# Structural Performance Validation of a Glass Fibre-Reinforced Composite Demonstrator for Wind Turbine Blades

Yadong Jiang
*Postdoctoral Researcher, MaREI Centre, Ryan Institute, School of Engineering, University of Galway, Ireland*

William Finnegan
*Lecturer, MaREI Centre, Ryan Institute, School of Engineering, University of Galway, Ireland*

Conor Kelly
*Project Engineer, ÉireComposites Teo, Galway, Ireland*

Ahmed El Bastawisy
*Project Engineer, ÉireComposites Teo, Galway, Ireland*

Tomas Flanagan
*CEO, ÉireComposites Teo, Galway, Ireland*

Ayaz Ahmad
*Doctoral Researcher, MaREI Centre, Ryan Institute, School of Engineering, University of Galway, Ireland*

Jamie Goggins
*Established Professor, MaREI Centre, Ryan Institute, School of Engineering, University of Galway, Ireland*

ABSTRACT: As the world moves towards a more sustainable way of life, Ireland must invest significantly in creating a carbon-free energy system. Currently, in Ireland, the largest source of renewable energy is wind. According to the Irish government's Climate Action Plan 2021, the development of offshore wind will be facilitated, including connecting at least 5 GW of offshore wind, based on competitive auctions, to the grid by 2030. The typical life span of a wind turbine is 20 years but it can be extended to 25 years or longer depending on environmental factors and maintenance procedures. Therefore, the structural performance of rotor blades is vital to ensure a safe operation of a wind turbine during its life span. As the wind turbine blade can be considered as a slender structure, which mainly suffers flexural loading, its strength at the root region is critical in structural design.

This research focuses on validating the structural performance of a 5-metre composite demonstrator, which represents the spar cap at the root region of a 13-metre wind turbine blade. The demonstrator has a rectangular cross-section and is made of glass fibre-reinforced composite material. Physical tests are carried out to investigate the demonstrator's structural performance. A hydraulic actuator is used to simulate the extreme loading that acts on a wind turbine blade during operation. During testing, the tip deflection, strain on the external surfaces and structural integrity are monitored to ensure the blade root region can withstand extreme operational conditions. Besides de-risking the demonstrator, test results will be used to validate a finite-element model for the design of wind turbine blades. The model is intended to be utilised to assist future blade design iterations and structural performance optimisation.

## 1. INTRODUCTION

Wind energy, a clean and renewable energy source, provides a sustainable solution to the world's energy problems. According to SEAI Energy in Ireland Report (2022), wind energy is the second largest source of generated electricity, contributing 28% of all electricity generated in Ireland in 2021. With the boost of wind turbine size, wind energy development is now moving towards offshore. As stated in the Irish government's Climate Action Plan (2021), the development of offshore wind will be facilitated by connecting at least 5 GW of offshore wind to the grid by 2030. With the increase in wind turbine capacity, the structural design of rotor blades becomes critical considering a typical lifespan of about 20 years.

To investigate the structural performance of a wind turbine blade, two methodologies, namely physical testing and numerical analysis, are widely used. Physical testing is a straightforward way of validating the structural design of a wind turbine blade. By simulating and applying wind loading to a blade and monitoring its responses, in terms of deflection, stress and strain distributions, etc., the blade safety under extreme load conditions can be guaranteed. The structural performance of a 13-m wind turbine blade, manufactured using glass fibre-reinforced composite materials, was tested by Finnegan et al. (2021). The blade survived under a series of static and fatigue tests, permitting the safe operation of the wind turbine during its service life. In the research works of Fagen et al. (2018), two 15-kW wind turbine blades, designed and manufactured with carbon fibre and glass fibre, respectively, were tested under static loading. Test results revealed that material has a significant impact on the structural behaviour of a wind turbine blade.

