computational Fluid Dynamics for Micro-Air Vehicle Airfoil Validation and Prediction*. 2005.
- [30] Estela Barroso Barderas, Rafael Bardera Mora, Ángel Antonio Rodríguez-Sevillano, and Juan Carlos Matías García. Numerical analysis of bioinspired tails in a fixed-wing micro air vehicle. *Aerospace*, 10(9), 2023.
- [31] Sungjin Choi and Jon Ahn. *A Computational Study on the Aerodynamic Influence of a Pusher Propeller on a MAV*. 2010.
- [32] Zhaolin Chen and Fan Yang. Propeller slipstream effect on aerodynamic characteristics of micro air vehicle at low reynolds number. *Applied Sciences*, 12(8), 2022.
- [33] Andy P. Broeren, Philippe Giguère, and Michael S. Selig. Summary of low-speed airfoil data - vol. 3. 1996.
- [34] Cheerson. Cheerson cx-10 nano drone 2.4 ghz. https://www.cheerson.it/index.php?id_product=2&controller=product, 2023.
- [35] Mirko Tresoldi. Infrared thermography setup for boundary-layer transition controlled by dbd plasma actuator. Master’s thesis, Politecnico di Milano, 2023. Tesi Magistrale in Ingegneria Aeronautica.
- [36] Parker Hannifin Corporation. Ps/px gen2 gearhead econfigurator. https://webassistants.partcommunity.com/23d-libs/us_configurator/parker/v1/index.html?info=parkeremn/gearheads_and_gearmotors/ps_px_asmtab.prj&varset=%7B SERIES=PS/PX%20Series%7D,%7BC1=PX%7D,%7BC2=60-%7D,

%7BC3=040%7D,%7BC4=-S2%7D,%7BC5=+MU60-033%7D&email=sample@parker.com&ManufacturerPartID=PX60-040-S2+MU60-033;Make:-Model:&CPN=.

- [37] KERT Srl. Alimentatore stabilizzato kat10vd. <https://www.kert.it/soluzioni/alimentatori-acdc/kat10vd/>.
- [38] J. B. Barlow, W. H. Rae, and A. Pope. *Low-Speed Wind Tunnel Testing*. New York: John Wiley & Sons, 1999.
- [39] R. C. Pankhurst and D. W. Holder. *Wind-Tunnel Technique*. Sir Isaac Pitman & Sons, Ltd, London, 1952.
- [40] H. Glauert. *The Elements of Aerofoil and Airscrew Theory*. Cambridge Science Classics. Cambridge University Press, 1983.
- [41] Richard S. Figliola and Donald E. Beasley. *Theory and design for mechanical measurements, Fifth edition*. John Wiley & Sons, Inc, 01 2011.
- [42] G.E.P. Box, W.G. Hunter, and J.S. Hunter. *Statistics for Experimenters: An Introduction to Design, Data Analysis, and Model Building*. Wiley Series in Probability and Statistics. John Wiley & Sons, Inc, 1978.
- [43] A. Azuma. *The Biokinetics of Flying and Swimming*. AIAA education series. American Institute of Aeronautics and Astronautics, Incorporated, 2006.
- [44] Matteo Tugnoli, Davide Montagnani, Monica Syal, Giovanni Droandi, and Alex Zantotti. Mid-fidelity approach to aerodynamic simulations of unconventional vtol aircraft configurations. *Aerospace Science and Technology*, 115:106804, 2021.
- [45] Luigi Morino. Helmholtz decomposition revisited: Vorticity generation and trailing edge condition. *Computational Mechanics*, 1:65–90, 1986.
- [46] Luigi Morino and Ching-Chiang Kuo. Subsonic potential aerodynamics for complex configurations : A general theory. *AIAA Journal*, 12(2):191–197, 1974.
- [47] Thomas D. Economon, Francisco Palacios, Sean R. Copeland, Trent W. Lukaczyk, and Juan J. Alonso. Su2: An open-source suite for multiphysics simulation and design. *AIAA Journal*, 54(3):828–846, 2016.
- [48] Mark Drela. Xfoil: An analysis and design system for low reynolds number airfoils. volume 54, 06 1989.
- [49] Peter Sharpe. NeuralFoil: An airfoil aerodynamics analysis tool using physics-informed machine learning. <https://github.com/peterdsharpe/NeuralFoil>, 2023.

