AIAA-2004-1362

A THREE-TIERED APPROACH FOR DESIGNING AND EVALUATING PERFORMANCE CHARACTERISTICS OF NOVEL WECS

Brad C. Cochran, CPP Inc, Fort Collins, Colorado 80524
David Banks, CPP Inc, Fort Collins, Colorado 80524
Scott J. Taylor, Terra Moya Aqua, Inc, Cheyenne, Wyoming 82001

bcochran@cppwind.com

ABSTRACT

As the wind energy market matures and becomes more competitive, it is increasingly important that new entrants realize the full potential of their wind energy conversion system (WECS) early in their product development cycle. Traditionally, the evaluation of potential performance characteristics has been limited to full-scale prototype testing in a field environment. This method can be both costly and time consuming, and, unless conducted carefully, can lead to erroneous results. A more practical approach is to integrate multiple design tools and to use each to the extent of its capabilities. The three-tiered approach described in this paper integrates numerical assessment tools, reduced-scale testing in a controlled environment, and full-scale field-testing. Each of these tools provide unique opportunities and limitations, but properly integrated, they can provide a method for designing and evaluating performance characteristics at a cost and time frame significantly less than traditional methods. The exact integration of each these analysis techniques will tend to be device specific since the limitations of each tool will depend on the characteristics of the WECS. While each of these three tools have been used in the past, to the best of the authors' knowledge, this is the first time that this integrated three-tiered approach has been utilized specifically for designing and evaluating the performance characteristics of WECS.

INTRODUCTION

The vertical axis wind energy conversion system (VA WECS) under review is being developed by Terra Moya Aqua, Inc. (TMA) under U.S. patent 6,015,258; South Africa patent 98/9472; and Australia patent 749851s. The VA WECS consists of an internal rotor cage, an external set of stator blades, and a base unit. Power performance tests were carried out in CPP's closed-circuit atmospheric boundary layer wind tunnel, shown in Figure 1. Variables evaluated included: the number of stator blades; the angle of attack of the stator blades; the number of rotor blades; and the spacing between the rotor blades and the center shaft.

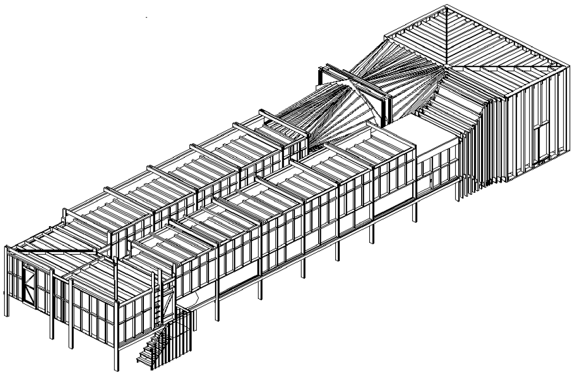

Figure 1. Isometric view of CPP's Closed Circuit Atmospheric Boundary Layer Wind Tunnel.

The results of this analysis, along with other published performance data for VA WECS, as described below, were used in the second phase of the study to validate the computational analysis. The computational analysis provided insight into the flow patterns in and around the rotors. This information was used to optimize the blade configuration. During the third phase, the reduced-scale model was placed back into the wind tunnel to confirm the finding of the computational analysis and to "tweak" various spacing parameters. The final phase of this analysis is to construct a full-scale unit and conduct power performance testing in a field environment. At this time, field tests have only been conducted on the original turbine configuration, as part of a validation study of the initial reduced-scale wind tunnel results.

A field test of the final configuration is on going. Results of this analysis will not only be used to fully-

---

describe the performance of the prototype, but to evaluate the effectiveness of the reduced-scale and computational analysis at predicting the full-scale performance. Combined, this information will provide valuable insight into the effectiveness of utilizing this three-tier approach for designing and evaluating performance characteristics of WECS.

COMPUTATIONAL ANALYSIS

Background

Computational Fluid Dynamics (CFD) models provide knowledge concerning the nature of the flow field through and around rotating turbine blades. This includes information about the velocity and pressure distributions and the overall blade efficiencies at each time step within the turbine's rotation. For this case study, the CFD simulations were performed using Version 6.0 of the Fluent, software program. The simulations were limited to a two-dimensional space domain. Solving the flow parameters in a 2-D domain simplified the solution considerably, allowing for reasonable computational times on a high-end desktop PC. Since the VA WECS is inherently a 2-D device, where the blades rotate in the same plane as the approaching wind, and the purpose of the analysis was to compare various turbine configurations as part of a performance optimization scheme, it was felt that a 2-D simulation was sufficient for this application.

In a CFD analysis, the fluid (in this case air) is divided into thousands of small cells. The equations governing fluid motion between adjacent cells are solved in order to generate a picture of the complete flow field. The manner in which the cells are arranged, and the type of approximations made when solving the equations of motion often require a high level of familiarity with the type of flow being studied and detailed knowledge of the CFD package being used. Insufficient grid points or the use of an inappropriate turbulence closure model can lead to erroneous results. The turbulence closure model is of particular importance in this application because of the complex airflow in and around the rotor blades. For example, the k-ε closure method, which is the most common method applied in most CFD applications, cannot accurately simulate the flow separation region that occurs at the tips of the rotating blades. This can lead to significant errors in the overall analysis because the forces that act at the leading edge of the blade can dominate the total torque transferred to the central shaft and, thus, the predicted power conversion efficiency. Without conducting validation studies, it would be difficult to determine the best turbulence closure method to apply for this application. Therefore, before any credence is given to any of the CFD simulations, a thorough validation study was completed.

