Numerical modelling of composite tidal turbine blades

William Finnegan[*,], Tomás Flanagan[**], Adrian Doyle[**], Michael Walls[*,**], Micheál Forde[**], Edward Fagan[*], Jamie Goggins[*]**

[*] Civil Engineering, College of Engineering and Informatics, National University of Ireland Galway, Ireland
[**] EireComposites Teo, An Choill Rua, Inverin, Co. Galway, Ireland

Abstract - Numerical modelling offers a relatively inexpensive method of testing and validation of early designs, compared to physical models. This paper presents a methodology for developing an advanced numerical model for designing and assessing the performance of composite tidal turbine blades. The model includes the determination of the loadings due to the tidal flow using computational fluid dynamics, the material properties of the fibre-reinforced polymer composite and a structural analysis of the blades. A hydrodynamic analysis of the main loading (thrust force) on the tidal turbine blade is presented, along with a validation of the accuracy of the composite model coupled with the structural analysis (finite element method) solver.

Keywords- composite materials, computational fluid dynamics, hydrodynamics, structural analysis, tidal energy.

I. INTRODUCTION

In recent years, there has been a growing interest in establishing renewable sources of energy to alleviate the global reliance on fossil fuels. Tidal energy, with its reliability and predictability, offers an excellent opportunity for a number of countries worldwide, such as Ireland, France, the UK and Canada. In addition, 2017 saw the global installed capacity of tidal current energy deployments increase to over 17 MW [1]. However, the harshest environments, where tidal energy devices will perform most efficiently, cause severe loading on the devices. A key component of a tidal energy device is the blades, which convert the kinetic energy of the current into useful mechanical energy. As a result, the blades experience very high thrust forces and need to be designed to deal with these loadings. In recent years, the horizontal axis tidal turbine (HATT) has become the most popular type of tidal energy device as, based on a EU report by Corsatea and Magagna [2], 76 % of research and development efforts in the tidal energy sector are related to HATT technologies.

Traditionally, the hydrodynamic modelling tool used for estimating the loadings on the blades of HATTs is the blade element momentum theory [3]. However, in recent years, computational fluid dynamics (CFD) is gaining ground due to increased computational capabilities and higher accuracy under a range of operating conditions. A summary of previously published studies exploring the use of CFD to examine the operation of horizontal axis tidal turbines has been included and discussed in Finnegan et al. [4]. Previously, a number of structural analysis studies related to tidal turbine blades have been completed. Bir et al. [5] presents a details description of the structural design of a tidal turbine composite blade. Grogan et al. [6] developed a hydrodynamic-structural design methodology for a tidal turbine blades in order to compare glass fibre and carbon fibre as blade construction materials for a full scale (1.5MW) tidal turbine. Davies et al. [7] experimentally explored different composite materials on a small-scale tidal turbine in order to evaluation of the durability of composite tidal turbine blades. Harper and Hallett [8] used numerical models to explore the interfacial cracking between composite material plies in tidal turbine blades in order to improve their design. Fagan [9] used advanced hydrodynamic and finite element method (FEM) models, along with a genetic algorithm, to optimise the mass of the blade and evaluate a number of near-optimum tidal turbine blade designs. Payne et al. [10] discussed the design and manufacture of the blades and the shaft for a bed-supported tidal turbine device, which is loaded by turbulent flow and waves. The accuracy of their assessment was validated against laboratory testing. Since HATTs are constantly and completed submerged in either seawater or fresh water, they should be constructed of a material that is suitable for this environment, while being able to withstand high loadings. Therefore, in this study, it is proposed that tidal turbine blades are constructed of a fibre-reinforced polymer composite.

In this paper, a methodology for developing an advanced numerical model, capable of accurately simulating the structural operation of a fibre-reinforced polymer composite tidal turbine blade within a tidal flow, is presented. This numerical model is being developed, within ANSYS Workbench, using a CFD model to derive the hydrodynamic loadings on the blade and a composite material model of each part of the blade. These models are then coupled within a FEM solver in order to perform a structural analysis of the tidal turbine blade. A concept HATT has been used to demonstrate the capabilities of the numerical model and aspects of the composite material model coupled with the FEM solver has been validated using experimental data.

