The Use of Computational Fluid Dynamics in the Optimisation of Marine Current Turbines

R. F. Nicholls-Lee, S. R. Turnock

Abstract—**The use of Computational Fluid Dynamics (CFD) in research and development in industry has become much more commonplace. Technological advances have improved the accuracy of codes although this is at the expense of computational power. CFD is a powerful tool if implemented correctly, and in order to do this it is important to understand when to use the different levels of code. This paper illustrates the relative merits of codes ranging from simple three dimensional panel codes to Reynolds Averaged Navier stokes equations with regards to the optimisation of marine current turbines. It goes on to discuss turbulence models, fluid structure interactions and ultimately design, search and optimisation.**

Index Terms—**Computational Fluid Dynamics, renewable energy, marine current turbine.**

NOTATION

σ	Cavitation number
P_O	Reference static pressure (N/m^2)
P_V	Vapor pressure (N/m^2)
ρ	Water density (kg/m^3)
V	Free stream velocity (m/s)
P_{AT}	Atmospheric pressure (N/m^2)
g	Acceleration due to gravity (m/s^2)
h	Head of water (m)
C_P	Pressure coefficient

I. 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 [1], but these can only be applied if the technology can be successfully developed to exploit such resources reliably and cost effectively.

Tidal energy has the advantage of invulnerability to climate change; whereas wind, wave, and hydro are all susceptible to the unpredictable 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 this category that a breakthrough is made. Figure 1 illustrates a typical horizontal axis free stream marine current turbine.

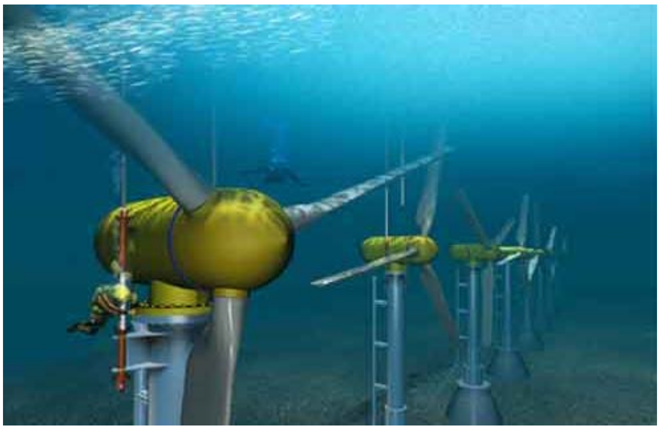

Fig. 1. A typical horizontal axis free stream marine current turbine.

Horizontal Axis Tidal Turbine (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 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 interesting dynamic effects on the rotor blades [2].

Many tidal sites are relatively bi-directional, however, some sites can have flow reversal of 20° or more away from 180° such as the flow around islands [3] and headlands [4] e.g.: Portland Bill, UK, where a swing upon flow reversal of around 35° from rectilinearity is apparent. It has been shown

---

R. F. Nicholls-Lee is with the University of Southampton, Southampton, SO17 1BJ, UK (phone: +44 (23) 8059 6626; fax: +44 (23) 8059 3299; e-mail: rnl@soton.ac.uk).

S. R. Turnock is also with the University of Southampton (e-mail: srt@soton.ac.uk).

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 [5].

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. CFD is a powerful tool which, when used either singly or in conjunction with other tools, can provide vital information as to the performance of a marine current turbine in varying flow conditions. As well as obtaining the turbine performance data, lift and drag that can be converted into thrust, torque and power estimates, and also pressure distribution on the device enabling computation of likely cavitation, CFD can give a detailed picture of the flow around the turbine enabling a more advanced outlook on possible environmental problems such as scour, erosion and the change in tidal magnitude to be understood and also provides vital data regarding the positioning of tidal device arrays.

This paper aims to discuss the benefits and disadvantages of the more common CFD techniques and turbulence models. It will then proceed to consider the further uses of CFD in conjunction with other analysis techniques such as fluid structure interactions. Ultimately a discussion into the relevancy of design, search and optimisation with respect to complex fluid modelling is undertaken.