For the VA WECS the validation was conducted by comparing predicted performance characteristics from various Savonius type rotors with published results from both field and wind tunnel tests. The validation study compared the CFD simulations with measured data obtained at Sandia National Laboratories [2], Kansas State University [3], at the University of British Columbia [4], and at CPP [5].

Once a CFD method has been determined to provide reliable results, i.e., results that have been thoroughly validated, it can provide valuable insight into the manner in which flow accelerates around, and exerts forces on, a bluff body such as a WECS. It can characterize the flow in and around the blades that would be either extremely difficult or impossible to obtain experimentally.

The reasonable amount of time and effort required to obtain a CFD solution to the 2-D simulation makes it a useful tool for evaluating changes to the WECS design. It is particularly useful in evaluating potential blade designs, where it has two distinct advantages over either reduced-scale or full-scale testing. First, it provides greater insight into the flow phenomenon in and around the blades. This leads to the potential for a more "educated" evaluation of the results. This allows the investigator to have a better opportunity to successfully identify future favorable blade designs. It also has the advantage in that a physical model of the blade is not needed. This can save time and effort and allow the investigator to evaluate novel blade shapes without the restraint of actually having to construct the blades.

CFD analysis, however, may not necessarily be the most effective method for evaluating small "tweaking" of a design. Running simulations associated with even the smallest design change can require many hours of processing time on a high-end desktop computer. While this time frame may be competitive with full-scale testing, such changes can be evaluated much more economically in a reduced-scale wind tunnel environment. Thus, while CFD has the potential to be a very useful design tool for evaluating performance characteristics of WECS, it should not be seen as a stand-alone method. Rather, it should be seen as an important contributor in an effective three-tiered evaluation approach.

Test Procedures

The CFD simulations assume that the blades are spinning at predetermined constant tip speed ratio. All simulations were assumed to take place in a wind tunnel with solid walls, so that the results could be compared to past and future experiments. The domain is essentially a horizontal cross section of the wind tunnel. The computational tunnel extends 10 turbine diameters in each direction. This allows the wake of the turbine to develop without being cut off by the tunnel exit, and similarly prevents the positive pressure zone in front of the turbine from encroaching upon the upstream entrance boundary. The sides of the domain are presented by walls that are also separated by 10 turbine diameters. This matches the width of the wind tunnel test section.

The approach velocity is set such that the blades complete a full rotation once per second at an optimal tip speed ratio. Typical efficiency vs. tip speed ratio curves for Savonius rotors display a broad peak at a ratio near 0.85.

For most of the simulations, the flow was assumed to have a 10% turbulence intensity (TI) at the inlet. In the absence of any trips or blocks in the upstream flow, the TI drops slightly, to just below 9%, by the time it reaches the blades. This is representative of the TI value for the CPP wind tunnel experiments. It is considerably higher than the TI used in the studies at Sandia [1] and UBC [3], which used smooth flow. It is also 5%-10% lower than what turbines are likely to experience in the field.

While turbulence is not generated upstream of the model, considerable turbulence is generated as the air flows around and through the model. Turbulence is known to affect the flow separation, which in turn controls flow patterns and air pressure, so that it is important that the production and dissipation of turbulence be modeled properly.

The rotation interface delineates the edge of the rotating portion of the mesh. The fluid inside the interface rotates at the same rate as the turbine blades and the shaft, and the acceleration of the fluid is affected by additional terms that appear in the momentum equations [1].

As mentioned above, CFD involves the division of the fluid into small cells. The edges of these cells form a grid. Cells are concentrated in regions of high fluid strain, such as along the rotating fluid interface and near the center openings by the shaft. The grids feature approximately 50,000 cells, half of which are inside the rotating blade zone. There are nearly 800 cells along the rotation interface.

The CFD program begins by assuming that the entire flow zone has the same velocity as the inlet. Simulation time is initially marched forward quickly, to allow the general flow field around the turbine to be established. The general flow field includes features such as the accelerated flow around the edges of the turbine and the low velocity wake behind it. Once the wake reaches the tunnel exit, the simulation time step size is reduced. To achieve a solution which was independent of the time step size, it was found that the time step has to be small enough so that a single step does not rotate the interface a distance greater than the smallest cell size. As a result, the time step used was quite small, typically on the order of 0.001 seconds, or 1/1000th of a revolution of the rotor blade.

In most cases, 8-10 seconds of simulation time are required to reach a stable solution. (Note, each second of simulation time requires 10 minutes to 1 hour of computational time on a 2 GHz PC, depending upon the size of the time step.) The solution stability is judged by plotting a running average of the blade efficiency, as shown in Figure 2. The running average is for half a second for a two-blade configuration, and over one third of a second for a three-blade configuration. The low frequency (~0.5 Hz) oscillations in the efficiency are the result of periodic changes in the wake structure.