II. RESEARCH ELABORATION

The schematic shown in Figure 1 illustrates the methodology proposed within this study and the interaction between the different design aspects: primarily, the CFD model; the composite material model; and the finite element method (FEM) analysis. Therefore, in order to develop this numerical model with ease, ANSYS Workbench has been used, which allows the transfer of data between a number of ANSYS modules (i.e. DesignModeler, CFX, ANSYS Composite PrepPost, Mechanical).

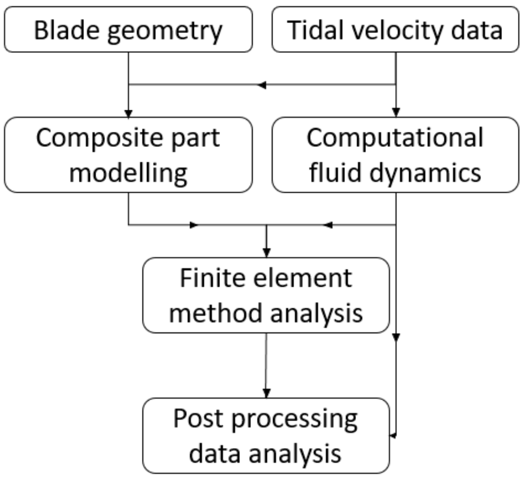

Figure 1 : Overview of the methodology used to analyse a composite tidal turbine blade

A. Tidal turbine blade geometry

The 2-blade tidal turbine geometry for a concept HATT that is used in this analysis is shown in Figure 2. The diameter of the tidal turbine is approximately 21 m, where the hub is 3 m in diameter and, in turn, is suitable for use in tidal turbine with a capacity of approximately 1MW . The blades are pitched so that the tip of the blade is near 0° so to design for a worst-case scenario.

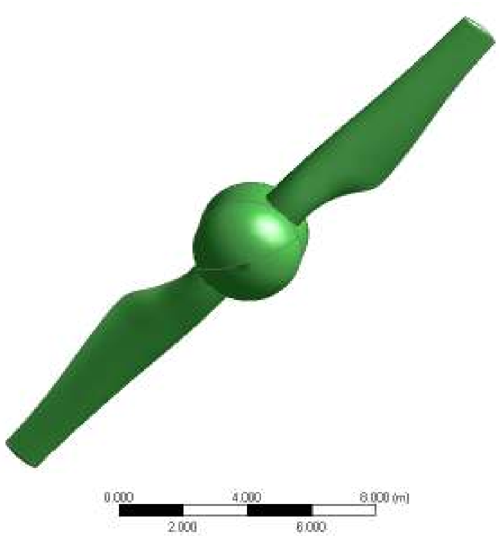

Figure 2 : Concept tidal turbine blade geometry used in the study

B. Computational fluid dynamics

The CFD model has been developed using the commercial software ANSYS CFX, which uses a finite volume method in order to solve the Reynolds-averaged Navier-Stokes equations. Further details on the governing equations used in the CFD model are given in Appendix A. The CFD analysis is used to calculate the hydrodynamic loading imposed on the tidal turbine blades, which is based on an incoming tidal current profile. A number of tidal current profiles, along with specified constant flow, have been used within the study in order to gain more meaningful results. The methodology used to develop the CFD model is similar to that used in Finnegan et al. [4].

The velocity profile of the tidal current (U_x) reduces near the seabed due to friction. Therefore, the velocity profile can be described using the following power law [11]:

$$U_x = \left(\frac{z}{\beta h}\right)^{1/\alpha} \mathrm{U} \qquad (1)$$

where, z is the height above the seabed, α is the power law, β is the bed-roughness coefficient, h is the water depth and U is the depth averaged velocity, which is 3 m/s for the purpose of this study. Based on the findings of Lewis et al. [12], on average the 1/7th power-law (α = 7) with a bed-roughness coefficient (β) of 0.4 was found to accurately represent the velocity profile.

