Hydrodynamic modelling of a vertical axis tidal current turbine using CFD

Gareth I. Gretton[1], Tom Bruce[2] and David M. Ingram[3]

[1]Institute for Energy Systems, School of Engineering, University of Edinburgh, King's Buildings, Edinburgh, EH9 3JL, UK.
Email: gareth.gretton@ed.ac.uk

[2]Affiliation and address as above. Email: tom.bruce@ed.ac.uk

[3]Affiliation and address as above. Email: david.ingram@ed.ac.uk

Abstract

Results from a computational fluid dynamics (CFD) model of a vertical axis tidal current turbine are presented. This CFD model has been implemented in the commercial code CFX and is a Reynolds-averaged Navier-Stokes solution with the k-ω SST turbulence model. The turbine simulated is a 15 kW small-scale prototype device being developed by Edinburgh Designs Ltd. with independently and continuously variable pitch blades.

A major focus of the present work is the thorough verification and validation of the numerical model. This is based on a series of progressively more complex simulations, beginning with fixed pitch hydrofoils (airfoils), progressing to oscillating pitch hydrofoils, and finishing with a complete time-dependent model of the turbine.

The paper concludes with a parametric study of turbine performance, comparing fixed pitch and variable pitch operation and a four-bladed variant of the turbine with the three-bladed baseline. It is found that variable pitch operation both increases the peak power coefficient and broadens the peak in the power coefficient versus tip speed ratio curve. An increase in the number of blades (maintaining solidity) leads to a slight drop in performance, commensurate with Reynolds number effects.

Keywords: tidal current turbine, vertical axis, CFD, verification and validation

Nomenclature

A	= blade planform area
c	= chord length
c_d	= drag coefficient
c_l	= lift coefficient
c_m	= moment coefficient
C_P	= (power)$/\frac{1}{2}\rho U_\infty^3 A$; power coefficient
C_Q	= (torque)$/\frac{1}{2}\rho U_\infty^2 AR$; torque coefficient
C_T	= (thrust)$/\frac{1}{2}\rho U_\infty^2 A$; thrust coefficient
F_N	= blade force normal to chord
F_N^+	= $F_N/\frac{1}{2}\rho U_\infty^2 cL$; blade normal force coefficient
F_T	= blade force tangential to chord
F_T^+	= $F_T/\frac{1}{2}\rho U_\infty^2 cL$; blade tangential force coefficient
L	= blade length
L_b	= length to far-field boundary
M	= moment on blade
M^+	= $M/\frac{1}{2}\rho U_\infty^2 c^2 L$; blade moment coefficient
Ma	= Mach number
R	= turbine radius
Re	= Reynolds number
U_∞	= freestream speed
ρ	= fluid density
ω	= turbine rotational speed
Δy^+	= non-dimensional distance to first node from wall

1 Introduction

A number of previous authors have presented results from the CFD analysis of vertical axis wind and tidal current turbines, an exhaustive listing of which is provided in [1]. Only one of these references, however, includes any verification of the numerical model, this being a paper by Simão Ferreira et al. [2] on the modelling of a laboratory scale wind turbine; the verification study therein is based on qualitative comparisons of the vorticity fields and the blade forces and moments. Many references do include validation of the numerical model, but the veracity of this is questionable given the lack of verification. (Note that the specific meanings of the terms verification and validation will be discussed in the next section.) A major aim of the present work, then, is to perform a thorough and quantitative verification of the CFD model and then to validate that model with the best experimental data available in the literature.

The turbine geometry studied in the present work is that of a small-scale prototype being developed by Edinburgh Designs Ltd. This turbine has a diameter of 2.5 m and three blades of 0.2 m chord, 2 m length, and NACA 0024 section profile. The rated power is 15 kW at a free stream speed of 2.5 m/s. The turbine blades may be pitched independently and continuously according to any given algorithm. In the current work only one algorithm (other than fixed pitch) is implemented, this being a 'limiting angle of attack' strategy. The essence of this is to limit the angle of attack to a certain maximum value by pitching towards feather when the relative flow angle exceeds this maximum value; the relative flow angle itself is calculated by assuming a velocity field.

The commercial CFD code ANSYS CFX, hereafter CFX, was selected for the present modelling work. This particular code was selected because of its know strength in modelling turbomachinery and free surface flows; in practice a number of other commercial codes could also have been successfully employed. More generally, the choice of a commercial code was driven by the desire to make the process by which the results were generated as useful as the results themselves in terms of their application to industry.

As will be discussed below, all of the modelling carried out was for 2D problems. This was achieved in CFX (a 3D solver) by using one-element thick meshes and applying symmetry boundary conditions as appropriate. When discussing the meshes used, all of the language will refer to the surface meshes that were extruded to create the volume meshes.

2 Verification and validation

While the terms verification and validation are frequently used interchangeably in common English usage, they have specific and broadly accepted definitions in the field of numerical analysis. These definitions are given by a number of sources e.g. [3–6]; perhaps the most terse of these is that given by Roache [3]:

Verification: Solving the equations right.

Validation: Solving the right equations.

