Simulation Based Optimisation of Marine Current Turbine Blades

Rachel F. Nicholls-Lee, University of Southampton, Southampton/UK, rnl@soton.ac.uk
Stephen R. Turnock, University of Southampton, Southampton/UK, srt@soton.ac.uk
Stephen W. Boyd, University of Southampton, Southampton/UK, swb@soton.ac.uk

Abstract

This paper discusses techniques for simulation based optimisation of marine current turbines, and the relative benefits and disadvantages of such methods. Blade Element Momentum codes, Computational Fluid Dynamics and Finite Element analyses, and subsequently the coupling of such techniques, are considered. The relevancy of design, search and optimisation with respect to complex fluid and structural modelling is discussed.

1. Introduction

The oceans are an untapped resource, capable of making a major contribution to our future energy needs. In the search for a non polluting renewable energy source, there is a push to find an economical way to harness energy from the ocean. There are several different forms of ocean energy that are being investigated as potential sources for power generation. These include thermal energy, wave energy, offshore wind energy, tidal energy and ocean current energy, *VanZwieten et al. (2006)*, but these can only be applied if cost-effective technology can be developed to exploit such resources reliably and cost effectively.

Tidal energy has the advantage of much less vulnerability to climate change; whereas wind, wave, and hydroelectric are more susceptible to changes in renewable fluxes brought about by shifts of climate regimes. An advantage of the tidal current resource is that, being gravitation bound, it is predictable and quantifiable both spatially and temporally. Devices designed for tidal energy extraction come in a plethora of shapes, sizes and forms although, principally, they are all harnessing either potential energy or kinetic energy from the tide, and converting it into electricity. It is the second group that renewed interest has been focused in the past few years, and it is expected to be in this category, in particular the horizontal axis tidal turbine (HATT) concept, Fig.1, that a breakthrough is made.

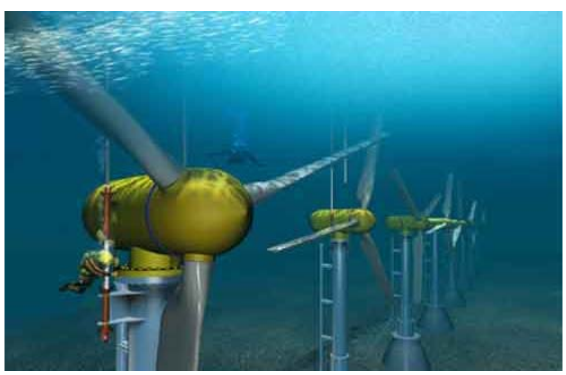

Fig. 1: A typical horizontal axis free stream marine current turbine,
(http://www.e-tidevannsenergi.com/ accessed 14/3/2008)

HATT design has to confront problems that do not occur when operating such a system in air, and as a result the blade topography will differ from those used on a Horizontal Axis Wind Turbine (HAWT). Due to differences in fluid density, for instance, the thrust on a HATT is typically over three times greater than that experienced by a HAWT of a given rated power, despite the tidal device having a significantly smaller swept area. Other forces present on a HATT include increased cyclic loads, cavitation, boundary layer interference and wave loading. The variation in static pressure and velocity across the vertical water column also impose dynamic effects on the rotor blades, *Fraenkel (2002)*.

Many tidal sites are relatively bi-directional, however, some sites can have flow reversal of 20^{o} or more away from 180^{o} such as the flow around islands, *Myers et al. (2005)*, and headlands, *Blunden et al. (2006)*, e.g.: Portland Bill, UK, where a swing upon flow reversal of around 35° from rectilinearity is apparent. It has been shown by experimentation and calculation that an increase in turbine yaw angle causes a consistent power decrease and thus a fully rectilinear flow is more desirable, *Batten et al. (2006)*.

The use of Blade Element Momentum (BEM) codes, Finite element Analysis (FEA) and Computational Fluid Dynamics (CFD) in research and development in industry has become much more commonplace. Technological advances have improved the accuracy of codes resulting in several powerful tools which, when used either singly or in conjunction with each other, can provide vital information as to the performance of a marine current turbine in varying flow conditions.

This paper aims to discuss available techniques for simulation based optimisation of marine current turbine blades, and the relative benefits and disadvantages of such methods. The use of BEM codes, CFD and FE analyses, and subsequently the coupling of such techniques, will be considered. Ultimately a discussion into the relevancy of design, search and optimisation with respect to complex fluid and structural modelling is undertaken.

2. Blade Element Momentum Methods

The basis of turbine performance can be considered in terms of the performance of an infinitely thin disk that acts to convert the kinetic energy of an onset wind or current into rotational motion. The actuator disk represents the influence of an infinite number of rotating blades which function, for an energy extractor, by causing a step change in static pressure, and hence total pressure along a streamline, whilst maintaining continuity of flow speed. The disk can be analysed in terms of the work done to convert axial momentum into rotational momentum.

This momentum conversion is controlled by the shape and orientation of the blade sections. The blade is divided into strips at a fixed radius. The effective onset flow containing the axial free stream and rotational flow determine the effective angle of attack. The blade element analysis, which uses the 2D section performance, including the influence of stall and/or cavitation, requires knowledge of the deceleration of the free stream and the imposed reverse spin (circumferential/tangential component of velocity).

Coupling together the momentum analysis and the blade element analysis using an iterative approach allows the performance at a given tip speed ratio, λ and a given radius to be calculated. A spanwise integration produces the generated torque, axial thrust and power. Blade Element Momentum (BEM) provides a rapid technique for analysis and is therefore suitable for blade geometry optimisation.

