Recent advances in modeling of wind turbine wake vortical structure using a differential actuator disk theory

Christophe Sibuet Watters and Christian Masson
École de technologie supérieure, Mechanical Engineering Department, Montreal H3C 1K3, CANADA
E-mail: christophe_watters@hotmail.com and christian.masson@etsmtl.ca

Abstract. This paper presents the recent developments of a new CFD-based method aimed at predicting wind turbine aerodynamics, where velocity and pressure discontinuities are used to model the vortical system that creates lift on the turbine blades. To illustrate the ability of the present model to predict induced wake effect, the case of the taper wing is thoroughly analyzed and effects of both domain discretization and convection scheme are presented. Results are mitigated regarding predicted performance of induced drag, but accurate induced and upstream flow angles values are obtained. The method is even shown to be a useful calculator for the relationship between inflow angle measured upstream and effective angle of attack of a wing section. Interesting results for the NREL phase VI rotor have been obtained showing improvement of the method upon actuator-disk approach in handling tip vortices effect on the blade aerodynamics.

1. Introduction
The accurate prediction of wind turbine aerodynamics is a problem that has received considerable attention since the beginning of the aerodynamic science. Apparently, the art of predicting wind turbine performance has not reached a consensus yet, as showed by the blind code comparison lead by NREL [1] that reported important discrepancies between predicted power curves for 15 types of computer-based analysis. Especially in operation where the blade is partly, or completely stalled, predictive models meet major difficulties to ensure accuracy: reasons for this poor performance can be attributed to the model itself, or to the inputs provided to the model, like the airfoil aerodynamics characteristics. Amongst the sources of discrepancies, the way models handle blade tip vortices is recognized as a key issue by many workers in the field [2, 3, 4]. Theoretically, only vortex or full CFD analysis naturally models the tip vortices and its interaction with the flow. Actuator disk representation of the wind turbine suffers for lack of physical representation of the tip vortices and make use of a tip factor to compensate for their influence.

This paper presents new advances towards the development of a numerical method that models the flow around a wind turbine by surfaces carrying pressure and velocity discontinuities, with the objective to improve predictions of blade tip vortices effects on performance. A similar approach but only with pressure discontinuity has been developed by Massouh et al. [5]. Previous papers [6, 7] have demonstrated the validity and pertinence of this method with applications to
problems having analytical solutions and to the case of the actuator disk. Compared to methods that represent the actuator disk by a set of body forces only [8, 9, 10, 11], no spurious oscillations of the flow parameters happen.

First, mathematical and numerical aspects of the method are briefly reviewed. New results will be discussed for the case of the tapered wing in uniform translation for different taper and aspect ratios. As per the last contributions, a control-volume finite-element method will be used to solve the flow with the introduction of a new, second-order, convection scheme. The computation of the induced drag will be discussed together with the influence of the domain and wing discretization, showing that care must be taken to achieve accuracy with the price of a high computational cost. Preliminary results for the NREL phase VI turbine will also be presented together with an insight into the difficulties met to achieve a solution for the rotating blade.

2. Mathematical model

Theoretical aspects exposed in the next sections are taken from classical concepts of incompressible fluid mechanics; the original idea presented regards the introduction of porous surface of velocity and pressure discontinuities in the flow, and focus is made on the consequences of these discontinuities on the flow, as well as to their evaluations.

2.1. Surfaces of velocity discontinuities - implications

In terms of vorticity, a lifting surface can be decomposed in a set of vorticity surface distributions (vortex sheets), and the flowfield is solely determined by this set. From the point of view of an approach based on flow primitive parameters (velocity and pressure), vortex sheets can equally be modeled as surfaces of velocity discontinuities: it is therefore interesting to study their implications in the context of the Eulerian description of the flow with velocity components \((u,v,w)\) and pressure \(p\) measured in a classical Cartesian \((x,y,z)\) system of axes.

