



# Investigating wake reproduction of a model-scale wind turbine: experimental measurements versus Large Eddy Simulation with actuator line

Emmanuel Gillyns<sup>1,2,3</sup>, Sophia Buckingham<sup>2</sup>, Jeroen van Beeck<sup>1</sup>, and Grégoire Winckelmans<sup>3</sup>

<sup>1</sup>von Karman Institute for Fluid Dynamics, 1640 Sint-Genesius-Rode, Belgium

<sup>2</sup>ENGIE Laborelec, 1630 Linkebeek, Belgium.

<sup>3</sup>Institute of Mechanics, Materials and Civil Engineering (IMMC), Université catholique de Louvain (UCLouvain), 1348 Louvain-la-Neuve, Belgium

**Correspondence:** Emmanuel Gillyns (emmanuel.gillyns@engie.com)

**Abstract.** Accurate modeling of wind turbine wakes is essential for understanding turbine performance, wake interactions, and structural loading in wind energy applications. This work presents both numerical and experimental investigations aimed at evaluating the strengths and limitations of the Actuator Line Method (ALM) for wind turbine simulations in a controlled environment. Wind tunnel experiments were conducted using a purpose-built small-scale turbine TWIST (Turbine for Wind-tunnel Investigation & Scaled Testing) to provide high-resolution measurements of near wake velocity and blade deflection. In parallel, a numerical study was carried out using an ALM model developed with a force distribution designed to avoid averaging along the blade length in order to better capture variations in the near tip and hub regions. The inflow conditions in the simulations were carefully matched to the experimental profiles of mean velocity and turbulence intensity, ensuring that discrepancies would be mostly attributed to the modeling rather than to differences in inflow representation. Comparative analyzes between simulations and experiments were performed to assess the model's ability to capture key near-wake features. Additionally, the influence of blade pitch angle on wake development was explored numerically. The blade deflection was analyzed both numerically and experimentally to evaluate the coupling between aerodynamic loading and structural response. The results show that the ALM, combined with distributing the hub drag over a Gaussian ellipsoid, is able to reproduce the main characteristics of the wake at 1.4 rotor diameter downstream, including the velocity deficit and the sharp transition between the wake and the free stream. Wake results at 4.35 diameter are also presented and are in line with the experiment, except very near the ground. Overall, the combined experimental and numerical approach provides valuable insight into the predictive capability of the ALM and its applicability for wind turbine modeling in controlled conditions.

## 1 Introduction

Accurate modeling of wind turbine wakes is essential for predicting power losses, structural loading, and wake interactions within and between wind farms. These wakes can also affect nearby structures, making reliable wake characterization important for both design and operational decisions. Large-eddy simulation (LES) has become a valuable tool for studying these flows, as it resolves the dominant turbulent structures that govern wake evolution and recovery. However, resolving full turbine



geometries within LES remains computationally prohibitive, motivating the use of efficient modeling approaches such as the Actuator Line Method (ALM). To ensure predictive capability, ALM-based simulations must be carefully validated against experimental data obtained under controlled conditions, providing a benchmark for assessing accuracy and guiding improvements.

Building on this motivation, the ALM is a widely used tool in the numerical simulation of wind turbines, thanks to its ability to represent a wind turbine without the computational cost of resolving the full blade geometry (Troldborg, 2008). It has become a widely adopted approach in LES frameworks for wind energy research, offering a practical balance between accuracy and efficiency. However, its performance depends strongly on the details of its implementation. Differences in how the ALM is formulated and applied (such as the smoothing kernel or velocity sampling) can lead to significant variations in the simulated turbine behavior and wake characteristics (Jha et al., 2014; Martínez-Tossas et al., 2017). For this reason, careful validation against experimental data remains essential to assess and improve the reliability of ALM-based simulations (Spyropoulos et al., 2021). Several studies have investigated the optimal smoothing length scale for the Gaussian body force distribution used in ALM. Martínez-Tossas et al. (2017) proposed a theoretical framework to determine the optimal kernel width, showing that improper smoothing can lead to over- or under-prediction of aerodynamic forces. Building on this, Martínez-Tossas and Meneveau (2019) introduced a filtered lifting line theory to better understand the impact of force regularization and to improve the physical consistency of ALM implementations. More recent work by Stanly et al. (2022) applied a filtered ALM in LES to better capture wake dynamics and reduce numerical artifacts, while Jha and Schmitz (2017) proposed the Actuator Curve Embedding (ACE) method to improve the representation of blade geometry. Despite these advances, validation remains a critical step. The SWiFT benchmark campaign (Doubrawa et al., 2020) provided valuable field-scale lidar measurements of turbine wakes under varying atmospheric conditions. However, the uncontrolled nature of the inflow affected by shear, veer, turbulence intensity, and thermal stability limits the ability to isolate model performance.

A growing body of research has focused on wind tunnel experiments using scale turbines, where inflow conditions can be precisely controlled (Bastankhah and Porté-Agel, 2017; Han et al., 2022; Hodgson et al., 2023). Small-scale turbines ( $\varnothing \simeq 15\text{cm}$ ) have been used to study wake dynamics such as meandering (Coudou et al., 2017), but their low Reynolds numbers introduce scale-dependent effects that distort performance metrics such as the power curve. Larger model-scale experiments, such as the MEXICO project (Schepers et al., 2014; Boorsma and Schepers, 2016), used a 4.5m rotor to generate high-fidelity validation data, including wake profiles and blade pressure distributions. More recently, Huang et al. (2022) conducted experiments with a 40 cm swept-blade turbine, measuring wake evolution up to  $8D$  downstream. However, this study used a uniform inflow, which, while useful to isolate blade effects, does not replicate the complexity of an atmospheric boundary layer.

Building on this foundation, this study implements a variant of the actuator line model that introduces targeted modifications to aspects of the conventional approach and evaluates its performance through a direct comparison with wind tunnel measurements. A key feature of this work is the integrated experimental and numerical approach: a custom wind turbine, TWIST (Turbine for Wind-tunnel Investigation & Scaled Testing), was specifically designed, built, and tested under controlled wind tunnel conditions to provide a robust and consistent data set for validation at  $1.44D$  and  $4.35D$  downstream. The turbine was operated within a turbulent boundary layer designed to replicate the characteristics of an atmospheric boundary layer on a full



scale, ensuring that the experimental conditions are relevant for real-world applications. To enable a meaningful comparison, substantial effort was invested in reproducing the experimental inflow conditions in the LES environment using Nek5000. This included carefully characterizing and replicating the turbulent inflow to ensure that any discrepancies between simulation and experiment can be attributed to the ALM itself rather than differences in the underlying base flow. By closely aligning the experimental and numerical setups, this study provides a robust platform to evaluate the capability of a newly proposed ALM to produce wake dynamics under realistic flow conditions.

The paper is structured as follows.

- Section 2 presents the experimental setup, including a description of the wind tunnel used during the campaign and the instruments used to measure the flow, along with an assessment of their associated uncertainties. This is followed by a detailed description of the wind turbine model designed for the study.
- In Section 3, the numerical setup is introduced, covering the computational code utilized, the mesh generation process, and related simulation parameters.
- A dedicated Section 3.2 addresses inflow generation, with the objective of accurately reproducing the experimental inflow conditions in both mean and fluctuating components.
- The ALM implemented in the context of this work is then described in Section 3.3, also highlighting the differences relative to conventional ALM.
- Section 4 presents a comparative analysis of the experimental and numerical results at two angles of attack. Those results are analyzed at two locations downstream of the wind turbine.
- Section 5 compares the results of the blade deformation obtained numerically and experimentally.
- A comprehensive discussion of the results is provided in Section 6, evaluating the agreement between experimental and numerical findings and highlighting key insights.
- Finally, the paper concludes with a summary of findings and reflections on the implications of this work.

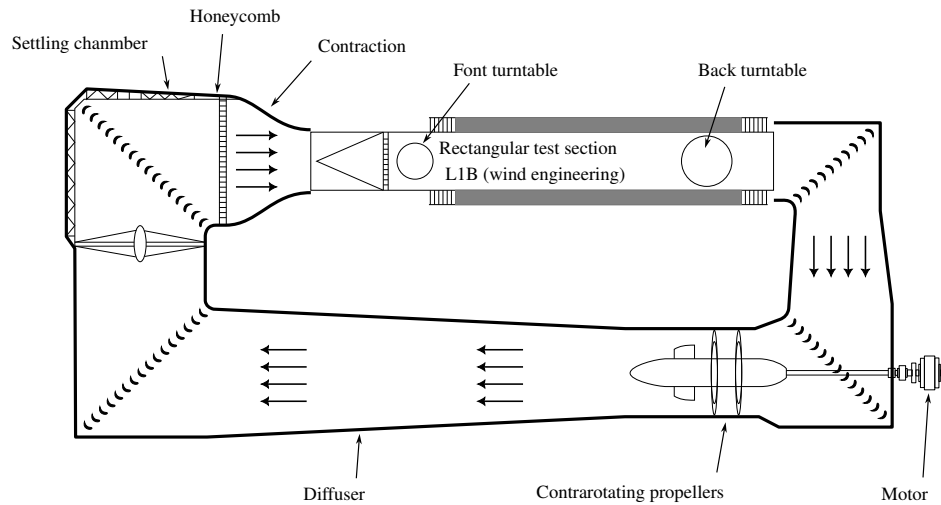
## 2 Experimental setup

The wind turbine wake experiment is conducted in the controlled environment of a wind tunnel. By testing in stable flow conditions, the wind tunnel provides an ideal setting to study the performance of the wind turbine under well-defined and repeatable conditions. This controlled setup not only allows for precise testing, but it also facilitates a numerical reproduction of the flow regime. As a result, the discrepancies observed between the experimental and numerical results can be attributed primarily to the modeling of the wind turbine itself, rather than to uncertainties in replicating ambient environmental parameters. This level of control and precision is essential for evaluating the behaviour of the ALM used to model the wind turbine in Section 3.3.



## 2.1 L1-B wind tunnel

In order to perform experiments in a controlled environment, we are using the L1B wind tunnel of the von Karman Institute:  
 90 a low speed, closed-loop wind tunnel with rectangular test section. The flow is driven by a fan with contra-rotating blades, powered by a 580 kW DC motor with variable speed controller. In the center of the test section, the flow velocity can range from 1 to 60  $\text{ms}^{-1}$  in the 1000 hp configuration, and from 1 to 22  $\text{ms}^{-1}$  in the 100 hp configuration. A schematic of the facility is provided in Figure 1.

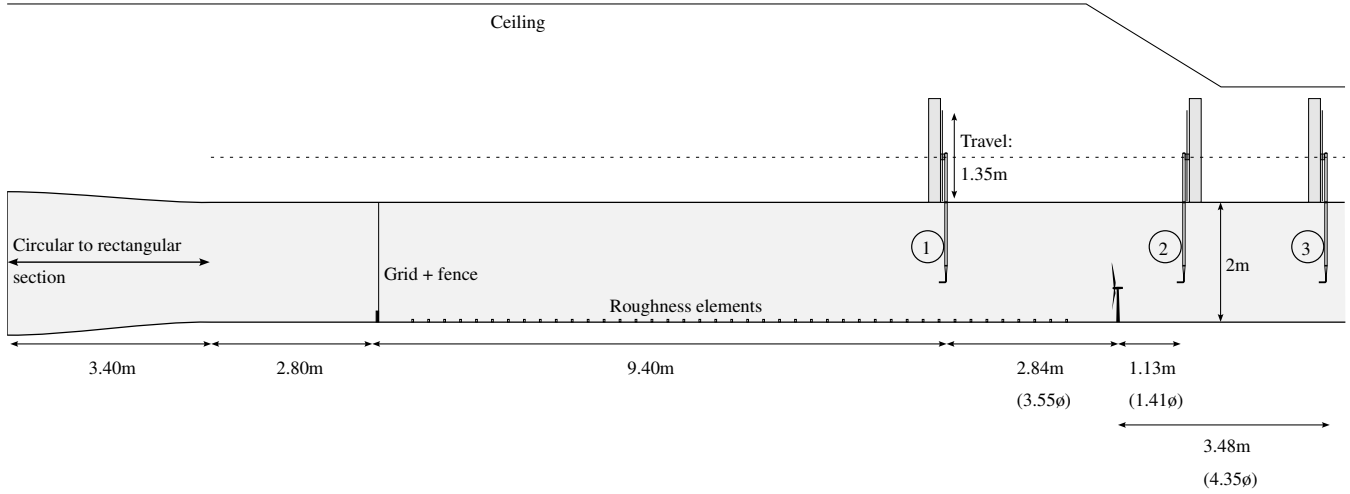


**Figure 1.** Top view of the VKI-L1 wind tunnel in wind engineering configuration.

The test section is 2 m high, 3 m wide, and  $\simeq 20$  m long; more detailed dimensions being provided in Figure 2. This  
 95 closed test section is used to immerse the experimental wind turbine in a turbulent boundary layer, similarly to the atmospheric boundary layer in which full-scale wind turbine are exposed. An atmospheric turbulent boundary layer is generated in the wind tunnel using a set of roughness elements (Barlas et al., 2015) on the bottom floor, complemented by a grid and fence at the inlet, to create velocity gradients and turbulence. This allows to reduce the streamwise distance needed to obtain a well-developed velocity profile at the back turntable, where the experiment is performed. A detailed verification of the wind tunnel's suitability  
 100 for atmospheric flow modelling can be found in Conan (2012). The position of the wind turbine is defined to be sufficiently downstream for a well-developed flow, while allowing enough streamwise distance to measure the wake. In Figure 1, the wind turbine is positioned at the back turntable, with a more detailed schematic in Figure 2.