Information gained from a Blade Element Momentum (BEM) analysis consists of power, thrust and torque data for the tidal device. Computational times are very quick, of the order of fractions of seconds, and as such use of this type of analysis is commonplace at the preliminary design stage. In order to demonstrate the relevancy of the results of BEM analysis a three bladed, horizontal axis, free stream tidal turbine with a diameter of 20m and hub/diameter ratio of 0.2 have been considered, *Nicholls-Lee (2007)*. In this design scenario, the use of adaptive composite blades is being assessed – i.e. blades that are able to change shape in response to the fluid flow. Fig.2 illustrates the variation of non-dimensional power coefficient C_{pow} as a function of λ for a rigid ‘base’ rotor compared to two variable pitch and two passively adaptive blades. Two configurations were analyzed for twist as a function of blade position - constant with span (effectively a variable pitch blade) and linear with span (effectively the passively adaptive blade). Two different twisting distributions were imposed on both of these twist configurations – the first involved a linear variation with wind speed and the second a square root variation with wind speed.

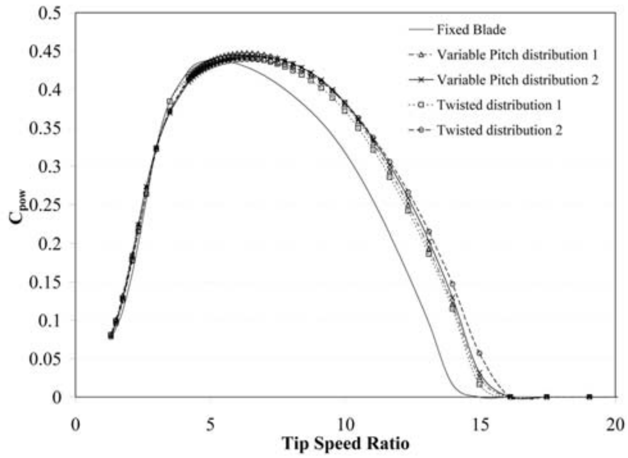

Fig. 2: Power coefficient vs tip speed ratio for the various twist distributions and configurations overlaid on the reference power curve

For a fixed RPM machine the area under the C_{pow}-λ curve is essentially a measure of the amount of power that can be generated by the turbine. All of the alternative blades cause the C_{pow}-λ curve to broaden and higher values of C_{pow} are maintained at the lower flow velocities - higher λ. All of the alternative blades show an increase in the maximum C_{pow} from the base value, representing an increase in power generation and thus annual energy capture.

The thrust force on the rotor is directly applied from the blades and hub through the support structure of the device, and thus considerably influences the design. Fig.3 illustrates the change in thrust coefficient, C_T, as a function of tip speed ratio over the various twist distributions and configurations.

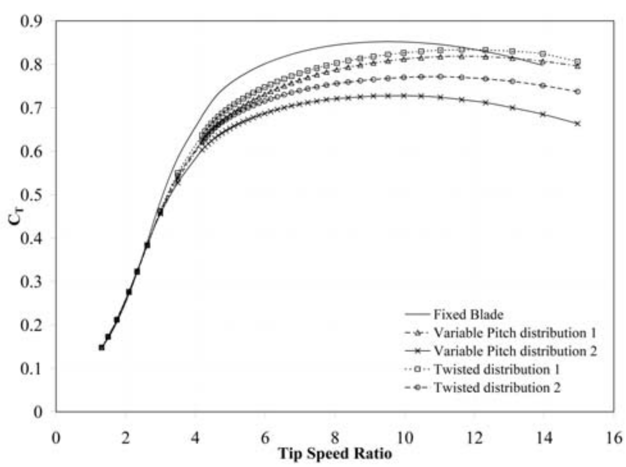

Fig. 3: Thrust coefficient vs. tip speed ratios for the various twist distributions and configurations

At moderate to high values of λ, equating to flow velocities under 3 m/s, values of C_T tend to lessen indicating less thrust produced by the turbine. Each blade configuration has increased the annual energy capture and decreased the unfavourable thrust loading. A compromise between an increase in annual energy capture and a decrease in loading on the device needs to be reached in order to produce an effective design due to the non-linear coupling between the increase in C_{pow} and the decrease in C_T.

If the tidal cycle is assumed to be a double sinusoid - one with a period of 12.4 h representing the diurnal cycle, and the other a period of 353 h representing the fortnightly spring-neap period – the flow velocity of the tidal current can be predicted, *Fraenkel (2002)*. This information, when coupled with the power data attained from the BEM analysis can be used to compute a value for the annual energy capture of the device.

Whilst overall performance data for the turbine is essential for design, it is also necessary to gain a more detailed understanding of the characteristics of the fluid flow around the device in order to optimise efficiency and energy capture.

3. Computational Fluid Dynamics

Computational fluid dynamics (CFD) uses numerical methods to solve equations that define the physics of fluid flow. CFD is a powerful tool which, when used either singly or in conjunction with other simulation tools, can provide detailed information about the local flow and hence performance of a marine current turbine in varying flow conditions. As well as obtaining the turbine performance data - local section lift and drag can be converted into integrated thrust, torque and power estimates and the surface pressure distribution on the device enabling computation of likely cavitation - CFD can give a detailed picture of the flow around the whole turbine enabling an assessment of possible environmental problems such as scour, erosion, and change in local tidal magnitude and direction, as well as providing fundamental data regarding the positioning of arrays of turbines.

3.1 Panel Methods

The fundamental basis of CFD methods are the Navier-Stokes equations defined for each fluid in a multiphase flow. These equations can be reduced by neglecting viscosity to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the irrotational, inviscid, potential flow that satisfies Laplace's equation. Use of Green's functions allows the fluid flow to be expressed as a solution of on the bounding domains of a three dimensional flow. Numerically this is implemented by a distribution of panel elements.