C. Fibre-reinforced polymer composite part modeling

The tidal turbine blades are made of fibre-reinforced polymer composites, which are modelled separately using ANSYS Composite PrepPost (ACP). In this study, ACP is used to define and design the layered composites, including ply fibre type, thickness and orientation. The resulting material properties of the part can then be used within the FEM solver. When modelling the composite part, the ply material properties and ply orientation need to be defined. The Young's modulus of the ply (E_{ply}) is estimated using the Rule of Mixtures as follows [13]:

$$E_{ply} = \kappa V_f E_f + V_m E_m \qquad (2)$$

where κ is a correction factor that accounts for the fibre area, the fibre diameter distribution, the interface and the fibre orientation distribution, V_f is the fibre volume faction (0.52), E_f is the Young's modulus of the fibre (72.4 GPa), V_m is the matrix volume fraction and E_m is the Young's modulus of the matrix (3.89 GPa).

D. Finite element method analysis

The FEM analysis is performed using ANSYS Mechanical. The pressure distributions on the blade are obtained from the CFD model to provide the loading on the blade due to presence of the tidal current. The material properties are calculated within the composite material model (using ACP). A static analysis is performed in order to determine the

deformation of the blades, including the maximum deflection and the von Mises elastic stress and the equivalent strain.
For the purpose of validation within this initial study, a composite part is used, which is detailed in Section 2.E.

E. Validation specimen manufacture

In order to validate the accuracy of aspects of the composite material model coupled with the FEM solver, it was necessary to manufacture fibre-reinforced polymer composite part specimens, which can be physically tested. The specimens were manufactured from E-glass bi-axial 45°/135° material (AHLSTROM 62042) that was prepared with a quasi-isotropic lay-up and infused with epoxy resin (Gurit's Ampreg 22 [14]) with a slow hardener using the Vacuum Assisted Resin Transfer (VART) method at ÉireComposites Teo. The quasi-isotropic lay-up for 16 ply panels was specified as [(45°/135°, 0°/90°)2]s and the specimen was cured for 48 hours at room temperature (21°C), followed by a post cure at 75°C for 5 hours. An example of the specimens being tested is shown in Figure 3.

Figure 3 : Static testing of fibre-reinforced polymer composite part specimens, where the results are used to validate part of the numerical model

III. RESULTS & FINDINGS

The main results and findings of the study relate to (1) the hydrodynamic analysis, using the CFD model, of a tidal current profile on a tidal turbine, which is supported by a support structure at 45° to the water surface, and (2) the validation study for the composite material model coupled with the FEM solver that examines the static structural behavior of a fibre-reinforced polymer composite part.

A. CFD model results

During the CFD analysis, both the thrust force (kN) and torque (kNm) on the tidal turbine blades are monitored. However, the most significant loading is associated with the thrust force and, therefore, only this loading is presented in this paper for analysis. A previous publication [4] includes additional details on torque and thrust force on a 3-blade tidal turbine with a support structure. Figure 4 illustrates the thrust force on various components, including each blade, the hub and the support structure, along with the total thrust force on the tidal turbine. As the blades rotate, an oscillatory effect on the thrust force is created due to the velocity profile of the tidal flow, which was described in Eq. (1), and shadow effects from the support structure. Consequently, a far more significant shadow effect is observed on the support structure as the blades move past, shadowing the flow. This induced fatigue loading on the support structure should be taken into account in the design stage in order to ensure the structure will survive through its operational lifespan.

