A Method for Analysing Fluid Structure Interactions on a Horizontal Axis Tidal Turbine

R.F. Nicholls-Lee#1, S.R. Turnock#2, S.W. Boyd#3

#Fluid Structure Interactions Research Group, University of Southampton
Southampton, Hampshire, SO17 1BJ, U.K.

1rnl@soton.ac.uk

2srt@soton.ac.uk

3swb@soton.ac.uk

Abstract — **Free stream tidal turbines are rotating bodies in fast flowing tidal currents, and as such are exposed to fluctuating loads from the surrounding fluid. These time varying forces will cause the blades to deform dynamically, potentially deflecting the blade shape away from the optimum orientation as well as exciting resonant responses that may enhance fatigue loading. It is important to understand this hydroelastic response for all but the stiffest blades. A loosely coupled, modular approach to fluid structural interactions (FSI) has been developed for the analysis of horizontal axis tidal turbine blades (HATTs). This paper discusses the methodology behind the FSI process and illustrates the technique through a case study of a 20m diameter, three bladed, horizontal axis tidal turbine, in which the deflection of the blades is examined through the iterative procedure.**

Keywords — **Fluid Structure Interactions, Hydroelasticity, Renewable Energy, Tidal Turbine.**

I. INTRODUCTION

Due to the complex loading scenario experienced by a tidal turbine, knowledge of the hydroelastic behaviour of the blades, hub, nacelle, and also the support structure is required so that structural constraints can be analysed. Additionally if the blade deformation can be designed to respond favourably to the loads the turbine is subjected to then there exists the possibility of bringing the efficiency of the device ever closer to the Betz limit [1].

The approach usually adopted, with regards to turbine blade design, is to assume that the blade can be made sufficiently rigid that the effects of blade deflection are negligible. However, the load regime experienced in areas of high tidal current (with a maximum spring peak current in excess of 2.5m/s) can require a significant increase in blade thickness/chord ratio, and corresponding fall in maximum achievable power coefficient, C_{pow}. In addition, the use of fibre reinforced composites results in structures that, although able to withstand the load, will deform significantly.

In this paper a loosely coupled method for capturing tidal turbine blade fluid structure interactions, developed at the University of Southampton, is described. The intention of this method is that it is suitably rapid that it can be used for parametric composite blade design optimisation. The use of loose coupling allows selection of either a rapid surface panel based code, or more computationally intensive Reynolds averaged Navier Stokes flow solver. The application of the process is illustrated through consideration of a three bladed 20m diameter tidal turbine.

II. FLUID STRUCTURE INTERACTIONS

Fluid-structure interactions (FSI), that is interactions of some movable or deformable structure with an internal or surrounding fluid flow, are among the most challenging multi-physics problems.

In order to gain an insight into the hydroelastic behaviour of a horizontal axis tidal turbine, practical experimentation may be used to quantify and visualise the response of the device to a multitude of environments. Experimental investigations of tidal devices, however, are extremely costly, time consuming and difficult to design in order to gain credible and useful information from the test device. Alternatively, computational investigations have the ability to be much less expensive, with regards to both time and money – although there is a propensity for both of these variables to increase with an increase in accuracy of results. It is therefore necessary to use both processes: the computational methods for the relative speed and ease of use, and the experimental work to validate the numerical modelling.

Fluid-structure interactions (FSI) occur when a fluid interacts with a solid structure, exerting pressure that causes deformation in the structure subsequently altering the flow of the fluid itself. If a situation involving structure flexure is to be analysed it is highly beneficial to couple both the fluid dynamics and the structural analysis programs to produce iterative solutions for complex problems.

There are three methods of joint fluid-structural modelling in the time domain which involve solving the governing equations in a coupled, uncoupled or integrated manner. Uncoupled simulations are computationally inexpensive but are limited to small perturbations and minimal non-linearities. Integrated methods solve the fluid and structural equations simultaneously, while coupled methods solve the equations separately but in an iterative manner [2].

Coupled methods are further subdivided into strongly or fully coupled and loosely coupled approaches; where the loosely coupled methodologies can be either integrated or modular. The integrated scheme modifies the source code of either the CFD or the FEA programs to include the coupling schemes; whilst the modular approach leaves both the FEA and CFD codes untouched, effectively making use of a "black box" which can manipulate the output of the CFD and/or the FEA and feed it into the other program respectively, thus allowing for a variety of software to be used. The key difficulty is ensuring the conservation of energy between frames of reference; from the work done by the fluid flow to the potential energy contained within the stress field of the deformed turbine.

In a loosely coupled approach the two domains are discretised to better suit the problem; as the CFD mesh will tend to require greater refinement in different areas of the structure than the FEA mesh, and vice versa. This ultimately leads to the main problem with loosely coupled analyses, the need to find a method to accurately pass boundary information from one simulation to the other.