3.1.1 Two-Dimensional Analysis

In 2-d, a number of Panel Codes have been developed for foil analysis and design. These codes typically include viscous effects through coupling of a solution of thin boundary equations to define the outer viscous flow. Codes such as XFOIL use a conformal transformation and an inverse panel method for airfoil design. XFOIL, *Drela et al. (2001)*, is a linear vorticity stream function panel method with a viscous boundary layer and wake model and has been found to be suitable for producing section performance data and cavitation criteria for a marine current turbine at the preliminary design stage, although care should be taken to recall the apparent underestimation of drag and the overestimation of leading edge pressure coefficient, *Molland et al. (2004)*.

2-d analyses can be achieved using most CFD programs. Section performance data at this stage includes the lift and drag coefficients of differing sections from which estimates of the power, thrust and torque on the turbine rotor and structure can be attained.

Evaluation of ventilation and cavitation of marine current turbine blades are required in the design process. Cavitation inception is assumed to occur when the local pressure on the section falls to, or below, that of the vapour pressure of the fluid. Cavitation tends to occur towards the ends of the blades on the face and near the tip reducing the efficiency of the blades and thus the turbine as a whole, as well as possible erosion of the blade material. Experimental evidence suggests that tidal turbines may experience strong and unstable sheet and cloud cavitation, and tip vortices at a shallow depth of shaft submergence, *Wang et al. (2006)*. Fig.4 illustrates a model turbine in a cavitation tunnel exhibiting both sheet and cloud cavitation, and tip vortices.

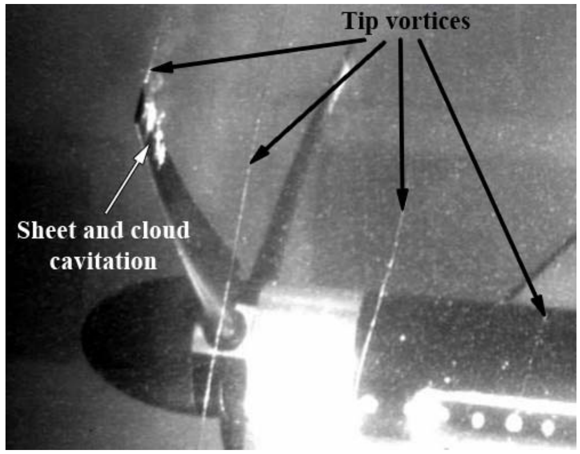

Fig. 4: Cavitation on a model turbine on test in a cavitation tunnel, *Bahaj et al. (2007)*

Cavitation number is defined as:

$$\sigma = \frac{P_O - P_V}{0.5\rho V^2} = \frac{P_{AT} + \rho g h - P_V}{0.5\rho V^2} = -C_P \qquad (1)$$

Cavitation inception can be predicted from the pressure distribution since cavitation will occur when $P_L = P_V$, or the minimum negative pressure coefficient, $-C_P$, is equal to σ. Fig.5 illustrates a typical pressure distribution over a changing foil section as the result of a two dimensional analysis.

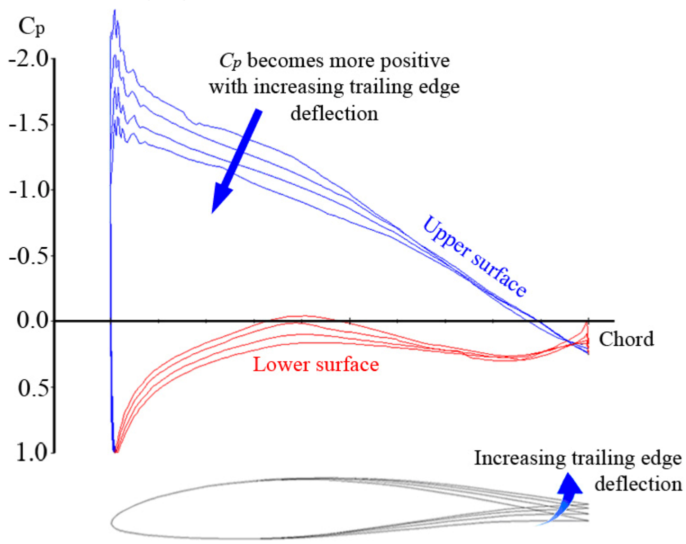

Fig. 5: Pressure distribution over the NACA 63-815 section with a variation in the deflection of the latter part of the foil at an angle of attack of 8°

The greater the pressure peak on the surface of the foil the more likely cavitation is to occur at this point. It can be observed that as the section trailing edge deflection is increased, the pressure peak decreases thus reducing cavitation inception at this angle of attack.

Some two dimensional analysis codes also provide fundamental section structural characteristics such as second moment of area, with minor modifications to the base section made within the program. This data can be used for basic structural analysis of the turbine blade which is important at this stage in the design process. One of the most prominent features of two dimensional CFD codes is the rapid computation time required to calculate section performance data – in the order of seconds. This is useful, as lift and drag data for the blade sections is required for BEM analysis, and using a process which is computationally intensive to obtain such information would be counter productive due to the swift calculations times of the BEM analysis.

The process is easy to parameterise and hence optimise due to its simplicity. Two dimensional section analyses are a powerful tool at the preliminary design stage for a tidal turbine, and should not be underestimated. It is apparent, however, that for more integral design information, a more complex code able to model more complex situations in three dimensions is required.

3.1.2 Three-Dimensional Analysis

Surface panel codes allow a more complete analysis of the performance of the turbine to be attained. Such codes calculate the characteristics of each panel over the surface of the body under analysis to produce a pressure distribution and lift and drag data for the panel, and ultimately the body as a whole. The codes can be used as a more detailed prediction of cavitation inception on the turbine blades and also as a source of detailed distribution of blade loading for further structural calculations.

