

Determining Diffuser Augmented Wind Turbine performance using a combined CFD/BEM method

J E Kesby, D R Bradney and P D Clausen

School of Engineering, Faculty of Engineering and the Built Environment, The University of Newcastle, Australia

Abstract. The optimisation of a Diffuser Augmented Wind Turbine has traditionally focused on maximising its power output. Optimising the design of the blade and the shape of the diffuser for maximum turbine power over a range of wind velocities is a complex process, as each will influence the others flow regime. In this paper we propose a method that combines the predictions of flow through a diffuser, using computational fluid dynamics, and the flow from a turbine blade using a modified blade element theory to predict the power output of a diffuser augmented wind turbine. Good agreement was found between the predictions from this new method and experimental data from the literature.

1. Introduction

The addition of a diffuser to a wind turbine, known as a Diffuser Augmented Wind Turbine (DAWT), has been found to increase power output for a given wind speed [1, 2] and to maintain the power production capabilities in turbulent conditions [3, 4], making a DAWT more suited to small scale energy production in an urban environment than a traditional Horizontal Axis Wind Turbine (HAWT). When combined with 3D printing technology to manufacture the turbine blades and diffuser, the Levelised Cost of Energy (LCoE) of a small scale wind turbine could be significantly reduced, making them more attractive to the consumer for use either independently or in a wind/solar hybrid generation system, providing greater energy security for the household, business or community.

The design of both the turbine and diffuser in DAWT must be done simultaneously as the flow from the turbine is informed by the performance of the diffuser and vice versa. It is technically possible to simulate the flow from the turbine and through the diffuser using CFD, however, the solve times would be prohibitively long and would still require an iterative approach to find the optimal turbine design and diffuser shape for maximum Annual Energy Production (AEP). Here we propose a process that combines Computational Fluid Dynamics (CFD) and Blade Element Momentum (BEM) theory to determine the performance of a DAWT. The new method will be validated against experimental results for a DAWT from the literature of van Dorst [5].

2. Theory

2.1 Effects of thrust and swirl on diffuser performance

Theoretical analysis shows that the increase in power output of a DAWT over a standard HAWT is directly proportional to the increase in mass flow at the blade plane [6-8]. Therefore the ultimate aim in designing a diffuser for use in a DAWT is to maximise the augmentation of mass flow through the diffuser. For an empty diffuser this augmentation is relatively straightforward to maximise: the geometry of the diffuser is altered to give the greatest wake expansion whilst minimising inner surface separation. However, it has been shown experimentally that introducing swirl through a diffuser can improve its



effectiveness over a diffuser with only axial flow [9, 10]. There are two primary mechanisms for this: firstly, swirl can reenergise the boundary layer on the inside surface of the diffuser, delaying separation in an environment of predominantly adverse pressure gradients, and secondly the increase in kinetic energy of the flow due to swirl results in a decrease in static pressure and a lower sub-atmospheric pressure at the diffuser exit. The lower exit pressure has the effect of drawing more mass flow through the diffuser thereby increasing the energy available for extraction. However, too much swirl can result in a recirculation region in the core flow [10], a phenomenon known as wake stall. Through Computational Fluid Dynamics (CFD) analysis, Hjort and Larsen [11] have shown that wake stall can become a major impediment to mass flow augmentation under certain blade geometries and flow conditions. In light of this, an optimisation process is required to maximise the mass flow through the diffuser when designing a DAWT, a process well suited to CFD simulations.

2.2 CFD and the Actuator Disc

The unfortunate drawback of CFD is the time needed to undertake simulations where the flow and geometry are complex. Determining the power output of a DAWT with a specific blade geometry can be very time consuming as the rotating blades need to be modelled, resulting in an extremely long simulation time. As the design process is iterative to determine the optimal wind turbine/diffuser design, the time taken would be prohibitive using only a CFD approach.