Whilst being a highly informative technique for the design analysis of a standard fixed blade device, FSI is essential when considering adaptive composite blades as the pressure loading alters the shape more significantly when compared to standard blades. A fixed blade can simply be analysed to maximise the stiffness of the structure. In the case of a passively adaptive blade, however, variables such as fibre orientation and number of plies in the laminate have a significant effect on the design as the blade shape alters in response to the incoming flow.

In this work, a loosely coupled, modular approach was used; the flow problem and structural problem are solved successively until the change is smaller than the convergence criterion. The CFD analysis is carried out using Palisupan [3] which has been shown to be adequate for predicting the pressure distribution across a HATT blade [4]. The FEA is performed in ANSYS 12.1. The "black box" coupling the results of the fluid analysis to the structural analysis is a program based in Matlab 2009b. This program alternately manipulates both the surface pressure distribution yielded from the fluid dynamics program and the displacement data output from the structural analysis, and feeds them back into the next relevant stage of the process. This is illustrated schematically by the flow chart in Fig. 1.

The left side of the vertical blue line is indicative of the actions carried out by the CFD and FEA software, the right hand side shows the actions carried out by the "black box". The points in the flow chart illustrate an overview of the process. Similar methods to this have been shown to be successful by both Turnock and Wright [2] and Casagrande [5], and as such the concept of the use of a loosely coupled modular approach for marine structure analysis does not need to be proven, rather modified in order to operate effectively for a tidal turbine.

This paper discusses the methodology behind the FSI process and illustrates the technique through a case study of a 20m diameter, three bladed, horizontal axis tidal turbine, in which the deflection of the blades is examined through the iterative procedure.

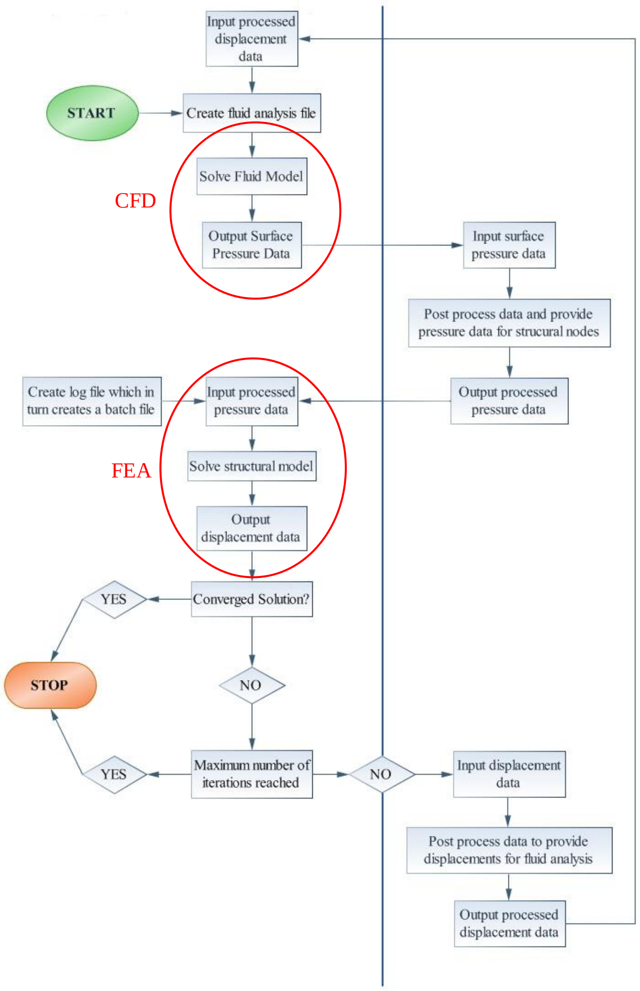

Fig. 1 Flow diagram of the Fluid Structure Interaction simulation process

III. Fluid Dynamic Analysis

The surface panel code used in this work was PALISUPAN (PArallel LIfting SUrface PANel) [3] which was initially developed to investigate ship rudder-propeller interaction. By describing a set of quadrilateral panels on the surface of a body, and distributing a set of surface sources and dipoles on the surface of the model, the code generates a numerical solution of the flow over the individual components. For a lifting body, a wake is described downstream of the body to solve for the Kutta Condition. This condition is imposed by iteration until the solution has converged to a defined maximum value.

The basic potential flow mathematical model was refined by the addition of an empirical skin friction calculation which enabled viscosity effects to be estimated. In these calculations, the actual influence of the boundary layer growth is not included although this is possible in Palisupan. However, the program does not model separation or stall. For rotating devices, such as propellers or turbines, this viscous component forms only a small amount of the total resistive torque; the majority of load arising from lift based processes. This code has been used to model the behaviour of a representative tidal turbine and good comparisons were obtained with published data, although significant problems were experienced in getting low twist sections to work due to the low pitch of the wake sheet [4].