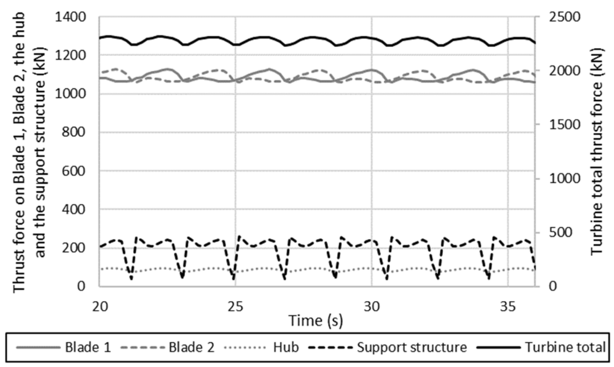

Figure 4 : Time-series displaying the thrust force (in kN) on each blade, the hub, the support structure and the total thrust force (in kN) on the turbine

The variation of thrust force on an individual blade as it rotates through 360° is shown in Figure 5, which is based on the same data that is shown in the time-series in Figure 4. As the blade moves past the support structure, which is at 45° on the diagram, a reduction in loading is observed. Additionally, there is a higher loading before and after the support structure, compared to the case where there is no support structure present. This causes a fatigue loading on the blade (of approximately 6% of the maximum thrust force on the blade) and needs to be included within the design of the blades. This variation in thrust force is lower than observed previously [4], which may be a result of the diameter of the supporting structure being proportionally less than the width of the turbine blade.

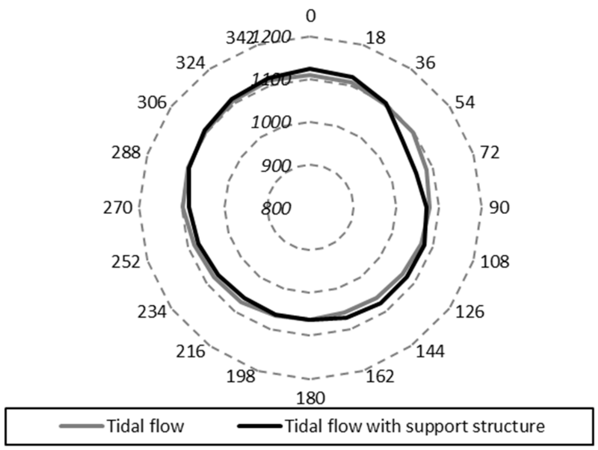

Figure 5 : Variation of thrust force on an individual blade as it rotates through 360° with and without a support structure

The absolute velocity profile of the flow as it moves past the rotating tidal turbine in shown in Figure 6, where the support structure is also present. A short range in velocities is used in order to clearly illustrate the variance in velocities around the turbine. The presence of the turbine causes the water to flow quicker around the turbine, while causing a wake behind it.

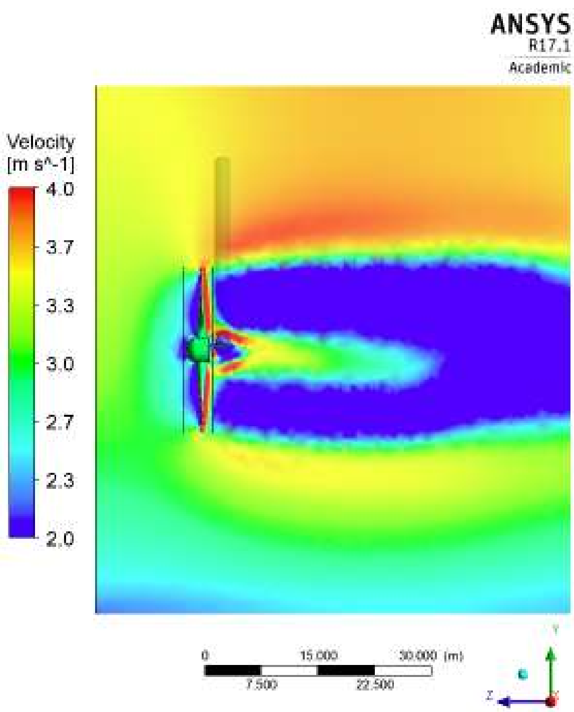

Figure 6 : Side elevation detailing the absolute velocity (in m/s) of the tidal flow as it moves past the tidal turbine

B. Composite part performance results

In order to validate the performance of structural FEM model for composite parts, where the composite material model coupled with the FEM solver, static tensile testing is carried out on the composite specimen, detailed in Section 2.E. This static tensile testing was carried out in accordance with ASTM D3039 (Standard Test Method for Tensile Properties of Polymer Matrix Composite Materials [15]) using a 250kN Zwick test machine with wedge grips at CTL Composites. The test composite specimens, which are detailed in Section 2.E, is loaded at a speed of 2mm/min and the stain is measured between 0.1-0.3% using a biaxial extensometer in order to calculate the Young's Modulus of the specimen.