To elaborate upon these definitions. Verification is concerned with the quantification of numerical errors, these being primarily due to insufficiently fine spatial and temporal discretization, insufficient iterative convergence and programming errors. In the context of external flow problems the (arbitrary) placement of the far-field boundary also introduces an error and this is investigated within the framework of verification. Validation involves comparing the results of a verified simulation with experimental data in order to assess the modelling error. This may be problematic if the problem under consideration is complex and the experimental data is limited; in addition there may be significant uncertainties associated with the model setup.

Where validation is problematic, the American Institute of Aeronautics and Astronautics (AIAA) "Guide for the verification and validation of computational fluid dynamics simulations" [4] recommends a 'building block' approach. This entails the simulation of a series of problems, grouped into four levels of complexity. These

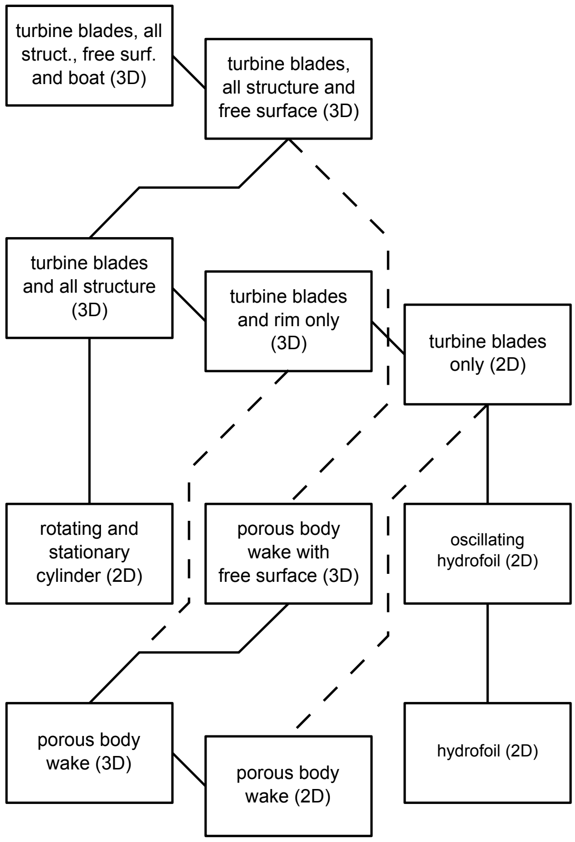

Figure 1: Problem decomposition. Solid lines indicate the flow of information within a phase or between two consecutive phases. Dashed lines indicate the flow of information between non-consecutive phases.

levels feature the complete system, subsystems, 'benchmark cases' and 'unit problems'. The expectation is that as the complexity decreases the quality of experimental data will increase while uncertainties associated with the model setup will decrease; additionally there may be analytical solutions for the unit problems.

Although not specifically highlighted in the AIAA guide, the building block approach is also relevant to the verification effort as verified grids from the less complicated problems can be used as the basis for the grids on the more advanced problems. Where further verification is carried out for the more advanced problems (as technically it should) it can be focussed and also based on a smaller number of grids than might otherwise be necessary.

For the present problem, the building block approach was defined as shown in figure 1. At the top-left is the most complicated problem featuring all fluid dynamic effects, while the bottom tier features the least complicated problems. The diagram also shows a hierarchy within each level of complexity. Only three problems are considered in the present work: 'turbine blades only (2D)', 'oscillating hydrofoil (2D)', and 'hydrofoil (2D)'. Clearly then the present work is both an end in itself and a clearly defined foundation for future work.

Validation data for the hydrofoil simulations were taken from the literature on wind tunnel tests of air-

Table 1: Lift, drag and moment coefficients from the four grids having different distance to far-field boundary. The 'RE 40-80' results were obtained by Richardson extrapolation from the grids having 40 and 80 chord lengths distance to their respective boundaries. The percentage changes are from these extrapolated results.

Grid (L_b/c)	c_l	c_d	c_m	% c_l	% c_d	% c_m
10	1.5887	0.02638	−0.06090	−0.74	26.24	1.90
20	1.5933	0.02404	−0.06839	−0.46	15.06	1.13
40	1.5968	0.02261	−0.06817	−0.24	8.18	0.81
80	1.5986	0.02182	−0.06801	−0.12	4.44	0.58
RE 40-80	1.6006	0.02090	−0.06762	–	–	–

foils. Such data are suitable if the Mach number is 'low' (Ma < 0.3) and are widely used for this purpose in the literature. Importantly, data for the correct Reynolds number are available. (Sources of such data were examined in a previous EWTEC paper by two of the present authors [7].)

No suitable 2D turbine data were found in the literature at the Reynolds numbers of interest, nor was it possible to carry out such experiments as part of the present work. High quality 2D turbine data were found in [8], but for a considerably lower Reynolds number. Given the disparity in the Reynolds number between this data and the present application, it is questionable to what extent validation with this data would serve to validate the high Reynolds number simulation. The requirements for the grid resolution in the boundary layer would also be very different. As such, it was decided to focus efforts on the validation of the hydrofoil/airfoil cases.

3 Airfoils

3.1 Verification

The primary verification effort was carried out for the case of the NACA 4412 airfoil at an angle of attack of $12.15°$ and a Reynolds number of 4.17×10^6. This case was selected for two key reasons: first, it was studied as part of a European Research project [9], from which the numerical grid was available, thus serving as an excellent starting point for the present work, and second, excellent experimental data were available [10] for subsequent validation. This section is also from the same series as that used for the turbine (the NACA 4-digit series) and so it has the same normalized thickness distribution. Only the verification work on this case will be discussed here.