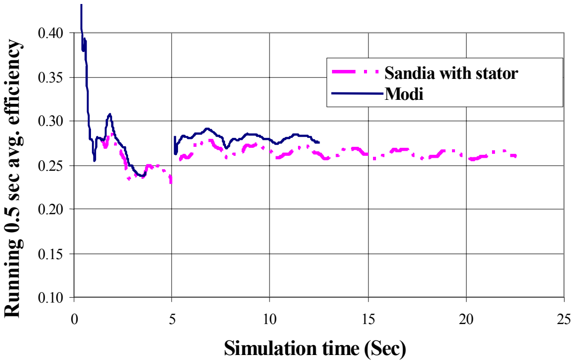

Figure 2. **Solution convergence: turbine efficiency as a function of simulation time.**

The structure of the cells used within the analysis is also critical. It is important to not only have sufficient number of cells, but also to have the orientation and placement of the cells properly defined. In these turbine cases, the need for small cells is greatest at the tips of the blades and at the tips of the stators. As a blade rotates past a stator, there is a large increase in flow velocity over a short distance (a large strain rate) as the flow is "squeezed" between the blade and the stator. To capture this phenomenon accurately requires numerous cells between the stator and the rotor, as is illustrated in Figure 3. Most simulations were performed using a mesh with just over 50,000 cells. The calculations were repeated with a finer mesh featuring over 110,000 fluid cells, and the results matched closely. This indicates that the reported solutions are grid independent.

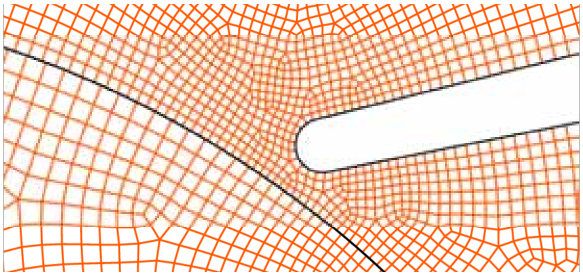

Figure 3 CFD grid features 50,000 cells, half of which are inside the rotating blade zone. There are roughly 800 cells along the rotation interface

Power Performance Calculations

Rotor power efficiencies were calculated using the formula:

$$Efficiency = \frac{P_t}{P_w} \quad (1)$$

P_w is the available power in the wind, which is given by:

$$P_w = A \cdot \tfrac{1}{2} \rho U^3_{freestream} \quad (2)$$

where A is the swept area of the turbine blades, ρ is the air density, and $U_{free\text{-}stream}$ is the approach flow velocity, which was set at the velocity inlet.

P_t is the turbine rotor power, which is defined for this study as the dot product of the local air pressure force and the local blade velocity.

$$P_t(s) = \vec{F}(s) \cdot \vec{v}(s) \quad (3)$$

where s is the position along the blade, $\vec{F}$ is the force vector (which is normal to the blade surface) and $\vec{v}$ is the velocity vector of the rotating blade. The fact that this is a vector dot product is important, since only the component of the pressure force on the blade that acts to rotate the blade contributes to the power. This is important for the analysis of highly curved blades, where the pressure force near the tips may act primarily to stress the blade, adding only a small portion to the total P_t of the blade.

The total power of the blade is calculated by integrating the power at each point along the blade (each value of s). Fluent does not perform this calculation. However, Fluent does calculate a normalized moment coefficient for each face of each blade, so that the instantaneous efficiency of each face of each blade can be readily calculated at each point in time

$$Efficiency_i(t) = \frac{Cm_i(t)}{2} \lambda C_f \quad (4)$$

where C_f is a correction factor based on the Fluent reference values (typically ≈ 1.2), λ is the tip speed ratio and $Cm_i(t)$ is the moment coefficient of the i^{th} blade face at each instant of time (i.e, the instantaneous efficiency of the front face of the 2^{nd} blade). To calculate the overall efficiency of the turbine, the efficiency of each blade face is averaged over an entire cycle and then the efficiencies of all of the blades faces are summed.

Results

During the CFD analysis 13 different configurations of the VA WECS were evaluated. This section highlights some general results noted during this phase of the study, specifically those that are best documented with the computational analysis. One of the major driving mechanisms for a WECS is the accelerated flow along the blades. This accelerated flow creates low-pressure regions that *lift* the blade through the rotation. The CFD analysis indicates that flow acceleration along the blades of the VA WECS is considerable, as evident in Figures 4 and 5. This is a phenomenon that is not typically attributed to Savonious type WECS. Figure 4, which shows the flow patterns in and around the Sandia [1] 2-bucket rotor, indicates area of flow acceleration passing through the small openings near the center shaft and along the leading face of the blade during its power stroke, between 9 o'clock (left hand side of Figure 4) and 6 o'clock. With the blade at 9 o'clock, the wind speed along the leading face of the blade is 1.5 to 2.0 times the "free-stream" wind speed. Figure 5 shows the flow patterns when stators are added to the rotor configuration. Here the flow passing between the side stator (located at 5 o'clock) and the adjacent blade reaches speeds in excess of two times the approach velocity.

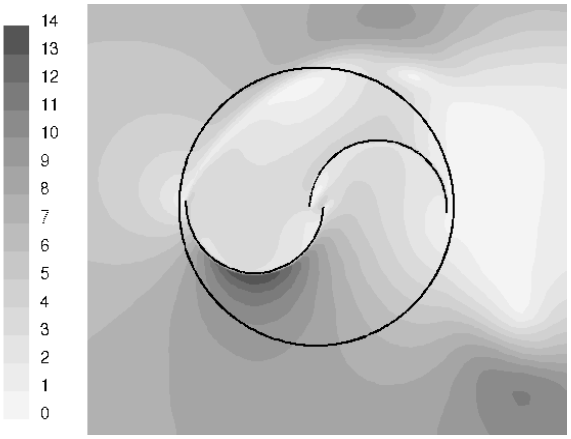

Figure 4 Contour plot of velocity (m/s) for Sandia [1] blades without stators, wind from left to right.