- [50] Peter D. Sharpe. *Accelerating Practical Engineering Design Optimization with Computational Graph Transformations*. PhD thesis, Massachusetts Institute of Technology, 2024.
- [51] Alain Pelletier and Thomas Mueller. Low reynolds number aerodynamics of low-aspect-ratio, thin/flat/cambered-plate wings. *Journal of Aircraft - J AIRCRAFT*, 37:825–832, 09 2000.
- [52] Fluid Mechanics 101. Fluid mechanics 101. <https://www.fluidmechanics101.com/>, 2021.
- [53] Bram van Leer. Towards the ultimate conservative difference scheme. ii. a second-order godunov scheme. *Journal of Computational Physics*, 32:101–136, 1979.
- [54] Larry Viterna and R. Corrigan. Fixed pitch rotor performance of large horizontal axis wind turbines. *Large Horizontal-Axis Wind Turbines*, -1:69–85, 01 1982.
- [55] Alberto Savino and Alex Zanotti. Aerodynamic interaction between wing and propeller in evtol airplane mode flight condition. *CEAS Aeronautical Journal*, 06 2025.

List of Figures

1.1	Effect of Reynolds number on airfoil maximum sectional lift-to-drag ratio[2].	1
2.1	Top view, front view and right view of the model with dimensions in millimeters.	10
2.2	Cheerson CX-10 drone and motor.	10
2.3	Propeller shape.	11
2.4	Front view of the full model with rotation direction of the propellers.	11
2.5	Five components of the model for production.	12
2.6	Complete model of the wing with pseudo-fuselage and nacelles.	14
2.7	Lateral view sketch of convergent section and test chamber, with pressure transducer position.	15
2.8	Metal support.	16
2.9	3D printed support for the balance.	17
2.10	Model, balance and supports.	17
2.11	Parker BE231DQ-NPSN high power density servo motor.	18
2.12	Parker PX60-040-S2-MU60-033 gearhead.	18
2.13	Parker servo motor and gearhead.	19
2.14	KERT KAT10VD power supply.	22
2.15	Laser tachometer and its support for wind tunnel operation.	23
2.16	Wind tunnel model.	23
2.17	Reference system for the wind tunnel results. The signs of the moments follow the right-hand rule.	24
2.18	OSU-001 sting balance.	25
2.19	Calibration support.	27
2.20	Calibration configurations.	28
2.21	First calibration loads.	29
2.22	Forces and moments reference axes with respect to the sting balance.	29
2.23	Second calibration loads.	31
2.24	Validation of calibration procedure.	32
2.25	Solid and wake blockage parameters.	33

3.1	DUST workflow [44].	37
3.2	Computational domain used in the simulations.	41
3.3	Structured mesh near the airfoil.	42
3.4	Complete mesh structure with refinement near the airfoil.	43
3.5	Grid convergence of SU2.	44
3.6	Velocity flow field in m/s around the E231 airfoil, with streamlines. Con- ditions are $Re=23500$, $AOA=0^\circ$	46
3.7	Pressure coefficient and y^+ distribution expressed as fraction of the chord.	46
3.8	Two different discretization models for the wing.	48
3.9	Panel convergence for wing.	49
3.10	DUST results for the wing at 10 m/s.	51
3.11	Propeller discretization.	52
3.12	Panel convergence for propeller.	54
3.13	Time step convergence for propeller.	55
3.14	Nomenclature of the two components of the numerical model; the combi- nation of both is defined as <i>Full model</i>	55
4.1	Lift coefficient curves against angle of attack at different airspeed for the clear wing.	59
4.2	Lift coefficient at different freestream velocities.	59
4.3	Drag coefficient curves against angle of attack at different airspeed for the clear wing.	60
4.4	Drag coefficient at different freestream velocities.	60
4.5	Pitch moment curves against angle of attack at different airspeed for the clear wing.	61
4.6	Pitch moment coefficient at different freestream velocities.	61
4.7	Polar curves with coefficients at different airspeed for the clear wing.	62
4.8	Polar curves at different freestream velocities.	62
4.9	Lift to drag ratio and pitch moment coefficient versus C_L	63
4.10	Isolated wing comparison between DUST and experimental.	64
4.10	DUST results for the wing at different airspeeds.	65
4.11	Verification of the linear relation between voltage supply and propeller rotational velocity in RPM.	65
4.12	Multiple runs measuring propeller's static thrust versus propeller rotational velocity in RPM.	66
4.13	Rotational speed of propeller versus wind tunnel freestream velocity at constant input power of 7 V.	66