Surface panel codes are more computationally intensive than two dimensional analysis methods. The selection of the correct panel distribution over the turbine model is important with relation to the accuracy of the results and the time taken for each calculation. During previous studies, however, it has been found that an optimum panel distribution can be achieved that maintains the accuracy of the result that comes with a finer distribution but reduces the calculation time to around twenty minutes. Paramaterisation and optimisation of surface panel codes is relatively simple, due to the low process times implementing multiple runs – over 30 at a time – is feasible. Using a frozen wake model it is possible to reproduce the helical wake characteristic of pre-stall marine current turbines. Checks as to local section behaviour are required as the onset of local stall is not captured by a potential flow analysis

As an example the performance of the alternative turbine blades assessed in the previous section was analysed by observing the variation of the minimum surface pressure coefficient at each blade span as a measure of likely cavitation inception. The turbines were modelled at a single tip speed ratio close to maximum (λ=5, V=2.1 m/s). In these calculations a frozen wake model was used with in excess of 2800 panels distributed over the blade and the hub.

Fig.7 shows the pressure coefficient, C_P, as a function of turbine radius for the various twist distributions and configurations.

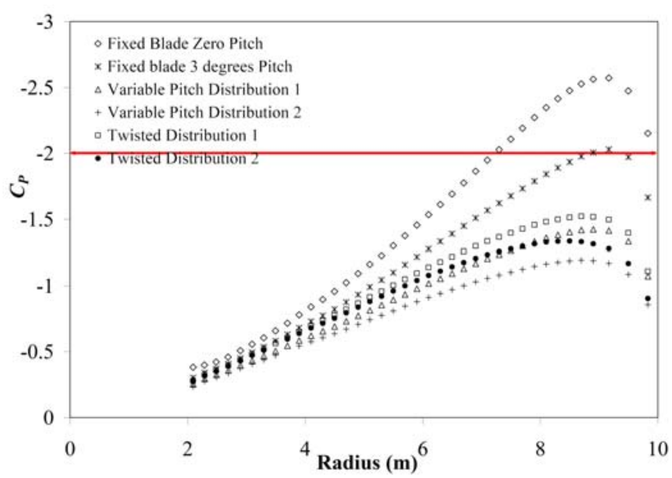

Fig. 7: C_p as a function of turbine radius for the various twist distributions and configurations

At this flow velocity, the cavitation number is approximately 2; indicating that cavitation inception will occur on areas of the blade where the pressure coefficient is less than -2. Both of the fixed blades exhibit a maximum negative C_P of less than -2, and thus these blades will cavitate at this flow velocity. Imposing a twist configuration on the blade causes the C_P over the outer two thirds of the blade to decrease significantly, and therefore the likelihood of cavitation has been significantly reduced.

Although it is possible to predict cavitation inception, once cavitation has occurred the use of surface panel techniques which include the presence of cavitation regions requires significant use of empirical information to define cavitation bubble shape, and analysis often becomes unstable unsuited to automated optimisation. Such codes also struggle to capture severe changes in the flow regime, i.e. separation, stagnation and recirculation, even when using coupled boundary layer approaches. It is therefore apparent that more advanced numerical simulation of the area around the turbine is necessary for a full design.

3.2 Reynolds Average Navier Stokes Equations

The Reynolds-averaged Navier-Stokes (RANS) equations are time-averaged equations of motion for fluid flow. They are primarily used while dealing with turbulent flows. These equations can be used with approximations based on knowledge of the properties of flow turbulence to give approximate averaged solutions to the Navier-Stokes equations.

The nature of RANS equations leads to the need for complex domain discretisation schemes as well as complex modelling with large numbers of elements or cells. This often leads to complex mesh structures on which the equations must be solved, and building such meshes is time consuming.

Turbulent flows contain many unsteady eddies covering a range of sizes and time scales. The RANS equations are averaged in such a manner that unsteady structures of small sizes in space and time are eliminated and become expressed by their mean effects on the flow through the Reynolds, or turbulent, stresses. These stresses need to be interpreted in terms of calculated time-averaged variables in order to close the system of equations thereby rendering them solvable. This requires the construction of a mathematical model known as a turbulence model, involving additional correlations for the unknown quantities, *Atkins (2003)*.

3.2.1 Turbulence Models

Most flows of practical engineering interest are turbulent, and the turbulent mixing of the flow then usually dominates the behaviour of the fluid. The turbulent nature of the flow plays a crucial part in the determination of many relevant engineering parameters, such as frictional drag, flow separation, transition from laminar to turbulent flow, thickness of boundary layers, extent of secondary flows, and spreading of jets and wakes.

It is possible to solve the Navier Stokes Equations directly without any turbulence model, using Direct Numerical Simulation (DNS). This requires that the whole range of spatial and temporal scales of the turbulence must be resolved, however this approach is extremely computationally expensive for complex problems, hence the need for turbulence models to represent the smallest scales of fluid motion.

The simplest turbulence modelling approach rests on the concept of a turbulent viscosity. Such models are widely used for simple shear flows such as attached boundary layers, jets and wakes. The one-equation models attempt to improve on the zero-equation models by using an eddy viscosity that no longer depends purely on the local flow conditions but takes into account the flow history, *Atkins (2003)*.
Two-equation turbulence models are frequently used. Models like the k-ε model, *Launder (1974)*, and

the k-ω model, *Wilcox (1998)*, have become industry standard models and are commonly used for most types of engineering problems. By definition, two-equation models include two extra transport equations to represent the turbulent properties of the flow. This allows a two-equation model to account for history effects like convection and diffusion of turbulent energy. In the field of renewable energy it is the k-ε model that has been found to be most useful, being able to be performed on most desktop PCs whilst coupling an acceptable level of accuracy with reasonable computational times.