Besides experiment testing, numerical analysis is also widely used in predicting wind turbine blade structural performance. Since wind turbine blade can be considered to be a thin-wall structure, it is usually modelled with shell elements (Serafeim et al. (2020), Maheri et al. (2020), etc.). To ensure the developed numerical model is reliable, its accuracy should be validated using the experimental testing data (Fagan et al. (2017), Fagan et al. (2018), Finnegan et al. (2021), etc.). A finite element model of a wind turbine blade was developed by Jiang et al. (2023) using shell elements. The predicted result was proved to have good agreement with the testing results. Besides shell elements, solid elements can also be used to model the wind turbine blade. In the research works of Finnegan et al. (2021), numerical results given by finite element models generated using solid elements and shell elements were compared. It was concluded that the results given by the two types of numerical models agreed well with the testing data.

However, with the increase in wind turbine capacities, it becomes difficult to perform full-scale physical testing on a rotor blade due to the size limits of the testing facilities. Hence, consideration can be to break the rotor blade into two or even more segments and identify the critical part to test. As the wind turbine blade can be treated as a cantilever structural under flexural bending, its root is subject to the highest bending force and can be the first region to fail. According to the performance analysis in various research works, it is found that the root part of a wind turbine blade has the highest strain values under both fatigue (Meng et al. (2018) and Sajeer et al. (2020)) and static loading (Fagan et al. (2017) and Finnegan et al. (2021)). Hence, it is considered to test the root segment of a wind turbine blade to explore the equivalent flexural behaviour under extreme wind conditions.

In this research, the structural performance of a 5-m demonstrator, which represents the root region of a 13-m wind turbine blade, was studied. A demonstrator was manufactured using glass fibre-reinforced composite material and a detailed testing plan was proposed for obtaining the structural responses of the composite structure under static loading. A finite element model was developed to predict the structural response of the demonstrator. The numerical results will help to determine the proper load amplitude that can be applied in the experimental tests.

## 2. MATERIALS AND METHODS

### *2.1. Aim, Objectives and Methodology*

The overall aim of this study is to explore the structural performance of the root segment of a 13-m wind turbine blade. However, in order to achieve this aim, the following objectives must be completed:

- Manufacturing a 5-m demonstrator to represent the root section of a 13-m wind turbine blade.
- Performing detailed experiment tests to study the structural responses of the demonstrator.
- Developing an accurate and efficient finite element model to predict the demonstrator structure performance.

The investigation of the performance of the demonstrator consists of three stages, namely the manufacture, testing and numerical modelling. This 5-m composite structure was manufactured by ÉireComposites using resin-infusion technology. The demonstrator then will undergo a series of static and dynamic tests to explore its stiffness and strength. A comprehensive finite element model is proposed to predict the structural responses of the demonstrator and other test setups.

### *2.2. Wind blade demonstrator*

The wind turbine blade can be considered as a slender structure. Under wind loading, the root of the blade suffers the largest bending moment, causing the root region at risk of failure. Hence, it is necessary to study the structural performance of a wind turbine blade root region under flexural loading. In this study, a 5-m glass fibre-reinforced composite demonstrator, which represents the structural part of the root section of a 13-m wind turbine blade, was manufactured and will be tested under a series of experimental tests. As shown in Figure 1, the shape of the demonstrator is simplified to a rectangular box. Conducting experiments will de-risk the root region of the full-scale wind turbine blade. Moreover, the testing data can be used to validate the numerical model, which will be used in the design of the next-generation wind turbine blade.

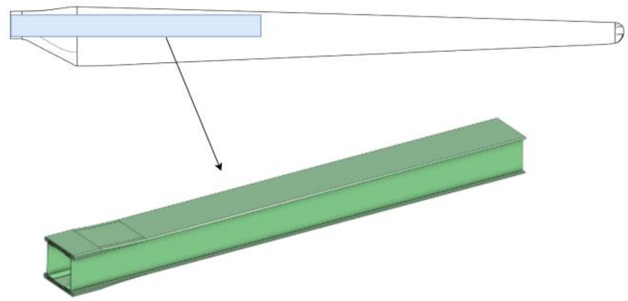

*Figure 1: Demonstrator simplified from the root region of a 13-m wind turbine blade*

#### *2.2.1. Structural Details of the Demonstrator*

Figure 2 demonstrates the structural details of the demonstrator, which consists of 2 spar caps, 2 webs and 22 steel inserts. The spar caps were manufactured using unidirectional glass fibre fabric infused with epoxy resin (UD). The spar caps are the main components to resist the flexural force so their thickness is adopted as 15 mm. However, to embed the steel inserts, the thickness of the spar cap root is increased to 40 mm and 11 holes were drilled at the rear surface using a CNC machine. The steel inserts were then embedded in the spar cap via adhesives. The webs are channel shape components made from tri-axial glass fibre-reinforced composite materials (TX). It has a uniform thickness of 6 mm along the demonstrator length. The spar caps and webs were assembled using adhesive as shown in Figure 3.