\[ \frac{\delta (\Delta v)}{\delta x} - \frac{\delta (\Delta u)}{\delta y} = 0 \]
Leclerc and Masson [6] have shown the following results: At a given location on a surface of velocity discontinuity, the velocity discontinuity is necessarily tangent to the surface, and provided that no energy is withdrawn from or added to the flow, then a system of force attached to the surface of discontinuity naturally arises and the components of this external force, per unit area are given by:

\[ f_x = \rho w_{av} \Delta u \quad f_y = \rho w_{av} \Delta v \quad f_z = -\rho (u_{av} \Delta u + v_{av} \Delta v) \]  

(2)

where \( u_{av}, v_{av}, w_{av} \) are the component of the mean velocity at the surface of discontinuity, given by:

\[ u_{av} = (u + (u + \Delta u))/2 \quad v_{av} = (v + (v + \Delta v))/2 \quad w_{av} = (w + (w + \Delta w))/2 \]  

(3)

It is interesting to note that the scalar product of the mean velocity and the force vector is null, which is another way to show that the system of force attached to the surface of discontinuity does not exert any work on the flow. A second, direct consequence of this system of force is related to the normal component \( f_z \) which implies that a pressure jump must be prescribed at the surface of discontinuity, whose value exactly equals to \( f_z \).

2.2. Blade-element theory for the finite wing

Figure 2 presents a sketch of a wing section with incoming, constant and uniform flow \( U_{\infty} \). The aerodynamic section of chord length \( c \) is pitched with an angle \( \beta \) towards the incoming flow. In the case of an infinite wing, i.e. an airfoil, the force produced by a constant air flow would consist in a lift force perpendicular to the direction of the flow only. Finite wing theory however, teaches that the system of downwind trailed vorticity induces a downwash of the flow around the wing, measured by the induced angle of attack \( \alpha_i \). Actual lift force is therefore inclined and its projection on the direction of the incoming flow is classically denoted \( D_i \), the induced drag. \( L \) is the lift of the wing being defined as the force perpendicular to the incoming flow.

The finite wing is modeled numerically as a flat surface parallel to the incoming flow which bears velocity discontinuities. \( x \) is the axis in the direction of the incoming flow and \( y \) the axis in the spanwise direction of the wing, as shown in Fig. 1.

For a section of the wing, the lift of the wing is given by the equation:

\[ L = \frac{1}{2} \rho U_{\infty}^2 C_L c \]  

(4)

where \( C_L \) is the lift coefficient of the airfoil defining the wing section, function of the effective angle of attack \( \alpha = \beta - \alpha_i \) and \( \rho \) is density of the air. Let \( \Gamma \) be the circulation of velocity around the airfoil, from the Kutta-Jukowski law, it is also possible to write:

\[ L = \rho U_{\infty} \Gamma \]  

(5)

Combining Eq.(4) with Eq.(5), \( \Gamma \) may be rewritten as:

\[ \Gamma = \frac{1}{2} \rho c U_{\infty} C_L \]  

(6)

Once the circulation \( \Gamma \) around an airfoil is known, the task consists in distributing this circulation along the airfoil chord in the most appropriate way. The constant distribution \( \Delta u = \Gamma/c \) is the simplest but performs poorly since it sets rapid variations of the \( \Delta u \) field, hard to manage numerically without unstabilities in the flow solution. Distributions respecting continuity of the \( \Delta u(x, y, z) \) are more appropriate. In this work, results will be produced using the simple parabolic distribution:
\[ \Delta u_P = \frac{6\Gamma}{c^3} x_P (c - x_P) \]  

where \( x_P \) is the distance from the point P to the leading edge of the wing along the \( x \) axis direction, as shown in Fig. 3. The distribution of \( \Delta v \) is deduced from Eq. (1) and the boundary condition \( \Delta v = 0 \) at \( x_P = 0 \):

\[ \Delta v_P = \frac{6\Gamma}{c^3} \frac{\partial c}{\partial y} \left( x_P^3 - cx_P^2 \right) + \frac{1}{c^3} \frac{\partial \Gamma}{\partial y} \left( 3cx_P^2 - 2x_P^3 \right) \] 

All results shown in the present work are performed using the above distributions of \( \Delta u \) and \( \Delta v \). The solution algorithm is simple: (i) initialize \( \alpha_i \) distribution to zero, (ii) calculate the spanwise distribution of effective angle of attack \( \alpha = \beta - \alpha_i \), (iii) calculate \( C_L \) coefficient across the wing using tables and \( \alpha \) distribution, (iv) set \( \Delta u \) and \( \Delta v \) using Eqs. (6)-(8), (v) solve for the flow and compute induced angle of attack \( \alpha_i \), (vi) iterate by going back to operation (ii) unless convergence is attained.