The flow measurements are performed using hot wire anemometry along vertical lines in the test section. Three locations have been used for the measurements, labeled 1, 2 and 3 in Figure 2. Due to the probe holder's maximum displacement of  
 105 1.35 m, the inlet measurement (labeled 1) was performed in two parts to capture the full height. Locations 2 and 3 are limited to the first 1.25 m due to space constraints, yet sufficient to cover the region of interest. All point measurements were sampled at 20 kHz for 5 min each. The uncertainty quantification is described in Section 2.1.1.





**Figure 2.** Test section for the L1-B wind tunnel and the three measurement locations.

### 2.1.1 Measurement uncertainties

The uncertainty on the measurements depends on a chain of uncertainties that can be split in two types: the static uncertainty (bias of the calibration), and the random/statistic uncertainty (distribution of the measurement). The static uncertainty, denoted  $\delta U$ , represents the confidence interval attributed to the initial imperfections of the devices used for the calibration. It is computed in the following way:

$$U = \sqrt{\frac{\Delta P * 298.058 * T}{0.5 * P}} \quad (1)$$

$$\delta U = \sqrt{\left| \frac{\partial U}{\partial \Delta P} \right|^2 \delta \Delta P^2 + \left| \frac{\partial U}{\partial P} \right|^2 \delta P^2 + \left| \frac{\partial U}{\partial T} \right|^2 \delta T^2}, \quad (2)$$

where  $U$  represents the wind speed,  $\Delta P$  the pressure difference of the manometer,  $T$  the temperature,  $P$  the atmospheric pressure, and  $\delta \Delta P$ ,  $\delta T$ ,  $\delta P$  the respective uncertainty from each measurement device.

Conversely, random uncertainties arise from other factors such as noise and turbulence. Recording for longer time and taking more points for the calibration curves and for the measurements reduces the random uncertainties on the final measurements. The methodology proposed by Vugrin et al. (2007) is applied here to propagate the random uncertainty of a linear fit from each step of the calibrations to the final measurement. With this method, one standard deviation of the linear fit is given by the square root of the diagonals of the covariance estimate of the fit. All calibrations in the performed measurement campaigns were linear, except the final hot wire calibration curve, for which the maximum difference in wind speed is taken from the calibration curves of each point. Finally, to ensure a 95 % confidence interval on the measurement, the number of independent samples  $N$  is given by:

$$N > \left( \frac{3.92 \sigma}{0.01 \bar{U}} \right)^2 \simeq 1.5 \cdot 10^5 (TI)^2, \quad (3)$$



where  $\sigma$  is the standard deviation, and  $\bar{U}$  is the mean of the wind speed, which together translate to a turbulence intensity (T.I). For a turbulence intensity initially assumed conservatively at 20 %, this leads to 6000 uncorrelated points. The autocorrelation time has been recorded to be between 0.035 and 0.045 s, leading to a minimum sampling time of 3.5 to 4.5 min. Each measurement has therefore been recorded for 5 min per point.

130 Between 2 and 6  $\text{ms}^{-1}$  (the operating condition range of interest), the static uncertainty ranges from 0.14 to 0.43  $\text{ms}^{-1}$ . The random uncertainty from the calibration curves from the pressure manometer and the rotational speed of the wind tunnel rotor together add up to values between 0.20  $\text{ms}^{-1}$  and 0.24  $\text{ms}^{-1}$  for the same range of velocities. Most of the random uncertainty comes from the calibration curve of the hot wire which accounts for  $\simeq 0.5 \text{ ms}^{-1}$ . This uncertainty has been reduced by repeating the measurement over multiple days, which has reduced it to  $\simeq 0.05 \text{ ms}^{-1}$ . This method was only possible for the  
 135 calibrations and measurements at location 1 and 2, which leaves location 3 with higher random uncertainties.

One can argue that, since the static uncertainty was applied on both profiles (with and without wind turbine), this bias would cancel out when examining the relative difference; which then only leaves the random uncertainties. This is the principle applied for the error bars in all the graphs of the present paper.

### 2.1.2 Quantification of the difference between numerical and experimental results

140 To evaluate the difference between the numerical and experimental results, we use several error metrics. For all error quantification, the two data sources are resampled at uniformly distributed heights, the zone close to the wall  $< 0.1\text{m}$  being ignored as it lies outside of the region of interest.

- RMSE (Root Mean Square Error): Quantifies the average magnitude of the differences between measurements and simulations, with larger discrepancies weighted more heavily.
- 145 – MAPE (Mean Absolute Percentage Error): Expresses the average absolute difference between measurements and simulations as a percentage of the measurements, helping to assess relative accuracy.
- Bias: Measures the average deviation between simulations and measurements.
- $R^2$  (Coefficient of Determination): used for assessing the quality of a calibration curve, values closer to 1 indicating a better quality.

### 150 2.2 Model-scale wind turbine TWIST (Turbine for Wind-tunnel Investigation & Scaled Testing)

Previous research (Bastankhah and Porté-Agel, 2017) has shown that scaled-down wind turbines can produce a similar wake as full-scale turbines, given that the main aerodynamic parameters that affect wake behaviour remain similar. For this purpose, a few main dimensionless numbers should be matched to ensure that a full-scale wake is reproduced at scale (Schmitz, 2019). First, the Tip Speed Ratio (TSR), that is the ratio between the incoming flow speed unaffected by the wind turbine, and the  
 155 blade tip speed ( $V_t = \Omega \cdot R$ ). This TSR determines the advection speed of the vortex generated by the blade tip, which in turn triggers turbulent structures in the wake, which are key contributors to wake breakdown and recovery. Then, we consider the



thrust coefficient ( $C_t$ ) defined as

$$C_t = \frac{T}{\frac{1}{2}\rho V^2 A}, \quad (4)$$

with  $T$  the thrust force exerted streamwise by the wind turbine and opposing the flow,  $\rho$  the air density,  $V$  the upstream wind velocity and  $A = \pi D^2/4$  the area swept by the wind turbine. This thrust is responsible for the velocity deficit in the wake:

Finally, there is the power coefficient ( $C_p$ ) defined as

$$C_p = \frac{P}{\frac{1}{2}\rho V^3 A}, \quad (5)$$

with  $P$  the mechanical power extracted from the flow by the wind turbine, as considered at the shaft of the turbine.

Scaled-down experimental wind turbines are typically designed to have a similar TSR and  $C_t$  as compared to the full-scale wind turbine, as those parameters are the main factors influencing the resulting wake. At this smaller scale, the wind turbine blades operate at lower Reynolds number ( $Re$ ), by about two orders of magnitude, which impacts the flow behavior locally and the capability to provide lift. Airfoils composing the blade of a model-scale wind turbine must be tailored for low- $Re$  applications. Small-thickness airfoils are selected with a good lift-to-drag ratio and reduced sensitivity to the flow separation at low Reynolds numbers.

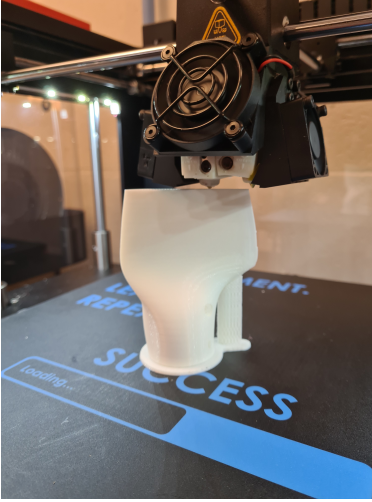
In addition, scale model testing introduces a blockage effect due to the closed environment of the wind tunnel. The blockage ratio is defined as the ratio of rotor swept area  $A$  divided by the wind tunnel cross-sectional area. Although it cannot be avoided, it is minimized such that its effect on the flow is reduced. According to McTavish et al. (2013), the blockage ratio should be  $< 10\%$ , and the minimum distance to a wall should be  $> 1D$ . The wind turbine designed in our work has  $D = 0.80$  m and a hub height of 0.61 m. According to the above definition, the blockage is thus  $\frac{A}{(2 \times 3)} \simeq 8.4\%$ . A more appropriate blockage concept in the present case consists of ignoring the region below the hub height, because this region is where the ground effect on the wind turbine is intentional. That blockage is obtained as  $\frac{A/2}{(1.39 \times 3)} \simeq 6.0\%$ . Above the wind turbine, the clearance to the top wall is 1.0 m (or  $1.25D$ ), and is the closest wall (excluding the ground).

We also wish to produce a wake whose statistics are independent of the Reynolds number which, for turbulence quantities is reached, above  $Re \simeq 9.3 \times 10^4$  as shown in Chamorro et al. (2011) (with  $Re$  based on the rotor diameter and free stream velocity). The present wind turbine model operates at  $Re \geq 35 \times 10^4$ , which should ensure Reynolds independency of the wake for the mean velocity and second-order statistics.

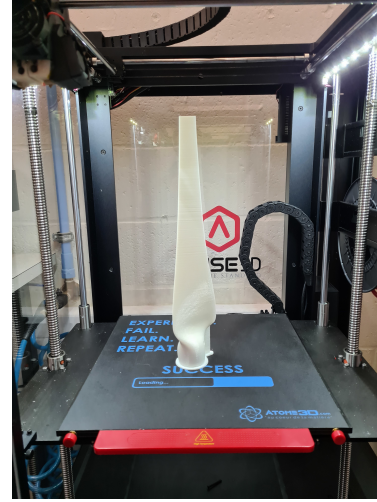
The blades were produced using a 3D printer of UCLouvain, as illustrated in Figure 3. The material is a PLA+ plastic, a more ductile variation of PLA. The ductility of the material allows the blade to flex under load without breaking. The airfoil used for the blades is the SD7037-092-88 airfoil, shown in Figure 4. In the operating conditions used for this experiment, the Reynolds number of the airfoil based on the chord is  $\simeq 5.0 \times 10^4$  throughout the length of the blade. Indeed:

At the tip of the blade, the Reynolds number is:

$$Re_c = \frac{Uc}{\nu} = \frac{29.9 \times 0.0259}{1.54 \times 10^{-5}} \simeq 5.0 \times 10^4 \quad \text{with} \quad U = \sqrt{(73.3 \times 0.40)^2 + 6.0^2} \simeq 29.9 \text{ ms}^{-1} \quad (6)$$



(a) During the printing process.

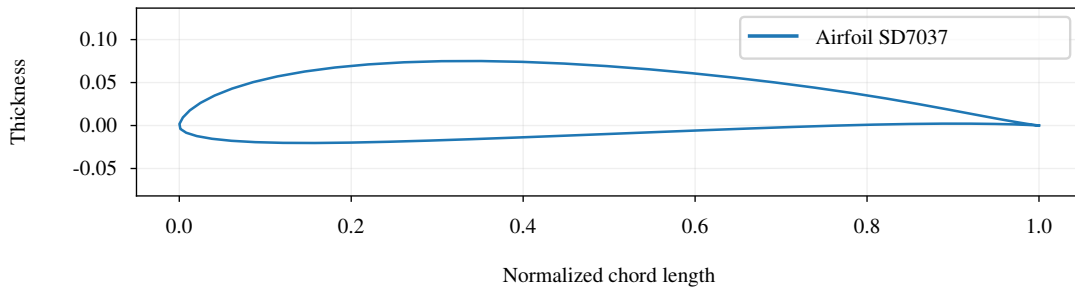


(b) Result of the printed blade.