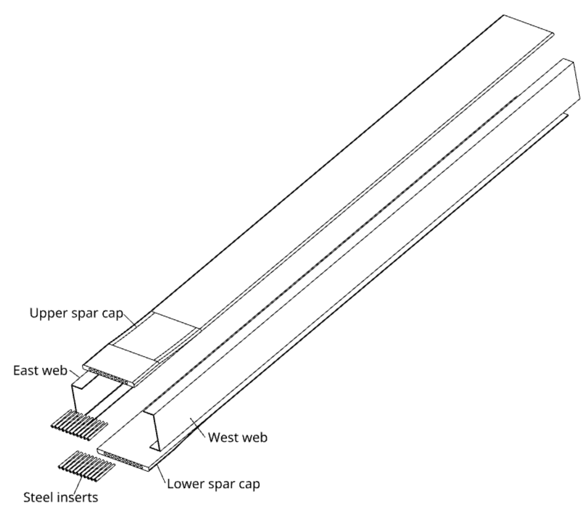


*Figure 2: Structural details of the demonstrator*

*Figure 3: Demonstrator manufactured using glass fibre-reinforced composite materials*

### 2.3. Structural Test Program

#### 2.3.1. Test Setup

A series of experimental tests will be carried out to validate the structural performance of the demonstrator. The main objective of the physical testing is to ensure the demonstrator can withstand the maximum bending moment introduced by the wind. Besides that, the test results can be used to verify the accuracy of the developed finite element model. Tests will be conducted in the Large Structures Testing Laboratory of the University of Galway. Two fixtures, namely the steel adaptor and support frame, are manufactured aiming to constrain the demonstrator to the strong reaction floor. As shown in Figure 4, the root of the demonstrator is mounted to the front surface of the adapter via 22 M10 bolts. The rear surface of the adapter is constrained to the support frame through 30 M40 bolts. The adaptor and support frame are manufactured using S355 steel.


*Figure 4: Test setup*

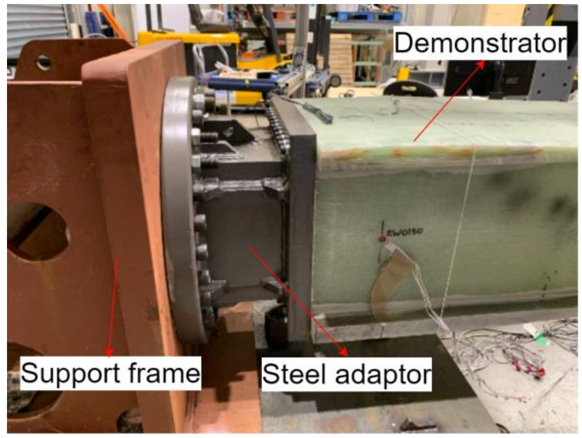


*Figure 5: Details of the demonstrator root connection*

#### 2.3.2. Test Methodology

The testing program includes natural frequency tests and static tests. As the demonstrator is manufactured to represent the root structure of a wind turbine blade, the IEC 61400–23 (2014) is adopted as the testing standard. In the natural frequency tests, the demonstrator will be vibrated in the horizontal and vertical directions, respectively. This is achieved by giving a transient impact to the tip of the demonstrator. Accelerometers are installed along the demonstrator to monitor the vibrations. The first two vertical, first two horizontal and first torsion-wise natural frequencies are of interest.

A 250-kN hydraulic actuator will be used to apply vertical load to the tip region of the demonstrator. A steel extender beam is employed to distribute the load uniformly to the loading area. Four load cases, which represent 25%, 50%, 75% and 100% of the testing load, will be applied in sequence during the static testing. The natural frequency tests will be carried out before and after the static tests to check if there is any change in the demonstrator's natural frequencies. Any variation in natural frequencies indicates that the stiffness of the demonstrator changes and damage is possibly introduced during the static testing.