3. Numerical method

To solve the set of partial differential equations describing the flow evolution (the Navier-Stokes equations), a 3D Control-Volume Finite-Element Method (CVFEM) is used. This method is typical of modern commercial CFD softwares using control volume principles to solve for the flow like CFX Ansys and Fluent. One advantage of this method lies in its ability to easily access meaningful physical quantities like mass or momentum fluxes and, through interpretation of results, understand the limitations of the numerical method.

The calculation domain is discretized using a structured mesh and tetrahedral finite elements. In every tetrahedral element, 3 control surfaces are defined joining the middles of the tetrahedron sides, so that upon assembly of all elements, complete control volumes are formed about each node in the calculation domain, as shown in Fig. 4.

Viscous diffusion and gradient of pressure are evaluated at the control surfaces using linear distributions of velocity and pressure within one tetrahedral element. Convection fluxes however, are evaluated using the first-order Mass-Averaged-weighted (MAW) scheme developed by Saabas [12] or the more recent second-order MAW scheme of Tran et al. [13]. Assembling the contributions of all tetrahedral element of one control volume for mass and momentum balance builds the sets of discretized equations to solve numerically; Mass fluxes (the \( \dot{m} \) in Fig. 4 are related to pressure gradients such that the discretized pressure equations are elaborated from the mass conservation principle.
Boundary conditions typically used in the problems under study are either of the Dirichlet (fixed values) or Neumann (fixed gradients) type. For the case of the finite wing in rotation, an original aperiodic condition has been implemented as discussed in Section 5.

Handling properly the occurrence of velocity discontinuities in the 3D CVFEM necessitates modifications in the numerical code and the introduction of new source terms in the discretized momentum equations. This subject has been explained in previous contributions [14] and will not be further addressed in the present one. It should be noted that commercial codes, at the present time, are unable to cope with velocity discontinuities, while some are designed to handle pressure jumps across surfaces (as a model to porous media for example).

4. Tapered wing problem
The problem of the finite wing in uniform translation is a necessary step to verify the proposed method capabilities to model wake induction on a lifting device. If the method performs adequately for the wing in translation problem, then it is expected to perform as well for the wing in rotation. Furthermore, the influence of scheme order and mesh discretization will be studied to provide useful information for the design of a numerical solution to the problem of the wind in rotation.

4.1. Description of the problem
24 forms of the tapered wing have been studied; They were elaborated from the combinations of 6 taper ratios \( c_t \) (0.1;0.2;0.4;0.6;0.8;1.0) and 4 Aspect ratios \( AR(4;6;8;10) \). The same span length \( b \) of 10m, inflow velocity of 50 m/s and 5 degrees pitch angle has been set for all 24 cases. \( AR \) is defined as the ratio of \( b^2 \) to the wing area. Aerodynamic characteristics of every wing section is identical and based of the S809 airfoil [15]. Only lift characteristics are considered and the flow viscosity is set null for all simulations of the tapered wing.

