

# CFD Analysis on Layerwise Modelling of Wind Turbine Blade by Viscoelastic Material Over Composite Material

Mr D Siva Akash,  
PG Scholar, Department of Mechanical Engineering,  
Government College of Technology,  
Coimbatore, India.

Dr N Nandakumar, Professor,  
Department of Mechanical Engineering,  
Government College of Technology,  
Coimbatore, India.

**Abstract** - Enhancing the efficiency of wind turbines largely depends on improving the aerodynamic performance of their blades. Composite materials are widely preferred for blade fabrication because of their high strength, low weight, and excellent fatigue life. Applying a viscoelastic coating on the blade surface can modify the airflow pattern, reduce drag, and delay flow separation, which collectively improve aerodynamic stability. The present study focuses on conducting a Computational Fluid Dynamics (CFD) analysis of airflow over a composite wind turbine blade treated with a viscoelastic surface layer. The blade geometry, derived from airfoil sections such as DU series, will be designed using CAD modeling tools and analyzed through ANSYS Fluent. A fine mesh will be generated close to the blade wall to resolve boundary-layer flow accurately. Simulations will be performed at different wind speeds using turbulence models like SST  $k-\omega$ . The outcomes are expected to reveal how viscoelastic coatings influence lift, drag, and pressure distribution, leading to improved aerodynamic performance and reliability of composite wind turbine blades.

**Keywords** - Wind energy, Wind turbine blade, CFD, Composite blade, Aeroelastic analysis, Viscoelastic material.

## I. INTRODUCTION

Industrialized nations have committed to reducing greenhouse gas emissions by promoting the development and deployment of environmentally sustainable alternative energy sources. This global initiative has significantly accelerated research interest and investment in renewable energy technologies as viable, low-impact solutions to meet future energy demands while mitigating climate change [1]. A detailed study on a hybrid control methodology for an aero-servo-viscoelastic wing model. Their research integrates passive viscoelastic damping with active vibration suppression using piezoelectric actuators operated through an optimal control algorithm. The combined technique effectively reduces flutter, oscillations, and structural deformation in flexible aircraft wings. Results from the simulation and experimental validation show that the hybrid system provides superior damping and stability compared to individual control schemes. Moreover, it enhances energy efficiency and adaptability under varying aerodynamic loads. The study emphasizes that employing both control mechanisms leads to improved aeroelastic performance, reduced fatigue, and extended service life of lightweight aircraft structures. This work contributes significantly to the development of advanced aerospace

control systems for safer and more reliable flight operations [2].

A comprehensive study on active aero-visco-elastic flutter suppression and layerwise modelling for supersonic smart sandwich panels. Their work integrates piezoelectric actuators and sensors, viscoelastic damping materials, and variable stiffness composite laminates to enhance flutter resistance and vibration control in lightweight aerospace structures. By employing a layerwise high-order shear deformation approach, the study investigates proportional and derivative feedback control systems for both elastic and viscoelastic panel configurations. The findings reveal that proportional feedback performs better in mitigating coupled-mode flutter, whereas derivative control improves stability in single-mode flutter cases. The model exhibits strong predictive accuracy for thin and moderately thick panels. Overall, the research highlights that hybrid active-passive control combined with curvilinear fiber designs significantly boosts aeroelastic performance, damping efficiency, and reliability of smart aerospace structures subjected to supersonic aerodynamic loads [3].

comprehensive investigation into the aerodynamic characteristics of floating offshore wind turbines (FOWTs) through advanced computational fluid dynamics (CFD) simulations. Their work explored how wave-induced platform motions, particularly pitch and surge displacements, influence aerodynamic loads and rotor performance. Using the NREL 5MW reference turbine model, they analyzed variations in thrust, generated power, and wake development under dynamic sea conditions. The findings indicated that while the average thrust was only slightly affected, aerodynamic power increased during forward motion phases due to enhanced inflow velocity. The study also highlighted the emergence of complex unsteady phenomena, such as vortex interactions, wake meandering, and flow instabilities, which conventional prediction tools like the blade element momentum (BEM) theory could not accurately capture. Overall, the research demonstrates the importance of CFD-based high-fidelity modelling in enhancing aeroelastic stability, performance prediction, and design optimization of modern floating wind turbine systems [4].