The two-equation turbulence models are reasonably accurate for fairly simple states of strain but are less accurate for modelling complex strain fields arising from the action of swirl, body forces such as buoyancy or extreme geometrical complexity. Several alternative models have been proposed, for example, Reynolds stress transport models, Large eddy simulation (LES), and Detached-eddy simulation (DES), *Spalart (1997)*), though these are used infrequently due to the long computational times and the requirement of exceptionally powerful computing hardware in order to process the data.

It should be considered that there is no universally valid general model of turbulence closure that is accurate for all classes of flows. Validation and calibration of the turbulence model is necessary for all applications. In the context of marine current turbines this can be achieved through wind tunnel testing, tank testing and open water tests.

4 Structural Analysis

The hydrodynamic loading on the tidal turbine causes shear forces and bending moments to increase from tip to root. Depending on the magnitude of the blade buoyancy, the blade will experience a periodic variation in vertical force and associated moment. Centrifugal loading will also be present although as rotational speeds are small, this will not play a significant role. If the turbine axis is yawed relative to the onset flow the blade will experience a periodically varying, axial, circumferential and radial load. Blade fatigue will be an important design consideration as typically a blade would experience $1x10^8$ cycles over a 20 year device life. Fig.8 illustrates a simplified distribution of shear force and bending moment over the length of a single free stream tidal turbine blade.

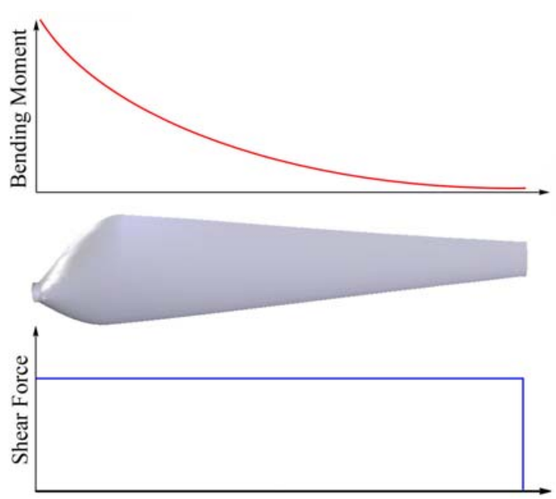

Fig. 8: Shear force and bending moment over the span of a turbine blade

The largest moments on the turbine are in the vicinity of the hub, and the magnitude of the local stresses critically depends on whether a controllable pitch or fixed pitch system is implemented. As a controllable pitch device is required to rotate, a blended zone is necessary between the outboard hydrodynamic section and a circular cross-section, and associated pitch mechanism and support bearings. Experience from wind turbines would suggest that the sizing of the pitching mechanism and bearing design should be driven by imposed axial loading, i.e. thrust, rather than the hydrodynamic moment about the blade generator. The cross-section of the blade is not equally thick in all directions and thus the direction of the loading will be of great importance; this, relative to the cross-section, will depend on the direction of the net force, and on the angle of the section, both of which vary along the span of a turbine blade. The direction in which the blade will bend is dependent upon the direction of the net loading and the second moment of area of the blade section.

With a fixed pitch blade the loading scenario is much less complex, due to the lack of machinery required in the hub which is present in the variable pitch blades. A simple structural analysis is sufficient to give a basic understanding of the loads encountered at this point.

The subject of Finite Element Analysis (FEA) has been used in the field of engineering for over half a century and has been continuously developed throughout this time. It has only been in the last few decades, with improvements in computational efficiency and power, that use of FEA has become more commonplace. FEA is based on subdividing the structure under investigation into elements, each element being of simpler geometry and thus easier to analyse than the whole structure. In essence, the complex solution is approximated by a model consisting of piecewise-continuous simple solutions, *Cook et al. (1989)*.

Various different types of code exist, although these tend to differ with respect to the discipline under examination (i.e. structural, thermal, electromagnetic and so forth) with additions to the equations, rather than in the mathematical formulations on which the code is based. The main advantage of this type of analysis is that complex design forms can be studied whereby an analytical solution is either not possible or not feasible. It is possible to undertake dynamic analyses, in order to simulate complex problems such as unsteady loading and time varying excitations.

At the preliminary design stage, uncomplicated FE analyses can be carried out by imposing a point load the tip of the blade, and subsequently a uniformly distributed load over the length of the beam. This will ensure that the structure can be assumed to be a good basis from which improvements can be made. For more accurate modelling, the pressure distribution over the blade due to the fluid flowing over it can be attained using CFD and then input into the FEA code. The more complex the loading, and hence the model, becomes the more computationally intensive the problem. However, the structural mesh will differ considerably from a fluid mesh, and is likely to be simpler and therefore use less time to create.

As for any body, the tidal turbine is subjected to pressure form the surrounding fluid. In addition to this it is a lifting body with rotational motion; hence it is subjected to further loading because of the lift it produces, and because of its rotation. 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 under this regime could lead to a more thorough understanding of structural constraints and how performance of the turbine could be improved in order to bring the efficiency of the device ever closer to the Betz limit of 0.593.

5 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 important and, with respect to both modelling and computational issues, the most challenging multi-physics problems.

FSI occurs when a fluid interacts with a solid structure, exerting pressure that may cause deformation in the structure and, thus, alter the flow of the fluid itself. If a problem involving structure flexure, or possibly adaptive materials 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.