4.14	Comparison between numerical and experimental fixed point thrust.	67
4.15	Thrust coefficient, torque coefficient, efficiency.	68
4.16	Single propeller loads.	69
4.17	Multiple propeller loads.	70
4.18	Full model lift and drag coefficients.	71
4.19	Full model vs. clear wing.	72
4.19	Comparison of full model.	75
4.20	Velocity in m/s around the mid-span airfoil at 5 m/s freestream and at 7.5°.	75
4.21	Velocity in m/s around the mid-span airfoil at 10 m/s freestream and at 7.5°.	76
4.22	Axial velocity in m/s around the propeller in static conditions at 0°.	77
4.24	Propeller flow fields at 10 m/s, 60000 RPM.	77
4.23	Propeller flow fields at 5 m/s, 60000 RPM	78
4.25	Top view of velocity flow field around the six propellers in 5 m/s at 7.5° and RPM = 60000.	79
4.26	Top view of velocity flow field around the six propellers in 10 m/s at 7.5° and RPM = 60000.	79
4.27	Full model flow fields at 5 m/s, 60000 RPM.	80
4.28	Full model flow fields at 10 m/s, 60000 RPM.	81
4.29	Full model velocities flow field top view.	82
4.30	Turbulence representation using Q-criterion at 5 m/s freestream velocity, 60000 RPM and 7.5°AOA.	83
4.31	Turbulence representation using Q-criterion at 10 m/s freestream velocity, 60000 RPM and 7.5°AOA.	83
4.32	Comparison between full model and isolated wing.	86

List of Tables

2.1	Parameters of the Green Wind Tunnel.	15
2.2	Technical data of Parker PX60 gear head.	20
2.3	Tachometer main characteristics.	22
2.4	Sting Balance Specifications.	26
2.5	Calibration matrix of the sting balance.	30
2.6	Values of t for 95% confidence interval as a function of the number of degrees of freedom.	35
3.1	Comparison for lift and drag coefficient, with relative errors, between SU2 simulations and experimental results, for different angles of attack.	45
3.2	$\bar{\Omega}$ and relative advance ratios for the different freestream velocities with constant propeller speed of 60,000 RPM.	56
4.1	Wing's Re numbers based on the chord and dynamic pressure for the air- speed settings of the wind tunnel.	57
4.2	Propeller efficiency as a function of advance ratio and Reynolds number computed at 2/3 of the propeller radius, at 60000 RPM.	68
4.3	Force components generated by the propellers for the two velocities tested at 7.5°.	70
4.4	Comparison of wing performance between the Clear Wing and the Full model, for the two freestream velocities analyzed at 0° AOA.	74
4.5	Force coefficients in wind axes, divided by component, for the two freestream velocities analyzed.	84
4.6	Comparison of wing performance between the Clear Wing and the Full model wing component, for the two freestream velocities analyzed.	85

List of Symbols

Variable	Description	SI unit
AOA	angle of attack	deg
AR	aspect ratio	-
C_L	net stream normal force coefficient	-
C_D	net stream tangential force coefficient	-
C_m	pitching moment coefficient	-
C_Q	torque coefficient	-
C_T	thrust coefficient	-
J	propeller advance ratio	-
q	dynamic pressure	Pa
Re	Reynold number	-
RPM	revolution per minute	-
U_∞	free stream velocity	m/s
U_{corr}	corrected free stream velocity	m/s
ϵ_{sb}	solid blockage correction	-
ϵ_{wb}	wake blockage correction	-
ϵ_{prop}	propeller blockage correction	-
η	propeller efficiency	-
$\bar{\Omega}$	adimensional fan speed	-

Acknowledgments

I would like to sincerely thank Professor Alex Zanotti for his constant availability, support, and invaluable expertise throughout the development of this project. I am deeply indebted to Professor Roberto Albertani, whose knowledge, dedication, and generous commitment of time made this work possible. I would also like to extend my heartfelt thanks to Alberto Savino, Donato Grassi, and Luca Riccobene. Their expertise, guidance, and continued support were essential to the successful completion of this work. I am truly grateful for their insights, constructive feedback, and time. I sincerely appreciate the mentorship and encouragement of all the individuals mentioned above.