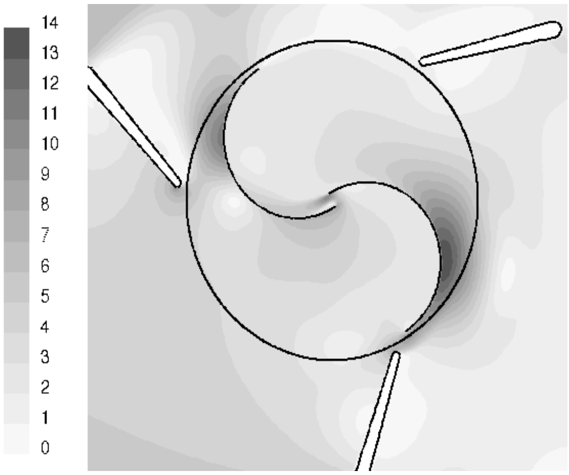

Figure 5 Contour plot of velocity (m/s) for Sandia [1] blades with stators, wind from left to right.

Based on the Bernoulli equation, this increased flow speed would be expected to produce suction forces four times as great as the direct drag force of the free-stream wind impacting a similarly sized surface. The pressure produced by stopping the 6.7 m/s free-stream wind with a flat plate or a wall is given by the free-stream flow head, q:

$$q = \tfrac{1}{2}\rho U^2 = 30\ \text{Pa} \qquad (5)$$

where once again ρ is the air density and U is the approach wind speed. The pressure increase on the back side of the blade at the 5 o'clock position in this simulation is well below -100 Pa. This means that the suction or lift force on the blade is three to four times greater in magnitude than the drag force at this stage. Further analysis shows that the lift or suction force on the leading face is greater than the drag or pushing force on the trailing face throughout the blade's entire power stroke.

One notable feature of the flow that was observed in several cases is the presence of a strong vortex that forms at the back side of the blade during the return cycle. This vortex is present during the last half of the power loss portion of the cycle, which is roughly defined as the region between 1 o'clock and 9 o'clock (corresponding to the passage between the rear and the front stator). The vortex has the effect of increasing the trailing edge suction force that inhibits the blade from moving forward during the power loss phase.

The vortex is a product of the arrangement of the openings at the center of the blades. In general, these openings appear to exert a significant influence on the pressures experienced by the blades during their return stroke (roughly defined as the passage from 2 o'clock to 10 o'clock for the cases with stators). It is not clear from these tests what the best center opening would be; it appears that the optimal opening depends on the blade shape. Modi's [3] wind tunnel experiments showed optimal performance for the 2-bucket style blades with no central opening at all. Yet of the cases tested in this study, the most efficient configuration during the return stroke was based on the Kansas State University study [2], which featured a very large opening between 2 bucket style blades.

Validation

The results described above are of great benefit to the investigator if it can be shown that the flow phenomenon described by the CFD analysis is correct. It is quite difficult to directly validate the flow patterns in and around the rotor and stators because this information is nearly impossible to obtain from either a reduced-scale or a full-scale evaluation. Therefore, rather than trying to validate the actual flow structure, the performance characteristics of the turbine, which are driven by the flow structure, were compared against data obtained from the reduced-scale model and full-scale experiments.

Wind tunnel power performance test results obtained by Sandia [1] and CPP [4] were used as the primary basis for validating the CFD simulations. The major areas of concern were the turbulence modeling and the mesh refinement. The standard 2-equation, k-epsilon turbulence closure model and the 5-equation Reynolds stress model (RSM) were evaluated. The RSM model is an anisotropic Reynolds-Average Navier-Stokes (RANS) turbulence model that solves transport equations for Reynolds stress [8]. It is recommended for cyclone flows, highly swirling flows, and rotating flow passages.

Initial simulations were conducted using the 2-equation k-epsilon turbulence closure model. The resulting simulation predicted the strong vortex that forms at the backside of the blade during the return cycle, as described above. However the magnitude of this vortex was such that the overall efficiency of the VA WECS was negative, indicating that, if anything, the blades should rotate backwards. Since this vortex is formed by separated flow occurring at the trailing edge of the blade, a phenomenon that the 2-equation k-epsilon model has been known to poorly simulate, this solution method was considered unreliable for this application. After trying various closure methods, the Reynolds-stress 5-equation turbulence model was determined to perform adequately for these simulations.

Using the 5-equation Reynolds stress model the efficiency of the classic half-barrel Savonius design tested by Sandia [1], Case A, was calculated with approach turbulence intensities (T.I.) of 1%, 7%, and 10%. The Sandia tests were conducted with an approach turbulence intensity of approximately 0.5%. The calculated efficiencies ranged from 23% with a 10% T.I. to 26% with a T.I. value of 1%. This compares very favorably with the stated efficiency of 26.5% in the Sandia study. Similar results were obtained with two configurations of the TMA VA WECS, Cases B through D. Table 1 shows comparisons between efficiencies obtained with the two methods, expressed as the ratios of the measured power efficiency in the wind tunnel or in the field, η_M, to the calculated efficiency from the corresponding CFD simulation, η_{CFD}.

The results shown in Table 1 indicate there is a reasonably good agreement between the measured and computed power conversion efficiencies for those configurations tested at Sandia and CPP. The measured and computed efficiencies are within ±10%. There is greater variation between the values computed using the 2-D CFD simulation and those measured at UBC and KSU. A review of these two data sets provides some indication of why there may be more uncertainty associated with their reported data.