This verification study considered the effect on the solution of the following four 'numerical parameters':

- Spatial discretization
- Distance to far-field boundary
- Alignment of grid in the wake of the airfoil
- Iterative convergence

The spatial discretization study used a series of five C-grids, with each grid having a linear coarsening/refinement ratio of $\sqrt{2}$ relative to the next. As the grids were 2D (one element thick in the third dimension), this represented a halving/doubling of the number of nodes. Richardson extrapolation was used to calculate asymptotic values for the lift, drag and moment coefficients, and these asymptotic values were in turn used to calculate the spatial discretization error for each of the grids. The errors in the drag coefficient were found to be the largest, with the 'medium' grid reducing the error to less than 5%. This grid, having 268×82 cells and $\Delta y^+ \sim 1$, was selected.

All of the grids used in the spatial discretization study had the far-field boundary at a distance L_b of 20 chords from the airfoil. Given that the intention of these simulations is to simulate the behaviour in freestream conditions (as opposed to that of a wind tunnel for example), the sensitivity of the solution to the position of this boundary must be examined. This was approached in a manner similar to the spatial discretization study with a series of grids being created having 10, 20 (as above), 40 and 80 chord lengths to the far-field boundary. Each of the smaller grids (10, 20 and 40) were effectively a crop of the larger grid such that the spatial discretization remained the same. As noted by Roache [3] in his analysis of the results of Zingg [11], Richardson extrapolation may again be used to estimate asymptotic values for the solution variables. The results of this are shown in table 1. Again, the errors in the drag coefficient were found to be the largest, with a grid having $L_b/c = 80$ being required to reduce the error from the asymptotic value to less than 5%.

When creating a structured grid for the simulation of an airfoil at a fixed angle of attack the grid lines in the wake may be aligned with the flow in order to best capture the physics, as was the case in the two studies discussed above. Without resorting to adaptive meshing, this is of course not possible for the case of oscillating airfoils and turbines. In order to study the effect of non-aligned grids, a new grid was created having grid lines behind the airfoil aligned with the chord line, but otherwise having the same spatial discretization and extent as the 80 chord lengths grid above. This gave estimates of the lift, drag and moment coefficients less than 1% different from the aligned grid, but significantly higher (RMS) equation residuals. On inspection it was found that the elevated residuals were confined to a small number of cells in the wake behind the trailing edge of the airfoil and it is speculated that the cause of this may be due to a Kelvin-Helmholtz instability. These localized residuals were considerably reduced by creating a series of grids with 'fanned' grid lines in the wake. Such a grid arrangement was therefore adopted; nevertheless, it is clear that the solution is insensitive to the alignment of the grid in the wake.

The effects of iterative convergence on the prediction of the lift, drag moment were also examined. This suggested that reducing the RMS residual to less than 5×10^5 reduced the error to less than 5%.

Further verification was carried out for simulations of the NACA 0012 and 0015 airfoil sections at angles of attack up to 20 degrees and for both fixed pitch and oscillating cases. This confirmed the generality of the conclusions drawn from the simulations of the NACA 4412 airfoil and also investigated the effect of the time step and the total time; these simulations are not discussed in this paper.

3.2 Validation

Simulations of the NACA 0012 and 0015 airfoils, as introduced above, represented the primary validation effort in the current work. These cases were chosen because of the wealth of experimental data available for both fixed and oscillating pitch.

Preliminary simulations of the oscillating case on a C-grid as above showed that the CFX solver produced spurious solutions when the grid was rotated. This was attributed by CFX support to a problem with moving inlet boundaries, for which no solution was available. Two alternatives were available: the use of grid stretching and the use of a multi-domain grid where the inner domain rotates and the outer domain is stationary. As it was desired to develop a turbine model capable of arbitrarily large pitching angles, for which the grid stretching option would not be suitable, it was decided to pursue the two domain solution.

The grid structure used for the inner domain is shown in figure 2 and is termed an ‘O-C-grid’ because it features both grid structures. This allows for an airfoil with a sharp trailing edge and an orthogonal grid in the boundary layer as with the C-grid while the O-grid structure allows for a circular domain interface. This domain interface is located at a radius equal to two chord lengths. Gridding parameters are, as far as possible, equivalent to those on the ‘medium’ C-grid discussed above.

Results from the present simulations are compared with the experimental results of Sheldahl and Klimas [12] in figure 3 for the NACA 0015 airfoil at a Reynolds number of 0.68×10^6. As is seen, the prediction of lift stall is very good while the prediction of moment stall is also acceptable. That the numerical simulation shows higher drag than experiment at low angles of attack is readily explained by the fact that the numerical simulation is run as fully turbulent while the experimental case is untripped. Predictions of the zero-lift drag coefficient for the NACA 0012 (not shown) showed excellent agreement with a curve fit to a large body of tripped experimental data derived by McCroskey [13] - within 1% for some Reynolds numbers.

Figure 4 shows a comparison with experiment for an oscillating case. The section is again a NACA 0015, this time at a Reynolds number of 2×10^6. Here the comparison with experiment for the lift and the moment is again good, but the drag shows poor agreement. Note that the experimental data here are for pressure drag, this being calculated by integration of surface pressure data from pressure taps. It has been shown by McCroskey et al. [14] that this method of calculating the pressure drag may be inaccurate, especially at low angles of attack. Nevertheless, other authors [15] have shown good agreement between CFD prediction and the experimental data of Piziali [16], and so the poor agreement shown here cannot be entirely explained by criticizing the experimental data(!).