The primary objective of this research is to evaluate the synergistic effects of integrating a viscoelastic damping layer

within a multi-layer composite wind turbine blade through high-fidelity CFD and structural modeling. This study aims to bridge the gap between advanced material science and aerodynamic efficiency by implementing a layerwise modeling technique that accurately captures the interlaminar shear stresses often overlooked in simplified structural models. By utilizing Computational Fluid Dynamics, the project seeks to quantify the aeroelastic response of the blade, specifically focusing on how the energy-dissipative properties of the viscoelastic core mitigate structural vibrations induced by turbulent wind loads. Furthermore, this analysis serves to establish a performance benchmark, comparing the hybrid viscoelastic-composite architecture against conventional composite designs to determine its viability in extending the fatigue life of the turbine and maintaining aerodynamic stability under extreme operational conditions.

## II. METHODOLOGY

Methodology refers to the systematic approach, techniques, and procedures used in conducting research or solving a problem, it outlines the framework, tools, and strategies employed to collect, analyse, and validate information, ensuring scientific accuracy and reliability. A well-defined methodology ensures that research findings are based on logical, repeatable processes, allowing others to replicate or verify the results. Here the methodology followed.

The methodology adopted for this project on the design and analysis of a flying car follows a structured, simulation-driven engineering approach that integrates conceptual design, computational modelling, and performance optimization. The process began with the development of a 3D CAD model that encapsulated a streamlined fuselage, rotor configurations, and energy-efficient blade architecture. This model was then subjected to aerodynamic evaluation, using Computational Fluid Dynamics (CFD) simulations in ANSYS Fluent, where pressure distribution, power generated, and rotor-induced airflow were analysed under defined boundary conditions and turbulence models.

A fine computational mesh was developed to accurately capture near-wall flows and rotor blade interactions. Simultaneously, structural integrity was ensured through material selection and Finite Element Analysis (FEA), focusing on load bearing capacity and stress distribution. Post-processing of simulation data provided key parameters such as pressure coefficient ( $C_p$ ), viscous force, and total aerodynamic force, which were compared with theoretical benchmarks for validation. Iterative design modifications were made to improve lift efficiency and minimize drag, resulting in a refined flying car model that satisfies both aerodynamic and structural performance criteria. This methodology not only enables effective lift generation but also provides a scalable framework for future prototype development.

The below diagram is the representation of flow chart of one CFD analysis of wind turbine blade.

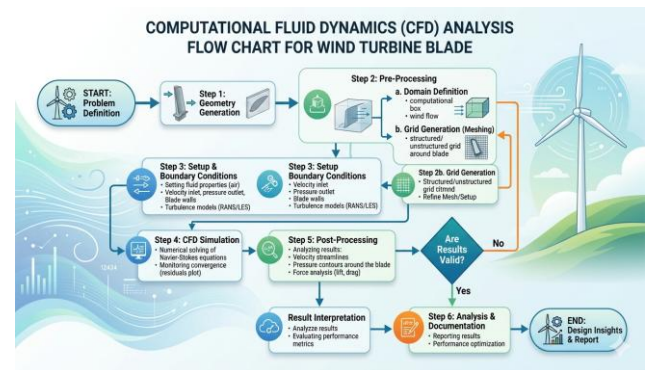


Fig. 1. Flowchart of CFD analysis of wind turbine blade

## III. DESIGN ON LAYERWISE MODELLING OF 5MW/61.5M WIND TURBINE BY VISCOELASTIC MATERIAL OVER COMPOSITE BLADE

SolidWorks 2023, developed by Dassault Systèmes, is an advanced computer-aided design (CAD) software widely utilized across industries such as mechanical engineering, product development, and architecture. It offers an extensive range of tools for creating precise 3D models, performing simulations, generating visualizations, and producing technical documentation. The software enhances design accuracy and efficiency through its user-friendly interface and integrated design-to-manufacturing workflow. SolidWorks 2023 also supports collaborative design, allowing engineers and designers to optimize product performance while reducing development time and errors.

### 1. Airfoil data

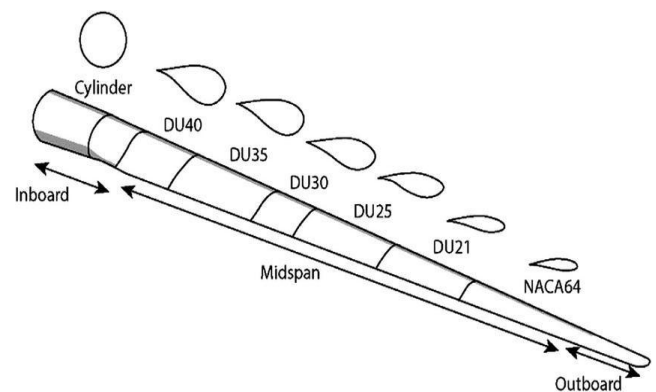


Fig 2 Schematic representation of NREL 5MW blade , showing airfoil and main sections [11]