To reduce computational time, the current research utilises CFD simulations using ANSYS CFX in conjunction with the Actuator Disc methodology as described by Shives and Crawford [12] to simulate the thrust and swirl that a turbine induces on the airflow through a diffuser. This method utilises a force per unit volume in the axial and tangential directions to simulate the effects that rotating blades impart on the airflow based on a desired thrust and tip speed ratio. The specific geometry of the blades are not modelled. The axial force is given by:

$$F_a = \frac{C_T \rho U_0^2}{2t_d} \quad (1)$$

where C_T is the thrust coefficient, ρ is air density in kg/m^3 , U_0 is freestream velocity in m/s and t_d is the thickness of the actuator disc in m . The tangential force is given by:

$$F_t = F_a \frac{U_a - \frac{d}{l}(r\Omega - U_t)}{(r\Omega - U_t) + \frac{d}{l}U_a} \quad (2)$$

where U_a is the axial velocity at the disc in m/s , d/l is the drag to lift ratio, r is the local radius in m , Ω is the angular velocity of the disc in rad/s and U_t is the tangential velocity in m/s . The value of Ω will be dependent on the nominated tip speed ratio (TSR) which is defined as the tangential velocity of the blade over the freestream velocity. For the interested reader, Shives and Crawford [12] provide a good discussion of the method of obtaining equations (1) and (2).

2.3 Modified BEM method

BEM is a computationally efficient method of calculating the power output of a wind turbine that is widely used in wind energy industry to optimise HAWT blade design. Tavares Dias do Rio Vaz *et al* [13] have proposed the use of a modified BEM methodology to predict the power output of a DAWT. The modification implements an augmentation factor, γ , which is defined as the velocity increase at the

blade plane for an empty diffuser normalised by the freestream velocity. γ is a radially varying parameter as the velocity increase is not uniform across the blade plane and can be determined experimentally, or as in the case of the current research, through CFD. Figure 1, obtained from [13], shows the forces applied to the blade element by the airflow for a DAWT along with relevant angles and velocity relationships.

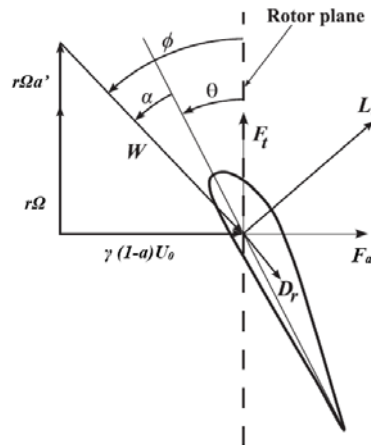


Figure 1. Forces applied to the blade element by the airflow. [13]

The augmentation factor, γ , is implemented into the standard BEM method by including it in the calculation for the inflow angle, ϕ . From Figure 1, the equation for ϕ is given by:

$$\phi = \tan^{-1} \left[\frac{\gamma(1-a)V_0}{(1+a')\Omega r} \right] \quad (3)$$

where a' is the tangential induction factor and a is the axial induction factor taking into account the diffuser effect, that is, the fractional decrease in axial velocity when compared to the empty diffuser. In equation form, a is defined by:

$$a = \frac{\gamma V_0 - V_1}{\gamma V_0} \quad (4)$$

The inclusion of γ in the inflow angle calculation results in the axial induction equation becoming:

$$\frac{a}{(1-a)} = \gamma^2 \frac{\sigma C_a}{4 \sin^2 \phi} \quad (5)$$

where C_a is the axial force coefficient, given by:

$$C_a = C_L \cos \phi + C_D \sin \phi \quad (6)$$

where C_L and C_D are the lift and drag coefficients respectively. The tangential induction equation remains as for standard BEM and the normal iteration process is employed for each blade element until the axial and tangential induction factors are found, from which the thrust and torque can then be calculated.

The method proposed by Tavares Dias do Rio Vaz *et al* [13] is a very useful and computationally efficient manner of determining the power output for a given DAWT. However, the method has the following three shortcomings: the first, is the assumption that there are no losses through the diffuser, which is an idealised situation unlikely to be achieved in a real diffuser; the second is that there is no provision for the improvement in mass flow augmentation or diffuser efficiency through the application of swirl or thrust, meaning that the diffuser is essentially decoupled from the turbine, and lastly is the method's dependence on the freestream velocity at which γ is determined as the diffuser's performance is affected by the velocity of the airflow. The proposed combining of the CFD/BEM method aims to address these shortcomings in a computationally efficient manner.