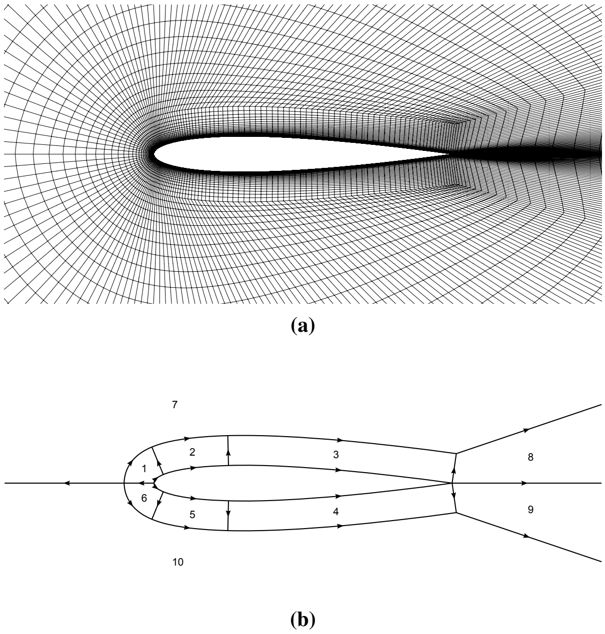

Figure 2: View of the medium O-C-grid (a) and the block structure used to create the grid (b) in the vicinity of the airfoil.

4 Turbine

The verification and validation work on airfoils described in the previous section served as the foundation for the turbine modelling work. In particular the verification studies on spatial discretization formed the basis for the grids of the turbine while the validation work showed the capacity of the current modelling approach to correctly predict the behaviour of airfoils i.e. turbine blades.

The multiblock structure of the turbine mesh is shown in figure 5 while a detailed view of part of the mesh is shown in figure 6. This incorporates the inner domain of the oscillating airfoil grids for the three domains immediately around the three blades. The other two domains are the rotating domain around the blade domains, formed by an unstructured tri mesh, and the stationary domain outside of this, formed by a structured O-grid.

4.1 Verification

As with the airfoil simulations, a thorough verification study was conducted for the turbine simulations.

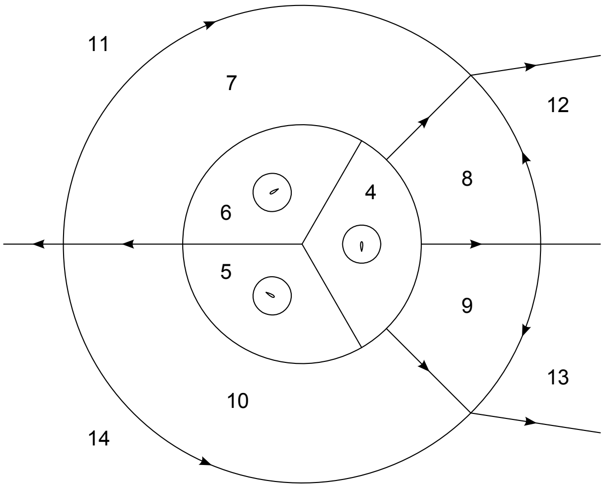

Figure 5: Block structure of the mesh used for the turbine. The sub-division of the blade domains (blocks 1–3) is not shown; for this see figure 2.

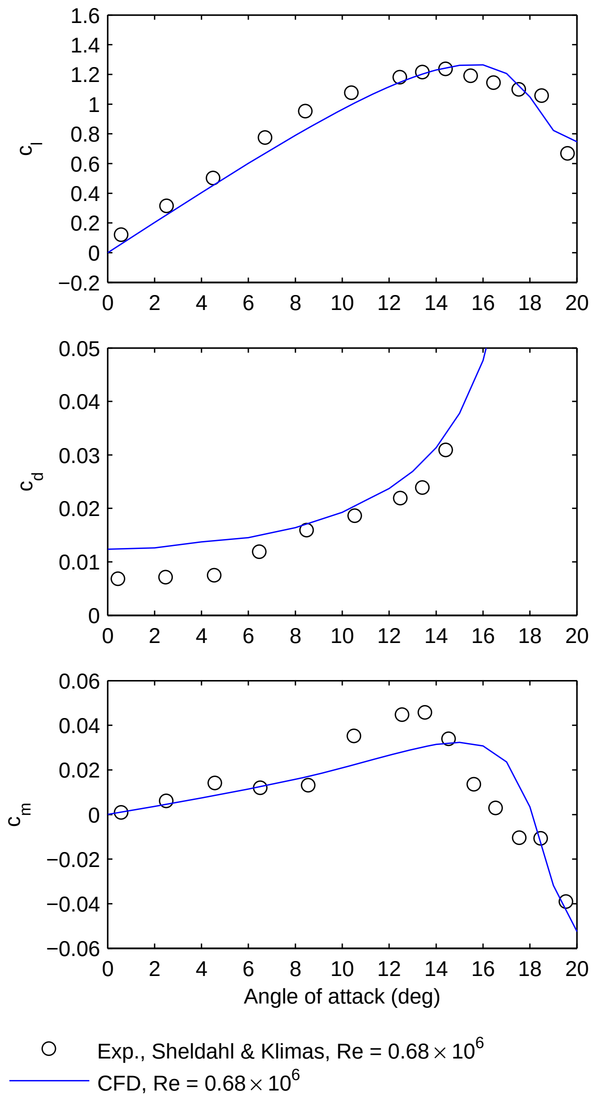

Figure 3: Validation of NACA 0015 simulations with the results of Sheldahl and Klimas [12].