#### 2.3.3. Instrumentation and monitoring

Instruments, including strain gauges, displacement transducers and accelerometers, will be used to monitor the responses. As shown in Figure 6, for each spar cap, 5 linear strain gauges are installed along the demonstrator length to record the strain distribution along the fibre direction. Considering that the root is subjected to the highest bending moment, 2 rosette strain gauges are installed in the root region of each web to monitor the shear strains (Figure 7). There are 4 single axial accelerometers attached to the demonstrator, three on the spar cap and one on the web, to record the vibration responses of the demonstrator. The recorded acceleration will be used to calculate the natural frequencies, the damping ratios and modal shapes of the demonstrator. 3 displacement transducers are installed along the lower spar cap to monitor its vertical deflections. Besides the transducers on the demonstrator, two Linear Variable Differential Transformers (LVDTs) will be installed on the front surface of the adapter. The recorded displacement can indicate the rotation of the fixtures. This rotation should be taken into consideration when correcting the deflection data of the demonstrator. Besides the conventional instruments, non-contact measurement techniques will be used to provide additional data for the tests. A 3D laser scanner will be utilised to record the shapes of the demonstrator before and during loadings. Digital image correlation techniques will be used to monitor the strain distribution at the root of the demonstrator using a dual-camera system.

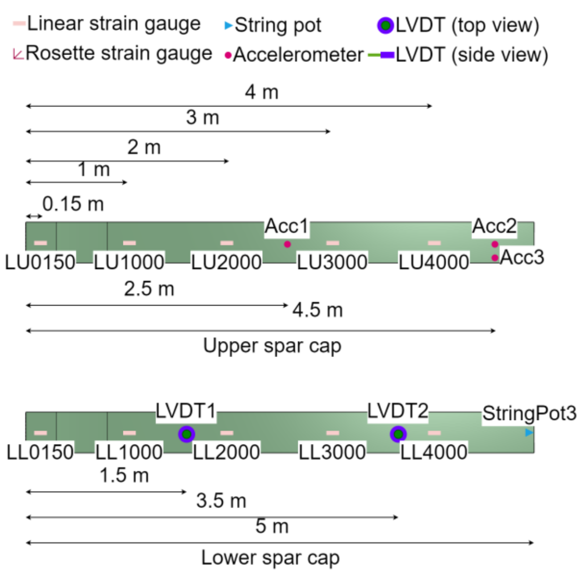


*Figure 6: Instrumentation layout of the spar caps*

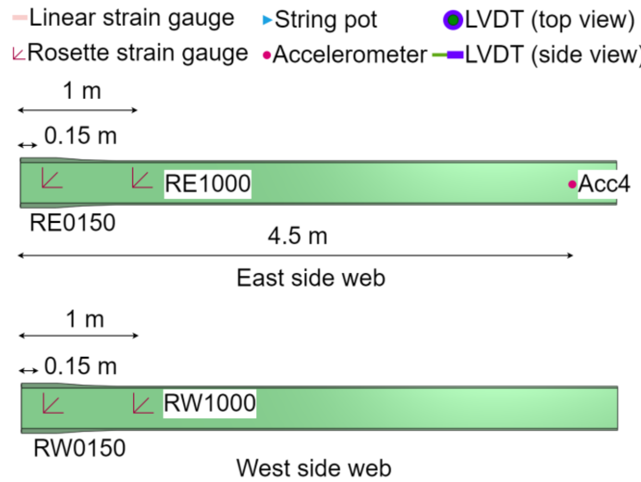


*Figure 7: Instrumentation layout of the Webs*

## 3. NUMERICAL ANALYSIS

Numerical analysis was performed to study the structural performance of the demonstrator and the steel fixtures under static loading. The numerical results can help to determine the maximum testing load that can be applied to the demonstrator. Considering that the test objective is to study the responses of the demonstrator, the loading amplitude should not cause damages of the steel fixtures and the bolts. Hence, the numerical analysis should not only include the responses of the demonstrator but also simulate the behaviour of the steel inserts, bolts and fixtures. However, to reduce the complexity of the analysis, the performances of the demonstrator and the steel fixtures were analysed separately.