The panel distribution over the turbine model becomes very important with relation to the accuracy of the results and the time taken for each calculation. During previous studies [4] it has been found that an optimum panel distribution can be achieved that maintains the accuracy of the result obtained with a finer distribution, but reduces the calculation time to around twenty minutes. Parameterisation and optimisation of surface panel codes is relatively simple, due to the low process times when implementing multiple runs – over 30 at a time – being highly feasible. Using a frozen wake model it is possible to reproduce the helical wake characteristic of tidal turbines. The number and distribution of panels in the wake is also very important for accurate modelling of device performance. Codes have been developed that generate optimal panel distributions over complex shapes, such as a propeller or tidal turbine, and the associated wake panel arrangement [6].

In order to run the CFD analysis, the initial blade geometry must be developed. A geometry creation program, Propgen [6], was used for this. Propgen requires the input of blade section data and section offsets, alongside the number of panels and distribution of panels over the body and wake of the geometry under analysis. The geometry is constructed from a set of section curves. These sections are generated for any given radius by defining the chord, thickness, skew, rake, pitch and 2-D section shape. The 2-D section is then mapped onto a cylindrical surface according to the specified variables using a transformation matrix. The program is constructed such that when the geometry is intended for use with surface panel codes, including a frozen wake model, an automatic helical wake sheet is generated. The appropriate pitch of the wake can be found through consideration of the axial and radial momentum changes in the flow due to the tidal turbine. It is also possible to expand the wake dependent on the energy extraction although this was found to only have a limited effect on the blade forces and has not been applied in the subsequent case study.

The final geometric output is illustrated in Fig. 2. This assembly shows the options for having a duct, hub and cap alongside the blade geometry.

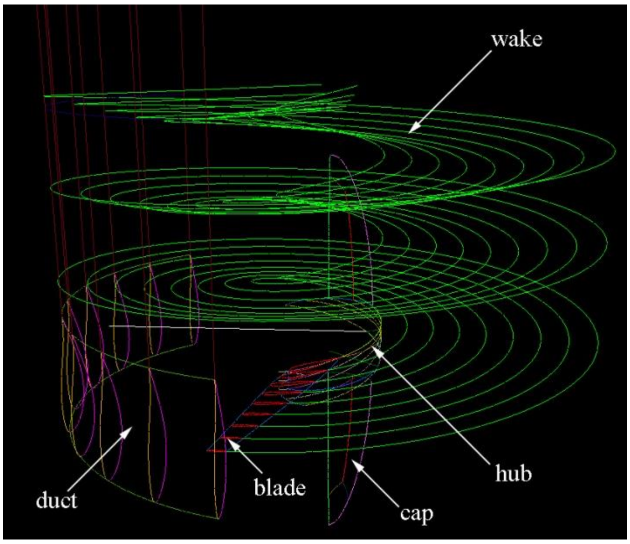

Fig. 2 Propgen geometry output

Once the blade geometry was generated it was then exported to Adaptflexi [7] in which a surface is lofted through the sections to generate the blade and the turbine is rotated such that the flow is in the positive x-direction. In Adaptflexi other variables are defined for the HATT scenario such as flow velocity, RPM, number of blades, water density, and panel discretisation; and subsequently a surface mesh created consisting of quadrilateral panels. In the case of a turbine blade, it is necessary to refine the surface panel mesh around the leading and trailing edges of the control surface in order to most accurately capture the geometry of the body around which the fluid is flowing. The number and distribution of panels in the wake is also important for accurate modelling of device performance [8]. Fig. 3 illustrates a single blade (colour) and wake panel distribution for analysis in the CFD code of a 20 metre diameter, three bladed HATT; there are in excess of 2800 panels in the blade of the turbine alone. For uniform inflows the rotational symmetry can be used and only the single blade is required to be modelled.

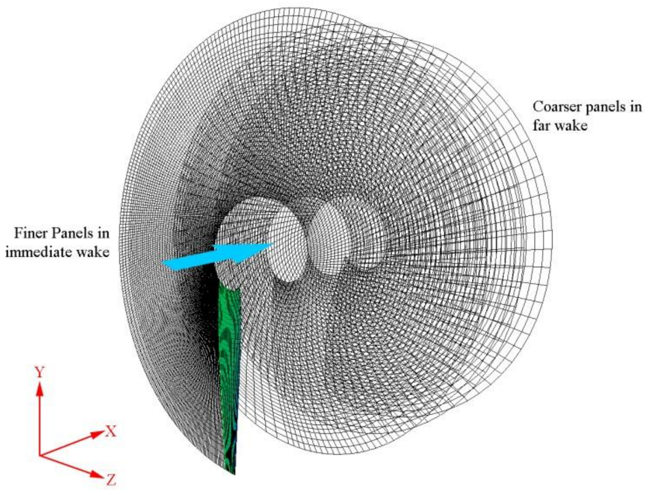

Fig. 3 Panel distribution over blade and wake

A panel sensitivity analysis was carried out for a 20m diameter, 3 bladed HATT [4]. In the immediate wake a finer panel distribution can be observed when compared to the far wake downstream. This captures the flow more precisely leading to a more accurate CFD solution. The turbine is rotationally symmetric, and therefore the body coefficients are rotated and copied twice more to simulate the whole entity. The output from Adaptflexi is in the form of a *.uns* file which contains the flow data for the simulation along with a list of panels throughout the turbine and wake model and the corresponding corner points.