Table 1.
Comparison of Measured and Computed Power Conversion Efficiencies for Various VA WECS

CASE	η_M / η_{CFD}
A Sandia Blades at 1% T.I.	1.02
B TMA Blades and Stators Config. 1 (Confidential)	1.10
C TMA Blades and Stators Config 2a With stators in CFD analysis not at optimum orientation (Confidential)	0.93
D TMA Blades and Stators - Config 2b With Stators at optimum orientation (Confidential)	0.91
E Modi Blades (Tested at UBC)	1.14
E Modi Blades with Stators (Tested at CPP)	0.93
F KSU Blades with Stators (Compared against KSU field data)	1.15

The results from Modi [4] for the UBC data include a correction factor for blockage effects. The correction factor, 36% in this case, may be a source of considerable uncertainty in his stated efficiencies (actual measured efficiencies without the correction factor applied were around 50%).

The results from Johnson [3] for the KSU field study also may have a larger than expected uncertainty. The results indicate a strong dependence in the VA WECS conversion efficiency between the approach wind speed and the optimum tip speed ratio. This could be attributed to inaccurate measurement techniques, or unaccounted for mechanical losses (high bearing friction, for example).

Despite the larger uncertainties associated with the UBC and KSU date, these results provided reasonable assurance that the CFD code is providing reliable information about the nature of the flows in and around the blades. The results further validate that the trends in the WECS power performance predicted by the codes (improved or diminished performance) are generally accurate within an uncertainty of ± 10%.

REDUCED-SCALE WIND TUNNEL ANALYSIS

Background

Reduced-scale wind tunnel modeling provides a means for time and cost sensitive analysis of WECS performance characteristics. Because the wind environment can be controlled, power performance curves can be generated in short order. Also, the results from a well-designed and carefully executed wind tunnel modeling study are inherently more accurate than a numerical assessment because there is no need for any closure assumptions, nor problems with invicid potential flow that cannot separate from a body, let alone a sharp corner [6]. A wind tunnel is, in effect, an analog computer and compared with numerical solutions, has "near-infinitesimal" resolution and "near-infinite" memory [7].

The task thus becomes one of defining a "well-designed" and carefully executed study, to fully understand the limitations of such a study, and to interpret the results accordingly. The primary limitation associated with wind tunnel studies has to do with the fact that the presence of the wind tunnel walls produces a closed environment that can alter the airflow characteristics in and around the WECS. The commonly referred to "blockage effect" has been studied for quite some time for bluff bodies placed in a wind tunnel [9] and more recently for Savonius WECS [4]. The results of Modi [4] indicate that blockage ratios in excess of 5 percent can lead to dramatic increases in the measured power performance, primarily due to an increase in the local velocity associated with the blockage. This 5% blockage limitation will either significantly increase the required cross-sectional area of the wind tunnel test section or reduce the allowable size of the WECS model.

It is not likely to be feasible to conduct such studies in a wind tunnel that is large enough to test full-scale units rated at much more than 1kW. Assuming a 30 percent power coefficient, a 1 kW unit has a projected area of approximately 2m^2 (21.5ft^2), requiring a test section cross-sectional area of approximately 40m^2 (430ft^2). Similarly, a 10 kW unit would require a 400m^2 (4300ft^2) test section, or roughly 20m by 20m in cross-section. There are wind tunnels available of this size, however, they are prohibitively expensive to operate for this type of application. Therefore, it is more likely that a reduced-scale model of the WECS is necessary for most applications.

Conducting tests on reduced-scale models also has its inherent difficulties. The main problem is that the Reynolds number associated with the model-scale simulations are generally one to two orders of magnitude less than those present in the full-scale environment, depending upon the length scale applied to the model. The rotational motion created by propeller type WECS is a result of a lifting force produced by airflow over cambered airfoils. This lift force is highly dependent upon Reynolds number. Therefore, to accurately model the performance characteristics of a propeller type WECS requires that the model and full-scale Reynolds numbers are similar. This can only be achieved by increasing the ratio of wind speed to kinematic viscosity in proportion with the scale reduction. Based on a "reasonable sized" wind tunnel, scale reduction may be on the order of 1:100 or greater for WECS rated at 10 kW and above. If the WECS in placed in a water flume instead of a wind tunnel, the kinematic viscosity is reduced by approximately a factor of 10, leaving an additional factor of 10 to be "made-up" in flow velocity. Rated wind speeds for WECS are typically around 14 m/s (or 30 mph). Therefore, just to achieve Reynolds number equivalency at the rated wind speed, the water flume velocity would have to be in the neighborhood of 140 m/s (300 mph). This velocity of water flow would likely be highly destructive to a WECS model. Thus, modeling propeller type WECS, particularly large units, is outside the limits of most wind tunnels.

This Reynolds number limitation, however, does not apply to all types of WECS. Units that are primarily drag devices or wake derived lift devices can be modeled quite accurately in a reduced-scale environment. The drag coefficient for a plate normal to the approach flow obtains Reynolds number independence at a value of around 1000. Similarly, studies of separation regions behind sharp-edged bodies [10] indicate that Reynolds number independence is achieved for Reynolds numbers between $2x10^4$ to $2x10^5$. Assuming a 15 m/s wind speed and a kinematic viscosity of air of approximately 1.5 x 10^{-5}, the characteristic length is on the order of 20 cm. Therefore, a WECS which generates torque through either drag forces, or wake derived lift can be accurately modeled in wind tunnels with cross sectional areas as small as approximately 1m^2.