In the context of a tidal turbine blade, the flow interaction and flow field from the components of the device are complex, and produce a unique time-varying load distribution. Coincidentally, the complex shapes and joining of the individual parts of the turbine, creates a highly non-linear stress distribution. Due to these complexities the hydrodynamic performance of the turbine cannot be determined from the application of foil theory, nor can the structural solution be gained from standard closed-form analytical solutions.

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, 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 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.

There are three methods of joint fluid structural modelling in the time domain. These involve solving the governing equations in a coupled, an uncoupled or an integrated manner. Uncoupled simulations are computationally inexpensive but are rarely used as they are limited to small perturbations and minimal non-linearites. For non-linear flows and large perturbations a fully coupled or integrated method should be used. Integrated methods solve the fluid and structural equations simultaneously, while coupled methods solve the equations separately but in an iterative manner, *Turnock and Wright (2000)*.

Coupled methods are subdivided into strongly or fully coupled (single-domain) and loosely coupled (independent domains) approaches. 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 kinetic energy of the fluid flow to the potential energy contained within the stress field of the deformed blade.

The main advantage of the loosely coupled approach is that advanced purpose designed codes can be used for the tackling of specific problems. In additions to this 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.

For the analysis of horizontal axis tidal turbines, a loosely coupled, modular approach is likely to be most successful. With such an approach, ultimately it is immaterial which CFD and FEA programs are used, as the pre and post processing of results is carried out in the "black box" phase of the simulation, and this may be tailored to suit any combination of software. Similar methods to this have been shown to be successful for propeller design by both *Turnock and Wright(2000)* and *Casagrande (2000)*, and as such the concept of the use of a loosely coupled modular approach for marine structure analysis does not need to be proven, but can be modified in order to operate effectively for a tidal turbine.

To carry out high-quality trade-off studies, designers must synthesize and analyze alternative design configurations. To do this cost effectively and quickly requires tools that support automation, evolutions and innovation. Automation stems mainly from the desire to reduce the high costs associated with professional manpower and, at the same time, to reduce design cycle times. A variety of technologies are coming together in providing a new class of tool that automatically optimizes designs based on multiple variables. Mechanical design synthesis is a next-generation solution combining optimization technologies with CAE simulation methods and parametric CAD into an integrated solution. These types of tools find that optimal part dimensions for resonant frequency is below a certain level, for example, or weight and stress are minimized.

Automated design is now usable (with appropriate care) for relatively straightforward, single discipline problems, however improvements are needed in automatic meshing of complex geometries. CAD geometry parameterization is likely to offer benefits for multidisciplinary optimisation. Engineering judgment in the modelling assumptions, design parameters and design targets is crucial, *Chew et al. (2006)*.

6 Design, Search and Optimisation

Design search and optimisation is the term used to describe the use of formal optimisation methods in design, *Keane et al. (2005)*. The phrase 'to optimise' is the process of finding the solution to a problem, which gives the best results with respect to certain decisional criteria, through varying a limited number of variables, and respecting certain constraints. Generally, the optimisation process is the search for the absolute maximum (or minimum) of a function, which depends on certain variables, respecting certain constraint equations, *Campos et al. (2006)*. Fig.9 illustrates the "classical" optimisation problem, where the global optimum needs differentiating from the local optimum.

Often optimising the design for one variable adversely affects the configuration according to other variables, e.g. minimizing weight and resulting material costs could lower durability. The traditional trial and error approach requires that numerous loops of the design spiral are undertaken which, when using CFD and especially FSI, are both computationally expensive and time consuming. There is therefore an increasing need to use advanced optimisation software to help achieve an optimum design or solution with the minimum effort.

Optimisation algorithms can be classified in different ways. Firstly a distinction can be made between gradient based algorithms and stochastic algorithms, a second between mono-objective algorithms and multi-objective algorithms. Each type of algorithm is applicable to certain design problems, and it is essential to use the correct algorithm for each case in order to determine accurately the global optimum and not any number of local optima that may be present. For example in Fig.9, a gradient approach is as likely to solve to the local optimum as it is to the global optimum, whereas a multi-objective algorithm can differentiate between the two.

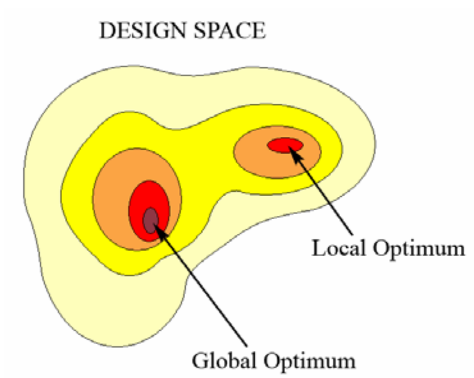

Fig. 9: The classical optimisation problem

The accuracy, robustness and convergence velocity of algorithms are also important. Robustness is the algorithm's capability to find the absolute maximum of the objective function. The accuracy is the algorithm's capability to reach a value as close as possible to the real value of the objective function maximum. The convergence velocity is the number of iterations required to reach the convergence, *Campos et al. (2006)*.

Other important concepts of the optimisation theory are 'Design of Experiment' (DOE), Statistical analysis and Response surfaces. The first two are useful in every optimisation process and particularly if they are used together. Relationships among different variables or among variables and objectives can be selected and the most interesting areas of the objective functions domains may be localised, thus reducing the optimisation calculation time. Response Surfaces are very powerful tools when the calculation time of each single design in an optimisation process is high, a key feature of complex CFD calculations and most FSI coupled problems. A Response Surface approximates the real behaviour of the objective function within its domain and so the total optimisation time decreases.