2.4 Combined CFD/BEM method

The aim of the combined CFD/BEM method is to adjust the velocity augmentation factor, γ , to account for the increase in mass flow from the improvement in diffuser performance due to swirl and thrust. It uses CFD to account for both the losses through the diffuser and the increases in mass flow under a range of thrust, swirl and velocity conditions, the data of which is then utilised in a specifically written BEM code.

Figure 2, from [13], shows the important locations of the airflow through a DAWT.

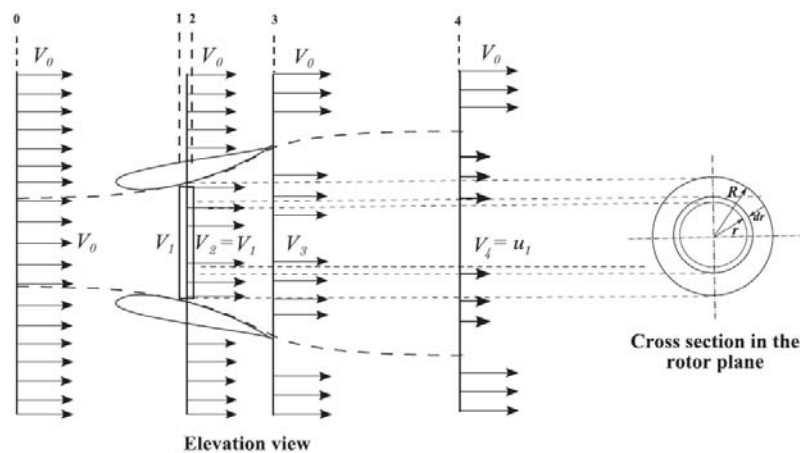


Figure 2. Important locations on a DAWT. [13]

From Figure 2, Point 0 is far upstream of the DAWT (the freestream); Points 1 and 2 are respectively immediately before and immediately after the blade; Point 3 is the diffuser exit, and Point 4 is in the far wake. These locations will be denoted by subscripts in the following equations.

Mass continuity dictates that the mass flow crossing the blade plane of the diffuser must equal the mass flow at the diffuser exit. Assuming incompressible flow, this relationship is given by:

$$A_2 V_{2a} = A_3 V_{3a} \quad (7)$$

where A is the area at the denoted location, and V_a is the axial velocity at the denoted location.

From Figure 1 it can be seen that $V_{2a} = \gamma V_0 (1-a)$, which allows Equation (7) to be rewritten as:

$$A_2 \gamma V_0 (1 - a) = A_3 V_{3a} \quad (8)$$

Equation (8) makes the assumption that the flow at the diffuser exit occurs across the entire exit area *i.e.* there is no separation on the inner surface of the diffuser. To account for separation, an effective exit area term, A_{3eff} , is introduced replacing A_3 in the above equation.

A_{3eff} is defined in the current method as being the cross sectional area of the non-stalled airflow at the diffuser's exit. Assuming there is no separation in the diffuser at the blade plane, the effective area is given by equation (9):

$$A_{3eff} = \left(\frac{V_{2ar} A_2}{V_{3ma} A_3} \right) A_3 = \frac{V_{2ar} A_2}{V_{3ma}} \quad (9)$$

where V_{3ma} is the mass flow averaged axial velocity at the diffuser exit, and V_{2ar} is the area averaged axial velocity immediately behind the actuator disc.

The combined CFD /BEM method aims to adjust γ to account for the improvement in the diffuser performance due to the presence of thrust and swirl. To do so, the γ value is split into two components:

$$\gamma = \gamma_i \gamma_{adj} \quad (10)$$

where γ_i is the initial augmentation factor in the 'baseline' case and γ_{adj} is the adjustment that is to be made to the base augmentation factor.

Combining Equations (8) and (10) and rearranging yields:

$$\gamma_{adj} = \frac{V_{3a}}{\beta_{eff} V_0 \gamma_i (1 - a)} \quad (11)$$

Here β_{eff} is the increase in the effective exit area ratio over the baseline case and is given by:

$$\beta_{eff} = \frac{A_2 A_{3i}}{A_{3eff}^2} \quad (12)$$

where A_{3i} is the effective area of the baseline case.