In total 5 specimens were tested. The results of this physical testing, including an upper and lower bound for the test results and the average based on the mean Young's Modulus from this physical testing (calculated to be 19,567.2 MPa for the material) are compared to the results from the numerical model, which estimated the Young's Modulus of the material as 18,926.5 MPa. Therefore, the results are found to be in good agreement, as shown in Figure 7, with the numerical model underestimating by approximately 3% but very close to the results of one of the specimen physical tests. The Young's Modulus obtained here is also in line with that of Kennedy et al. [16], who reported a value of 19,300 MPa for a similar epoxy infused E-glass material.

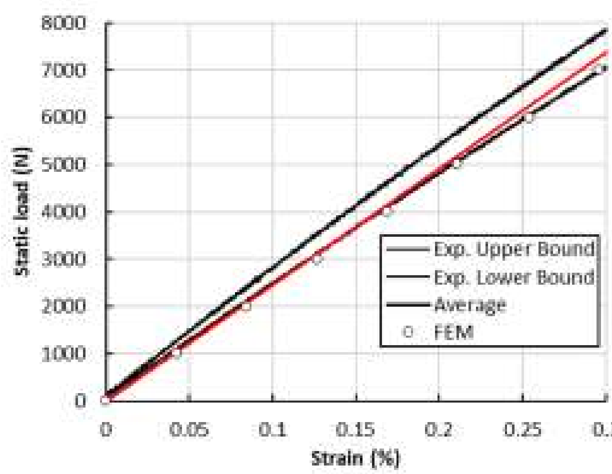

Figure 7 : Comparison of the results from the static physical testing of a composite test specimen with the results of a FEM analysis of the same part

IV. CONCLUSION

The proposed methodology for developing an advanced numerical model for designing and assessing the performance of composite tidal turbine blades has been demonstrated for a concept HATT. The hydrodynamic loadings on the tidal turbine blades in operation has been determined and the results of an experimental validation study, which consists of a structural analysis of a fibre-reinforced composite part using ANSYS ACP and ANSYS Mechanical, has been detailed.

These types of detailed coupled hydrodynamic-structural analysis of blades will be an essential tool for design engineers and manufactures alike. The success of tidal stream turbines will depend largely on their long-term operation with minimal maintenance. Since the blades comprise a significant portion of the capital, their robust design is a key factor. The accurate modeling of these complex scenarios are vital in both design and in operation as it allows for prediction of failure in extreme scenarios. Consequently, this will aid in lowering the levelised cost of tidal energy, which will be vital in defining the success of the sector.

APPENDIX

A. CFD model governing equations

In this study the CFD model is developed using the commercial software, ANSYS CFX, where its solver is based on the finite volume technique [17]. This technique divides the region of interest into sub-regions and discretises the governing equations in order to solve them iteratively over each sub-regions. Therefore, an approximation of the value of each variable at points throughout the domain is achieved.

The governing equations that need to be solved by the ANSYS CFX solver are the mass continuity equation, which is given as:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u_1)}{\partial x}+\frac{\partial(\rho u_2)}{\partial y}+\frac{\partial(\rho u_3)}{\partial z}=0 \qquad (3)$$

and the 3-dimensional Navier-Stokes equations, which are given as:

$$\begin{aligned}\rho\left(\frac{\partial u_1}{\partial t}+u_1\frac{\partial u_1}{\partial x}+u_2\frac{\partial u_1}{\partial y}+u_3\frac{\partial u_1}{\partial z}\right) &= -\frac{\partial p}{\partial x}+2\mu\frac{\partial^2 u_1}{\partial x^2} \\ &+\frac{\partial}{\partial y}\left(\mu\left(\frac{\partial u_1}{\partial y}+\frac{\partial u_2}{\partial x}\right)\right) \\ &+\frac{\partial}{\partial z}\left(\mu\left(\frac{\partial u_1}{\partial z}+\frac{\partial u_3}{\partial x}\right)\right)+F_1\end{aligned} \qquad (4)$$