**Figure 3.** 3D printing process of the blade at UCLouvain.

Close to the root of the blade (at 10 % of the radius):

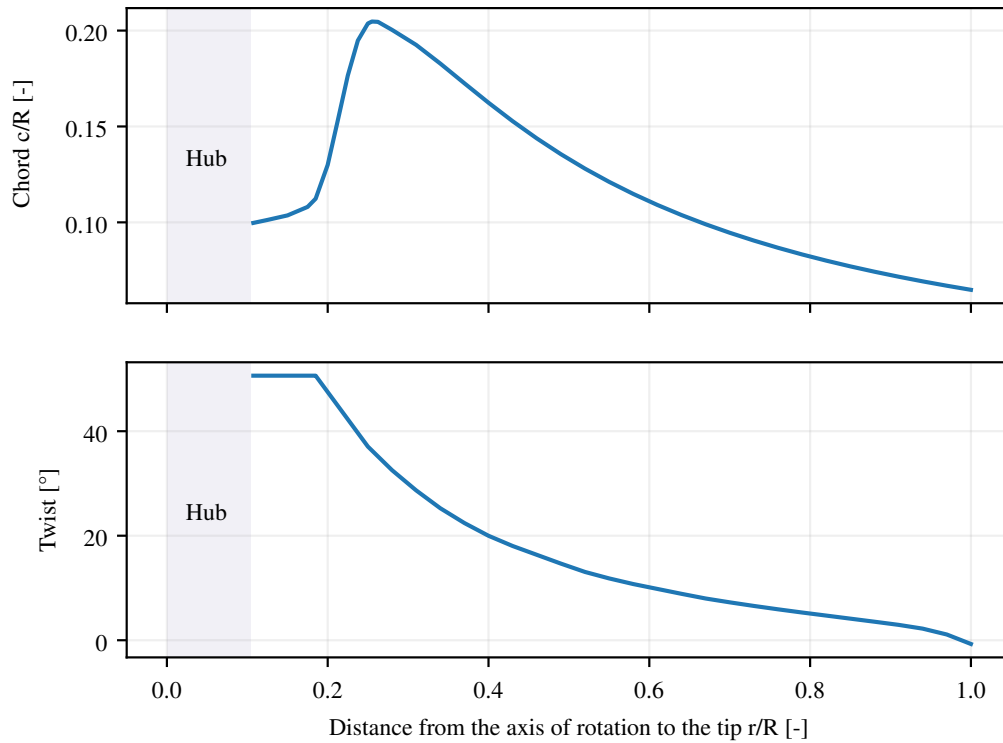
$$Re_c = \frac{Uc}{\nu} = \frac{9.5 \times 0.0814}{1.54 \times 10^{-5}} \simeq 5.0 \times 10^4 \quad \text{with} \quad U = \sqrt{(73.3 \times 0.10)^2 + 6.0^2} \simeq 9.5 \text{ ms}^{-1} \quad (7)$$



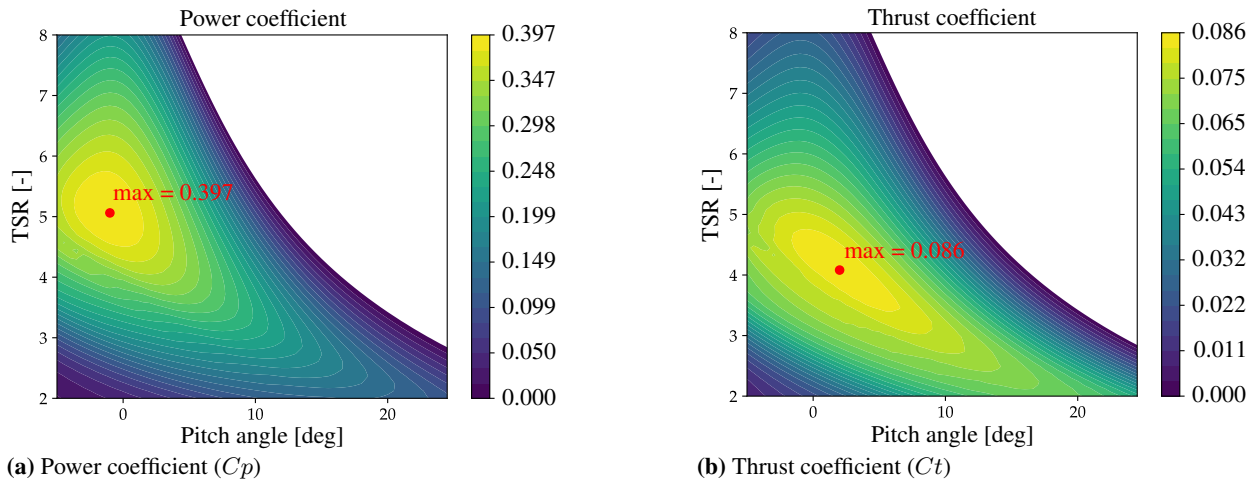
**Figure 4.** Airfoil SD7037-092-88 used for the 3D printed wind turbine blades.

190 The twist angle and chord distribution of the blades have been designed using a BEM approach. A second iteration of the blade design could improve the total power extraction, mostly to improve the aerodynamics in the region close to the blade root, but the current design can already serve as a validation case if the main parameters are matched in the ALM model. This design has been simulated using OpenFAST to obtain the  $C_p$  and  $C_t$  plots shown in Figure 6, for varying blade pitch angle and TSR, in order to be compared with experimental values.

195 Experimentally, the power curve was derived from electrical power measurements, at a pitch angle of  $0^\circ$ . As expected, a discrepancy arises between this measured electrical power and the theoretical aerodynamic power output predicted by OpenFAST. This deviation is attributed to several sources of mechanical and electrical inefficiency: bearing friction, losses



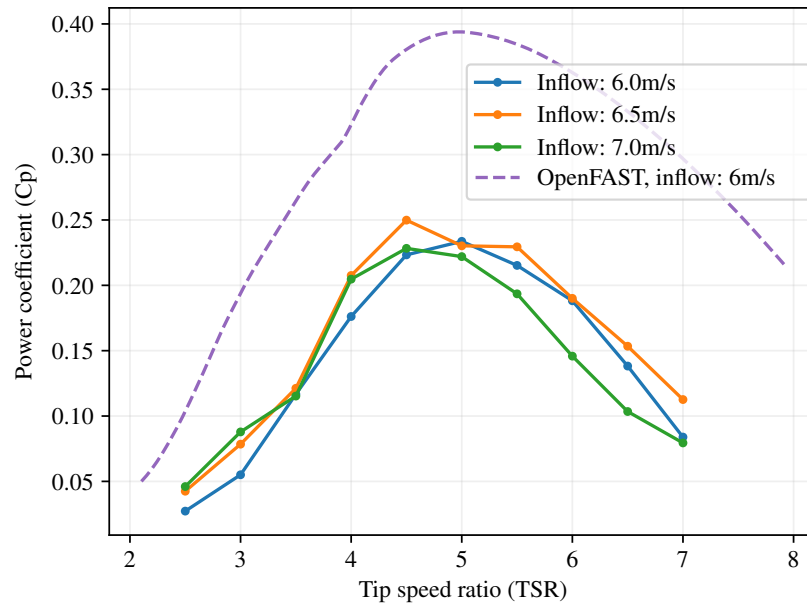
**Figure 5.** Blade design parameter for the TWIST wind turbine.



**Figure 6.** Power and thrust coefficients computed using OpenFast and the design parameters of the TWIST wind turbine.



in the bevel gear transmission to the generator, suboptimal generator efficiency (particularly below its rated operating speed) and additional losses incurred during the AC-to-DC conversion where electrical measurements are taken. This power curve



**Figure 7.** Experimental power curve of the TWIST wind turbine.

is shown in Figure 7. This corresponds to a vertical line in Figure 6. The measurement has been repeated at various inflow velocities close to the operating conditions of this experiment to assess the sensitivity to the Reynolds number. The results indicate no significant deviation of the curve across different inflow velocities.

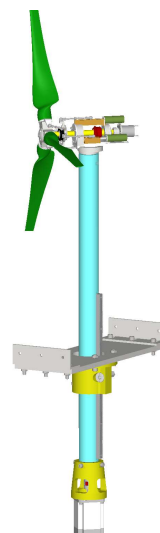
As mentioned before, The turbine model diameter is  $D = 80$  cm. The wind turbine support, indicated in Figure 8, allows adjusting the yaw and the hub-height of the model (set here to 61 cm). In an attempt to maintain a realistic ratio between nacelle size and swept diameter, the generator is placed under the wind tunnel. The power is then transmitted from the horizontal shaft to a vertical shaft through a set of spiral bevel gears, as shown in Figure 9. Other components are placed within the nacelle, resulting in a diameter of 8 cm, thus 10 % of the swept diameter. The total length of the nacelle, from the base of the blade support to its furthest part downstream, is 23.1 cm, thus approximately 3 times the nacelle diameter. The blades can operate in cyclic and collective pitch control. In the scope of this work, the pitch angle remains constant throughout the experiment.

The aerodynamic power is transferred to the 3-phase BLDC generator, to convert to electrical energy. This electrical energy is then rectified to DC using a full bridge rectifier and then used in a series of resistances where the load can be controlled by changing the total resistance. Changing the load is the easiest way to change the reactive torque on the wind turbine, hence control the rotational speed. Twelve resistances are used, each having an ohmic resistance value double of the previous one. This results in a 12-bit binary code for the load of the wind turbine, which provides 4096 possible load values; hence a set of loadings that can be considered continuous. The measurement of power is done on the AC side (before the rectifier to DC) by measuring the voltage and current on one line. In this configuration, the electrical power is measured before the losses in the





(a)



(b)

**Figure 8.** TWIST wind turbine model installed in the L1-B test section and its CAD.



(a)



(b)

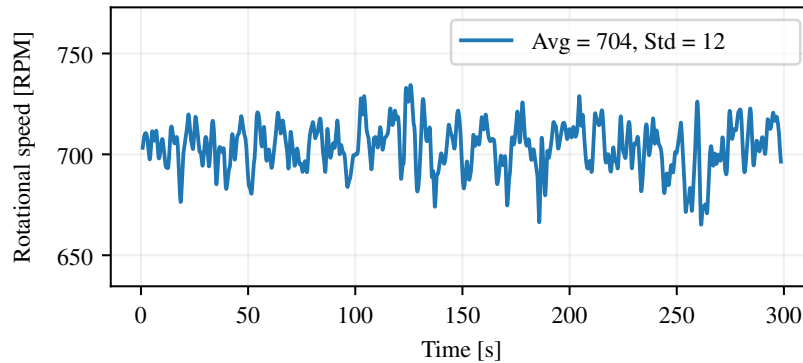
**Figure 9.** Mechanism in the nacelle, and generator below the wind tunnel floor.





conversion to DC. The electrical power is computed as three times the average power of one line; leading to  $P = \frac{3}{\sqrt{3}} V_L I_L$ , where  $V_L$  is the instantaneous line voltage, and  $I_L$  is the instantaneous line current.

In the scope of this study, the TWIST wind turbine is operated at constant rotational speed, as it experiences a steady inflow. The encoder of the generator is used to obtain the rotational speed of the wind turbine during operation, connected to an Arduino microcontroller which also controls the set of relays to act on the resistive load. An integral controller was added to maintain the rotational speed, as it was found to vary otherwise due to changes in resistance caused by temperature fluctuations. Nevertheless, some variability in the rotational speed is still observed as shown in Figure 10. Indeed, the controller was tuned to compensate for the heating of the electrical resistances, which is the slowest process, but still some fluctuations are present due to low inertia of the rotor and flow turbulence.



**Figure 10.** Time series example of the rotational speed of the designed wind turbine for the present experimental campaign.

### 3 Numerical setup

In this paper, the code chosen for the LES simulations is Nek5000 (Argonne National Laboratory, 2023), a high-order spectral element code developed by Argonne National Laboratory in the USA. It presents the advantage of being open-source, with excellent scalability for large parallelization, and minimal numerical dispersion and dissipation errors. In addition, its GPU version NekRS would allow our implementations to be ported to this new computational paradigm with relative ease.

The spatial domain is discretized into elements that then use a spectral method to obtain the numerical solution, as described in Patera (1984). This combines the advantages of the spectral method to compute the partial derivatives with high efficiency, together with the local grid refinement of the finite element method. The solution is computed on each element using a high-order weighted residual technique with Gauss-Lobatto-Legendre polynomials (Deville et al., 2002). The order of accuracy of the code is dictated by the order of the polynomials used as a basis function in the discretized elements. The main advantage of a higher order method is to produce smaller dispersion errors as depicted in Wang et al. (2013), for a similar cost compared to lower order methods. For a fair comparison with the literature, the computational cost of high-order methods is usually expressed in terms of Degrees of Freedom (DoF), which compares directly to the number of cells of a finite difference solver.

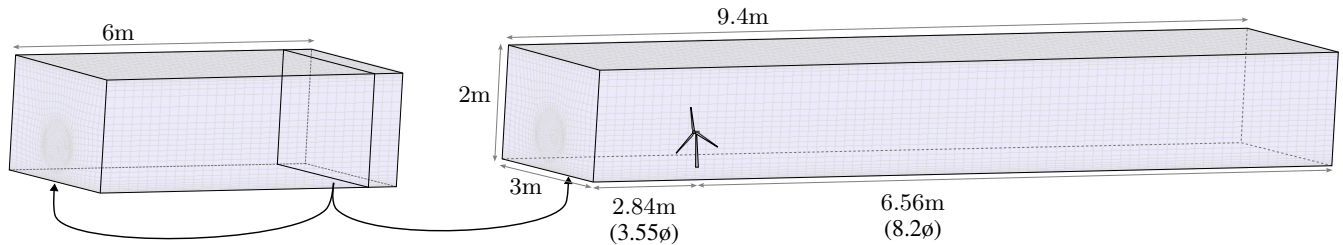


In the case of Nek5000, the DoF is calculated by multiplying the number of elements in the computational domain by the number of independent nodes in each element (equivalent to the polynomial order used in each element). Based on previous research using Nek5000 Fischer (2013), it appears that for most cases the code exhibits optimal performance when using a 7th order polynomial.

### 3.1 Numerical domain

In order to reproduce blockage effects numerically, the first computational domain corresponds to the wind tunnel test section dimensions and it includes both the bottom and top walls. It is used to obtain a fully developed turbulent flow with properties similar to those of the experiment. This is achieved by using a recycling method with some added forcing, and described later. To reduce the computational cost, the side walls are not represented, as they are sufficiently far away from the wind turbine not to affect the flow in the mid-vertical plane of interest. Instead, a periodic boundary condition is applied to the sides.

The second domain, which is the main domain, is continuously fed at its inlet using the turbulent flow field sampled at the recycling plane of the first domain simulation. This second domain contains the wind turbine, with blades modelled using an ALM. Both domains have the same cross-section but different lengths, as shown in Figure 11.