Most DOE methods seek to efficiently sample the entire design space by building an array of possible designs with relatively even but not constant spacing between the points. In contrast to interpolating data to find results, the data in RSM is regressed to find the global optimum. Traditional methods tend to be less capable of distinguishing between the myriad of local basins and bulges that can occur in more complex engineering problems. A Kriging approach allows the user to control the amount of regression as well as accurately model the user data. It also provides measures of probable errors in the model being built that can be used when assessing where to place any further design points. It also allows for the relative importance of variables to be judged, *Keane et al. (2005)*.

Fig.10 illustrates a relatively simple composition of trigonometric functions with imbedded polynomial arguments. Under such circumstances, it is essential to use a proper global search strategy. Furthermore, instead of 'exact' solutions, most typically one has to accept diverse numerical approximations to the globally optimal solution (set) and optimum value.

It is thought that for the case of a tidal turbine, where in excess of twenty variables need to be optimised, the use of a Genetic Algorithm (GA) would be the most effective form that design optimisation could take. For initial cases, where the number of variables can be reduced, the more simplistic approach of DOE may be utilised, however for greater accuracy and a more thorough optimisation the GA is favoured.

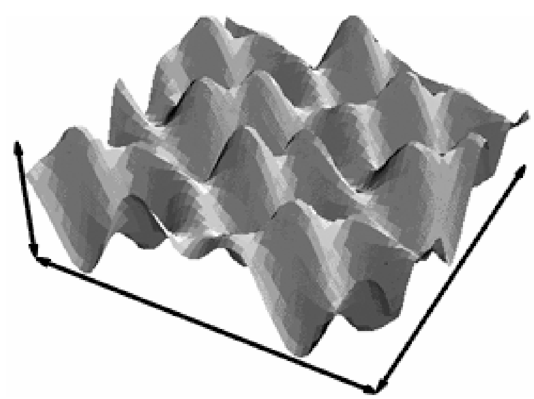

Fig. 10: More realistic design space for an engineering problem with many local and global maxima and minima

GA's are based on models of Darwinian evolution, that is, survival of the fittest. Essentially a pool of solutions, the *population*, is built and analysed and then used to construct a new, improved, pool by methods that mimic those of natural selection, *Keane et al. (2005)*. Key among these ideas are those of fitness (or mutation), crossover, and selection; that is, designs that are better than the average are more likely to contribute to the next generation than those that are below average and that new designs are produced by combining ideas from multiple "parents" in the population, which provides a mechanism for exploring the search space and taking successful ideas forward.

Sobey (2007) researched the use of GAs to optimise the design of stiffened panels. The stochastic method allows a large design space to be searched, although with the disadvantage that only a near optimum result is sometimes found. Through the use of combining the elite results from a given generation and taking these on to the next it is ensured that the algorithm will evolve its way towards the optimum result. The level of mutations and crossover can be changed to increase the spread of the search space, or decreased so that the optimum value can be found as quickly as possible. It is the use of mutation that ensures that the GA will not search a local optimum, but will gradually investigate results from the entire search space to find the global optimum. The GA has developed by *Sobey (2007)* is still undergoing further refinement. It is intended that, with some minor alteration, this algorithm be tried and assessed for use in the optimisation of horizontal axis tidal turbine blade design.

7 Overview

This section illustrates the manner in which the computational analysis methods discussed previously in the paper can be utilised with regards to the optimisation of a free stream marine current turbine. Expected timings are based on calculation times on a typical desktop PC workstation.

1) Using previous work and background knowledge choose initial airfoil sections for the blade with a selection based on an expected thickness/chord variation form root to tip.
2) Analyse these sections using a two dimensional coupled panel code in order to determine which of these sections performs most efficiently and obtain lift and drag data for the foil for use with BEM analysis – 5 minutes
3) Carry out a BEM analysis, using an automated process to optimise the performance turbine with regards to number of blades, diameter, pitch, and chord and twist distribution, for a range of tip speed ratio etc. – undertaking approximately 32,000 blade shape variations, the complete C_{pow}-λ performance can be determined in around 10 minutes.
4) For the best case for design C_{pow} use a three dimensional panel code at a single tip speed ratio to check the likelihood of cavitation inception – 10 minutes.
5) Use a two dimensional panel code to make modifications to the blade tip sections. Repeat steps 2-4 as necessary.
6) Carry out simple FEA with the blade under a point load at the tip and subsequently a uniformly distributed load in order to assess the structural stability of the initial device – 1 hour. Steps 2-4 may need to be carried out again in order to meet initial structural constraints.
7) Undertake steady RANS simulation with around 1.5 million cells in order to check the flow effects due to the presence of a three dimensional hub, the behaviour of the blade tips, and check possibility of cavitation inception – 6 hours.
8) Detailed design of the internal blade structure. More thorough FEA simulation of the blade, using the surface pressure data calculated during the CFD simulation for the load distribution. Repeat until blade is determined to be sufficiently strong.
9) Couple surface panel and FE analyses to run through a series of spatially varying inflow fields with variations due to waves, support structure, boundary layer, proximity of the free surface etc. Identify fatigue loadings present in the design.
10) Final design check using a full unsteady RANS calculation of the design running beneath the free surface within the actual bathymetry of a proposed channel site at the maximum spring peak current.

At stages 8 and 9 it could be beneficial to make use of a GA to optimise the design for a number of variables as calculations times at this point are substantial.

8 Conclusions

With the need for renewable energy sources becoming ever more important, a focus is being brought to predictable and quantifiable marine sources such as marine currents, or tides. The design and optimisation of tidal energy extraction devices is paramount, to ensure they are rugged, robust and reliable and yet effective at capturing maximum available energy in the hostile sub sea environment.