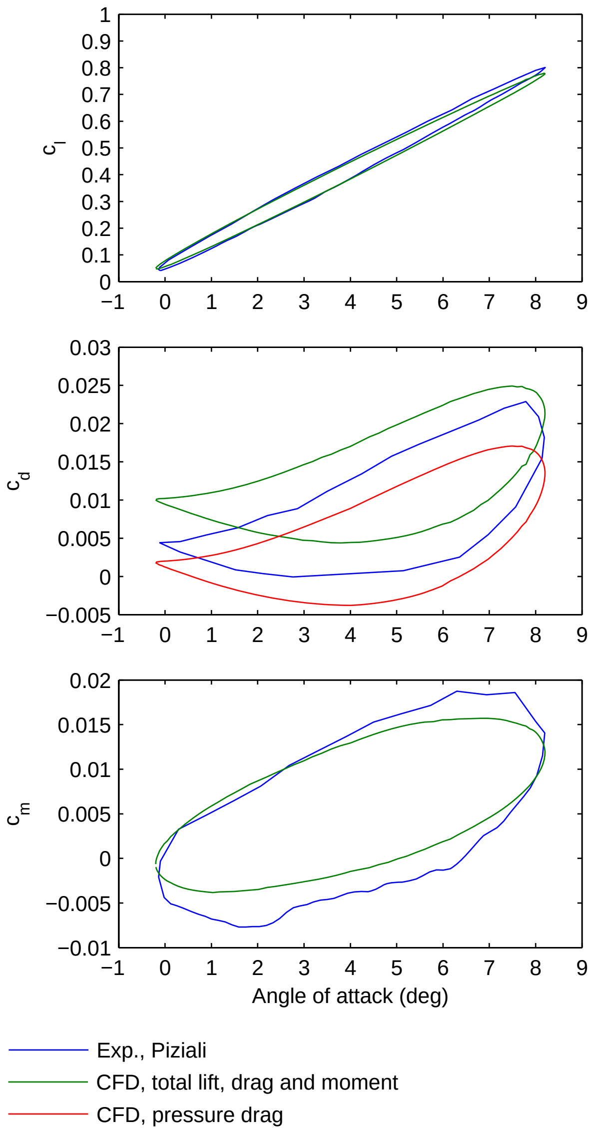

Figure 4: Validation of NACA 0015 simulations with the results of Piziali [16, figure 24(b)].

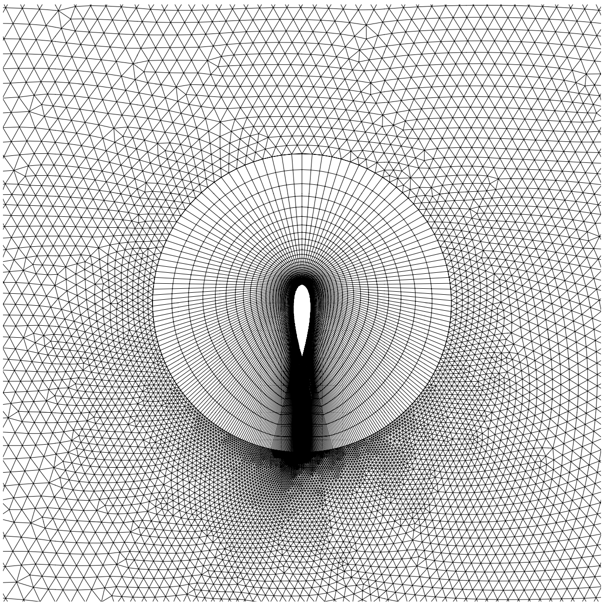

Figure 6: View of the O-C-grid grid around the turbine blade and a partial view of the rotor mesh.

This considered the following aspects of the numerical setup:

- Spatial discretization
- Time step and total time
- Iterative convergence

The effect of varying these numerical parameters was assessed both qualitatively and quantitatively, with the qualitative comparisons being based on the following:

- Blade force and moment versus azimuth angle
- Velocity profile across the wake at various streamwise positions
- Velocity profile along the wake

and the quantitative comparisons being based on:

- Maximum, minimum and mean blade force and moment during a cycle
- RMS deviation in the blade force and moment over a cycle

The most significant variations in the predictions were seen when the time step and the total time were varied and so we focus here on these two parameters. Considering first the time step, it was found that the blade moment showed the greatest variation between simulations and that a time step of $2°/\omega$ was required to reduce the errors (in the quantities noted above) to less than 5% relative to the best solution (having a time step of $2°/\omega$). It might be noted that simulations having time steps of $4°/\omega$ and $8°/\omega$ did show qualitatively correct behaviour.