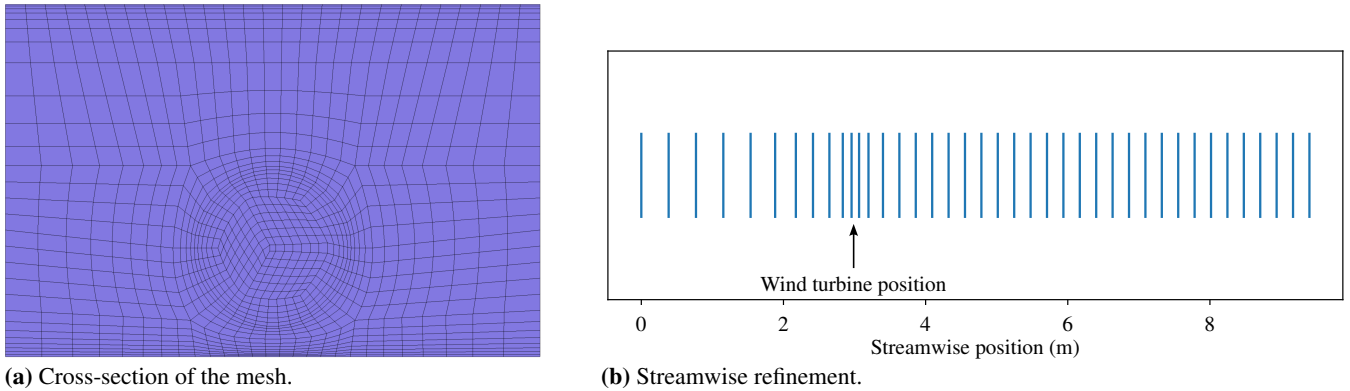


**Figure 11.** Co-simulation domains with their dimensions and mesh, as used in the Nek5000 setup.

The mesh is refined vertically close to the bottom and top walls to capture the wall-induced shear; and also in the zone of the wind turbine, especially near the tip, see Figure 12. In the cross-section, the mesh is composed of 133 elements in the inner part of the wind turbine region, 360 elements around the tip region, 216 elements close to the bottom wall, 189 elements close to the top wall, and 252 elements in the remaining zones, which totals 1150 elements. In length, the mesh of the recycling domain is composed of 32 equally spaced elements. The recycling plane is at 5.4 m from the inlet, which is sufficiently far from the outlet to avoid spurious numerical effects. This mesh was created using the GMSH software (Geuzaine and Remacle, 2009).

For the main domain, the length is 9.4 m with 80 elements. The element size streamwise starts identical to that of the recycling domain in the zone upstream of the wind turbine, then gets finer at the location of the wind turbine, and is then only slightly coarsened to still properly capture the wake. The wind turbine represented using the ALM is placed at 2.84 m ( $3.55D$ ) from the inlet.

The bottom and top wall of both domains use a shear stress boundary condition, where the value of the shear is obtained from the experimental profile. The value for  $\tau_{wall}$  used in the LES simulation was obtained by fitting a log law from the experimental profile.



**Figure 12.** Mesh details: cross-section and streamwise refinement.

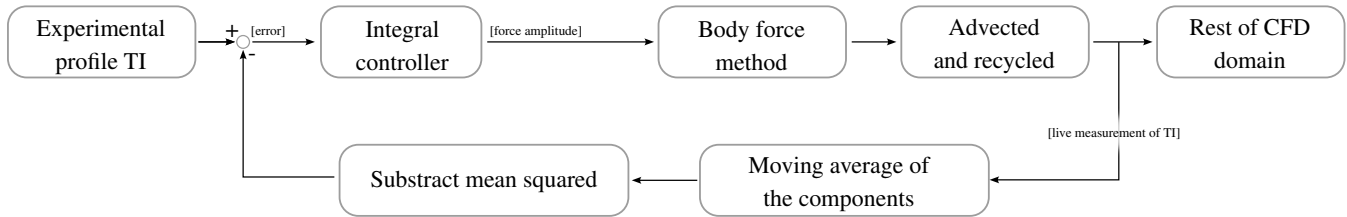
265 The velocity field at the inlet of the recycling domain is updated at each timestep, using a rescaling of the velocity field measured at the recycling plane. Finally, the outlet of both domains is a Neumann open boundary condition with specified pressure.

### 3.2 Inflow generation

270 The simulation uses the co-simulation described in Section 3.1 to generate an inflow for the main domain matching as closely as possible that of the experiment, both in terms of mean velocity profile and of turbulence level; more detailed results having been presented in Gillyns et al. (2024).

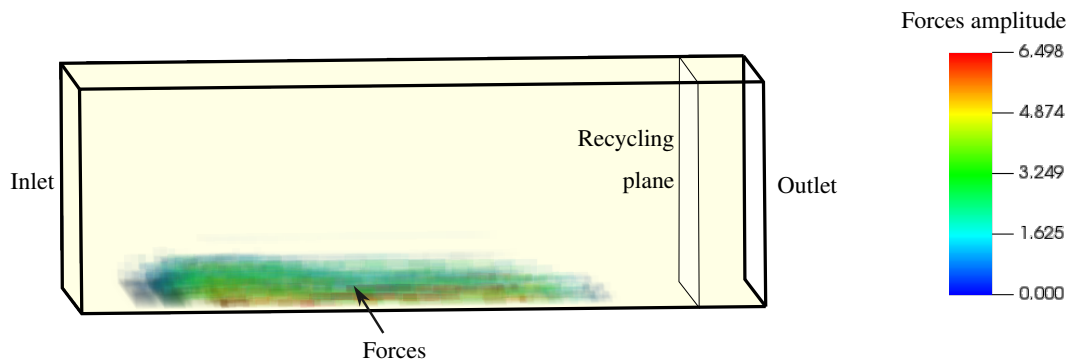
The mean flow is reconstructed using a modified version of the “Recycle and Rescale Method” (R2M) (Xiao et al., 2016), also described in Gillyns et al. (2024). Unlike the original approach, this method does not apply a vertical scaling to match the boundary layer height. Instead, a scaling is performed at every grid point. A moving average of the simulated flow is calculated and divided by the experimental mean velocity profile to generate a spatial scaling factor field. This factor is then applied to the instantaneous flow at the recycling plane before re-introducing it at the inlet. The technique ensures that the inlet profile matches the experimental profile in its time-averaged value, while allowing the turbulence to naturally develop within the recycling region.

280 However, the turbulence level which naturally develops in the domain remains lower than what is observed experimentally in some regions. This is attributed to the upstream grid and fence which are not reproduced numerically, and generate additional turbulence. In an attempt to provide an inflow that matches the experiment as closely as possible, a set of forces have been added where appropriate to increase the level of turbulence in the recycling domain. Those forces are distributed in space as a set of 10 moving sine waves in  $x$ ,  $y$ ,  $z$ , pointing in the (positive and negative) vertical direction. Their amplitude is controlled using an integral controller at every grid cell, with the control strategy illustrated in Figure 13. A smoothing function is used to remove the forcing near the inlet and the recycling plane, also for numerical stability. The main advantage of the present method is that it allows to increase the level of turbulence in needed regions, without having to tweak any parameters, since this action is handled by the controller.



**Figure 13.** Control strategy for the forces in the recycling domain.

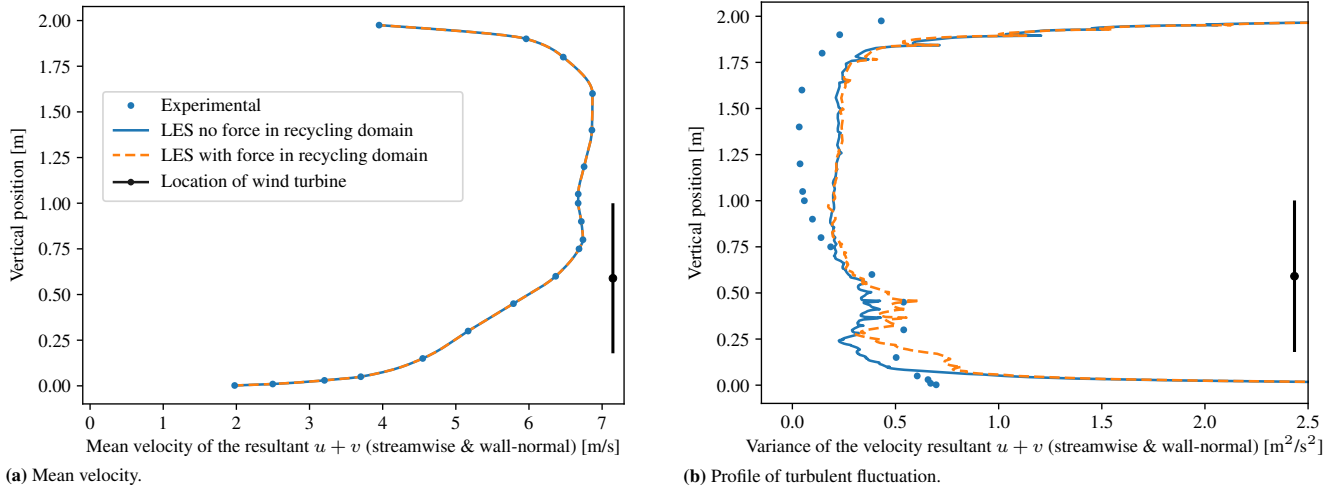
The amplitude of the forcing is illustrated in Figure 14. In this illustration, it appears that the forces are needed only in



**Figure 14.** Amplitude of the forces used in the recycling region of the numerical setup.

the lower region, which is confirmed by the vertical profile in Figure 15. The use of a smoothing function near the inlet and recycling plane is clear as the amplitudes vanish there. Within the recycling region, where the forcing is applied, the amplitude of the forces gradually decreases along the streamwise direction. This trend arises from the controller's local operation at each grid point: in the upstream part, stronger forces are initially required to generate the desired turbulence level but, as the flow propagates downstream, the incoming fluctuations begin to resemble those of the experimental profile, hence reducing the need for additional forcing. An important consequence of this approach is that, by the time the flow reaches the recycling plane, the imposed perturbations have largely dissipated. As a result, their influence on the turbulent spectrum becomes negligible, allowing the flow to naturally exhibit the desired turbulence characteristics downstream without contamination from artificial forcing.

The resulting mean velocity profile shown in Figure 15 matches exactly the experimental profile; which is expected since it is imposed in the recycling method. No significant difference is observed in the turbulence fluctuation profile in regions where the experimental turbulence is lower than the numerical prediction (that is, above 0.75 m); which is also as anticipated. Below this height, the turbulence level is increased by the forcing method. To quantify this improvement, the Mean Absolute Percentage Error (MAPE) between the experiment and LES in this region was calculated: 31% without volume forcing, and 16% with it, representing a 50% reduction in error.



**Figure 15.** Profile of wind speed at location 1 (Inlet of the main domain).

With the successful alignment of the mean inflow profile and the enhancement of the fluctuating velocity field, the baseline  
 305 flow now replicates the experimental conditions quite closely. This minimization of inflow-induced discrepancies better ensures  
 that remaining differences observed between the experimental and simulation results can be more likely attributed to the  
 modeling approaches themselves, rather than to inflow mismatches.

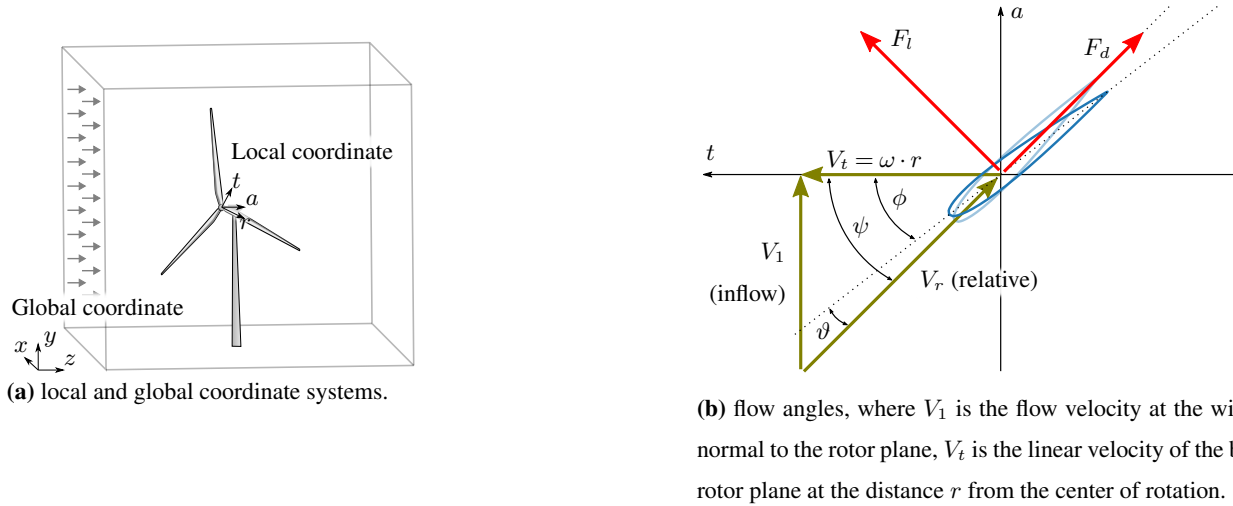
### 3.3 Extended implementation of ALM

The core principle of the ALM is to compute and apply the force induced by the wind turbine on the flow, without explicitly  
 310 representing it. This method is typically used for the blades but, in this work, we expand the same principle for all three parts  
 of the wind turbine, namely the blades, the nacelle, and the tower.