2.5 Discrete blade effects

Discrete blade effects, also known as hub and tip loss, will effect DAWT performance. It has been shown that tip loss will be lower than those of a HAWT as the proximity of the diffuser surface to the blade tips will interact with and partly suppress the development of blade tip vortices [14]. This suppression effect will be dependent on a range of factors, including blade tip geometry, proximity of the blade tip to the diffuser surface, flow velocity and tip speed ratio. Hub losses will be as for a standard HAWT as there is no hub suppression effect inherent in a diffuser.

The current work makes use of the Prandtl tip and hub loss as utilised in *AeroDyn*, the software package developed for the National Renewable Energy Laboratory in the United States [15]. Hub and tip loss can be independently included or excluded in the BEM program written for the current work.

3. Methodology

To utilise the proposed method, there are three components that are required from CFD which are then exported to the BEM program:

- 1) The velocity profile at the blade plane for the 'baseline' case, namely the empty diffuser at the base freestream velocity, which in the present instance is 5 m/s. The velocity profile is normalised to the freestream velocity to obtain the relative velocity. The relative velocity for the base freestream velocity is implemented as γ_i in the BEM program.
- 2) The effective exit area of the empty diffuser at the base freestream velocity.
- 3) The relationship between the total velocity immediately after the actuator disc (blade wake velocity, calculated by area averaging) and the axial diffuser exit velocity (calculated by mass flow averaging).

The current work has investigated a large number of thrust, swirl and freestream velocity combinations to establish a relationship between the velocity after the actuator disc and the exit velocity. Two different freestream velocities have been investigated: 5 m/s and 20 m/s. C_T has ranged from 0 (a diffuser with no actuator disc) to 1 (the actuator disc applying full airflow thrust). Finally, three levels of swirl have been applied: no swirl (TSR of 0), low swirl (TSR of 5) and high swirl (TSR of 2)

The process of implementing the combined CFD/BEM method for the current work is as follows:

- 1) Model the diffuser in Creo Parametric and export the model to ANSYS CFX.
- 2) Obtain the velocity data through the empty diffuser at the two different freestream velocities. Determine the velocity profile at the blade plane and the effective exit area for the baseline case (empty diffuser at 5 m/s freestream velocity).
- 3) Apply a thrust at the blade plane using the actuator disc method for the different freestream velocities.
- 4) Apply swirl in combination with thrust using the actuator disc method.
- 5) Establish a relationship between the total blade wake velocity and the axial exit velocity.
- 6) Import the three components required from CFD into the BEM program (written in MATLAB).
- 7) Input the freestream velocity to be investigated along with the blade geometry data into the BEM program.
- 8) Calculate axial and tangential induction using the standard BEM iteration process and hence the total blade wake velocity.
- 9) Adjust γ at each blade element and recalculate axial and tangential induction.
- 10) Repeat steps 8) and 9) until convergence is reached, that is there is no adjustment to the γ values between iterations.
- 11) Calculate the thrust and torque for each blade element and hence power output of the DAWT.

4. Results

To ensure that the results from the proposed combined CFD/BEM method replicate real world performance, validation against experimental results is required. In the current work, the proposed method has been compared to experimental data published by van Dorst [5], who undertook wind tunnel testing of three different blade configurations in a diffuser. The blade geometries consisted of:

- a) A blade design that was originally manufactured for the diffuser (the Original blade).
- b) A blade that van Dorst optimised for the diffuser which had a chord length of up to 230mm (the Optimal blade).
- c) A variation of the optimal blade that had a maximum chord length constraint of 130mm to accommodate mass production manufacturing techniques. This resulted in a large section of the

blade nearest the hub having a constant chord length, giving the blade a more linear appearance and is referred to by van Dorst as the Linear Blade, a convention which will continue in this paper.

4.1 CFD implementation

The experimental data used for validation was obtained from wind tunnel testing, and to allow a direct comparison, the DAWT was modelled in a wind tunnel, the geometry of which was obtained from the published experimental data [5]. To reduce the number of elements and therefore the simulation run time, a 10° section of the total wind tunnel and DAWT was modelled with rotational periodicity boundary conditions applied to the section surfaces in a manner similar to that employed by Hansen *et al* [6]. A mesh independence study was undertaken and showed that a mesh consisting of 924443 elements would give mesh independent results. The model had 20 inflation layers for the diffuser with a resultant average y^+ value of 0.75. A Shear Stress Transport (SST) turbulence model (a two equation, Reynolds-averaged Navier-Stokes (RANS) based model) with curvature correction was utilised due to its sensitivity to adverse pressure gradients. The Reynolds number for the diffuser was calculated to be 3.3×10^5 at 5 m/s freestream velocity and the flow was taken to be fully turbulent along the diffuser.