$$\begin{aligned}\rho\left(\frac{\partial u_2}{\partial t}+u_1\frac{\partial u_2}{\partial x}+u_2\frac{\partial u_2}{\partial y}+u_3\frac{\partial u_2}{\partial z}\right) &= -\frac{\partial p}{\partial y}+2\mu\frac{\partial^2 u_2}{\partial y^2} \\ &+\frac{\partial}{\partial x}\left(\mu\left(\frac{\partial u_1}{\partial y}+\frac{\partial u_2}{\partial x}\right)\right) \\ &+\frac{\partial}{\partial z}\left(\mu\left(\frac{\partial u_2}{\partial z}+\frac{\partial u_3}{\partial y}\right)\right)+F_2 \\ &-\rho g\end{aligned} \qquad (5)$$

$$\begin{aligned}\rho\left(\frac{\partial u_3}{\partial t}+u_1\frac{\partial u_3}{\partial x}+u_2\frac{\partial u_3}{\partial y}+u_3\frac{\partial u_3}{\partial z}\right) &= -\frac{\partial p}{\partial z}+2\mu\frac{\partial^2 u_3}{\partial z^2} \\ &+\frac{\partial}{\partial x}\left(\mu\left(\frac{\partial u_1}{\partial z}+\frac{\partial u_3}{\partial x}\right)\right) \\ &+\frac{\partial}{\partial y}\left(\mu\left(\frac{\partial u_3}{\partial y}+\frac{\partial u_2}{\partial z}\right)\right)+F_3\end{aligned} \qquad (6)$$

where t is time, ρ is the fluid density, x, y, z are Cartesian coordinates (as shown in Figure 6), u_1 is the flow velocity in the x-direction, u_2 is the flow velocity in the y-direction, u_3 is the flow velocity in the z-direction, F_1 is the body force on the fluid in the x-direction, $F_2-\rho g$ is the body force on the fluid in the y-direction (vertical), F_3 is the body force on the fluid in the z-direction, p is pressure and μ is viscosity.

ACKNOWLEDGMENT

This publication has been emanated from research supported in part by a research grant from Science Foundation Ireland (SFI) under Grant Number 16/IFB/4512. This research was carried out with the support of the Marine Institute under the Marine Research Programme with the support of the Irish Government. Additionally, this research was funded by Science Foundation Ireland (SFI) through the Marine and Renewable Energy Ireland (MaREI) research centre (Grant no. 12/RC/2302) and the European Commission under the FloTEC project (Floating Tidal Energy Commercialisation, Funded under: Horizon 2020, Low Carbon Energy (LCE), Grant Number: 691916). The last author would also like to acknowledge the support of Science Foundation Ireland through the Career Development Award programme (Grant No. 13/CDA/2200). The authors would like to express their gratitude to the Irish centre for high-end computing (ICHEC) for the computation processing power provided.

REFERENCES

[1] Ocean Energy Systems, Annual Report: An Overview of Ocean Energy Activities in 2017, in: A.B.e.M.a.H. Jeffrey (Ed.) The Executive Committee of Ocean Energy Systems, Lisbon, Portugal, 2017.

[2] T.D. Corsatea, D. Magagna, Overview of European innovation activities in marine energy technology, European Commission, Brussels, Belgium, 2013.

[3] A.H. Day, A. Babarit, A. Fontaine, Y.P. He, M. Kraskowski, M. Murai, I. Penesis, F. Salvatore, H.K. Shin, Hydrodynamic modelling of marine renewable energy devices: A state of the art review, Ocean Engineering 108 (2015) 46-69.

[4] W. Finnegan, E. Fagan, T. Flanagan, A. Doyle, J. Goggins, Operational fatigue loadings on tidal turbine blades using computational fluid dynamics, Renewable Energy (2018).