Both the blades and the tower have a slim aspect ratio, hence it is assumed that using the coefficients of lift ( $C_l$ ) and drag  
 ( $C_d$ ) from a 2D simulation representing the local geometry constitutes a good approximation over most of the span. As for the  
 nacelle, it is assumed to be a bluff body with an associated drag coefficient.

315 The different steps of the ALM as implemented for the blades are presented hereafter. At each time step and every grid  
 point, the wind speed components are sampled and transformed into a local coordinate system for each blade, as shown in  
 Figure 16 (a). This local coordinate system is positioned in the domain at the location of the wind turbine based on the hub  
 height and yaw position. The rotation of the wind turbine blades around the axis of rotation is applied on this coordinate system,  
 which makes it easy to change between using a controller or a fixed rotational speed. In this case, a fixed rotational speed is  
 320 used.

In Figure 16 (b), the airfoil composing the blade is shown with a non-zero angle of attack. The forces of lift  $F_l$  and drag  
 $F_d$  are, by definition, those aligned with the flow and perpendicular to the flow, respectively. The angle between the effective



**Figure 16.** Coordinate systems (a), and local wind speed and directions around the blade (b).

position of the airfoil and the flow direction is the angle of attack. As shown in the figure, the airfoil is angled in space, labeled  $\phi$  as the sum of the pitch angle and the twist angle (as a function of  $r$ ). Based on this, the angle of attack labeled  $\vartheta$  is  $\vartheta = \psi - \phi$ .

325 The forces of lift and drag are obtained using

$$dL = \frac{1}{2} \cdot \rho \cdot V_r^2 \cdot C_l \cdot c \cdot dr \quad (8)$$

$$dD = \frac{1}{2} \cdot \rho \cdot V_r^2 \cdot C_d \cdot c \cdot dr, \quad (9)$$

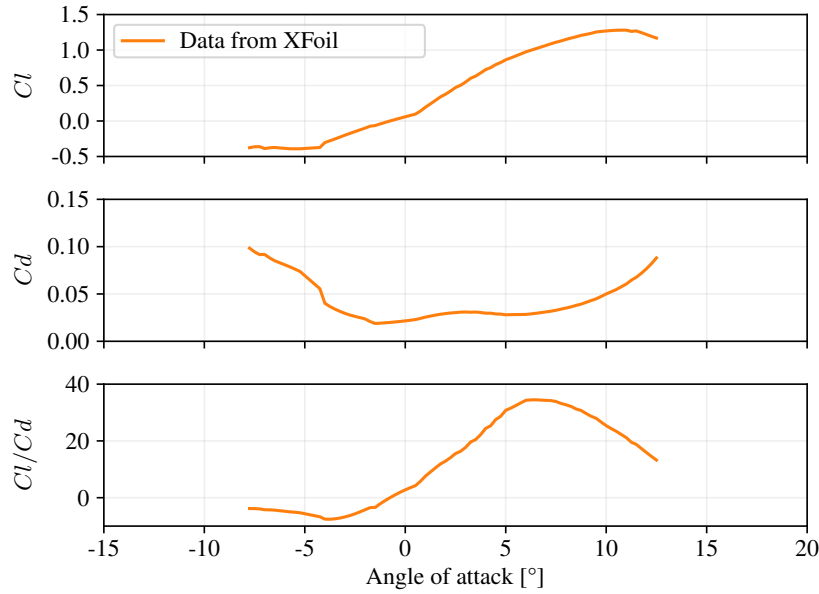
with  $dL$  : the discretized fraction of lift force (N),  $dD$  : the discretized fraction of drag (N),  $V_r$  : the measured relative velocity ( $\text{ms}^{-1}$ ),  $c$  : the chord length (m),  $dr$  : the discretized fraction of radius (m). The lift and drag coefficients,  $C_l$  and  $C_d$ , are  
 330 obtained from the tabulated data using the measured angle of attack and Reynolds number  $Re_c = V_r c / \nu$ . These data were obtained using XFOil at  $Re_c \simeq 5 \cdot 10^4$ , and for the range of angles  $-7.5$  to  $12.5^\circ$ , which is sufficient for the simulation in normal conditions.

Close to the blade root, there is a transition from the airfoil to an ellipse, and then to a circle. The lift and drag coefficients for the ellipse were computed using OpenFoam, and the drag coefficient for the circle was taken from Munson et al. (2012). It  
 335 is worth stressing that, in this region, there is a sharp transition in geometry, and the main hypothesis of the ALM to use 2-D polar curves can be questioned. The coefficients used for the lift and drag are shown in Figure 17.

It is worth noting that here, the forces of lift and drag are computed locally at every grid point in the domain, unlike most ALM implementations that consider the center of the actuator line. This is one of the difference with most implementations of the ALM.

340 We also use a continuous distribution along the actuator line which avoids any smoothing in the radial direction Trigaux et al. (2024c, b), hence providing a better radial resolution. The Gaussian spreading of the force solely acts in a plane perpendicular





**Figure 17.** coefficients of lift and drag obtained from XFOIL

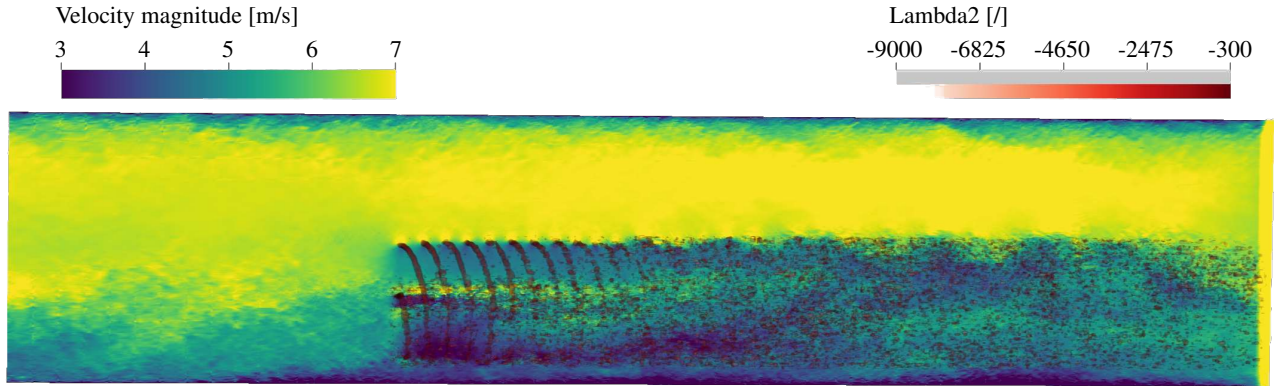
to the blade direction. The 2-D Gaussian kernel is thus

$$f_{(x,y)} = \frac{1}{2\pi\sigma^2} e^{-\frac{d_p^2}{2\sigma^2}}, \quad (10)$$

with  $\sigma$  its core size and  $d_p$  the distance from the actuator line to the point  $(x,y)$  in that plane. The value of  $\sigma$  is chosen as a  
 345 function of the mesh created for the wind turbine (Jin, 2013; Kleusberg, 2017):  $\sigma = 2.5l/n$  where  $l$  is the blade length, and  $n$   
 is the number of GLL points along the blade: the number of elements times the degrees of freedom per element (= polynomial  
 order).

Because a two-dimensional spreading formulation may lead to numerical instabilities at the blade tip when the local  
 mesh is insufficiently refined, an optional tip-spreading correction was implemented. This correction is based on a modified  
 350 Heaviside function with a smoothing parameter consistent with that used in the two-dimensional Gaussian kernel. In the present  
 simulations, the mesh resolution near the blade tip was sufficiently fine, and the tip spreading was therefore disabled, allowing  
 for a more physically realistic formation and advection of the tip vortex in the wake.

As a first qualitative view of the resulting flow field, a plot of the velocity magnitude in the vertical symmetry plane is  
 provided in Figure 18. The differences in boundary layer height between the two walls are clearly visible, due to the increased  
 355 roughness at the bottom. At the outlet, a flow acceleration that is introduced for numerical stability reasons is also visible,  
 although its effect remains very localized. The tip and root vortices shed by the blades are clearly noticeable. The near wake is  
 coherent, but it quickly loses its coherence and becomes fully turbulent, due to the interaction of the tip and root vortices with  
 the tower and the turbulent inflow.



**Figure 18.** Field of instantaneous velocity magnitude in the vertical symmetry plane passing through the hub of the wind turbine, on which a lambda2 criterion isocontour is overlaid, highlighting the vortices from the wake.

The tower is represented by a non-moving, vertical actuator line at a distance of 13 cm from the center of the blades which is the distance of the experimental wind turbine design. It is a cylinder of constant diameter ( $\phi 60$  mm), for which the drag coefficient is taken from the literature (Munson et al., 2012). Its Reynolds number being of the order of  $10^4$  the drag coefficient is taken as  $C_d = 1.4$ .

At the top of the tower, the actuator line also stops sharply, the numerical effects are not as significant as those at the blade tip because of the much lower relative velocity. Nevertheless, a similar spread as for the tip has been used for consistency.

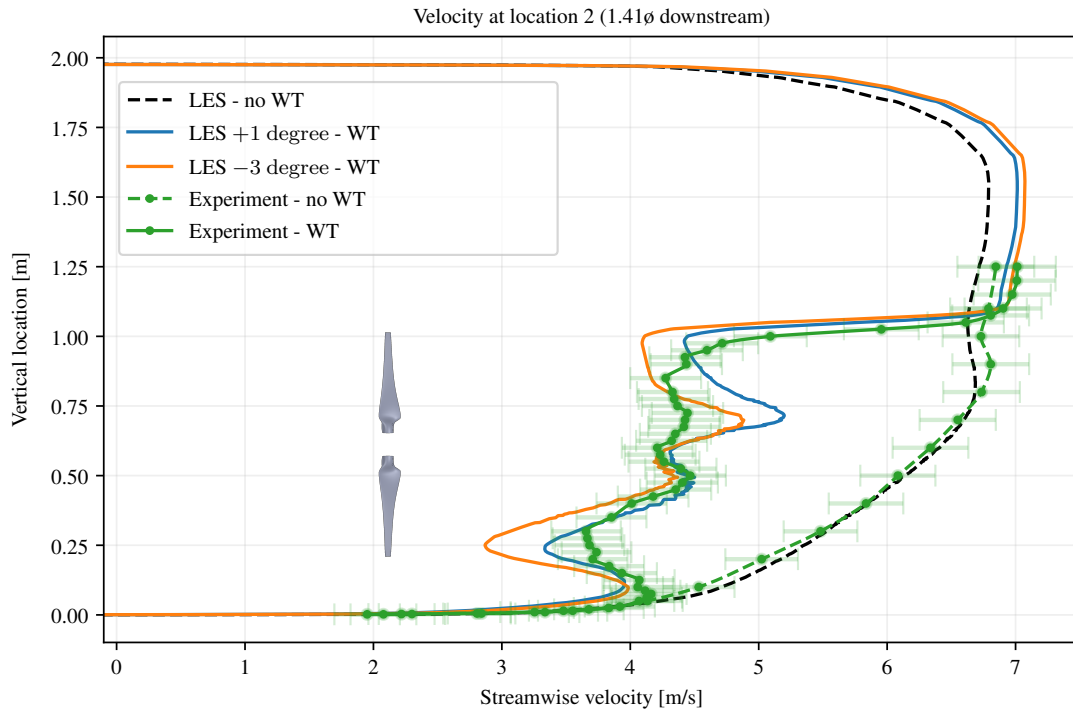
The nacelle is a bluff body without longitudinal extend; hence its effect cannot be represented using an actuator line. The drag of the nacelle is computed using the local flow velocity and a drag coefficient, and the resulting force is distributed using a 3-D Gaussian ellipsoid. Its effective frontal area is approximated as a circle with a radius equal to the distance from the turbine center to the blade root  $A = \pi \cdot 0.043^2 = 5.8 \cdot 10^{-3} \text{ m}^2$ . Determining the drag coefficient of the nacelle is not trivial, as it is not an enclosed body. Therefore, we have adjusted the coefficient empirically until the wake in this region appeared to match observations, resulting in a value of  $Cd = 2$ .