4.2 Empty diffuser velocity profiles

The relative velocity for the 'base' condition of an empty diffuser at 5 m/s freestream velocity was obtained from ANSYS CFX and compared to results from [5]. Figure 3 shows the results obtained from CFX along with the published experimental data.

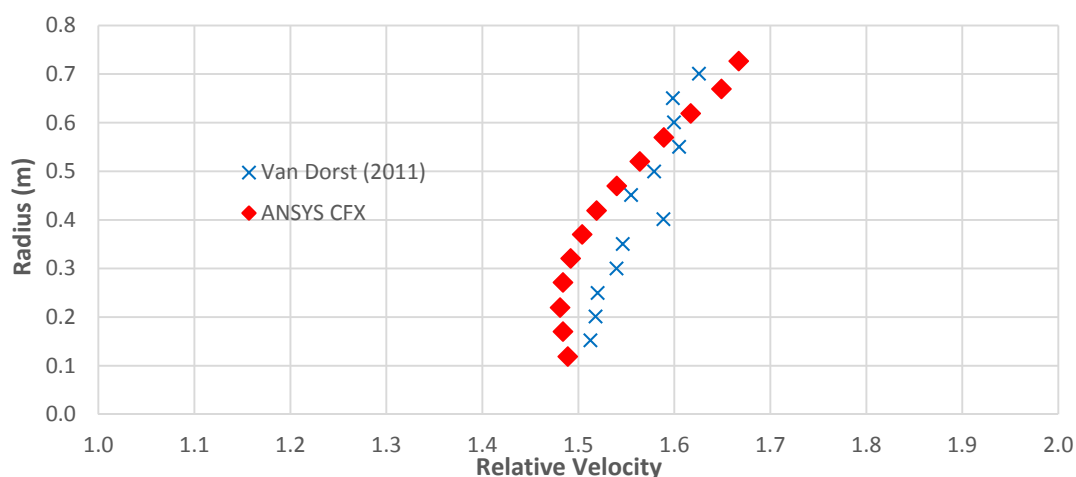


Figure 3. Velocity profile at the blade plane of the empty diffuser.

It can be seen from the results shown in Figure 3 that the velocity profile determined from ANSYS CFX has good correlation with the experimentally obtained profile from van Dorst [5]. The largest discrepancy between the two velocity profiles is at a radius of 0.4m, where ANSYS CFX under predicts the velocity by around 4.9%, although it does appear that the experimental velocity at this point is an outlier. The next largest difference, at 0.35m radius, has an under prediction of around 3.2%. This velocity profile was exported to the BEM program to be utilised as the initial augmentation factor, γ_i . The initial effective exit area, A_{3i} , was calculated to be 2.618 m^2 .

As discussed previously, the combined CFD/BEM method requires the mass flow averaged exit velocity to calculate the effective exit area of the diffuser. This data was obtained from ANSYS CFX at freestream velocities of 5 m/s and 20 m/s for a range of thrust coefficients and tip speed ratios. The data was plotted against the area averaged total velocity at the blade plane, the results of which are shown in Figure 4.