The solution variation with total time, for a simulation having a time step of $2°/\omega$, is show in figure 7. This shows that the error from the final value follows a pattern that approximates to exponential decay. Of prime importance is the prediction of the average value of the tangential force coefficient, as this determines the power coefficient for the turbine. (A small contribution also comes from the moment.) Approximate errors from the assumed asymptotic value (taken as the value after 100 turbine revolutions) are 10% after 10 revolutions, 5% after 20 revolutions and 1% after 40 revolutions. Given the clear importance of this effect it is surprising that no study of this was found in the literature.

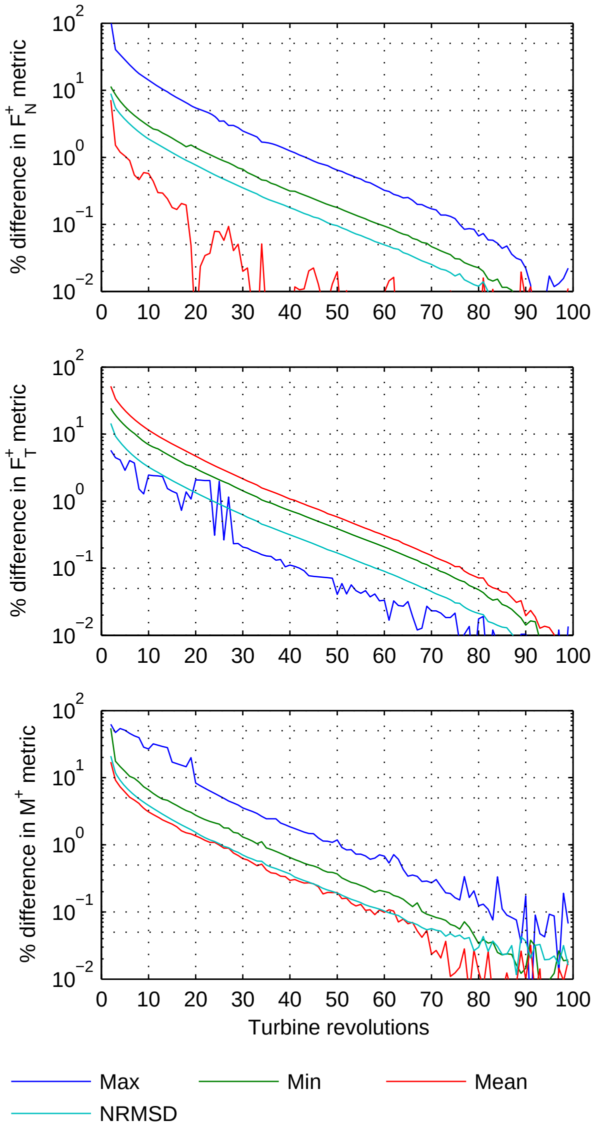

Figure 7: Quantitative analysis of the development of the forces and moment on blade A. Values shown are the percentage deviations from the result during the final (100th) turbine revolution.

The velocity field in the wake was also (unsurprisingly) found to be highly dependent upon the total time. In the immediate wake, taken as extending about 10 diameters downstream of the turbine, about 40 turbine revolutions were required for a qualitatively accurate picture of the wake profile, while at the edge of the computational domain, at 40 diameters, at least 60 turbine revolutions were required.

Based on this verification study, the following solution procedure was devised for investigating the physical parameters of interest: the model was first run on a 'coarse' mesh and with 'loose' iterative convergence for 60 turbine revolutions in order to produce a solution for the wake velocity profiles. This coarse mesh was based around a blade domain grid that had one quarter of the number of nodes of the medium grid used for the inner domain of the airfoil grids (and therefore had $\Delta y^+ \sim 2$). The coarse mesh solution was then interpolated onto the medium mesh (based around the medium grid used for the airfoil simulations), and advanced for a further three turbine revolutions and with 'good' iterative convergence to produce a solution for the blade forces and moments. This second stage provides a more accurate solution for the forces and moments, but introduces a perturbation into the wake solution, hence why the coarse mesh solution is taken for the wake velocity solution. Overall, this strategy minimizes the computational time required to produce a solution of the required accuracy for both the wake velocity profiles and the blade forces and moments.

4.2 Investigating physical parameters

The variation in blade normal force, tangential force and moment with azimuth angle is shown in figure 8 for five different values of the tip speed ratio. Fixed pitch operation is used in each case. The sharp drop in the moment at the lowest three tip speed ratios and for azimuth angles around 180–225° suggests moment stall. Lift stall is shown for the lowest tip speed ratio by a sharp drop in the tangential force at an azimuth angle of 180°. At the highest tip speed ratio the turbine is presenting a considerable blockage to the flow, with the effect that during the downstream pass of the blades the tangential force remains positive (giving a negative shaft torque).

The velocity field in the wake is analysed by plotting the variation in the streamwise velocity along a line parallel to the freestream direction and normal to the turbine axis, figure 9. The five cases considered are the same as those above. As expected, higher tip speed ratios lead to lower velocities at the turbine and in the near wake (up to about 6 or 7 diameters downstream of the turbine axis). The pattern of wake recovery differs somewhat, with the cases that have higher tip speed ratio showing more rapid and more complete recovery within the domain studied.