#### 4 Influence of the blade pitch angle

In this section, Nek5000 simulations with the extended ALM implementation are performed at two different pitch angles ( $+1^\circ$  and  $-3^\circ$ ) to investigate the effect that this parameter has on the resulting wake. The parameter that has the most influence on the power and thrust, hence also on the wake, is the angle of attack that each section of the blade makes relative to the local flow. As shown previously in Figure 16 (b), the angle of attack  $\vartheta$  depends on the twist angle (from the blade geometry) and the pitch angle (variable, based on the controller operation). The twist angle distribution used in the simulation is the same as that used for the 3D printing of the blades, see Figure 5



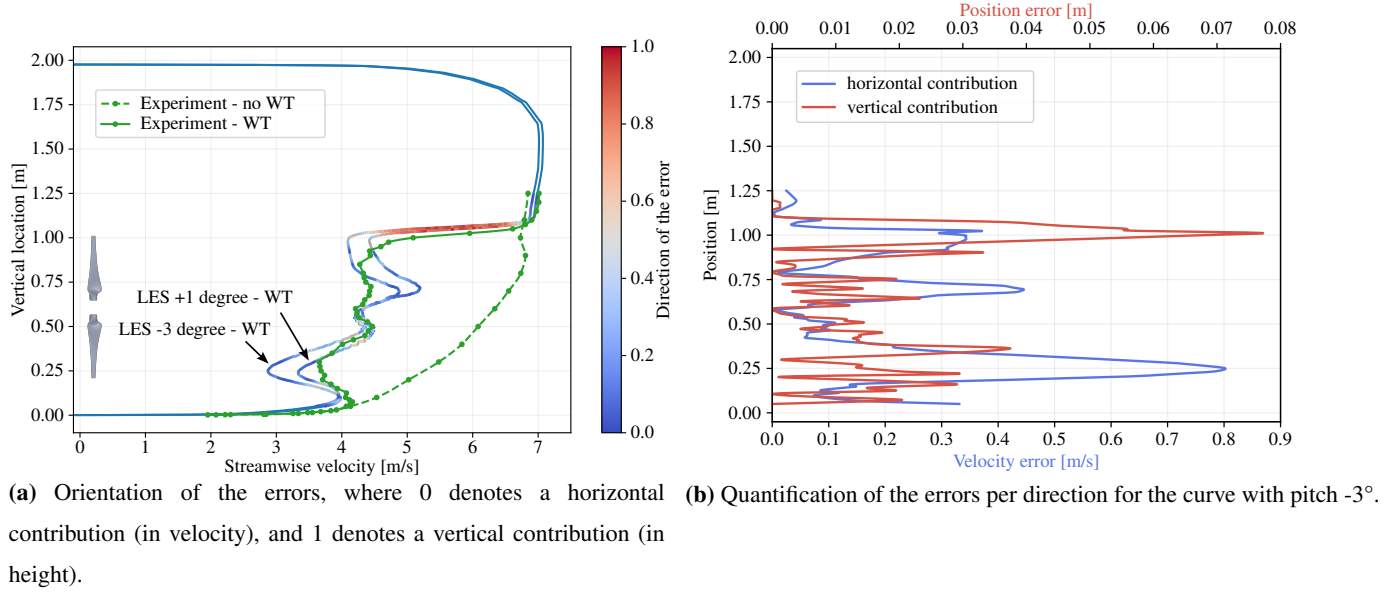
As shown in Figure 19, the velocity magnitude is extracted along a vertical line at position 2 (see Figure 2), located 1.13 m (1.41  $D$ ) downstream of the wind turbine, for comparison with the hot wire measurements. The dashed lines correspond to the results without the wind turbine, while the solid lines are obtained with it. As indicated in the plot, the experimental results are point by point hot wire measurements, and a spline interpolation was used here to connect those values. The base flow (without



**Figure 19.** Velocity profile comparing the LES results for two pitch angles with the experimental results at location 2 (1.41  $D$  downstream of the wind turbine).

wind turbine) at this location of both simulations show a good agreement with the experiment, with an error on the mean profile characterized with a RMSE of  $0.11 \text{ ms}^{-1}$ , a BIAS of  $-0.0037 \text{ ms}^{-1}$ , a MAPE of 1.55 %, and an  $R^2$  value of 0.98 thanks to our previous effort to match the mean profile and turbulence level of the inlet (Gillyns et al., 2024). With this small error in the base flow, it is expected that the observed differences when the wind turbine is present can be attributed to the modeling of the wind turbine itself (i.e., the simplified modeling of the blades and tower effects on the flow using the ALM, and of the nacelle effects) rather than to inaccuracies in the base flow.

We also observe, in Figure 19, that the experiment exhibits an intricate change in the velocity gradient caused by the interaction between the turbine wake and the ABL, and that this is fairly well captured by the simulation. The ability of the model to reproduce such detail in regions of high gradient near the tips can be attributed to the present ALM implementation, made more precise by not averaging in the blade direction. We cannot claim that the method is accurate near the blade root as



**Figure 20.** Error quantification for the velocity profile from Figure 19 at location 2.

**Table 1.** Errors for the velocity profile at location 2.

| Case | Velocity RMSE          | Velocity MAPE | Position RMSE | Position MAPE |
|------|------------------------|---------------|---------------|---------------|
| +1   | $0.26 \text{ ms}^{-1}$ | 4.0 %         | 0.02 m        | 2.6 %         |
| -3   | $0.30 \text{ ms}^{-1}$ | 5.1 %         | 0.02 m        | 3.7 %         |

the aerodynamics of the transition from an airfoil to an ellipse and then to a circle cannot be well represented by an ALM. The same comment applies to the nacelle, which is modeled using a drag force spread over some ellipsoid.

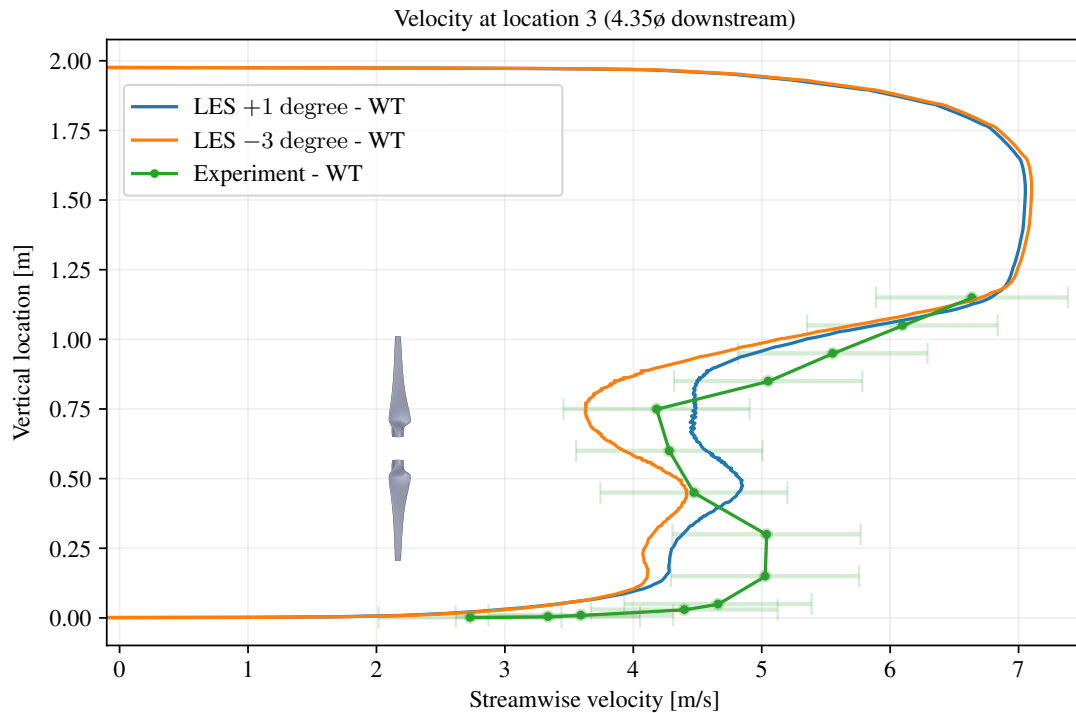
The simulation also shows a good agreement in slope with the experiment in the transition zone between the freestream and the wake, where tip vortices are most prominent, particularly at a height of 1 m. The accuracy of the ALM in capturing this slope largely depends on the tip spreading. When the tip spread is minimal (i.e., a sharp tip transition), the ALM represents more accurately the blade tip, resulting in a better agreement with the experimental data, as seen in Figure 19. However, achieving this requires a sufficiently refined mesh near the tip of the blade. In the velocity deficit induced by the blades, the case with  $+1^\circ$  appears to agree better near the tip, while the case with  $-3^\circ$  appears to agree better near the root of the blade. This trend is observed both on the upper region (0.6 m to 1 m) and the lower region (0.2 m to 0.6 m).

To quantitatively compare each numerical result with the experimental data, the metrics described in Section 2.1.2 are employed, as presented in Table 1. However, this alone does not fully explain the source of the discrepancies. To better understand them, the error is broken down into two categories: positional errors (vertical) and velocity errors (horizontal).



These are illustrated in Figure 20 where it appears that most of the errors are in velocity, except around the height of 1 m where  
 405 the curve shows mostly errors in position.

Similarly, results are extracted at location 3 and compared in Figure 21. For this position, measurements could not be repeated to reduce the random uncertainties due to the limited availability of the wind tunnel. Both experimental and numerical



**Figure 21.** Velocity profile comparing the LES simulation using two pitch angles with the experiment at location 3 ( $4.35D$  downstream of the wind turbine).

curves show smoother gradients in velocity, as expected due to the enhanced mixing of the wake at this further downstream position. As a result, the slope of the transition between the free flow and the wake is not as sharp. In the lower region  $< 0.4$  m,  
 410 a noticeable difference is observed between the experiment and both numerical cases. This is likely due to the change in ground roughness in the wind tunnel, which is not accounted for in the simulation. In the bulk of the wake, there is a difference of  $\approx 0.9 \text{ ms}^{-1}$  between the two numerical cases, and how well these compare with the experimental data remains unclear due to the large uncertainty in the measurement.

## 5 Deformation of the blades

415 The deformation of wind turbine blades plays a crucial role in the overall aerodynamics and performance of the turbine. Under the influence of aerodynamic forces, gravitational loading, and centrifugal forces, the blades experience elastic deformations



**Table 2.** Errors for the velocity profile at location 3.

| Case | Velocity RMSE          | Velocity MAPE | Position RMSE | Position MAPE |
|------|------------------------|---------------|---------------|---------------|
| +1   | $0.38 \text{ ms}^{-1}$ | 6.1 %         | 0.04 m        | 8.8 %         |
| −3   | $0.47 \text{ ms}^{-1}$ | 8.2 %         | 0.05 m        | 11.5 %        |

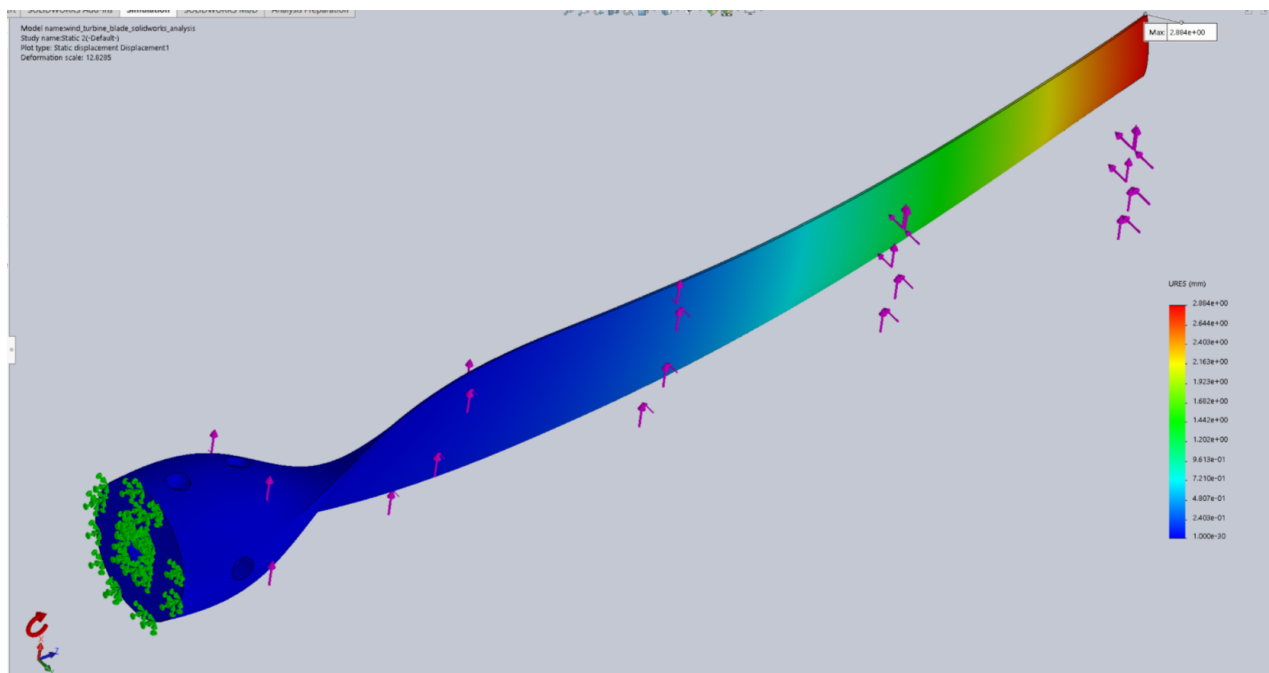
that can significantly alter the flow characteristics around the turbine. This deformation can affect key parameters such as angle of attack, lift and drag of the blades sections, leading to changes in the aerodynamic efficiency and power production of the turbine. In the case studied here, the experimental wind turbine has printed blades made of plastic (PLA) that is highly flexible  
 420 and hence can lead to a different flow behavior compared to a rigid configuration. The methodology used to explore blade deformation of the current experimental model uses both numerical simulation and experimental comparison.