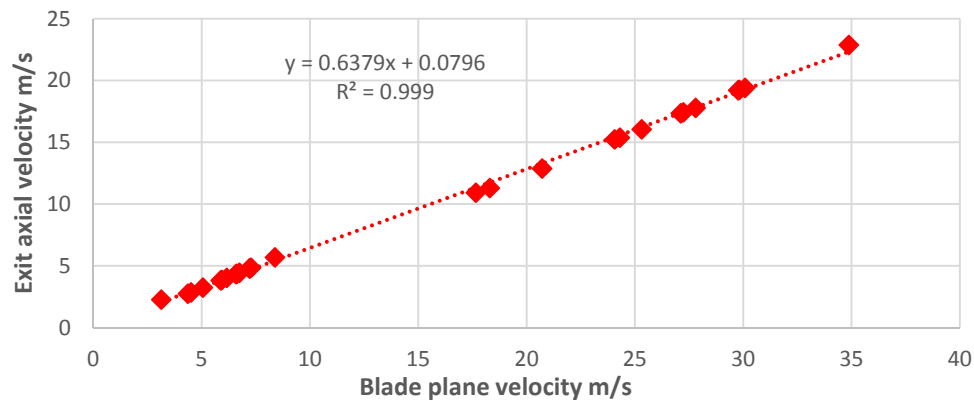


Figure 4. Total blade wake velocity versus axial diffuser exit velocity

The results in Figure 4 exhibit an excellent linear relationship between the total velocity at the blade plane and the axial velocity at the diffuser exit. The equation displayed in Figure 4 was utilised to calculate the effective exit area for the diffuser in the BEM program for the differing blade geometries and free stream velocities. The current work has investigated a wide range of thrust, swirl and freestream velocity combinations to establish a relationship between the velocity after the actuator disc and the exit velocity. However, given the very strong linear relationship between the two, it is anticipated that future work will require fewer data sets, resulting in a reduced total simulation time in predicting the power output for a DAWT over a wide range of blade geometries and freestream conditions.

4.3 Power output

Figure 5 shows the published power output of the various blade geometries from [9] and the power output obtained from the combined CFD/BEM method with and without tip loss effects. Hub loss was included in both cases.

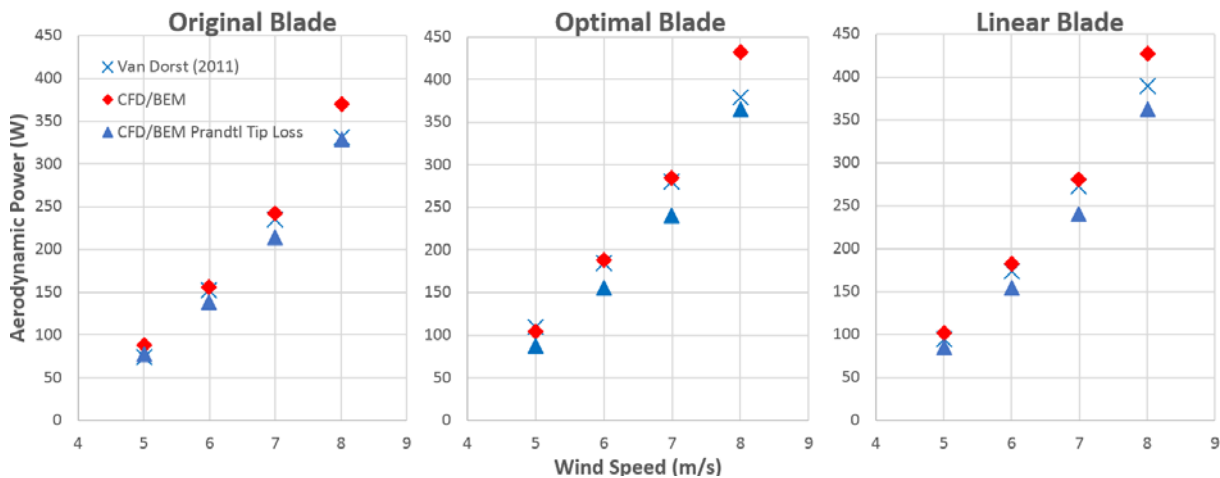


Figure 5. Experimental and predicted power output for the three blade geometries

The results in Figure 5 show a good correlation between the simulated and experimental power output for the DAWT with the various blade geometries. With the exception of the power output at 5 m/s for the optimal blade, the experimental power outputs are bounded by the predicted values, with the upper bounds given by the combined CFD/BEM method without tip loss effects, and lower bounds provided by the proposed method with Prandtl tip loss effects. These results indicate that tip loss needs to be accounted for when predicting the power output of a DAWT, although the method developed by Prandtl substantially over predicts the effect when applied to DAWTs. The largest over prediction in power output for the combined method without tip loss effects is 44 Watts for the optimal blade at 8 m/s, equating to an over prediction of approximately 12%. The largest under prediction with tip loss effects is 40 Watts for the optimal blade at 7 m/s, an under prediction of 14 %. At 8 m/s, the experimental data actually shows the power output for the optimal blade is less than for the linear blade, a result not anticipated by the proposed method.