The problem is then solved using Palisupan [3] where the turbine blade generator is in the positive y direction, with the onset current travelling in the positive x direction. The blade is rotating in the negative direction (anti-clockwise about the x-axis). In effect the moment about the y-axis is a measure of the torque required to rotate the blade about an axis passing through the origin. The resulting pressure loading across the surface of the blade was then calculated for input to the FEA model.

IV. Finite Element Analysis

The blade was then modelled in ANSYS 12.1 [9]. The models were meshed using the SHELL 281, 8 node, finite strain, shell element and solved using a large displacement static non-linear analysis. The SHELL 281 element is suitable for analysing thin to moderately-thick shell structures. It is well-suited for linear, large rotation, and/or large strain nonlinear applications. It is suited to layered applications for modelling laminated composite shells or sandwich construction. The accuracy in modelling composite shells is governed by the first order shear deformation theory. Each layer of the laminate is defined by the material properties and thickness – they were modelled from the inner skin out to the outer roving. Each model consisted of in excess of 200000 quadrilateral elements. This was a very fine mesh and was such that discretisation issues were not significant. This is illustrated in the following mesh sensitivity analysis.

An example blade of the form detailed in Table I was solved structurally in ANSYS 12.1 for ten different mesh densities, ranging from under 25000 elements to over 225000 elements using the SHELL 281 element. The sensitivity of solution for both the maximum bend exhibited by the central spar at different mesh densities is illustrated in Fig. 4. The solution converged as the mesh density increased, with the optimum number of elements for both accuracy of solution and minimising computational time being 200,000. The computer used was a standard desktop PC with four intel Xeon processors and 12GB of RAM.

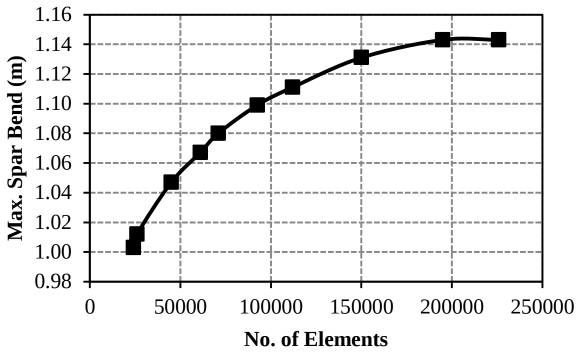

Fig. 4 Solution convergence for increasing mesh density

The geometry created for the CFD analysis is imported into ANSYS 12.1, and a central spar fitted through the blade, Fig. 5. This ensured that nodes existed at the corner points of each panel of the CFD mesh. The spar was fitted such that it supported the blade at one third of the local chord distance from the leading edge; this is where the centre of effort of the blade typically acts.

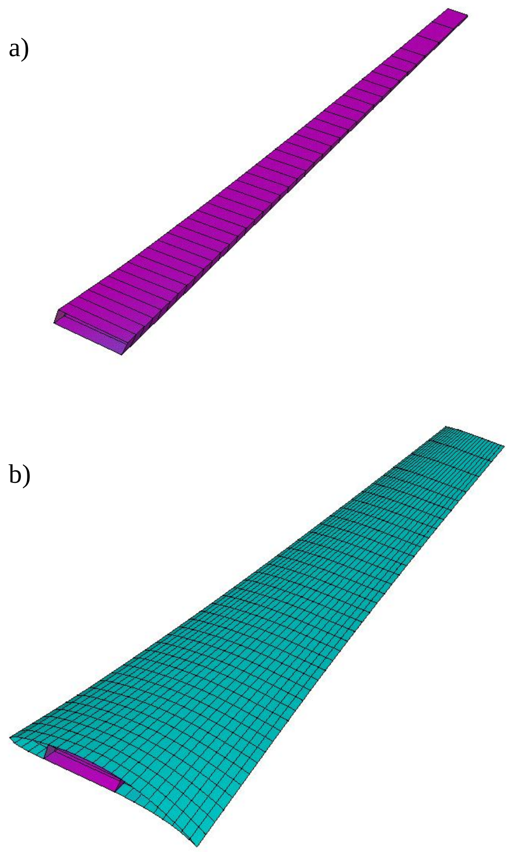

Fig. 5 Geometry of blade showing :
(a) the central spar and (b) the blade skin

The resulting geometry was then meshed and the pressure loads from the CFD analysis applied. This was achieved directly, without requiring interpolation, by virtue of the FEA geometry having the same areas mapped out as the surface panel CFD model; the pressure at the centre point of each panel is output from Palisupan and applied to the corresponding panel in the FEA model. The blade mesh was created in the smallest areas first to ensure that it was adequately dense in these regions, resulting in a higher quality of mesh overall.