Finally, we examine the variation in the coefficients of power, torque and thrust with tip speed ratio for fixed pitch operation with three blades, variable pitch operation with three blades and variable pitch operation with four blades (figure 10). The advantages of variable pitch operation over fixed pitch operation are clear, with variable pitch operation giving an increased $C_{P_{\max}}$ and a broader peak in the curve. Variable pitch operation also gives a desirable increase in the ratio of power coefficient to thrust coefficient. The four blade variable pitch results are seen to be very similar to the three blade variable pitch results, with the small decrement in the performance being readily attributable to the reduction in the

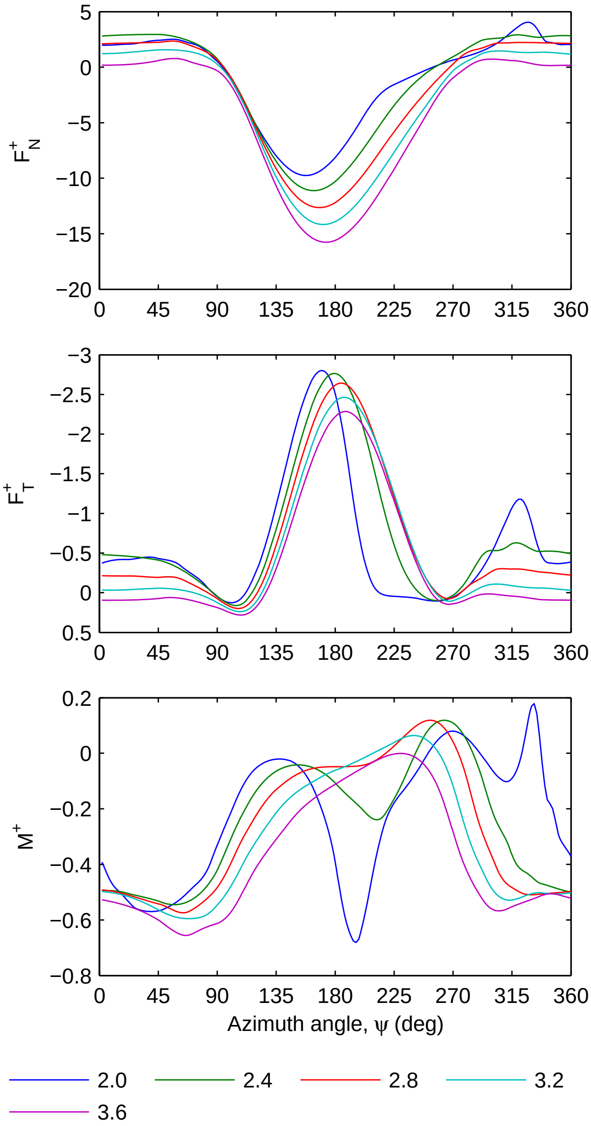

Figure 8: Forces and moment on blade A for five different tip speed ratios (see legend) with fixed pitch operation

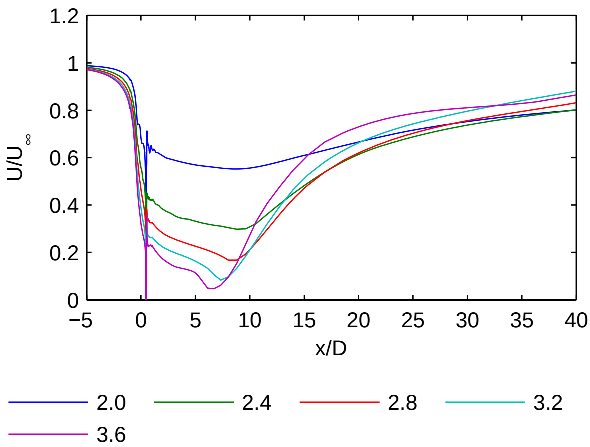

Figure 9: U-velocity along a line parallel to the free stream and normal to the turbine axis for five different tip speed ratios (see legend).

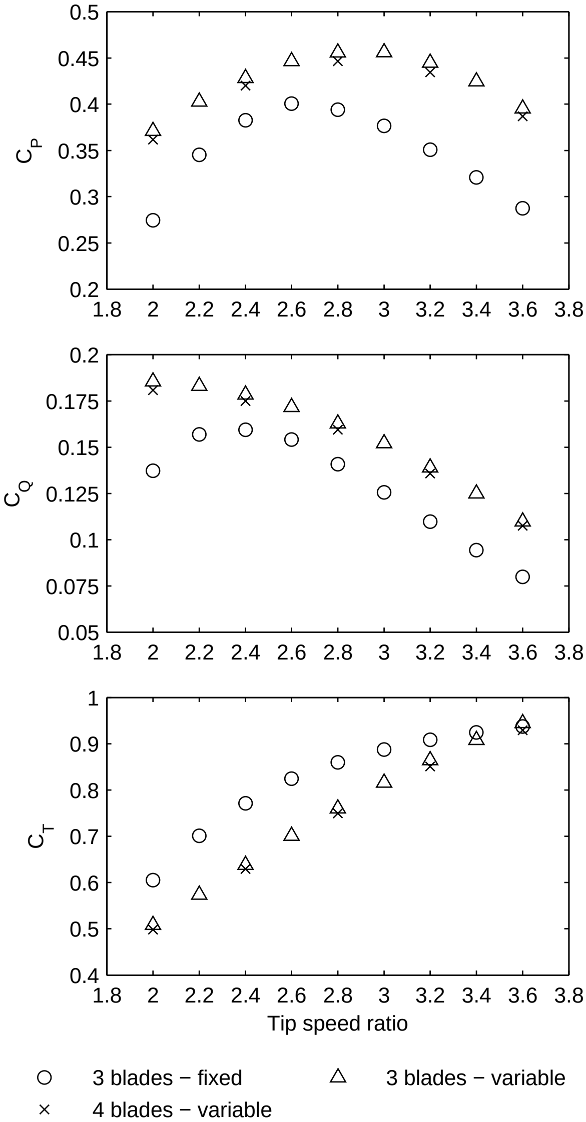

Figure 10: Coefficients of power, torque and thrust versus tip speed ratio.

blade chord Reynolds number.