II. PANEL METHODS

The fundamental basis of any CFD problem are the Navier-Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, these equations can be linearised to yield the linearised potential equations.

A. Two Dimensional Analysis

Historically, methods were first developed to solve the Linearised Potential equations. Two-dimensional methods, using conformal transformations of the flow about a cylinder to the flow about an airfoil were developed in the 1930s; the computer power available paced development of three-dimensional methods.

In the two-dimensional realm, quite a number of Panel Codes have been developed for airfoil analysis and design. These codes typically have a boundary layer analysis included, so that viscous effects can be modelled. Some incorporate coupled boundary layer codes for airfoil analysis work. Codes such as XFOIL use a conformal transformation and an inverse panel method for airfoil design. XFOIL 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 [6], although care should be taken to recall the apparent underestimation of drag and the overestimation of leading edge pressure coefficient [7].

Two dimensional analyses can be achieved using most CFD programs, although some are more suited to the technique. 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 [8]. Figure 2 illustrates a model turbine in a cavitation tunnel exhibiting both sheet and cloud cavitation, and tip vortices.

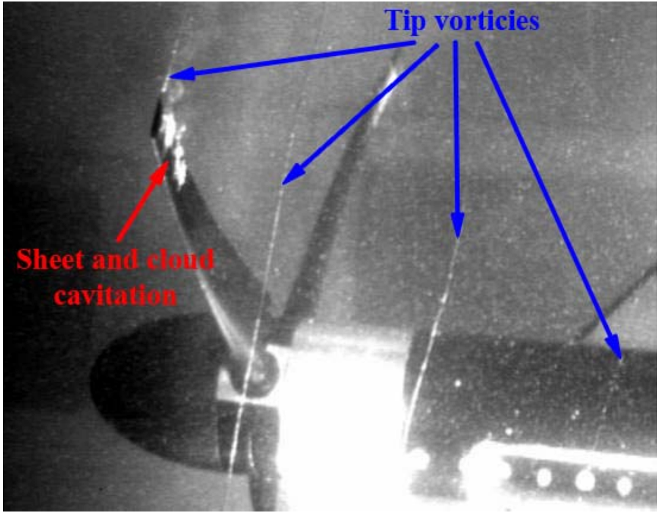

Fig. 2. Cavitation on a model turbine on test in a cavitation tunnel [9]

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. Figure 3 illustrates a typical pressure distribution over a changing foil section as the result of a two dimensional analysis. 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. Computational times are very short – in the order of seconds.

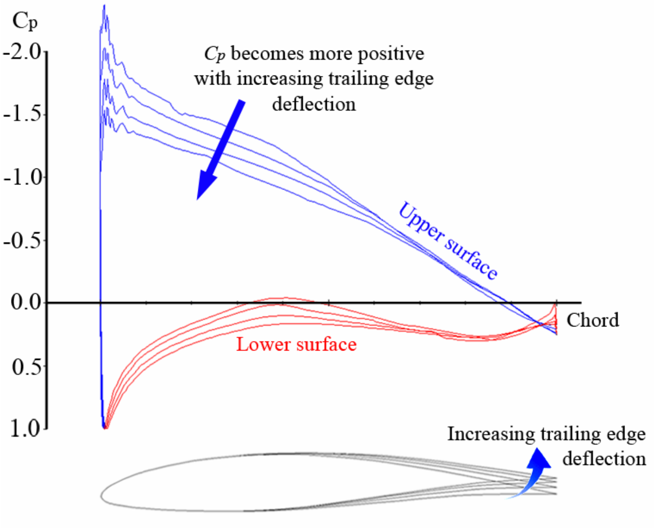

Fig. 3. Pressure distribution over the NACA 63-815 section with a variation in the deflection of the latter part of the foil at an angle of attack of 8°.

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

B. *Three Dimensional Analysis*

Surface panel codes allow a more thorough 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 blade loading data for further structural calculations. Since the panels are geometric shapes and are flat, an increased panel density will obviously model a three dimensional, complex curved shape such as a marine current turbine more efficiently.

Surface panel codes are more computationally intensive than two dimensional analysis methods. 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. However, during previous studies 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 very feasible. Using a frozen wake model it is possible to reproduce the helical wake characteristic of marine current turbines.