The Fig 4.1 shows the aerodynamic structure of a wind turbine blade, demonstrating how airfoil shapes change along its span. At the inboard section, the blade starts with a cylindrical root that provides mechanical strength and easy hub connection. As it moves outward, the blade transitions through various DU airfoil profiles (DU40, DU35, DU30, DU25, DU21), each designed to improve lift and minimize drag at specific radial locations. Near the outboard region, a NACA64 airfoil is used for greater aerodynamic efficiency and lower thickness. This

smooth 19 variation in profiles ensures optimal performance and structural stability throughout the blade.

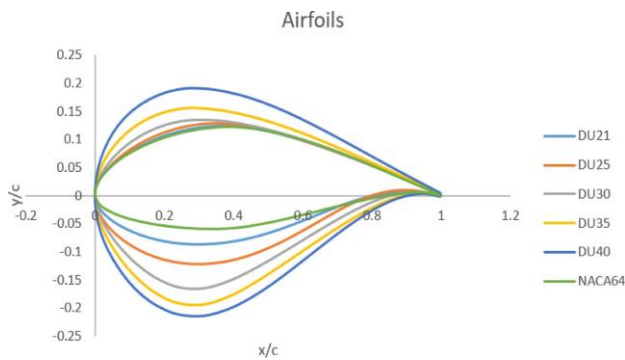


Fig 3 Range of airfoils used in blade in comparison to each other

The image compares several airfoil profiles typically used in wind turbine blades, including DU21, DU25, DU30, DU35, DU40, and NACA64. The plot represents the normalized coordinates ( $x/c$  and  $y/c$ ), highlighting variations in shape, curvature, and thickness across the chord. The DU-series airfoils have a thicker leading edge and greater camber, offering high lift and better structural strength, making them suitable for the inner sections of the blade where loads are higher. Moving outward, the blade transitions to the NACA64 profile, which has a thinner and smoother contour designed to reduce drag and improve aerodynamic efficiency. This gradual change in airfoil geometry from inboard to outboard regions ensures a proper balance between strength and performance. The figure effectively demonstrates how different airfoil sections are combined to optimize the wind turbine blade's efficiency, stability, and overall aerodynamic characteristics under varying wind and operational conditions.

## 2. SPECIFICATION

The table 4.2 shows the specification of the NREL 5MW/61.5m wind turbine blade model which is essential for calculating the loading condition of the wind turbine.

TABLE I specification of NREL 5MW/61.5m wind turbine blade model

Sl.No	Details	Value
1	Rated Power	5MW
2	Rotor Diameter	126m
3	Hub Height	90m
4	Cut-in wind speed	3m/s
5	Rated wind speed	11.4m/s
6	Rated rotor speed	12.1rpm
7	Cut-out wind speed	27m/s
8	Blades	3
9	Drivetrain	High-speed , multi-stage gearbox
10	Generator	Synchronous permanent magnet

## 3. CAD MODELING FOR 5 MW WIND TURBINE

The table 4.2 used to design the CAD modeling for 5 MW wind turbine. The data we are going to use in the current study is slightly modified form made by Brian R. Resor. This design also assumes the 0.4 m distance instead of 0.375 from leading edge for the pitch axis.

TABLE II CAD Modelling data

Station No.	Span location (Radius) m	Aero Twist ( $\phi$ )	Chord (m)	Pitch Axis in m (From leading edge for 1 m chord airfoil)	Airfoil section
1	2.8667	13.308	3.542	0.498774853	Cylinder1
2	5.6000	13.308	3.854	0.471815	Cylinder1
3	8.3333	13.308	4.3167	0.438315742	Cylinder2
4	11.7500	13.308	4.557	0.3964475	DU40_A17
5	15.8500	11.48	4.652	0.375	DU35_A17
6	19.9500	10.162	4.458	0.375	DU35_A17
7	24.0500	9.011	4.249	0.375	DU30_A17
8	28.1500	7.795	4.007	0.375	DU25_A17
9	32.2500	6.544	3.748	0.375	DU25_A17
10	36.3500	5.361	3.502	0.375	DU21_A17
11	40.4500	4.188	3.256	0.375	DU21_A17
12	44.5500	3.125	3.01	0.375	NACA64_A17
13	48.6500	2.319	2.764	0.375	NACA64_A17
14	52.7500	1.526	2.518	0.375	NACA64_A17
15	56.1667	0.863	2.313	0.375	NACA64_A17
16	58.9000	0.37	2.086	0.375	NACA64_A17
17	61.6333	0.106	1.419	0.375	NACA64_A17