Another common problem associated with testing WECS in a wind tunnel environment is that the turbulent flow structure within the wind tunnel does not adequately represent the full-scale environment. Aeronautical wind tunnels are designed to have a homogeneous, low turbulent flow environment. However, small WECS immersed in an atmospheric boundary layer may have hub height turbulent intensities on the order of 15 to 20 percent (typical wind farms may experience turbulent intensities in the neighborhood of 10 to 15 percent). For high lift devices, the presence of turbulence can adversely impact the lift characteristics of the cambered airfoils. Thus, WECS tested in a low turbulent wind tunnel will tend to over predict the performance of such a device. WECS with greater solidity, which rely more on drag forces than lift forces, are less susceptible to influences from turbulence. As such, their potential full-scale performance characteristics can be better simulated in a wind tunnel.

Test Procedures

For this case study, a 0.6m (2ft) diameter model of the VA WECS (a 1:6 scale model of the existing 3.7m diameter prototype) was placed within the test section of CPP's closed circuit wind tunnel. The test section is downwind of a 21m (70 ft) long atmospheric boundary layer development section. The tunnel, which is typically used to model the aerodynamic loading of structures, is capable of reproducing the atmospheric boundary layer velocity and turbulence profiles for model-scales ranging from 1:100 to 1:6000. For this study, all flow-conditioning devices were removed to minimize the boundary layer and the turbulence intensity at the test section. Power performance tests conducted in the wind tunnel during previous studies has indicated that exposing the VA WECS turbine to a velocity and turbulence boundary layer reduces the reproducibility of the results without significantly altering the measured performance characteristics. At some point, it may be desirable to further validate this finding.

The VA WECS model has a total projected area of approximately 0.68 m^2. The cross-sectional area of the wind tunnel is approximately 8.4m^2, 3.0m wide by 2.8m tall, resulting in a blockage ratio of approximately 8 percent. This is a slightly greater percentage than is typically acceptable in a fixed wall wind tunnel [4] without applying any sort of a blockage correction factor. However, the atmospheric boundary layer wind tunnel is equipped with a flexible roof structure that can be adjusted to maintain a zero pressure gradient along the test section. This somewhat mitigates the effect of blockage. Studies conducted by CPP with bluff bodies in the wind tunnel indicate that nominal effects are noted for blockage ratios up to and slightly above 10 percent. Therefore, the measured performance values obtained during this study were not corrected for any potential blockage effects due to the presence of the wind turbine and its wake.

The ultimate determination of whether or not blockage is an issue is done by comparing the power performance measurements obtained in the wind tunnel to those obtained in an unobstructed open airflow.

Data Acquisition

The VAWT was mounted on a 3m diameter turntable with the shaft extending below the rotor assembly. A Vibrac TQ3200 torque meter was placed in-line between the turbine and a 1 hp permanent magnet DC generator. A diagram of the instrumentation set-up for the reduced-scale power performance evaluation is provided in Figure 6.

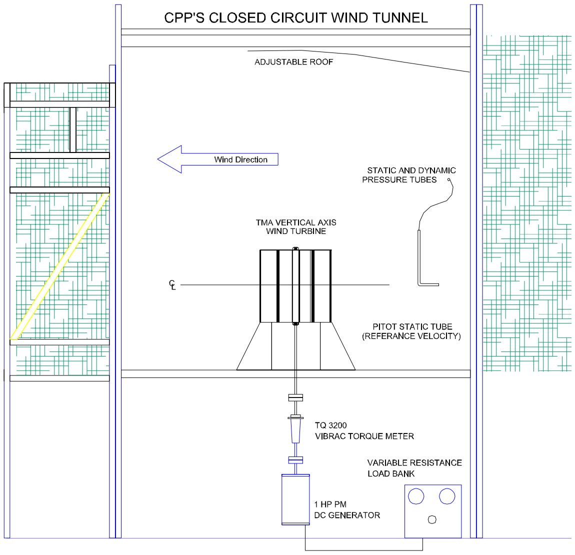

Figure 6 Instrumentation schematic for the reduced-scale testing in CPP's atmospheric boundary layer wind tunnel.

Power output from the wind turbine, P_t, was computed as the product of the applied torque and the angular rotation; both parameters were measured using the Vibrac torque and tachometer. The torque meter calibration was checked by a static torque calibration. The angular rotation calibration was checked with a pulse counter measuring the 900 pulses per revolution supplied from the tachometer. A mechanical load was applied to the turbine shaft using the DC generator to control the angular rotation of the turbine. A DC load bank with variable resistors (ranging from 1 ohm to approximately 250 ohm) was wired in series with the terminals of the DC generator. Increasing the resistance across the generator decreases the load on the turbine and increases the angular rotation and vise versa.

The wind speed approaching the model was monitored using a pitot-static tube positioned upwind of the turbine at the mid-rotor height of the VA WECS.

Results

Power performance tests with the reduced-scale wind turbine model were conducted both prior to and after the CFD analysis. Initial tests were conducted with a scale model of a 3.7m diameter by 3.7m tall prototype. The wind tunnel model was designed to accommodate changes in the number of rotor blades, the number of stator blades, the angle of attack of the stators, and the gap between the rotor blades and the central shaft. During the three days of testing 13 different configurations were evaluated. A power curve of efficiency versus tip speed ratio was developed for each configuration. The results indicate that the maximum power efficiency was increased by approximately a factor of 3.5 between the inventor's initial concept and the best performing configuration.