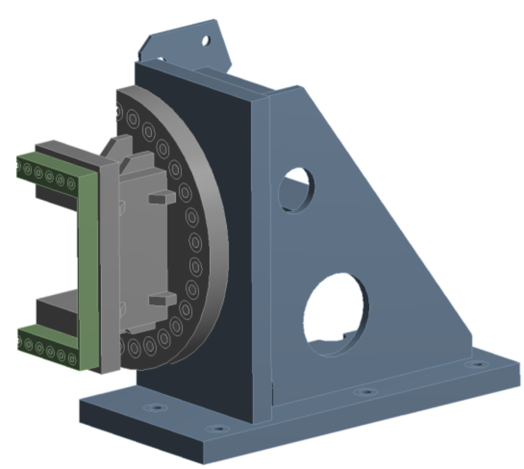

Figure 8: Finite element model of the adaptor and the support frame

### *3.1. Steel fixtures*

The capacity of the steel fixtures, namely the adaptor and the support frame, was determined by finite element analysis. As shown in Figure 8, the finite element model of the fixtures was created in Ansys mechanical (2022). Taking advantage of the symmetry, only half of the two components were modelled and analysed. Both the adaptor and support frame were generated using solid elements. The bolts connecting the two fixtures were simplified to line elements. The contacting surfaces between the two components were connected by contact elements. A remote vertical load is applied to the front face of the adaptor at a distance of 5 m. The load was controlled to increase in an interval of 2.5 kN until the yielding of the adaptor occurs. According to the analysis results, the steel adaptor is the first component to yield, under a load of 38 kN. Figure 9 shows the von-Mises stress distribution of the adaptor under this load. Hence, the testing load should not exceed 38 kN to avoid damaging the steel fixtures.

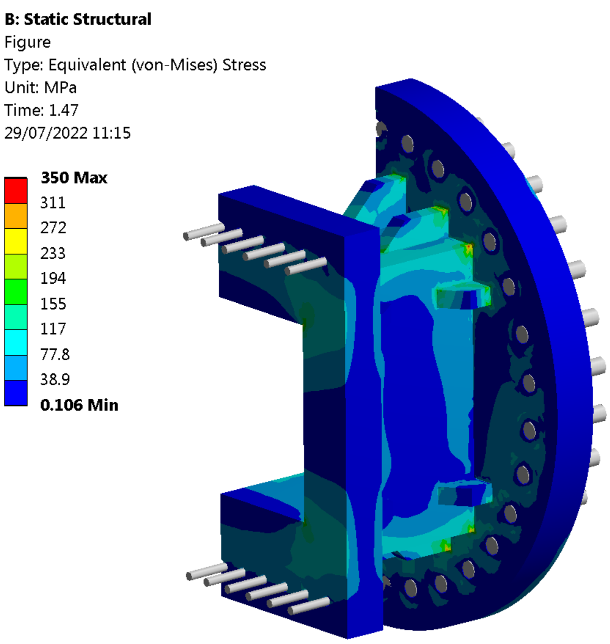


*Figure 9: Von-Mises stress distribution of the steel adaptor*

### *3.2. Demonstrator*

The structural response of the demonstrator is also predicted using the finite element software Ansys Mechanical (2022). Similar to the model of the steel fixtures, half of the demonstrator is generated, as shown in Figure 10. The finite element model includes two spar caps, one web, two adhesive layers, six steel inserts and six M10 bolts. The steel inserts and M10 bolts were both modelled with solid elements. To embed the steel inserts in the root, the spar caps were meshed into solid elements. The web can be considered as a thin-wall structure due to its 6 mm thickness. Hence, the layered shell elements were used to construct the web. The adhesives used to assemble the spar caps and webs were also modelled. To simulate the interaction between the spar cap and web, the adhesive layer was modelled as solid elements despite its 1 mm thickness. Static analysis was performed with a vertical force acting on the tip region of the demonstrator with the root surfaces of the M10 bolts constrained. The force was applied in 20 steps up to 38 kN which is the capacity of the steel fixtures. During the analysis, both the failure of composite and steel components were of interest. The failure of the composite material is analysed using Puck failure theory proposed by Puck and Schürmann (1998), which takes into account the multi-direction stress effect. The maximum stress method is employed to analyse the failure of the adhesive and steel.