### 5.1 Numerical approach: structural solver

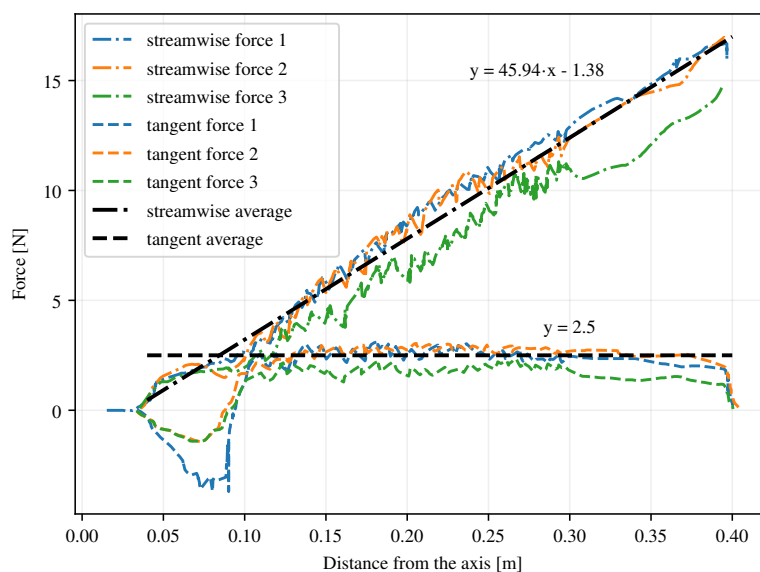
Starting with the numerical approach, we employ a structural solver to translate the aerodynamic loads applied to the wind turbine blade into resulting deformations. The solver calculates how the blade deforms under a defined loading condition. The  
 425 primary goal is to determine the deformation in the flapwise direction (perpendicular to the plane of rotation), the spanwise direction (in the direction of blade rotation), and the combined out-of-plane component of twist, which arises from the interplay of both flapwise and spanwise bending.

The applied load includes the aerodynamic lift and drag on each blade section, a centrifugal force due to the rotation (which is also significant considering the rotational speed). The component of gravity is here negligible. Based on previous work on  
 430 the topic for large wind turbine in turbulent wind with unsteady loads (Trigaux et al., 2024a), it appears that approximating the flow behaviour using the average aerodynamic load already provides good results for the blade mean deformation. In this study, we therefore use the average aerodynamic loads to compute the blade deformation.

The distribution of forces along the blade could not be obtained experimentally, so the values are exported from the previously shown LES. The distribution of forces along the blades is shown in Figure 23 for the axial (streamwise) direction,  
 435 and the tangential (rotation) direction. The axial forces increase linearly with radius, while the tangential forces remain relatively constant. This is a result of the blade design, and is expected to showcase this behavior. Each curve represents an individual blade; one of them experiences a lower amplitude of the force applied because it is in the lower part of the boundary layer where the flow velocity is lower. Some fluctuations due to the flow turbulence are also noticeable along the blades. Another particular aspect visible here is that, at 0.4 m, the loading stops abruptly: this is not typical of most actuator line  
 440 methods, and is again due to the way the ALM was implemented and which allows a sharp transition (when the mesh is refined enough at the tip). This provides a better flow representation of the tip vortex, and it reduces the error on the computation of the power produced. For simplicity, the distribution of forces is approximated using the two lines in Figure 23, which then represents the average load used in the structural solver.



**Figure 22.** 3D CAD of the deformed blade, with indications for the forces applied to it.



**Figure 23.** Distribution of forces along the blade in the stationary frame of reference.





Those forces are applied to the blade geometry, resulting in a deformation exposed in Figure 22. In this figure, the arrows represent the forces applied, with the vertical ones representing the constant tangential forces, the diagonal ones representing the axial forces, and the red arrow at the base indicated the rotation of the blade inducing a centrifugal force. The gravity component is ignored. The fixture of the part is represented with green arrows, for the anchor point.

The geometry used for the structural solver is the CAD used for printing the blade. The material properties used in the simulation are provided from Ltd (2021); Ferreira et al. (2017). There are however some differences between with the 3D printed blade. The blade was printed vertically, starting at the root. The outer layer, shaping the aerodynamic profile, is printed from a continuous filament, with the goal of having a better surface for aerodynamics. With this printing method, the inside of the blade consists of a honeycomb pattern, which is more compliant than a solid block. It is thus expected that the physical blade deforms a bit more than the simulated one.

From the view Figure 22, it appears clearly that the blade is deforming mostly in the flapwise direction, which is as expected since this is the direction where the geometry is most compliant and since it is also the direction where most of the axial force is applied. The maximum deflection of the blade is at the tip, where a deformation of 2.88 mm is predicted.

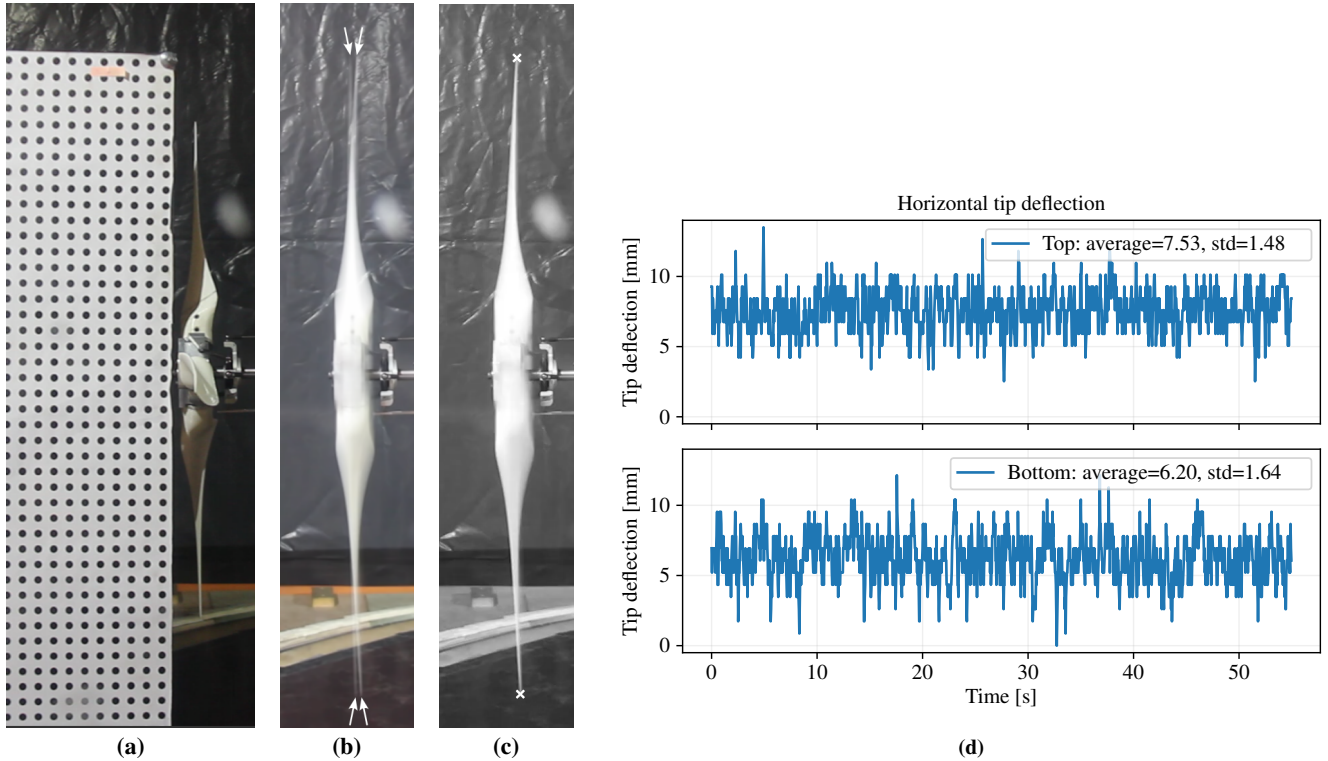
## 5.2 Experimental approach: camera measurement

A measurement of the experimental deformation of the blades provides a valuable insight, complementing the numerical prediction. Indeed, a significant uncertainty from the structural solver prediction comes from the fact the real blade is partly hollow, hence it deforms more than the simulation results. The goal of this experiment is to measure the flapwise deflection at the tip and compare to the simulation.

For this study, the wind turbine was placed in the wind tunnel, as described in Section 2, with a camera placed outside of the wind tunnel, perpendicular to the plane of rotation. First, a calibration pattern was placed in front of the wind turbine, as shown in Figure 24-a. This calibration pattern is placed at the center of the wind turbine to avoid distortion due to perspective. The wind turbine was then rotated slowly by hand, without any flow, to record a baseline (location of the blade tip when not loaded). The wind turbine is then operated with the wind tunnel turned on, as described in Section 2. The camera recorded a video with the blades loaded in their defined operating condition. In Figure 24-b, the two videos are overlaid. A ghosting effect is used on the unloaded case to show the blade tips at the top and bottom in their initial state, and a single image from the video of the loaded case is used to show the deformed state. The initial state is more faint, but outlined by an arrow on the image.

A detection algorithm was developed to measure the position of the blade tip at the top and at the bottom. A snapshot of the resulting video with the detected tips is shown in Figure 24-c, where an “x” indicates the detected tip. The algorithm for computing the blade deflection consists in the following steps:

- Convert the video to black&white, and increase the contrast such that the white blade is clearly identifiable on the black background.
- A horizontal line is used at the height of each tip (top and bottom) to restrict the detection zone. An additional restriction zone in the horizontal direction is used to avoid outliers of a reflection in the background.



**Figure 24.** Measurement of the blade deflection. (a): picture of the calibration of the camera. (b): overlay shown of the unloaded and loaded blades. (c): frame shown with the blade tip detection. (d): deflection of the tip at the bottom and at the top shown as a time series.

- On those two lines, the maximum brightness point determines the position of the blade.

This algorithm is applied on each frame. Finally, a manual check is done on the video to ensure that the algorithm correctly identified the tip at each frame.

At each frame, a distance is obtained from the unloaded point to the tracked point of the frame: this distance is in pixels. The calibration pattern then allows to convert from pixel to millimeter to quantify the deflection of the tip in the flapwise direction. The evolution of this deflection is provided in Figure 24-d. We see that the mean blade tip deflection is 7.5 mm at the top, and 6.2 mm at the bottom (a bit less, as less loaded); which is significantly more than the predicted 2.9 mm obtained when assuming that the blade is not partly hollow. Yet, it amounts to 2.0 – 2.4 % of the blade active span (which is  $\simeq 30$  cm); which is still moderate even though the PLA+ material used for the blade is quite flexible.

## 6 Discussion

The comparison between experimental measurements and LES using the ALM reveals both the strengths and the limitations in reproducing the loading and near wake dynamics of a model-scale wind turbine. In the results presented in Section 4, the wake at location 2 ( $1.41 D$  downstream, see Figure 19) exhibits more variation in the vertical direction as compared to the



490 location 3 ( $4.35 D$  downstream, see Figure 21), as expected. As location 2 is closer to the wind turbine and is more challenging to reproduce numerically, it was the main focus in this investigation. At this location, the numerical results capture well the intricate wake structure, with a notable agreement in the overall velocity deficit. This agreement is also to be attributed to the accurate reproduction of the inlet conditions (Section 3.2), which play a crucial role in shaping the near-wake behavior.

A key strength of the ALM implemented in this study, and described in Section 3.3, is its ability to replicate the sharp  
495 velocity gradient between the free stream ( $> 1.1$  m) and the wake ( $< 1$  m). This is made possible by disabling the tip force spreading in the ALM and by using a mesh that is sufficiently fine there. In the error quantification shown in Figure 20, this region appears with a deviation in height of 7.5 mm. While this error is greater than the measurement uncertainty, it remains small enough for most applications.

The multiple changes in the derivative around hub height (0.6 m) are also fairly well captured, highlighting the influence of  
500 the nacelle-induced drag and the limited energy extraction by the blade root.

However, a notable deviation near the blade root (0.7 m) underscores a key limitation of the ALM: the assumption that 2D airfoil polars can adequately represent the blade aerodynamics. In this region, the complex geometrical transition (from blade airfoil to ellipse to circle) and 3D nature of the flow invalidates this assumption, leading to discrepancies between simulation and experiment. This limitation is critical when analyzing the near-wake region.

505 The influence of the blade pitch angle was also investigated. Near the tip, the  $+1^\circ$  pitch case aligns more closely with the experimental near wake data while, near the root, the  $-3^\circ$  pitch case aligns better. This suggests that the ALM, which solely uses the airfoil chord direction for each blade section together with the local angle of attack (as measured on the aerodynamic center) and velocity amplitude, and also assumes a 2-D cross-sectional flow at all sections and uses a simple airfoil polar, still has difficulties to completely reproduce the experimental blade loading. It is however not possible to conclude here, as the  
510 experimental blade loading distribution was not measured.

The blade flapwise deformation was analyzed in Section 5. Experimentally, the tip deflection was moderate (6.2 mm bottom, 7.5 mm top), influenced by the sheared inflow profile. The structural model predicted an even smaller deflection of 2.9 mm, which is most likely due to the solid blade assumption in the simulation, whereas the experimental blade was partly hollow with a honeycomb structure. We also stress that PLA+ is a plastic that has more flexibility than a composite material. Still, the  
515 observed deformation amounts to only 2.0 – 2.4 % of the blade active span (which is  $\simeq 30$  cm).

## 7 Conclusion

In this study, we designed and constructed the TWIST (Turbine for Wind-tunnel Investigation & Scaled Testing) and tested it within the controlled environment of a wind tunnel. In parallel, we implemented an improved ALM in Nek5000, incorporating specific capabilities such as the ability to reduce the tip force spreading and avoid radial averaging. As described in Section 3.2,  
520 particular care was also taken to reproduce in the LES solver the experimental inlet flow conditions as accurately as possible, and for both the mean velocity profile and the turbulence intensity, so as to ensure a meaningful comparison between the experimental and the numerical results.