5 Conclusions

This paper has presented selected results from a recently completed PhD thesis [1]. A methodology for the verification and validation of a CFD model of a vertical axis turbine has been described which is based on a building block approach, and which is recommended for future studies. A major focus of this approach has been the simulation of hydrofoils at fixed and oscillating angles of attack; this has identified the setup necessary to reduce numerical errors to an acceptable level and it has demonstrated the accuracy of the current predictions in comparison to experimental data. Verification work on the turbine model has shown in particular the importance of running the simulations for a substantial period of time in order to generate accurate predictions for the power coefficient and for the wake velocity profiles. Finally, use of the model to investigate the variation in turbine performance with tip speed ratio for a number of configurations has shown the advantage of variable pitch operation based on limiting the angle of attack.

Acknowledgements

This work was supported by a CASE studentship between Edinburgh Designs Ltd. and the EPSRC (ref. 04302407). Support from the SuperGen Marine Consortium is also acknowledged, as well that from the Scottish Funding Council for the Joint Research Institute with Heriot-Watt University (part of the Edinburgh Research Partnership).

References

[1] Gareth I. Gretton. *The hydrodynamic analysis of a vertical axis tidal current turbine*. PhD thesis, School of Engineering, University of Edinburgh, 2009.

[2] C. J. Simão Ferreira, H. Bijl, G. van Bussel, and G. van Kuik. Simulating dynamic stall in a 2D VAWT: Modeling strategy, verification and validation with particle image velocimetry data. *Journal of Physics: Conference Series*, 75, 2007. Article number 012023.

[3] Patrick J. Roache. *Verification and Validation in Computational Science and Engineering*. Hermosa, Albuquerque, New Mexico, USA, 1998.

[4] AIAA. Guide for the verification and validation of computational fluid dynamics simulations. Guide G-077-1998, AIAA, 1998.

[5] Michael Casey and Torsten Wintergerste. *Special Interest Group on Quality and Trust in Industrial CFD: Best Practice Guidelines*. ERCOFTAC (European Research Community on Flow, Turbulence and Combustion), 2000.

[6] NPARC. *Resources for CFD verification and validation*, 2008. URL `http://www.grc.nasa.gov/WWW/wind/valid/resources.html`. [Accessed 15th April 2009].

[7] Gareth I. Gretton and Tom Bruce. Aspects of mathematical modelling of a prototype scale vertical-axis turbine. In *Proceedings of the 7th European Wave and Tidal Energy Conference*, Porto, Portugal, 2007.

[8] J. H. Strickland, T. Smith, and K. Sun. A vortex model of the Darrieus turbine: An analytical and experimental study. Technical Report SAND81-7017, Sandia National Laboratories, June 1981. Final report submitted on Sandia contract 13-5602.

[9] Werner Haase, Eric Chaput, Eberhard Elsholz, A. Leschziner, Michael, and Udo R. Müller, editors. *ECARP – European Computational Aerodynamics Research Project: Validation of CFD Codes and Assessment of Turbulence Models*, volume 58 of *Notes on Numerical Fluid Mechanics*. Vieweg, Brunswick, Germany, 1997.

[10] R. C. Hastings and B. R. Williams. Studies of the flow field near a NACA 4412 aerofoil at nearly maximum lift. *Aeronautical Journal*, 91(901):29–44, 1987.

[11] D. W. Zingg. Grid studies for thin-layer Navier-Stokes computations on airfoil flowfields. *AIAA Journal*, 30(10): 2561–2564, 1992.

[12] R. E. Sheldahl and P. C. Klimas. Aerodynamic characteristics of seven symmetrical airfoil sections through 180-degree angle of attack for use in aerodynamic analysis of vertical axis wind turbines. Technical Report SAND80-2114, Sandia National Laboratories, 1981.

[13] W. J. McCroskey. A critical assessment of wind tunnel results for the NACA 0012 airfoil. Technical Memorandum 100019, NASA, 1987.

[14] W. J. McCroskey, K. W. McAlister, L. W. Carr, and S. L. Pucci. An experimental study of dynamic stall on advanced airfoil sections volume 1: Summary of the experiment. Technical Memorandum 84245, NASA, 1982.

[15] J. A. Ekaterinaris and F. R. Menter. Computation of oscillating airfoil flows with one- and two-equation turbulence models. *AIAA Journal*, 32(12):2359–2365, December 1994.

[16] R. A. Piziali. 2-D and 3-D oscillating wing aerodynamics for a range of angles of attack including stall. Technical Memorandum 4632, NASA, 1994.