[5] G.S. Bir, M.J. Lawson, Y. Li, Structural Design of a Horizontal-Axis Tidal Current Turbine Composite Blade, (44373) (2011) 797-808.
[6] D.M. Grogan, S.B. Leen, C.R. Kennedy, C.M. Ó Brádaigh, Design of composite tidal turbine blades, Renewable Energy 57 (2013) 151-162.
[7] P. Davies, G. Germain, B. Gaurier, A. Boisseau, D. Perreux, Evaluation of the durability of composite tidal turbine blades, Phil. Trans. R. Soc. A 371(1985) (2013) 20120187.
[8] P.W. Harper, S.R. Hallett, Advanced numerical modelling techniques for the structural design of composite tidal turbine blades, Ocean Engineering 96 (2015) 272-283.
[9] E. Fagan, Design of fibre-reinforced polymer composite blades for wind and tidal turbines, Civil Engineering, National University of Ireland Galway, Galway, Ireland, 2017.
[10] G.S. Payne, T. Stallard, R. Martinez, Design and manufacture of a bed supported tidal turbine model for blade and shaft load measurement in turbulent flow and waves, Renewable Energy 107 (2017) 312-326.
[11] R. Soulsby, 1997. Dynamics of marine sands: a manual for practical applications. Thomas Telford.
[12] M. Lewis, S.P. Neill, P. Robins, M.R. Hashemi, S. Ward, Characteristics of the velocity profile at tidal-stream energy sites. Renewable Energy 114 (2017) p. 258-272
[13] B.D. Agarwal, L.J. Broutman, K. Chandrashekhara, Analysis and performance of fiber composites, John Wiley & Sons2017.
[14] Gurit, Ampreg 22: Epoxy Laminating System Datasheet. www.gurit.com/-/media/Gurit/Datasheets/ampreg-22.ashx. Accessed 20-04-2018.
[15] ASTM, D3039 / D3039M-17: Standard test method for tensile properties of polymer matrix composite materials, ASTM International, West Conshohocken, PA, USA, 2017.
[16] C.R. Kennedy, V. Jaksic, S.B. Leen, C.M.Ó. Brádaigh, Fatigue life of pitch- and stall-regulated composite tidal turbine blades, Renewable Energy 121 (2018) 688-699.
[17] ANSYS, ANSYS® CFX (Release 17.1): CFX-Solver Theory Guide, ANSYS Inc., 2017.

AUTHORS

First Author – William Finnegan, B.E. Ph.D. C.Eng. MIEI, Civil Engineering, College of Engineering and Informatics, National University of Ireland Galway, Ireland and EireComposites, An Choill Rua, Inverin, Co. Galway, Ireland, william.finnegan@nuigalway.ie.
Second Author – Tomás Flanagan, B.E. Ph.D., EireComposites, An Choill Rua, Inverin, Co. Galway, Ireland, t.flanagan@eirecomposites.com.
Third Author – Adrian Doyle, B.E. M.E., EireComposites, An Choill Rua, Inverin, Co. Galway, Ireland, a.doyle@eirecomposites.com.
Fourth Author – Michael Walls, B.E., EireComposites, An Choill Rua, Inverin, Co. Galway, Ireland, m.walls@ctlcomposites.com.
Fifth Author – Micheál Forde B.E., EireComposites, An Choill Rua, Inverin, Co. Galway, Ireland, m.forde@eirecomposites.com.
Sixth Author – Edward Fagan, B.E. Ph.D., Civil Engineering, College of Engineering and Informatics, National University of Ireland Galway, Ireland, edwardmichael.fagan@nuigalway.ie.
Last Author – Jamie Goggins, C.Eng., BA, BAI, Ph.D., Civil Engineering, College of Engineering and Informatics, National University of Ireland Galway, Ireland, jamie.goggins@nuigalway.ie.

Correspondence Author – Jamie Goggins, email: jamie.goggins@nuigalway.ie, tel: +353 91 492609.