The structural deformation problem was solved using a static non-linear analysis such that large deflection effects were taken into account. The maximum stress seen in the blade is monitored with the criteria that it must not exceed a third of the yield stress of the beam material; thereby incorporating a safety factor of three in to the design. While wind turbine aerodynamic loads are subjected to a safety factor of 1.35 [10], they tend to be designed using the method of partial safety factors, i.e. the overall safety factor consists of several partial factors multiplied together. Each of the partial factors accounts for one effect, e.g. uncertainty of loading, ageing of the material, manufacturing variations or material type. By using partial factors, the differences between one manufacturing method or another can be taken into account more accurately than with a simple global factor. A further factor may be added for the "consequence of failure"; thus if the integrity of a particular part is essential for the turbine's survival, that part's factor will be higher to reduce the probability of failure of the whole machine. In this work the partial safety factor method has not yet been integrated and therefore a safety factor of three was chosen in an effort to account for the through-life stresses present on the turbine and the inaccessibility for maintenance.

If this value is exceeded the number of plies in the mid-layer of the beam is altered, and the FEA rerun for the same pressure loading until a reasonable stress level is reached. The deflections are output at the nodes corresponding to the panel corner points in the CFD geometry. The deflection of the blade is then remapped into a new *.uns* input file for the CFD analysis and the problem rerun. This simulation is continued in an iterative manner until the deflection of the blade between successive runs is less than 10mm.

V. Coupling Design Tools

Coupling of both the FEA and CFD analyses has been carried out in Matlab. Fig. 6 illustrates a flow chart of the process. Initially tables of data for the range of variables under investigation are generated; these variables can include RPM, diameter, flow velocity, section shape and number of blades. The relevant data for the first run is then extracted from these tables and the Propoptions and Propeller input files for Propgen created. Propgen is then run to create the blade, hub, and wake geometry ready for CFD analysis.

A script is then generated to pass the geometry through Adaptflexi. The output required from the CFD analysis is governed through this script; pressure information, panel configuration data, and overall turbine loadings. Adaptflexi is used to generate the CFD input files and the fluid analysis solved in Palisupan. Each run through the CFD analysis takes in the order of 20 minutes.

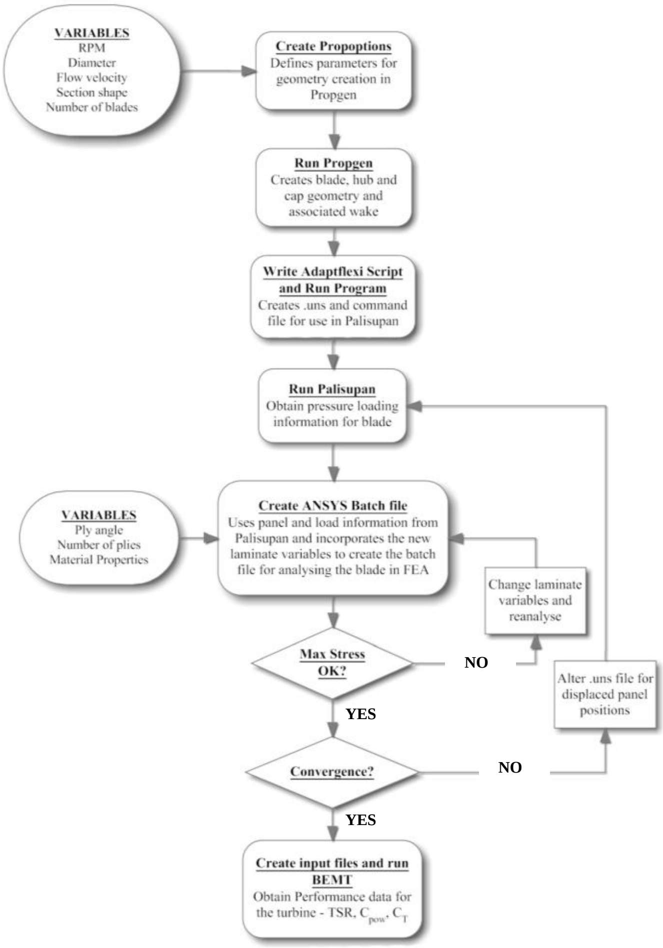

Fig. 6 Flow chart detailing the coupling process

Structural variables are now introduced in the form of composite ply angle, number of plies and material data. The intention being to create a design which maximises induced twist whilst minimising bend. The information from the CFD analysis is manipulated in order to create an ANSYS batch file. This file takes the geometry data and generates a blade within ANSYS that has a coupled box beam as the main central spar and applies the required lay-up information to the correct areas; the pressure loading is then applied and the problem solved through a non-linear static analysis to take into account large displacements. The problem takes around 30 minutes to solve. Once a solution is achieved a check is performed to make sure the maximum stress at any point in the structure does not exceed a third of the yield stress of the material of that section. If this check fails the FEA is rerun with different laminate properties until a positive result is achieved. The deformed positions of the loaded blade are then calculated through knowledge of the displacements of each node; and bend and induced twist determined.