The numerical representation of the finite wing is a flat plane parallel to the incoming flow whose shape is given in Fig. 3. The domain of solution is a cube with sides of length 30\( b \); the flat plate representing the wing (see Fig. 3) is located 10\( b \) downstream of the inlet and at the center of \((y,z)\) planes. The mesh is refined in the vicinity of the wing tip and leading and trailing edges, and nodes are placed on the mid-chord of the wing to compute the induced angle \( \alpha_i \). Table 1 summarizes the mesh characteristic along with total RAM needed to analyze the problem.

|                         | Mesh M1 | Mesh M2 | Mesh M3 |
|-------------------------|---------|---------|---------|
| Number of nodes for domain discretization | 41X31X49 | 81X61X97 | 161X161X193 |
| Number of nodes for wind discretization  | 6X16    | 12X32   | 24X64   |
| RAM needed (approximately)     | 100 Mb  | 800Mb   | 6.4 Gb  |

Since all computations were performed on a single computer, in the case of Mesh M3, RAM needs were so high that it was necessary to recourse to hard disk as a substitute to RAM for temporary storage of variables, therefore increasing very much the time of computations.

The evaluation of the proposed method is realized by comparing the numerical values computed for the induced drag to the ideal values found in classic literature [16]. Lift of the numerical wing is calculated from the surface integral of pressure jumps across the surface. Induced drag is determined from momentum consideration:

\[
D_i = - \int \int_S \Delta ud\bar{m}
\]
In these equations, \( \dot{m} \) is the mass flux passing through an elementary surface of area \( dx \, dy \), which is evaluated from the assembly of mass fluxes across all control surfaces on one side of a control volume located on the surface of discontinuity.

4.2. Results

Figure 5 shows in (a) a view from downstream of streamlines concentrated on the tips of a wing of \( AR = 10 \) and taper ratio of 1, and in (b) the same view, but with light inclination of the \( x \) axis to present downstream development of tip vortices. Marks are disposed at 10b and 20b downstream, so that vortices movements become measurable. In exact, analytic solution to this problem \([3]\), the trailed vorticity system exhibits natural tendency to roll up in singular, hardly predictable, behavior. Using vortex methods to analyze the aerodynamics of a lifting device, it is necessary to recourse to a specific model for the trailed vorticity system usually by setting, in advance the shape of this system. This shape can be set fixed or moving accordingly to the flow calculated by the method. When set moving, singular behaviors are noted as well and the solution becomes less tractable. Figure 5 teaches that vorticity creation and convection is well modeled by the present method: \( \Delta u \) and \( \Delta v \) discontinuities result in creation of vorticity within the flow (with major component along the \( x \) axis) whose evolution is naturally taken into account by the Navier Stokes equations, and therefore by the proposed method. If no unstability happen here, this is partly due to numerical inaccuracy to predict exactly the Navier Stokes solution, and therefore its singularities.

Figure 6 presents the ratio of calculated over ideal values of \( F \), where \( F \) denotes the ratio of lift to drag of the wing \( F = L/D_i \), as a function of taper ratio \( \frac{c_t}{c_r} \). On each of these 4 figures, calculation results performed either with the original MAW scheme (referred to as first order) or the new, second order scheme, for meshes M1, M2 and M3 are displayed. Few are available for mesh M3 since only \( \frac{c_t}{c_r} = 1 \) for \( AR = 4 \) and \( AR = 10 \) cases have been studied. Rapid inspection of the results show that calculated induced drag is always overestimated, since all values of \( F \) are below 1. It is evident from comparisons between results that the capacity of the method to predict the ideal value of induced drag (for a given lift) decreases when either \( AR \) increases or
\( \alpha_i \) decreases. Furthermore, the influence of the discretization and convection scheme is notable and clearly show that improving them leads to better performance of the method. Finally, it shall be noted that the method performs well, within 10% accuracy for induced drag, only in the case where \( AR = 4 \) and meshes count over 1 million nodes. Its performance is otherwise quantitatively deceiving although it reproduces some qualitative phenomenon like the fact that, for a given lift and aspect ratio, induced drag is minimum for a taper ratio between 0.2 and 0.3 (derivable, but not immediately available from Fig. 6).