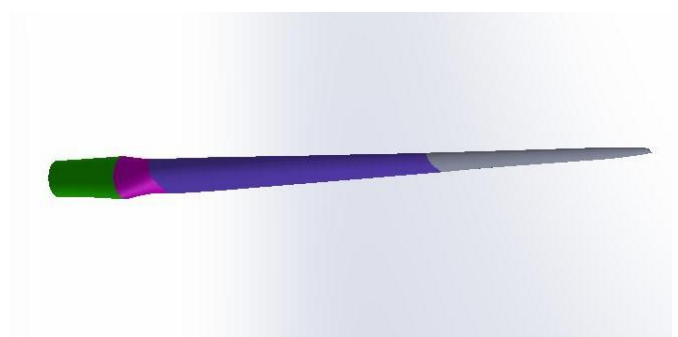


FIG. 4. Single solid blade

## IV. LAYERWISE MODELLING OF 5MW/61.5M WIND TURBINE BY VISCOELASTIC MATERIAL OVER COMPOSITE BLADE

After the composite wind turbine blade is created, the layerwise viscoelastic modelling process begins by defining each ply's geometry, thickness, and orientation within the finite element environment. The composite layers are assigned orthotropic material properties, while thin viscoelastic layers

are added to simulate damping behaviour. These viscoelastic layers are characterized by frequency-dependent shear and bulk moduli. Structural components such as spar caps, webs, and twist angles are accurately incorporated. Residual stresses and operational loads are applied for realistic performance analysis. Finally, modal and harmonic studies are conducted, and viscoelastic parameters are fine-tuned to optimize vibration damping, stability, and fatigue resistance. Fig 4.8 shows layerwise modelling over composite material

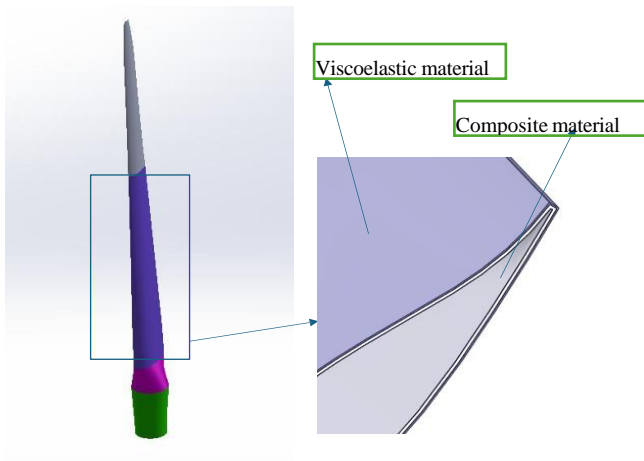


FIG. 5. layerwise modelling over composite material

## V. COMPUTATIONAL FLUID DYNAMICS

### 1. Inner domain and outer domain

In CFD simulations, the inner domain (Fig 5.1) defines the primary area where the actual flow or physical behaviour is analyzed, such as near an airfoil or blade surface. The outer domain (Fig5.2) surrounds this region, providing sufficient space for flow development and preventing boundary interference. Creating both domains carefully ensures reliable

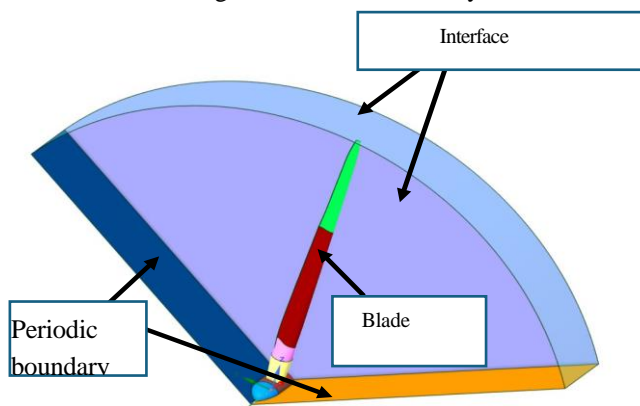


FIG.6. Inner domain rotating reference frame

simulation outcomes, enhances mesh refinement, and maintains numerical accuracy throughout the computational process, leading to more realistic and stable analysis results.