The nodal displacements are then transferred to the *.uns* file for the relevant panel corner points. The panel list in the *.uns* file is divided into trailing edge body panels, main body panels, wake panels attached to the trailing edge, mid wake panels and far wake panels. Each panel has four corners, and the points that represent these are listed in order. The displacements of each panel on the trailing edge of the main body are known from the FEA and are evaluated as axial, radial and angular displacements in cylindrical co-ordinates and then applied to the body keypoints in the *.uns* file. These displacements are then translated back through the wake panels to give a deformed wake to match the deformed blade. The 'black box' utilises the knowledge of the *.uns* file format to search for the panels associated with the trailing edge of the main body and apply the relevant displacement to each. In turn the subsequent panels in the wake are displaced in a similar manner until the whole wake reflects the deformation of the geometry.

The problem is then run through the CFD program again to determine a new surface pressure loading for the deflected blade. This loading is applied to the deflected blade in ANSYS and the structural analysis carried out. These stages are repeated until corresponding structural analyses converge – i.e. the maximum bend in the blade is less than 10mm different from the previous run.

In order to perform a final check on the data obtained, once convergence has occurred the input files for the Blade Element Momentum Theory (BEMT) code are created from the main blade parameters and performance characteristics calculated for the device. This data is compared to that gained through the final iteration of the CFD analysis to ensure that the thrust and torque estimates are similar. In this manner a composite, bend-twist coupled blade can be designed for use on a HATT.

VI. Example Case

In order to illustrate the FSI blade design method, an example design problem has been considered. The turbine in question is to be situated in the strong tidal race in the Bristol Channel, U.K. The maximum spring peak current here is around 2.5m/s, with both the ebb and flood stages of the tide having similar velocity profiles [11]. The turbine has a diameter of 20m, a hub-diameter ratio of 0.2 and three bend-twist coupled blades. The section shape to be used for the blade is the NACA 63815 which has been previously shown to perform well [3]. The blade is of the form detailed in Table I. A three-dimensional representation of the example turbine is illustrated in Fig. 7.

TABLE I
Example case blade design data

r/R	Chord (m)	P/D	t/c
0.2	2.304	0.3365	0.0858
0.3	2.033	0.2263	0.0753
0.4	1.798	0.1593	0.0653
0.5	1.576	0.1166	0.0616
0.6	1.367	0.0866	0.0581
0.7	1.170	0.0678	0.0546
0.8	0.986	0.0556	0.0511
0.9	0.814	0.0458	0.0476
1	0.655	0.0358	0.0441

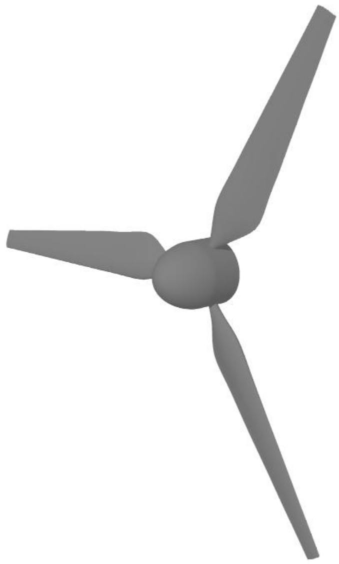

Fig. 7 3D representation of the example HATT

The bend-twist coupled spar is constructed of SE84LV UD carbon pre-preg (Table II). The SE84LV resin system was chosen as it is a toughened system, and offers good mechanical properties on a variety of reinforcing fabrics and fibres. It has a high compressive strength and is widely used in large heavily loaded components, such as yacht hulls, and spars. It has been selected for use by various America's Cup syndicates and boats racing in the Volvo Ocean Race and is, as such, suitable for a range of marine structures [12].

TABLE II
Mechanical Properties of SE84LV HEC 300/400

Property	Value
Fibre weight	300 g/m^2
Cured ply thickness	0.281 mm
E_X	129.2 GPa
E_Y	8.76 GPa
E_z	8.76 GPa
ν_{XY}	0.335
ν_{YZ}	0.0172
ν_{XZ}	0.0172
G_{XY}	5.76 GPa
G_{YZ}	5.76 GPa
G_{XZ}	5.76 GPa

The blade central spar initially has 20 plies in the mid-layer at 20^o, 30 plies in the inner skin at 45^o and 5 plies in the outer roving at 90^o – each ply has a thickness of 0.000281m. The number of plies in the mid-layer are then increased in steps of five plies until the minimum stress criteria is achieved (σ_y/3).