CFD is a powerful tool which, when used correctly, can provide valuable data regarding the performance of such devices. It is important not to underestimate the use of simpler CFD techniques, such as panel codes, at the preliminary design stage where an insight into cavitation characteristics and energy extraction can be achieved, justifying the need for further work. At a more advanced design stage RANS solvers are required to model the complex flow situations occurring around the turbines. Ultimately coupled fluid-structural analysis is required to better understand how the flow affects the structural integrity of both the rotor and supporting structure.

Structural simulation is essential in order to sound out stress concentrations, determine wall thicknesses, and areas that are susceptible to fatigue. FSI is particularly useful to both analyse and visualise how the blade will respond to the complex varying loads imposed upon it both through vertical and horizontal pressure and velocity fluctuations.

Design, search and optimisation play a key role in the use of computationally expensive processes such as CFD and FEA, and especially FSI. The proper use of optimisation algorithms could significantly reduce the number of design iterations required, producing optimal answers without the expense of huge amounts of both computational and human time.

An example of the manner in which all of the computational simulation methods can be brought to together to optimise a free stream, horizontal axis, marine current turbine has been suggested.

Whilst all the methods discussed in this paper require validation, be it through use of wind tunnel tests, towing tank data or open ocean experiments, ultimately the use of CFD, FSI and design, search and optimisation could cut design process times and allow more effective selection of model scale devices as well as greater confidence in the effectiveness of the blade structural design.

Acknowledgements

The authors gratefully acknowledge the support of the School of Engineering Sciences for part funding the PhD studentship of Ms. Nicholls-Lee.

References

ATKINS, W. (2003), *MARNET best practice guidelines for marine applications of computational fluid dynamics,* MARNET

BAHAJ, A.S.; MOLLAND, A.F.; CHAPLIN, J.R.; BATTEN, W.M.J. (2007), *Power and thrust measurements of marine current turbines under various hydrodynamic flow conditions in a cavitation tunnel and a towing tank,* J. Renewable Energy 32(3): pp.407-426

BATTEN, W.M.J.; BAHAJ, A.S.; MOLLAND, A.F.; BLUNDEN, L.S. (2006), *Yawed performance of horizontal axis marine current turbines*, in *conference on renewable energy in island maritime climate*, Dublin

BLUNDEN, L.S.; BAHAJ, A.S. (2006), *Initial evaluation of tidal stream energy resources at PortlandBbill, UK,* J. Renewable Energy 31, pp. 121-132

CAMPOS, F.; WESTON, S.; SCHUMACHER, T. (2006), *Automatic optimisation of CFD engineered designs*, Automated Design & Optimisation Techniques using CFD, IMechE, London

CASAGRANDE, A. (2000), *Coupled dynamic fluid structural model of a propeller at one time step*, in *school of engineering sciences*, University of Southampton

CHEW, J.; DOHERTY, J.; GILLIAN, M.; HILLS, N. (2006), *Practical applications of automated design and optimisation techniques using CFD*, Automated Design & Optimisation Techniques Using CFD, IMechE, London

COOK, R.; MALKUS, D.; PLESHA, M. (1989), *Concepts and applications of finite element analysis,* 3rd Edition, John Wiley & Sons

DRELA, M.; YOUNGREN, H. (2001), *XFOIL 6.94 User Guide*

DTI (2003), *Energy from tidal barrages technology route map*, Department of Trade and Industry

FRAENKEL, P.L. (2002), *Power from marine currents,* J. Power and Energy 216(1), pp.1-14

KEANE, A.; NAIR P. (2005), *Computational approaches for aerospace design the pursuit of excellence*, Chichester: John Wiley & Sons, Ltd.

LAUNDER, B.; SPALDING, D. (1974), *The numerical computation of turbulent flows,* Computer Methods in Applied Mechanics and Engineering 3, pp.269-289

MOLLAND, A.F.; BAHAJ, A.S.; CHAPLIN, J.R.; BATTEN, W.M.J (2004), *Measurements and predictions of forces, pressures and cavitation on 2-d sections suitable for marine current turbines,* Proc. Institute of Mechanical Engineers 218(M), pp.127-138

MYERS, L.; BAHAJ, A.S. (2005), *Simulated electrical power potential harnessed by marine current turbine arrays in the Alderney race,* J. Renewable Energy 30, pp.1713-1731

NICHOLLS-LEE, R.; TURNOCK, S. (2007), *Enhancing performance of a horizontal axis tidal turbine using adaptive blades*, Oceans'07, IEEE, Aberdeen

SOBEY, A. (2007), *Concurrent engineering in fibre reinforced boats*, 9 Month Report, Fluid Structure Interactions Research Group, School of Engineering Sciences, University of Southampton

SPALART, P.; JOU, W.; STRELETS, M.; ALLMARAS, S. (1997), *Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach*, in 1st AFOSR Int. Conf. on DNS/LES, Greyden Press: Rustin, LA

TURNOCK, S.R. (2000), *Technical manual and user guide for the surface panel code: PALISUPAN..* Ship Science Report No. 100, University of Southampton

TURNOCK, S.R.; WRIGHT, A.M. (2000), *Directly coupled fluid structural model of a ship rudder behind a propeller,* Marine Structures 13(1), pp.53-72

VANZWIETEN, J.; DRISCOLL, F.R.; LEONESSA, A.; DEANE, G. (2006), *Design of a prototype ocean current turbine - part i: mathematical modelling and dynamics simulation,* Ocean Engineering

WANG, D.; ATLAR, M. (2006), *Experimental investigation on cavitation performance, noise characteristics and slipstream wash of an ocean stream turbine*, World Maritime Technology Conference. IMarEST: London

WILCOX, D. (1998), *Turbulence modelling for CFD*. 2nd ed.: DCW Industries