These simple three dimensional analysis codes provide a much more detailed picture of the pressure distribution over the turbine blades and body therefore giving a much more comprehensive picture of areas of the blade at which cavitation will occur. Figure 4 illustrates the pressure distribution of a three bladed marine current turbine obtained from a surface panel code. The areas of red illustrate those parts of the blade where low pressure occurs, i.e. where the pressure coefficient is a minimum and cavitation is likely to occur. Those areas of green are those with a more even pressure, and those nearing blue are areas tending towards stagnation.

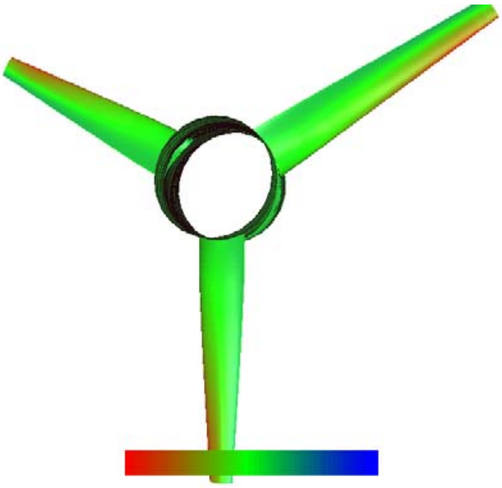

Fig. 4. Pressure distribution over a three bladed turbine obtained using a surface panel code.

Surface panel codes however, struggle to measure severe changes in the flow regime, i.e. stagnation and recirculation. Despite being a powerful tool to predict cavitation inception, once cavitation has occurred the analysis becomes unstable and is unable to complete. It is therefore apparent that more advanced numerical simulation of the area around the turbine is necessary for a full design.

C. *Reynolds Averaged 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 [10].

D. *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. This means that the whole range of spatial and temporal scales of the turbulence must be resolved. Direct numerical simulation (DNS) captures all of the relevant scales of turbulent motion, 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 choice of which turbulence model to use, however, is a far from trivial matter.

The simplest turbulence modelling approach rests on the concept of a turbulent viscosity. This relates the turbulent stresses in the RANS equations to the gradients of time averaged velocity similarly to the classical interpretation of viscous stresses in laminar flow by means of the fluid viscosity. Such models are widely used for simple shear flows such as attached boundary layers, jets and wakes. For more complex flows where the state of turbulence is not locally determined but related to the upstream history of the flow, a more sophisticated model is required [10].

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.

Two equation turbulence models are one of the most common type of turbulence models. Models like the k-epsilon model [11] and the k-omega model [12] 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. The performance of two-equation turbulence models deteriorates when the turbulence structure is no longer close to local equilibrium. Various attempts have been made to modify two equation turbulence models to account for strong non-equilibrium effects. For example, the SST (shear stress transport) variation [13], leads to marked improvements in performance for non-equilibrium boundary layer regions such as may be found close to separation.

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. The Reynolds stress transport models dispense with notion of turbulent viscosity, and determine the turbulent stresses directly by solving a transport equation for each stress component. This form of model can handle complex strain and can withstand non-equilibrium flows. However, it is complex, expensive to compute, can lead to problems of convergence and also requires boundary conditions for each of the new parameters being solved. For these reasons it has not yet been widely adopted as an industrial tool.

Large eddy simulation (LES) is based on an implication of Kolmogorov's theory of self similarity [14] is that the large eddies of the flow are dependant on the geometry while the smaller scales more universal. This feature allows one to explicitly solve for the large eddies in a calculation and implicitly account for the small eddies by using a subgrid-scale model. This method is more computationally expensive than a RANS model but less so than a DNS solution.

The difficulties associated with the use of the standard LES models, particularly in near-wall regions, has lead to the development of hybrid models that attempt to combine the best aspects of RANS and LES methodologies in a single solution strategy. An example of a hybrid technique is the detached-eddy simulation (DES) approach [15]. This model attempts to treat near-wall regions in a RANS-like manner, and treat the rest of the flow in an LES-like manner.