Fig. 8 illustrates a contour plot of the deformed blade indicating the undeformed edge, with displacement measured in metres. As expected the maximum displacement is at the tip of the blade, with a value of 1.48m. The changing contours can be observed to form diagonal stripes across the blade surface, indicating that the blade is twisting and the direction that twist is taking – clockwise around the axis down the centre of the blade (y-axis).

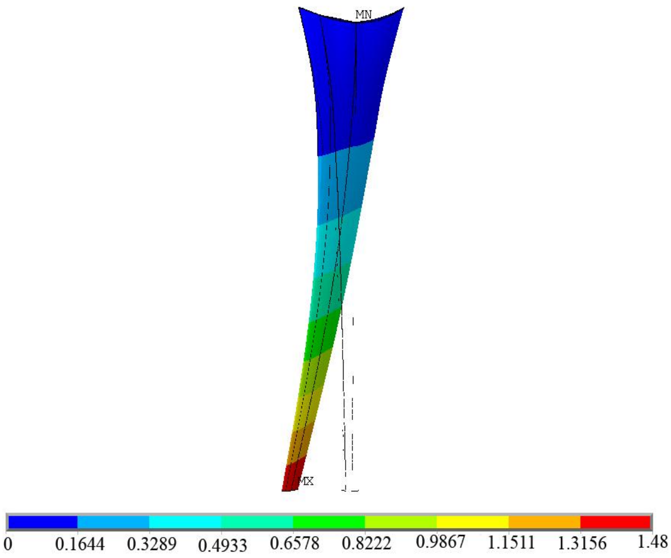

Fig. 8 Contour plot of blade displacement in metres

Fig. 9 illustrates the decrease in maximum stress in the beam with an increase in the number of plies. The maximum stress is located at the root of the blade. In this instance, each change increased the number of mid-layer plies by five.

It can be observed that, for the turbine in question, increasing the number of plies in the mid-layer over 35 ensures the structure of the beam is adequate. An increase increment of five was chosen to illustrate the procedure as each run is time consuming; however, in the case of a real life design scenario, the number of plies would be optimised to bring the maximum stress in close to the limiting criterion. This would prevent the structure being overdesigned and also minimise the amount of material (in this case carbon prepreg) that is required and hence structural weight, reducing costs and improving through-life performance. This is a relatively simple structural optimisation problem, and it is thought that further work could be carried out in order to optimise the structure fully in each layer of the beam and also the blade skin.

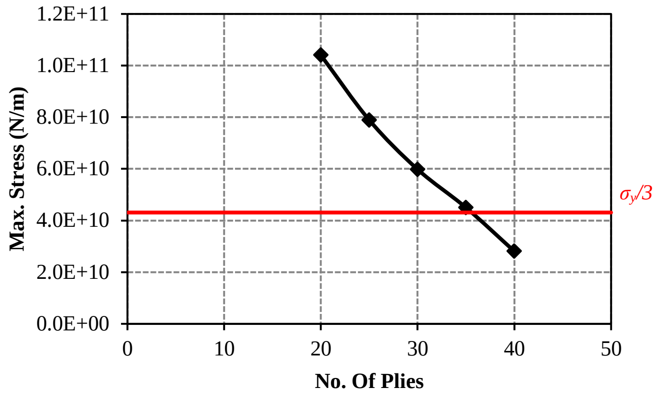

Fig. 9 Decrease in maximum stress in the blade with increase in number of mid-plies

Once the maximum stress criteria has been satisfied, the problem is then looped back through the CFD code and iterations continued until the maximum bend in the blade is less than 0.01m (10mm) greater than the previous loop. In this case this occurred after 16 iterations, Fig. 10.

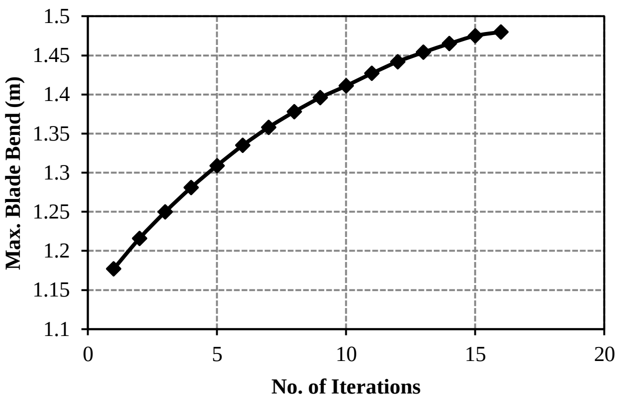

Fig. 10 Number of iterations to convergence of maximum blade bend

There is good correlation between the results from the final iteration of the CFD study and the BEMT analysis, with less than 1% difference in thrust coefficient and 2% difference in power coefficient. The main properties of the turbine, and performance results, are detailed in Table III.