5. Conclusions

This paper presents a combined CFD/BEM method of predicting the power output of a DAWT, following on from the work of Tavares Dias do Rio Vaz *et al* [13]. The proposed method uses CFD to determine a diffuser's performance under a variety of thrust, swirl and freestream conditions; these results are utilised by a modified BEM program to calculate the power output of the DAWT for a specific freestream velocity and blade geometry. In this manner, the proposed method harnesses both the power of CFD and the speed of BEM to predict the power output of a DAWT in a computationally efficient process.

The proposed method requires data showing the relationship between the velocity at the blade plane and the axial velocity at the diffuser exit to determine the effective diffuser exit area. The current work has investigated a large number of thrust, swirl and freestream velocity combinations to establish this relationship, however, given the very strong linear relationship between the two, it is envisaged that fewer data sets will be required when investigating diffuser performance in the future, resulting in a much reduced total simulation time.

The proposed method has shown good agreement with published experimental data for a range of freestream velocities and blade geometries with the results obtained indicating that blade tip loss needs

to be accounted for when predicting the power output of a DAWT, although the method developed by Prandtl substantially over predicts the effect when applied to DAWTs. A useful area of further research would be in developing a method that accounts for the wide range of factors affecting tip loss for a DAWT. To the author's knowledge, a satisfactory method has not yet been proposed.

6. References

- [1] Ohya Y , Karasudani T , Sakurai A , Abe K and Inoue M 2008 Development of a shrouded wind turbine with a flanged diffuser *J. of Wind Engineering and Industrial Aerodynamics* **96** 524-39
- [2] Bet F and Grassmann H 2003 Upgrading conventional wind turbines *Renewable Energy* **28** 71-8
- [3] Armfield S W and Fletcher C A J 1989 Comparison of k- ϵ and algebraic Reynolds stress models for swirling diffuser flow *Int. J. of Numerical Methods in Fluids* **9** 987-1009
- [4] Phillips D G 2003 An investigation on diffuser augmented wind turbine design (Ph.D. Thesis) The University of Auckland, New Zealand
- [5] Van Dorst F 2011 An Improved Rotor Design for a Diffuser Augmented Wind Turbine (Master's Thesis) Delft University of Technology, Netherlands
- [6] Hansen M O L , Sørensen N N and Flay R G J 2000 Effect of Placing a Diffuser around a Wind Turbine *Wind Energy* **3** 207-13
- [7] van Bussel G J W, editor An assessment of the performance of diffuser augmented wind turbines (DAWT's). *3rd ASME/JSME Joint Fluids Engineering Conf.*; 1999. San Francisco, CA: ASME.
- [8] van Bussel G J W 2007 The science of making more torque from wind: Diffuser experiments and theory revisited *J. Phys.: Conference Series* **75** 012010
- [9] Foreman K M and Gilbert B L 1979 Further investigations of diffuser augmented wind turbines. Part 2. *Technical report, Grumman Aerospace Corp., Bethpage, NY (USA)*
- [10] Clausen P , Koh S and Wood D 1993 Measurements of a swirling turbulent boundary layer developing in a conical diffuser *Experimental Thermal and Fluid Science* **6** 39-48
- [11] Hjort S and Larsen H 2015 Rotor Design for Diffuser Augmented Wind Turbines *Energies* **8** 10736 - 74
- [12] Shives M and Crawford C, ed. Ducted Turbine Blade Optimization Using Numerical Simulation 2011.
- [13] Tavares Dias do Rio Vaz D A , Amarante Mesquita A L , Pinheiro Vaz J R , Cavalcante Blanco C J and Pinho J T 2014 An extension of the Blade Element Momentum method applied to Diffuser Augmented Wind Turbines *Energy Conversion and Management* **87** 1116-23
- [14] Abe K , Kihara H , Sakurai A , Wada E , Sato K , Nishida M and Ohya Y 2006 An experimental study of tip-vortex structures behind a small wind turbine with a flanged diffuser *Wind and Structures, An Int. J.* **9** 413-7
- [15] Laino D J and Hansen A C 2002 User's guide to the wind turbine aerodynamics computer software AeroDyn Prepared for NREL Under Subcontract, University of Utah, Salt Lake City, Technical Report No TCX-9-29209-01