The comparison showed that the ALM approach is capable of capturing the essential features of the near-wake region, where the flow structures are highly sensitive to the modeling fidelity. At  $1.41 D$  downstream, both the experimental and the numerical results showed a complex velocity profile with significant vertical variation. The overall velocity deficit was fairly well captured, and the sharp transition between the free stream and the wake was also properly reproduced, due to the fine mesh resolution and the ability of the ALM to capture the blade tip effects with limited artificial smoothing. The effect of the hub was also modeled (using a drag force applied within an ellipsoid) as well as the blade root region. However, deviations near the blade root persist, due to the intrinsic limitations of the ALM which relies on 2-D airfoil polars, whereas the blade root geometry transitions from airfoil to ellipse to circle very rapidly so that the real flow is complex and 3-D in that region.

A sensitivity study on the blade pitch angle was also carried out in the LES-ALM. The blade deflection was also investigated numerically, and measured experimentally. It was measured as  $2.0 - 2.4 \%$  of the blade active span, which is still moderate even though the PLA+ material used for the blade is quite flexible. It is unlikely that such limited flapwise deformation has a significant effect on the blade loading, or that there is some added twist (due to some structural bend-twist coupling). It would be interesting to print new blades of identical geometry but using a stiffer material, and to test those. This is left to future work.

Overall, the results demonstrate the potential of LES combined with a refined ALM to model wind turbines, and to also produce the associated complex near wake dynamics. There is however still room for improvement. For instance, the airfoil polar used so far in the ALM could be refined, using experimental data or refined CFD conducted at the same  $Re_c \simeq 5.0 \times 10^4$ . A stiff blade could be manufactured, and also instrumented to measure the blade loading distribution. This would allow for a better comparison with the results obtained using LES with ALM. The inflow field representation used in the LES could also be further improved for its turbulence content (the mean flow was already near perfect).

#### . Code and data availability

The LES were performed using Nek5000 (Argonne National Laboratory, 2023), an open-source code for computational fluid dynamics. The implementation of the actuator line model used in this study has been developed within the Nek5000 framework and is available from the authors upon reasonable request. Similarly, the datasets generated and analyzed during the current study can be obtained from the authors upon request.

#### . Author contribution

EG: conceptualization (equal), data curation (lead), formal analysis (lead), investigation (lead), methodology (lead), software (lead), validation (lead), visualization (lead), writing – original draft preparation (lead), writing – review and editing (lead). SB: conceptualization (lead), formal analysis (equal), funding acquisition (equal), methodology (equal), project administration (lead), resources (equal), supervision (lead), writing – review and editing (equal). JvB: conceptualization (equal), funding acquisition (equal), methodology (equal), resources (equal), supervision (equal), writing – review and editing (equal). GW: conceptualization (equal), methodology (equal), supervision (equal), writing – review and editing (equal).



. Competing interests

555 The authors declare that they have no conflict of interest.

. Acknowledgements

The authors would like to thank Lorenzo Schena for performing the OpenFAST simulation shown in Figure 6, and for evaluating the performance of the wind turbine by measuring the power curve shown in Figure 7.



## References

- 560 Argonne National Laboratory: Nek5000 Theory, <https://nek5000.github.io/NekDoc/>, [Online; accessed 16-April-2024], 2023.
- Barlas, E., Buckingham, S., and van Beeck, J.: Roughness Effects on Wind-Turbine Wake Dynamics in a Boundary-Layer Wind Tunnel, *Boundary-Layer Meteorology*, 158, 27–42, <https://doi.org/10.1007/s10546-015-0083-z>, 2015.
- Bastankhah, M. and Porté-Agel, F.: A New Miniature Wind Turbine for Wind Tunnel Experiments. Part II: Wake Structure and Flow Dynamics, *Energies*, 10, 923, <https://doi.org/10.3390/en10070923>, 2017.
- 565 Boorsma, K. and Schepers, J.: Rotor experiments in controlled conditions continued: New Mexico, *Journal of Physics: Conference Series*, 753, 022 004, <https://doi.org/10.1088/1742-6596/753/2/022004>, 2016.
- Chamorro, L., Arndt, R., and Sotiropoulos, F.: Reynolds number dependence of turbulence statistics in the wake of wind turbines, *Wind Energy*, 15, 733–742, <https://doi.org/10.1002/we.501>, 2011.
- Conan, B.: Wind resource assessment in complex terrain by wind tunnel modelling, Ph.D. thesis, Orléans University, <http://www.theses.fr/2012ORLE2067>, 2012.
- 570 Coudou, N., Buckingham, S., Bricteux, L., and van Beeck, J.: Experimental Study on the Wake Meandering Within a Scale Model Wind Farm Subject to a Wind-Tunnel Flow Simulating an Atmospheric Boundary Layer, *Boundary-Layer Meteorology*, 167, 77–98, <https://doi.org/10.1007/s10546-017-0320-8>, 2017.
- Deville, M. O., Fischer, P. F., and Mund, E. H.: High-Order Methods for Incompressible Fluid Flow, *Cambridge Monographs on Applied and Computational Mathematics*, Cambridge University Press, <https://doi.org/10.1017/CBO9780511546792>, 2002.
- 575 Doubrawa, P., Quon, E. W., Martinez-Tossas, L. A., Shaler, K., Debnath, M., Hamilton, N., Herges, T. G., Maniaci, D., Kelley, C. L., Hsieh, A. S., Blaylock, M. L., Laan, P., Andersen, S. J., Krueger, S., Cathelain, M., Schlez, W., Jonkman, J., Branlard, E., Steinfeld, G., Schmidt, S., Blondel, F., Lukassen, L. J., and Moriarty, P.: Multimodel validation of single wakes in neutral and stratified atmospheric conditions, *Wind Energy*, 23, 2027–2055, <https://doi.org/10.1002/we.2543>, 2020.
- 580 Ferreira, R. T. L., Amatte, I. C., Dutra, T. A., and Bürger, D.: Experimental characterization and micrography of 3D printed PLA and PLA reinforced with short carbon fibers, *Composites Part B: Engineering*, 124, 88–100, <https://doi.org/10.1016/j.compositesb.2017.05.013>, 2017.
- Fischer, P. F.: Nek5000 users guide, <https://www.mcs.anl.gov/~fischer/Nek5000/examples.pdf>, [Online; accessed 18-June-2021], 2013.
- Geuzaine, C. and Remacle, J.-F.: Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities, *International Journal for Numerical Methods in Engineering*, 79, 1309–1331, 2009.
- 585 Gillyns, E., Buckingham, S., Van Beeck, J., and Winckelmans, G.: Wind tunnel inflow data assimilation in an LES digital twin for improving validation against an experimental wind turbine wake, *Journal of Physics: Conference Series*, 2767, 092 040, <https://doi.org/10.1088/1742-6596/2767/9/092040>, 2024.
- Han, Y., Wang, J., Li, X., Jin, K., Yang, B., Dong, X., and Wen, C.: Experimental study of turbulence intensity on the wake characteristics of a horizontal-axis wind turbine, *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, 44, 9545–9563, <https://doi.org/10.1080/15567036.2022.2128939>, 2022.
- 590 Hodgson, E. L., Madsen, M. H. A., and Andersen, S. J.: Effects of turbulent inflow time scales on wind turbine wake behavior and recovery, *Physics of Fluids*, 35, <https://doi.org/10.1063/5.0162311>, 2023.
- Huang, X., Yang, J., Gao, Z., Sha, C., and Yang, H.: Output Power and Wake Flow Characteristics of a Wind Turbine with Swept Blades, *Machines*, 10, 876, <https://doi.org/10.3390/machines10100876>, 2022.
- 595



- Jha, P. K. and Schmitz, S.: Actuator curve embedding – an advanced actuator line model, *Journal of Fluid Mechanics*, 834, <https://doi.org/10.1017/jfm.2017.793>, 2017.
- Jha, P. K., Churchfield, M. J., Moriarty, P. J., and Schmitz, S.: Guidelines for Volume Force Distributions Within Actuator Line Modeling of Wind Turbines on Large-Eddy Simulation-Type Grids, *Journal of Solar Energy Engineering*, 136, <https://doi.org/10.1115/1.4026252>, 2014.
- Jin, W.: Numerical Simulation of Wind Turbine Wakes based on Actuator Line Method in NEK5000, Master's thesis, KTH Mechanics, 2013.
- Kleusberg, E.: Wind turbine simulations, Master's thesis, KTH Mechanics, 2017.
- Ltd, I. B. L.: Polylactic Acid (PLA, Polylactide), <https://www.makeitfrom.com/material-properties/Polylactic-Acid-PLA-Polylactide/>, [Online; accessed 13-August-2024], 2021.
- Martínez-Tossas, L. A. and Meneveau, C.: Filtered lifting line theory and application to the actuator line model, *Journal of Fluid Mechanics*, 863, 269–292, <https://doi.org/10.1017/jfm.2018.994>, 2019.
- Martínez-Tossas, L. A., Churchfield, M. J., and Meneveau, C.: Optimal smoothing length scale for actuator line models of wind turbine blades based on Gaussian body force distribution, *Wind Energy*, 20, 1083–1096, <https://doi.org/10.1002/we.2081>, 2017.
- McTavish, S., Feszty, D., and Nitzsche, F.: An experimental and computational assessment of blockage effects on wind turbine wake development, *Wind Energy*, 17, 1515–1529, <https://doi.org/10.1002/we.1648>, 2013.
- Munson, B., Rothmayer, A., and Okiishi, T.: *Fundamentals of Fluid Mechanics*, 7th Edition, Blackwell handbooks in linguistics, Wiley, ISBN 9781118214596, <https://books.google.be/books?id=GQMCAAAQBAJ>, 2012.
- Patera, A. T.: A spectral element method for fluid dynamics: Laminar flow in a channel expansion, *Journal of Computational Physics*, 54, 468–488, [https://doi.org/10.1016/0021-9991\(84\)90128-1](https://doi.org/10.1016/0021-9991(84)90128-1), 1984.
- Schepers, J., Boorsma, K., Cho, T., Gomez-Iradi, S., Schaffarczyk, A., Jeromin, A., Shen, W. Z., Lutz, T., Meister, K., Stoevesandt, B., Schreck, S., Micallef, D., Pereira, R., Sant, T., Madsen, H., and Sørensen, N.: Final report of IEA Task 29, Mexnext (phase 1): analysis of MEXICO wind tunnel measurements, in: Report, p. 1, *Wind Energy*, 2014.
- Schmitz, S.: *Aerodynamics of wind turbines: a physical basis for analysis and design*, Wiley, Hoboken, ISBN 9781119405610, 2019.
- Spyropoulos, N., Papadakis, G., Prospathopoulos, J. M., and Riziotis, V. A.: Investigating the Level of Fidelity of an Actuator Line Model in Predicting Loads and Deflections of Rotating Blades under Uniform Free-Stream Flow, *Applied Sciences*, 11, 12097, <https://doi.org/10.3390/app112412097>, 2021.
- Stanly, R., Martínez-Tossas, L. A., Frankel, S. H., and Delorme, Y.: Large-eddy simulation of a wind turbine using a filtered actuator line model, *Journal of Wind Engineering and Industrial Aerodynamics*, 222, 104868, <https://doi.org/10.1016/j.jweia.2021.104868>, 2022.
- Trigaux, F., Chatelain, P., and Winckelmans, G.: Scale-resolved aero-servo-elastic simulation of large wind turbines using the actuator line, Ph.D. thesis, UCLouvain, <http://hdl.handle.net/2078.1/289594>, 2024a.
- Trigaux, F., Chatelain, P., and Winckelmans, G.: Impact of rotor size on the aeroelastic behavior of large turbines: a LES study using flexible actuator lines, *Journal of Physics: Conference Series*, 2767, 022062, <https://doi.org/10.1088/1742-6596/2767/2/022062>, 2024b.
- Trigaux, F., Villeneuve, T., Dumas, G., and Winckelmans, G.: Near-tip correction functions for the actuator line method to improve the predicted lift and drag distributions, *Journal of Fluid Mechanics*, 989, <https://doi.org/10.1017/jfm.2024.461>, 2024c.
- Troldborg, N.: Actuator line modelling of wind turbine wakes, Ph.D. thesis, Technical University of Denmark, 2008.
- Vugrin, K. W., Swiler, L. P., Roberts, R. M., Stucky-Mack, N. J., and Sullivan, S. P.: Confidence region estimation techniques for nonlinear regression in groundwater flow: Three case studies, *Water Resources Research*, 43, <https://doi.org/10.1029/2005wr004804>, 2007.





- Wang, Z., Fidkowski, K., Abgrall, R., Bassi, F., Caraeni, D., Cary, A., Deconinck, H., Hartmann, R., Hillewaert, K., Huynh, H., Kroll, N., May, G., Persson, P.-O., van Leer, B., and Visbal, M.: High-order CFD methods: current status and perspective, *International Journal for Numerical Methods in Fluids*, 72, 811–845, <https://doi.org/10.1002/fld.3767>, 2013.
- 635 Xiao, F., Dianat, M., and McGuirk, J. J.: An LES Turbulent Inflow Generator using A Recycling and Rescaling Method, *Flow, Turbulence and Combustion*, 98, 663–695, <https://doi.org/10.1007/s10494-016-9778-6>, 2016.