It is understood that the flow regime into a tidal device at any one time is not constant, with variations in the vertical water column apparent alongside effects from turbines in arrays, and thus the blades experience changing inflow depending upon where they positioned in the area of the rotor disk. This transience should be reflected in both the CFD and FEA analyses in the design tool and is thought to be an area of further work.

TABLE II

CASE STUDY TURBINE PRINCIPAL PARTICULARS AND PERFORMANCE DATA

Property	Value	Units
Current Velocity	2.5	(m/s)
Diameter	20	(m)
Number of blades	3	(blades)
Revolutionary speed	12	(RPM)
Mid-layer ply angle	20	(^{o})
Number of mid-layer plies	40	(plies)
Thrust	232.4	(kN)
Torque	792.3	(kNm)
Bend	1.48	(m)
Induced twist	8.6	(^{o})
Maximum stress	28.1	(GN/m^2)
Thrust coefficient	0.72	-
Power coefficient	0.40	-
Absorbed power	1248.9	(kW)

The primary advantage of the use of flexible composite blades in the production of HATTs is the ability to tailor the structure to deflect in response to load changes, allowing the shape of the blade to adjust automatically to a changing inflow velocity therefore not compromising the performance over a range of conditions. There are many parameters that require optimisation in HATT blade design; not least the ones considered in this section but also blade rake, skew, pitch, section shape, and material choice and layup of both the blade skin and other regions of the bend-twist coupled central spar [1, 13]. Ultimately the blade analysis becomes complex and involves numerous time-consuming iterations of the design tool to produce an optimal design. In order to enable many iterations to be carried out multiple variable optimisation may be carried out through the use of an optimisation algorithm. The use of evolutionary optimisation techniques, such as Genetic Algorithms, allows a wide search space to be considered to find the global solution to a problem without solving for one of the many local optima within this search space. It is thought that a GA should be combined with the design tool on order to optimise the design problem whilst decreasing computational effort.

VII. CONCLUSIONS

A fluid structure interaction simulation procedure has been developed for the design of composite HATT blades. Initially both a CFD and an FE analysis are coupled together to create a holistic FSI interpretation of the flow over the blade and subsequent deformation. The CFD and FE analysis are iterated until a suitable level of convergence of blade deformation is reached.

An example problem for a 20m diameter, 3 bladed, HATT operating in a 2.5m/s tidal stream has been analysed. There is good correlation between the FSI performance data and that obtained for comparison through the use of a BEMT program.

The method could be improved by incorporating analysis of the transient inflow to the turbine, and also by the addition of an optimisation algorithm allowing large number of design variables to be optimised without the need for numerous time-consuming iterations of the design tool.

REFERENCES

[1] Nicholls-Lee, R. and S. Turnock, *Enhancing Performance of a Horizontal Axis Tidal Turbine using Adaptive Blades*, in *Oceans '07*. 2007, IEEE: Aberdeen, Scotland.

[2] Turnock, S.R. and A.W. Wright, Directly Coupled Fluid Structural Model of a Ship Rudder Behind a Propeller. Marine Structures, 2000. 13(1): p. 53-72.

[3] Turnock, S.R., Palisupan User Guide. Ship Science Report No. 100, University of Southampton, 2000.

[4] Turnock, S.R. and R.F. Nicholls-Lee, Design of Three Bladed Tidal Turbine Blades for Bi-Directional Fixed Pitch or Azimuthing Variable Pitch Operation, in Economic Viability of a Simple Tidal Stream Energy Capture Device, Log+1/Alstom/WUMTIA, Editor. 2006.

[5] Casagrande, A., *Coupled Dynamic Fluid Structural Model of a Propeller at One Time Step*, in *School of Engineering Sciences*. 2000, MSc Thesis, University of Southampton: Southampton.

[6] Pashias, C., *Propeller Tip Vortex Simulation Using Adaptive Grid Refinement Based on Flow Feature Identification*, in *School of Engineering Sciences*. 2005, University of Southampton: Southampton.

[7] Rycroft, N. and S. Turnock, *Three Dimensional Multiblock Grid Generation: Fleximesh*. 1997, University of Southampton: Southampton.

[8] Molland, A.F. and S. Turnock, *Marine Rudders and Control Surfaces*. 2007: Butterworth-Heinemann. 448.

[9] ANSYS Inc., *ANSYS 12.1 Help*. 2009, Canonsburg, Pennsylvania.

[10] International Electrotechnical Commission, *Wind Turbine Generator Systems - Part 1: Safety Requirements*. 1999, International Electrotechnical Commission: Geneva.

[11] Daruvala, J., et al., *Seapower SW Review - Resources, Constraints and Development Scenarios for Wave and Tidal Stream Power in the South West of England*, M. plc, Editor. 2004.

[12] Gurit, *SE84LV - Low Temperature Cure Epoxy Prepreg System*, Gurit. p. 6pp.

[13] Nicholls-Lee, R.F., *Adaptive Composite Blades for Horizontal Axis Tidal Turbines*, PhD Thesis, May 2011, University of Southampton