The inner domain and the outer domain has an 120 degree sector in which the inner domain is the rotating reference frame and the outer domain is the stationary reference frame.

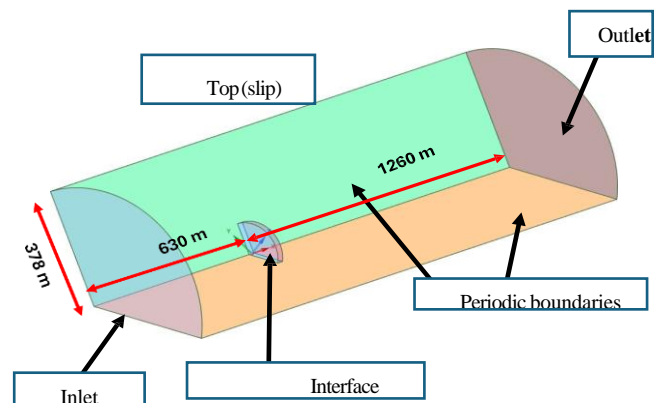


FIG. 7. Outer domain stationary reference frame

### 2. Computational modeling in wind tunnel testing

Computational modelling in wind tunnel testing is a critical approach used to analyse and optimize aerodynamic performance before real-world application. It involves replicating wind tunnel conditions through numerical simulations, allowing engineers to predict airflow behaviour around objects such as vehicles and aircraft. By combining Computational Fluid Dynamics (CFD) with experimental testing, computational models help refine designs, assess pressure distributions, and evaluate lift-to-drag ratios without the need for excessive physical prototypes. The process includes defining the test models such as k-w SST to simulate real airflow conditions accurately. geometry, generating high-resolution computational meshes, and applying turbulence. Fig5.3 shows the Computational modelling in wind tunnel. The inlet and the outlet condition was given according to the problem. The top of the wall is given as symmetry

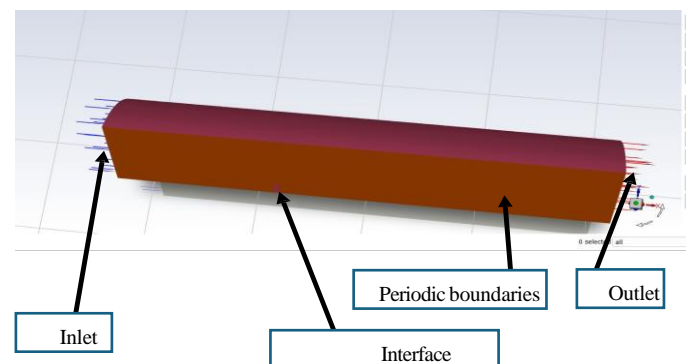


FIG.7. Computational Modelling in Wind Tunnel

### 3. MESH QUALITY AND QUANTITY

Mesh quality and quantity are crucial aspects of Computational Fluid Dynamics (CFD) simulations, directly influencing the accuracy, stability, and efficiency of

numerical results. High-quality mesh ensures smooth airflow calculations, minimizing numerical errors and improving convergence. Key factors that define mesh quality include aspect ratio, skewness, orthogonality, and boundary layer resolution. Low skewness and high orthogonality contribute to better numerical stability, while a well-refined boundary layer mesh accurately captures viscous effects near surfaces. Mesh quantity, on the other hand, refers to the density of grid elements within the computational domain. Excessively coarse meshes may lead to inaccurate results, whereas overly refined meshes increase computational cost without significant improvements.

## VI. STATIC PRESSURE CONDITIONS

The static pressure behaviour on the pressure and suction sides of a wind turbine blade is essential for creating aerodynamic lift and improving energy extraction efficiency. As wind flows around the blade, its shape and angle of attack generate a difference in pressure between both surfaces. The pressure side, which faces the incoming airflow, experiences a higher static pressure because the air slows down upon contact. Conversely, the suction side, located on the opposite face, encounters lower static pressure since the airflow accelerates over its curved surface. This pressure imbalance produces lift that acts perpendicular to the wind direction, enabling the rotor to spin and convert wind energy into mechanical power.

On the pressure side, airflow velocity decreases, increasing static pressure and exerting an inward force on the blade surface. This region contributes slightly to drag but helps maintain blade stability. The pressure side is generally flatter in profile to reduce turbulence and avoid early flow separation. The suction side, however, is more curved to enhance the airflow velocity, lowering the static pressure in accordance with Bernoulli's principle. The resulting suction generates the major portion of lift on the blade. However, excessive angles of attack can cause flow separation from the suction surface, leading to aerodynamic stall and reduced performance.