After the CFD analysis was completed, an additional set of wind tunnel tests were performed with the best performing blade design defined by the CFD analysis, to validate the CFD simulations. During this phase, minor adjustments were also made to further optimize the design of the VA WECS. During the second phase of wind tunnel testing, which lasted approximately two and a half days, an additional 23 configurations were evaluated. The results indicate that the maximum power efficiency was increased by 30 percent with the new blade design defined by the CFD analysis, and by an additional 30 percent by "tweaking" the VA WECS configuration in the wind tunnel. In total, the combined increase in efficiency realized utilizing the wind tunnel and the CFD analysis was more than a factor of six.

FULL-SCALE ANALYSIS

Background

The existing wind turbine testing standards [11 and 12] state that the only recognized method for conducting power performance testing is with a full-scale unit in a field environment. None of the standards recognize either wind tunnel tests or computational analysis as valid methods for measuring the power efficiency of a WECS. Therefore, any potential commercial WECS unit should undergo field-testing. However, in practice, obtaining accurate field results can be difficult, costly, and time consuming. Fifty different configurations were evaluated during the CFD and reduced-scale testing phases of this study. Had these only been evaluated in the field, the study would have required many years to conduct, and would have resulted in a much higher expense. Thus, when time and cost constraints exist, field-testing is probably not the best option for product development; rather it is preferable, where possible, to limit field testing to final validation and certification.

Test Procedures

A performance analysis was conducted on the existing 3.7m diameter by 3.7m VA WECS. The analysis was used to validate the results conducted in the wind tunnel and to determine if there is any "scale-up" factor due to increased efficiency at higher Reynolds numbers.

The VA WECS was instrumented with an Omega TQ503R-200 torque meter and a H1512-009 Airpax Halofax tachometer. A load was placed on the turbine with a 10 HP 2DM2332T Baldor Vector Drive Motor. The imposed load was controlled through a Baldor 2D22H410-EL Line Regen Vector Drive. The torque meter and tachometer were placed in line between a 1:16 ratio 90° gear reducer and the Vector Drive Motor. A schematic of the instrumentation layout for the 3.7m by 3.7m turbine is shown in Figure 7.

The climatic conditions were also monitored on the site. Approach wind speeds were measured using an R.M. Young Model 05103 Wind Monitor anemometer. The anemometer was placed at 5.5m above grade on a 10-meter tower located approximately 37m to the north-northeast of the VA

WECS. The predominant winds at the site come from the northwestern quadrant. Wind speed measurements from this anemometer were corrected to the hub height wind speed using a site calibration procedure. The ambient air temperature and relative humidity were monitored using a Campbell Scientific, Inc. CS500 probe. The CS500 probe was mounted on the 10m tower at 3m above grade. Barometric pressure was monitored with a Campbell Scientific, Inc. CS105 Barometric Pressure sensor. The barometric sensor was located within the housing at the base of the 3.7m by 3.7m turbine.

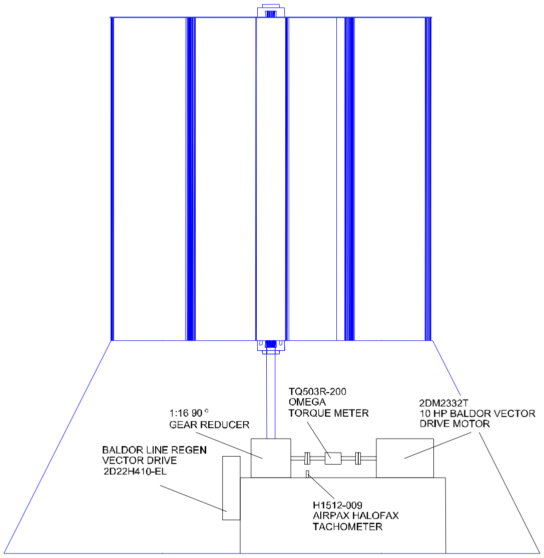

Figure 7 Instrumentation schematic for the full-scale field-testing.

The data acquisition was performed using a Campbell Scientific, Inc. CR10X measurement and control system. The climatic data, ambient temperature, relative humidity, and barometric pressure were recorded and stored on an hourly basis. Wind speed, wind direction, torque and shaft speed were sampled at a rate of 1 Hz and averaged over a 1 minute time period, following the guidelines in the draft annex to the IEC testing standard [12] for small WECS.

Comparison with Wind Tunnel Results

The power performance for the field unit was measured over a period of five months, following the procedures discussed above. The results were compared against the initial wind tunnel tests to validate the wind tunnel tests and to determine if there may be a scale-up factor associated with the turbine that would result in greater efficiencies at larger scales (typical of a lift driven device). Figure 8 below shows a plot of the measured performance in the field. Overlaid on the plot is the power performance curve for the same configuration measure in the wind tunnel. As in most field data, there is a great deal of scatter in the data, although the trend appears to match the wind tunnel data.

A better representation of the data is obtained by binning the field results. Typically, field data is binned by the hub height wind speed [11,12]. However, in this case the data was binned in intervals of tip speed ratio because the load induced on the generator was varied throughout the study. The results of binning by tip speed ratio are shown in Figure 9. These results indicate a very good agreement between the model-scale tests conducted in the wind tunnel and the full-scale field tests.

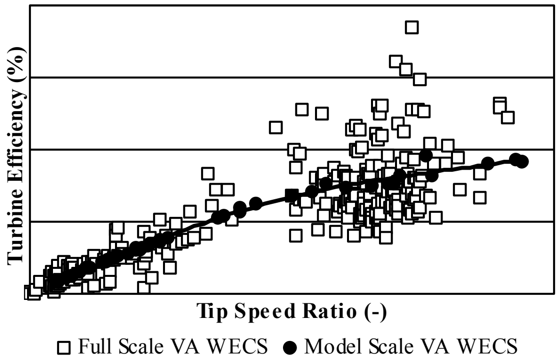

Figure 8 Measured power conversion efficiency in the field vs. the reduced-scale measurements in the wind tunnel.