It should be considered that there is no universally valid general model of turbulence 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.

E. *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 composite adaptive marine current turbine blade [16], 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.

FSI coupled problems are, however, very computational expensive to compute. For complex geometries calculations it is not yet feasible to use such a method, however for simpler problems it can be a very powerful tool when combined with wind tunnel and on site model tests.

III. DESIGN SEARCH AND OPTIMISATION

Design search and optimisation is the term used to describe the use of formal optimisation methods in design [17]. Literally “to optimise” means: find the solution to a problem, which gives the best results with respect to certain decisional criteria, 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 [18]. Figure 6 illustrates the “classical” optimisation problem, where the global optimum needs differentiating form the local optimum.

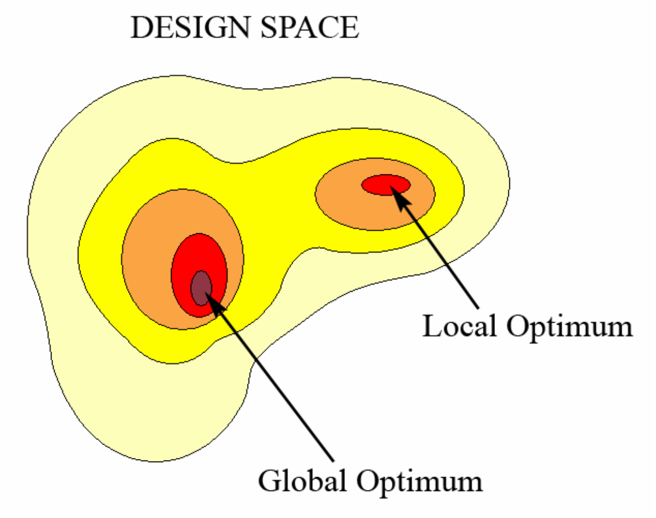

Fig. 6. The “Classical” optimisation problem

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 Figure 6, 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.

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 [18].

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 drastically 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 [17].

Figure 7 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.

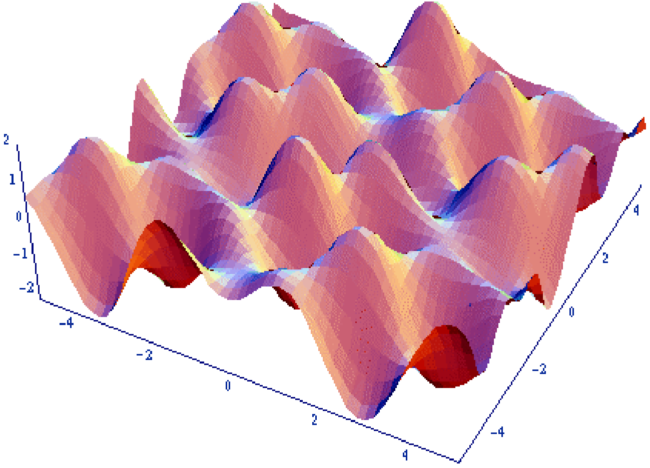

Fig. 7. A more realistic design space for an engineering problem illustrating many local and global maxima and minima.

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 [19].

IV. CONCLUSION

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, as they undergo intense forces in their hostile subsea 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.

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.

Whilst all the methods discussed in this paper require validation, be it using 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 negate the need for costly testing of model scale devices.

ACKNOWLEDGMENT

The PhD of R. F. Nicholls-Lee is supported in part by a 50% scholarship from the School of Engineering Sciences, University of Southampton, U.K.