- Maximum pressure = 3749.934 Pa
- Minimum pressure = -11365.82 Pa
- Difference in pressure = 15115.754 Pa

## VII. POWER GENERATED

The power output of a wind turbine blade is produced by transforming the kinetic energy of wind into rotational mechanical energy through aerodynamic lift and torque generation. When wind flows across the blade surface, it creates a pressure difference that results in torque on the rotor hub. This torque drives the blades to rotate at an angular velocity, represented by (radians per second). The mechanical power developed by the turbine is expressed as, where denotes the torque acting on the main shaft. The magnitude of torque

depends on factors such as wind speed, blade pitch angle, and aerodynamic design. The rotational motion generated is transferred to the generator, where it is converted into electrical energy. By optimizing blade shape, twist, and rotational speed, modern wind turbines maximize torque efficiency, ensuring smooth power generation and improved energy conversion from the available wind resource.

- Turbulence model = SST K-Omega
- For one blade: Torque =  $1.32 \times 10^6 N - m$
- For 3 Blades Torque =  $1.32 \times 10^6 \times 3 = 3.96 \times 10^6 N - m$
- Omega = 12.1RPM = 1.267 Rad/s
- Power = Torque  $\times$  Omega =  $3.45 \times 10^6 \times 1.267$
- Power = 5.017 MW

## VIII. RESULT

- Maximum pressure = 3749.934 Pa
- Minimum pressure = -11365.82 Pa
- Difference in pressure = 15115.754 Pa
- Maximum velocity = 0.02794 m/s
- Minimum velocity = 99.52552 m/s
- POWER GENERATED = 5.017MW

}

## REFERENCES

- [1] S. Shafiei and R. Salim, "A Non-renewable and renewable energy consumption and CO2 emissions in OECD countries: a comparative analysis," *Energy Policy* 66, 547-556 (2014).
- [2] Caroline Lienard, Ronan Boisard, Camille Daudin, 2019, Aerodynamic behavior of a floating offshore wind turbine, hal-02088810.
- [3] H. Nejadkhaki and J.F. Hall, "Control Framework and Integrative Design Method for an Adaptive Wind Turbine Blade," *J. Dyn. Syst. Measurement Control* (2020).
- [4] X. Lachenal, S. Daynes and P.M. Weaver, "Review of morphing concepts and materials for wind turbine blade application," *Wind Energy* 16(2), 283-307 (2013).
- [5] E. Gaertner, J. Rinker, L. Sethuraman, F. Zahle, B. Anderson, G.E. Barter, N.J. Abbas, F. Meng, P. Bortolotti, W. Skrzypinski, G.N. Scott, R. Feil, H. Bredmose, K. Dykes, M. Shields, C. Allen and A. Viselli, "Definition of the IEA 15-Megawatt Offshore Reference Wind" (National Renewable Energy Laboratory, Golden, CO, 2020). NREL/TP-500-75698J.
- [6] C.J. Faccio, "Modeling wind turbine blades by geometrically-exact beam and shell elements: a comparative approach", Master thesis, University of São Paulo, 2017.
- [7] Q. Wang, N. Johnson, M.A. Sprague and J. Jonkman, "BeamDyn: a high-fidelity wind turbine blade solver in the FAST modular framework", *Wind Energ.* (2017).
- [8] D.M. Sommers, "Design and Experimental Results for the S809 Airfoil" (N. p., United States, 1997).
- [9] R.R. Ramsay, M.J. Hoffman and G.M. Gregorek, "Effects of Grit Roughness and Pitch Oscillations on the S809 Airfoil" (N. p., United States, 1995).

- [10] R.R. Ramsay, M.J. Hoffman and G.M. Gregorek, "Effects of Grit Roughness and Pitch Oscillations on the S809 Airfoil" (N. p., United States, 1995).
- [11] John H. Chujutalli, Guilherme Pimenta da Silva and Segen F. Estefen, "Determination of the geometric and material properties of the NREL Phase VI wind turbine blade," Marine Systems & Ocean Technology 16, 63-83, (2021).
- [12] "Ansys® Academic Research ACP(pre), Release 19.0." .
- [13] "Ansys® Academic Research Mechanical, Release 19.0." .
- [14] "Ansys® Academic Research ACP(pre), Release 19.0." .
- [15] "U. S. Department of Defense—Composite Materials Handbook," in Polymer Matrix Composites Materials Properties. vol. 2, (B00U8DF23Y, ASIN, 2015).