Most of the reasons behind the method average performance can be understood by inspection of the predicted flow crossing the lifting surface. Figure 7 reveals the calculated crossflow for a wing of \( AR = 4 \) and a taper ratio of 1 obtained using either the first or the second order convection scheme with mesh M2. Along every section of the wing surface, the flow component \( w \) is observed to decrease from positive (upwash) to negative (downwash) values, a result in accordance with analytical vortex analysis. Intense lateral variations of \( w \) happens in the tip area section where the influence of tip vortices induction on the flow is the strongest. This behavior, to be linked with the fact that lift should cancel at the tip, is numerically hard to deal with. Improvements brought by the second order convection scheme are notable to help model the tip area. As underlined by Tran et al. [13], the second order scheme is more efficient in modeling regions with rapid velocity variation. Reduction of false diffusion is also evident between the two schemes, since the axial decrease in \( w \) component is less intense using the first order convection scheme. Note that considering the crossflows depicted in Fig. 7, and Eq.(9), it can be interpreted that induced drag is less when the first order scheme is used since the mass fluxes are less important, a result obviously opposite to the depicted calculations of Fig. 6. This apparent contradiction is explained by the difference in lift force as calculated by the two convection schemes. With the first order scheme indeed, values of induced angle \( \alpha_i \) measured at the wing mid-chord are higher (in absolute value) and therefore lift coefficient \( C_L \) is lower. Improvements with mesh refinement is finally evident from all previous considerations since the rapid variations in flow parameters are better taken into account using a finer discretization.

Figure 8 illustrates the same characteristics of the solution process by displaying the induced flow angle in the plane defined by the wing model at two stations along the flow direction: mid-chord and 0.8\( c_r \) upstream of leading edge. At the mid-chord position, the induced flow angle is simply the opposite of the induced angle \( \alpha_i \). Results produced using the three meshes and two convection schemes are reported on the figure with same notation than Fig. 6. Results at mid-chord were also produced using the Prandtl lifting line theory with the published method of Anderson [17] and 200 points discretizing the wing span. Spurious oscillations of the Prandtl solution happen in the vicinity of the tips and are not shown by zooming outside these positions. The differences in induced flow angle predictions between convection schemes lead to significant difference at mid-chord, thus less important at the 80% upstream position. Furthermore, improvements in mesh refinement do not show to cancel or diminish these differences. These remarks lead to a preliminary conclusion that regardless of the convection scheme or mesh employed, using the upstream inflow angle (plus a suitable relation between this angle and the effective angle of attack of the airfoil \( \beta - \alpha_i \)) instead of the effective flow angle is a better choice to estimate the loading of the wing. Finally, it should be noted that an excellent agreement exists between results based on Prandtl lifting line theory and the results from the new method, with second order scheme and mesh M3 : both approaches are therefore equivalent in the treatment of the vortical wake of a finite wing.

In the wake of the work of Whale et al. [18], the present method has been used to calculate the relationship between the inflow angle at the 80% upstream station and the effective angle of attack measured at mid-chord. Calculations have been performed for pitch angle \( \beta \) of the wing from 2 up to 9 degrees, using mesh M2 and the second order convection scheme. Figure 9 presents this relationship for two different spanwise stations: at mid-span and at 25% span.
Figure 7. Crossflow as a function of convection scheme for $AR = 4$, taper ratio=1.

Figure 8. Induced flow angle at mid-chord and 80% $c_r$ upstream.

from the wing tip. Tip effects on the relationship between effective angle of attack and inflow angle are well reproduced by the present method, and the similarity with the relations found by Whale et al. [18] is striking, although the problem addressed by these authors is the one of phase III, CER turbine blades (fixed pitch and chord) in rotation. The present method could therefore be used to correct inflow measurements when measuring blade sectional aerodynamics with the same improvement in 3D effects handling than the method of Whale et al.