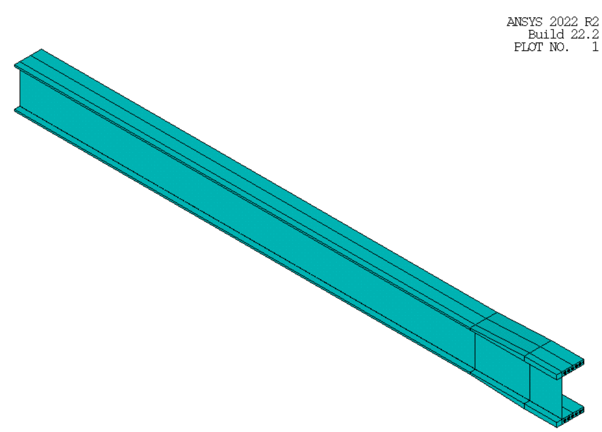


*Figure 10: Finite element model of the demonstrator*

Under a vertical load of 38 kN, the demonstrator has a tip deflection of 49 mm, as shown in Figure 11. The maximum von-Mises stress of the steel components (the inserts and M10 bolts) are 581 MPa, which is less than the yield stress of S12.9 grade steel. Regarding the adhesive, the maximum shear stress and tensile stress are 16 MPa and 2.1 MPa, which are both within the strength limits of the adhesive (37 MPa and 29 MPa, respectively). The test results indicate that the steel and adhesive components of the demonstrator are safe under this load. The failure risk of the composite materials were analysed using Puck failure theory. Inverse reserve factor distribution of the spar caps and web were calculated based on the stress components in different directions of each element. An inverse reserve factor larger than 1 indicates that the composite material may break at the corresponding location. The spar cap is observed to have matrix failure around the loading area and root drilling hole. By tracing the numerical results under each load step, it is found that the first failure occurred under a load of 32.3 kN. Figure 12 displays the inverse reserve factor distribution of matrix failure under this load. By comparing the capacity of the demonstrator, the bolts, the adhesive and the steel fixtures, it can be concluded that the maximum test load should be less than 38 kN to break the demonstrator while other components are protected. However, it should be noted that the analysis was performed without the consideration of safety factors. In the experiment testing, several safety factors should be applied to the load amplitude to consider the uncertainties of material and load according to the adopted testing standard.

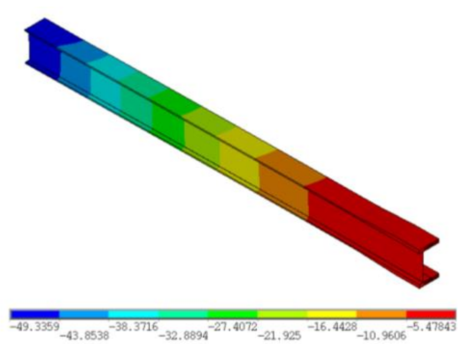


*Figure 11: Deflection of the demonstrator with a vertical load of 38 kN imposed at the tip*

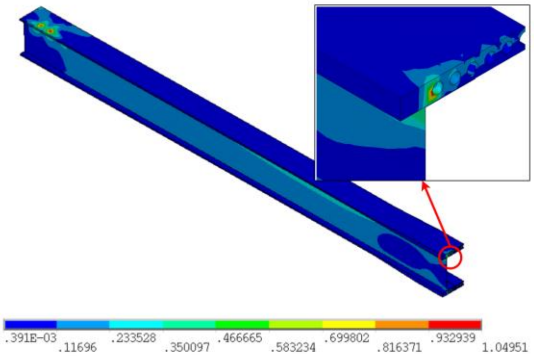


*Figure 12: Inverse reserve factor distribution for matrix failure (under force 32.3 kN)*

## 4. CONCLUSIONS

In this study, a glass-fibre reinforced composite demonstrator, which represents the root segment of a wind turbine blade, was manufactured. A detailed testing programme was proposed to investigate the demonstrator's structural performance. This includes a series of natural frequency tests and static tests. Steel fixtures were designed and manufactured to connect the demonstrator to the reaction floor. Instruments, including strain gauges, accelerometers, and displacement transducers, were planned to be installed on the demonstrator and steel fixtures. Numerical models were proposed to predict the structural responses of the demonstrator and steel fixtures. The maximum loading amplitude was determined to avoid steel component damage during testing. In the future, experimental tests will be conducted based on the testing program to verify the performance of the demonstrator. The test results will be used to validate and improve the proposed finite element model.