REFERENCES

1. VanZwieten, J., Driscoll, F.R., Leonessa, A., and Deane, G., *Design of a Prototype Ocean Current Turbine - Part I: Mathematical Modelling and Dynamics Simulation.* Ocean Engineering, 2006.
2. Fraenkel, P.L., *Power from Marine Currents.* Journal of Power and Energy, 2002. **216**(1): p. 1-14.
3. Myers, L. and Bahaj, A.S., *Simulated Electrical Power Potential Harnessed by Marine Current Turbine Arrays in the Alderney Race.* Journal of Renewable Energy, 2005. **30**: p. 1713-1731.
4. Blunden, L.S. and Bahaj, A.S., *Initial Evaluation of Tidal Stream Energy Resources at Portland Bill, UK.* Journal of Renewable Energy, 2006. **31**: p. 121-132.
5. Batten, W.M.J., Bahaj, A.S., Molland, A.F., and Blunden, L.S., *Yawed Performance of Horizontal Axis Marine Current Turbines*, in *Conference on Renewable Energy in Island Maritime Climate*. 2006: Dublin.
6. Molland, A.F., Bahaj, A.S., Chaplin, J.R., and Batten, W.M.J., *Measurements and Predictions of Forces, Pressures and Cavitation on 2-D Sections Suitable for Marine Current Turbines.* Proceedings of the Institute of Mechanical Engineers, 2004. **218**(M): p. 127-138.
7. Drela, M. and Youngren, H., *XFOIL 6.94 User Guide*. 2001.
8. Wang, D. and Atlar, M., *Experimental Investigation on Cavitation Performance, Noise Characteristics and Slipstream Wash of an Ocean Stream Turbine*, in *World Maritime Technology Conference*. 2006, IMarEST: London.
9. Bahaj, A.S., Molland, A.F., Chaplin, J.R., and Batten, W.M.J., *Power and Thrust Measurements of Marine Current Turbines Under Various Hydrodynamic Flow Conditions in a Cavitation Tunnel and a Towing Tank.* Renewable Energy, 2007. **32**(3): p. 407-426.
10. Atkins, W., *MARNET Best Practice Guidelines for Marine Applications of Computational Fluid Dynamics*. 2003, MARNET.
11. Launder, B. and Spalding, D., *The Numerical Computation of Turbulent Flows.* Computer Methods in Applied Mechanics and Engineering, 1974. **3**: p. 269-289.
12. Wilcox, D., *Turbulence Modelling for CFD*. 2nd ed. ed. 1998: DCW Industries.
13. Menter, F., *Zonal Two Equation k-w Turbulence Models for Aerodynamic Flows*. 1993, AIAA Paper.
14. Kolmogorov, A., *Dissipation of Energy in a Locally Isotropic Turbulence.* Doklady Akad. Nauk SSSR, 1941. **32**.
15. Spalart, P., Jou, W., Strelets, M., and Allmaras, S., *Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach*, in *1st AFOSR Int/ Conference on DNS/LES*. 1997, Greyden Press: Rustin, LA.
16. Nicholls-Lee, R. and Turnock, S., *Enhancing Performance of a Horizontal Axis Tidal Turbine using Adaptive Blades*, in *Oceans '07*. 2007, IEEE: Aberdeen, Scotland.
17. Keane, A. and Nair, P., *Computational Approaches for Aerospace Design The Pursuit of Excellence*. 2005, Chichester: John Wiley & Sons, Ltd.
18. Campos, F., Weston, S., and Schumacher, T., *Automatic Optimisation of CFD Engineered Designs*, in *Automated Design & Optimisation Techniques Using CFD*. 2006, IMechE: London, UK.
19. Chew, J., Doherty, J., Gillian, M., and Hills, N., *Practical Applications of Automated Design and Optimisation Techniques using CFD*, in *Automated Design & Optimisation Techniques Using CFD*. 2006, IMechE: London, UK.

Rachel F. Nicholls-Lee was awarded a Master of Engineering with First Class Honours in Ship Science (Yacht and Small Craft) at the University of Southampton, Southampton, United Kingdom in 2004. She worked as a Naval Architect from 2004-2006 and is currently studying towards a PhD researching adaptive materials and their use in free stream tidal turbine blade design at the University of Southampton.

Stephen R. Turnock is a Senior Lecturer in Ship Science in the School of Engineering Sciences. He joined the University of Southampton in 1988 to work on wind turbine section design and completed a PhD in ship rudder-propeller interaction in 1993. His main areas of research lie in the synthesis of computational and experimental fluid dynamics techniques for maritime design. Recent has been in the areas of autonomous underwater vehicles, tidal turbines and in sports engineering.