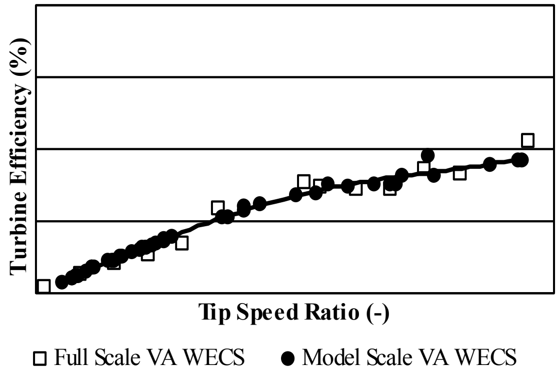

Figure 9 Binned data of measured power conversion efficiency in the field vs. the reduced-scale measurements in the wind tunnel.

While these results might suggest that there is only a slight, if any, “scale-up” factor, the results are not entirely conclusive. First, the power measurements on the full-scale VA WECS were made with a 90° gear reducer inline, while no gearbox was used in the wind tunnel tests.

Any inefficiency associated with this gearbox would reduce the measured efficiencies of the full-scale VA WECS. Second, although care was taken to avoid a significant effect, the reduced-scale wind tunnel measurements may overestimate the VA WECS performance due to blockage effects. Third, the wind tunnel data indicates that the performance of the VA WECS is somewhat dependent upon the approach wind direction. The data collected in the wind tunnel was all obtained at the optimum wind direction. The field data includes data from all wind directions. Since no effort was made to correct for wind direction effects, or to eliminate data from any particular wind direction, one would expect the results from the full-scale tests to be negatively biased when compared with those measured in the wind tunnel.

CONCLUSIONS

The results of this study indicate that a three-tiered approach to evaluating performance characteristics of novel WECS has been successfully implemented. A significant increase in the power performance of the TMA VA WECS was realized in a cost and time effective manner. Had only field testing been utilized, it is anticipated that a similar study, which included investigating over 50 different configuration, would have taken many years to conduct and would have been many times more expensive.

Each of the methods utilized, CFD, wind tunnel, and field study, have been shown to have their unique abilities and challenges. However working together they provide a cohesive design tool that provides invaluable insight into the flow phenomena that drives the WECS while also providing adequate validation to assure the investigator that the analysis truly represents the full-scale field environment.

ADDITIONAL RESEARCH

Since this investigation has shown that both CFD and wind tunnel based measurements can be used to adequately define the power performance characteristics of a novel VA WECS during the design process, the next step is to evaluate their capability for estimating other design parameters, such as structural loads. If this process proves to be successful, it will further enhance the usefulness of computational simulations and reduced-scale testing in the development and design of novel WECS.

REFERENCES

[1]Batchelor, G.K., “An Introduction to Fluid Dynamics,” Cambridge University Press, Cambridge, England, 1967.

[2]Blackwell, B. F., et al., Wind Tunnel Performance Data for Two- and Three-Bucket Savonius Rotors (SAND76-0131), Sandia Laboratories, Albuquerque, NM, 1977.

[3]Johnson, G. L., A modular Savonius Wind Turbine (148), Kansas State University, Manhattan, Kansas, 1981.

[4]Modi, V. J., and Fernando, M. S. U. K., "On the Performance of the Savonius Wind Turbine," *Journal of Solar Energy Engineering*, Vol. 111, pp. 71-81, 1989.

[5]Cochran, B. C., and Peterka, J. A., Performance Evaluation of the TMA, Inc. Vertical Axis Wind Turbine (01-2190), Cermak Peterka Petersen, Inc., Fort Collins, Colorado, 2002.

[6]Snyder, W.H., “Guideline for Fluid Modeling of Atmospheric Diffusion,” Meteorology and Assessment Division, Environmental Sciences Research laboratory, U.S. EPA, Research Triangle Park, NC, 1981

[7]Corsin, S., “Turbulent Flow,” *Am. Scientist*, v. 49, no. 3, p. 300-25, 1961.

[8]Launder, B.E., G.J. Reece, and W. Rodi, “Progres in the Development of a Reynolds-Stress Turbulence Closure,” *J. Fluid Mech.*, 68 (3):537-566, April 1975.

[9]Maskell, E.C., “A Theory of Blockage Effects on Bluff Bodies and Stalled Wings in a closed Wind Tunnel,” Ministry of Aviation, Aeronautical Research Council Reports and Memoranda, R&M No. 3400, Her Majesty’s Stationery Office, London, UK, 1965.

[10]Smith, E.G., “The Feasibility of Using Models for Predetermining Natural Ventilation,” Report No. 26, Texas Engineering Experimental Station, Texas A&M College, College Station, Texas, 1951.

[11]AWEA, “AWEA 1.2-2000, Performance Testing of Small Wind Energy Conversion Systems - Draft 1,” American Wind Energy Association, Washington D.C., May 20, 2000.

[12]IEC, “IEC 61400-12, Wind turbine generator systems - Part 12: Wind turbine power performance testing,” First Edition 1998-02, International Electrotechnical Commission, Geneva Switzerland, 1998.