The study carried out in this paper, relating to manufacturing, testing and numerical analysis, will benefit the wind turbine blade developer. The experimental and numerical analysis results will help designers have an in-depth understanding of the performance of different rotor blade components. Moreover, it will also benefit the composite structure manufacturers, as it verifies the reliability and strength of the demonstrator.

This guarantees the wind turbine blade's ability to withstand high wind loadings, resulting in a reduction in maintenance costs and allowing for uninterrupted energy generation.

## 5. ACKNOWLEDGEMENTS

This work was funded by the Sustainable Energy Authority of Ireland (SEAI) under the SEAI Research, Development, and Demonstration Funding Programme 2021 (grant numbers 21/RDD/630, 21/RDD/655 and 21/RDD/601). The research was also supported by Science Foundation Ireland (SFI) through the MaREI Research Centre for Energy, Climate, and Marine (grant no 12/RC/2302_2). The authors would also like to acknowledge the support of SFI through the SFI Industry Fellowship Programme (grant no. 19/IFA/7417) and the Marine Institute, funded under the Marine Research Programme by the Government of Ireland (PDOC/21/03/01).

## 6. REFERENCES

Ansys® Academic Research Mechanical, Release 2022 R2 (2021). *Help System, Mechanical APDL*. ANSYS, Inc.

Fagan, E. M., De La Torre, O., Leen, S. B., and Goggins, J. (2018). "Validation of the multi-objective structural optimisation of a composite wind turbine blade." *Composite Structures*, 204, 567–577.

Fagan, E. M., Leen, S. B., Torre, O. d. l., and Goggins, J. (2017). "Experimental investigation, numerical modelling and multi-objective optimisation of composite wind turbine blades." *Journal of Structural Integrity and Maintenance*, 2(2), 109–119.

Finnegan, W., Jiang, Y., Dumergue, N., Davies, P., and Goggins, J. (2021). "Investigation and Validation of Numerical Models for Composite Wind Turbine Blades." *Journal of Marine Science and Engineering*, 9(5), 525.

Government of Ireland (2021). "CLIMATE ACTION PLAN 2021 Securing Our Future." *Department of the Environment, Climate and Communications*, Dublin, Ireland.

Jiang, Y., Finnegan, W., Flanagan, T., and Goggins, J. (2023). "Optimisation of Highly Efficient Composite Blades for Retrofitting Existing Wind Turbines." *Energies*, 16(1), 102.

Maheri, A. (2020). "Multiobjective optimisation and integrated design of wind turbine blades using WTBM-ANSYS for high fidelity structural analysis." *Renewable Energy*, 145, 814–834.

Meng, H., Lien, F.-S., Glinka, G., and Geiger, P. (2019). "Study on fatigue life of bend-twist coupling wind turbine blade based on anisotropic beam model and stress-based fatigue analysis method." *Composite Structures*, 208, 678–701.

Puck, A. and Schürmann, H. (1998). "Failure Analysis of FRP Laminates by Means of Physically Based Phenomenological Models." Composites Science and Technology, 58(7), 1045–1067.

Sajeer, M. M., Mitra, A., and Chakraborty, A. (2020). "Spinning finite element analysis of longitudinally stiffened horizontal axis wind turbine blade for fatigue life enhancement." Mechanical Systems and Signal Processing, 145, 106924.

SEAI (2020), "Energy in Ireland 2020 Report." *Sustainable Energy Authority of Ireland (SEAI)*, Dublin, Ireland.

SEAI (2022). "Energy in Ireland 2022 Report." *Sustainable Energy Authority of Ireland (SEAI)*, Dublin, Ireland.