5. Wind turbine problem

The application of the proposed method to the case of the finite wing in rotation is still under development and meets numerous modelization problems. As an optimal computational effort, it has been chosen to model the rotor in the rotating system of coordinates where its blades appear fixed and inertia (centrifugal and Coriolis) forces occur. For the analysis of two-blade rotors, considering symmetry of the problem, the domain of solution is a half cylinder and periodic and aperiodic boundary conditions are set at the border of the mesh, depending on the flow variable solved. As shown in the middle detail of Fig. 10, the structured mesh is composed of radial lines extending from an inner radial position (set small compared to the blade dimensions) to an outer position far from the blade. Let $I,J$ and $K$ be the indices pointing a specific node of the structured mesh, then the previously mentioned radial lines correspond to $J = Cte$ lines, and providing that $u,v$ and $w$ are the flow components in the $x$, $y$ and $z$ axis shown in Fig. 10, then periodicity is expected for $u$, and aperiodicity for $v$ and $w$ components between surfaces $J = 1$ and $J = NJ$ as summarized by the formulas below the mesh picture. The necessary modifications in the solution algorithm have been implemented in the CVFEM based on the works of Sebben and Baliga [19].

The method was employed to study the NREL phase VI rotor [20] using following
effective angle of attack (deg.)

Results for the Finite Wing:
- Mid-Span
- 25% span from tip

Results of Whale et al.:
- Mid-Blade
- 30% span from tip

Figure 9. Inflow angle as a function of effective angle of attack.

simplifications:

- Uniform, rotationless inflow ($u = U_\infty$, $v = w = 0$).
- Blade loading is deduced from blade element theory (see for example [6]), adding the rotational velocity to the calculated flow to determine the relative incoming velocity at a blade section. Although viscous forces are not used in the determination of the flow, their contribution (significant in stall operation) is taken into account for the calculation of power output. Static measurements [15] of lift and drag of the S809 airfoil are used.
- A uniform viscosity of 10 Pa.s is taken. This level mimics atmospheric turbulence effect on flow entrainment, as well as it helps stabilize the solution process.

Figure 10. Mesh for the blade in rotation (only half part shown).

Figure 11 presents the power curve of the NREL phase VI rotor using two meshes of 80000 and 640000 nodes, with 11X16 and 22X32 nodes for blade discretization, respectively. To discern the 3D contribution brought by modeling the rotor with surfaces of discontinuity, results from the previously developed 2D model using the actuator disk concept [7] (without tip loss corrections) are also shown. It is first noted that discretization is a critical factor in the analysis because a reduced number of integration points is not accurate enough to evaluate the rapid varying loading of a wind turbine blade. Comparing results using the actuator disk concept with those produced from the new method developed, it is observed that power predictions are essentially the same, until 9 m/s. Between 9 m/s and 15 m/s, the new method exhibits higher levels of power, closer to experimental measurements. To understand the differences in power predictions between the two concepts, Fig. 12 presents the distribution of effective angle of attack along the blade for inflow velocities of 7.5 m/s and 12.5 m/s. At 7.5 m/s distributions obtained with the two models are almost identical, although predicted angles by the actuator disk concept are a bit higher. This light tendency is however significantly increased for the 12.5 m/s case where effective angles of attack are identical between methods only at the tip and root of the blades, but otherwise differ from up to 4 degrees. The differences between predicted angles are comparable to the induced flow angles of the tapered wing problem with magnitude and type of distribution of the same order. Reasons behind the differences observed between the two approaches are necessarily found in the difference in flow modeling, since all other parameters (blade geometry and aerodynamic characteristics) are identical. Since actuator disk analysis (without tip loss corrections) are notably unable to model the effects of a finite number of blades, it is thought
that the difference in predictions of the angle of attack can be linked with the wake induced flow effects, and constitute therefore a proof that the present method is able to model wake induction on the blades. The fact that for wind velocities lower than 9 m/s, predicted power outputs coincide with those of the actuator disk is related to the lower lift on blade sections, a fortiori to a weaker vortical system. Once stall is initiated on every blade section (at around 10 m/s) induced drag effects become important and provoke a shift up and to the right of the power curve produced using the actuator disk concept.

**Figure 11.** Power Curves for NREL Phase VI Wind Turbine.

**Figure 12.** Effective angle of attack along NREL Phase VI Rotor Blade.

### 6. Conclusion

Modeling the flow structure of lifting devices using vortex sheets is not new. Originality of the present method lies in the introduction of these types of singular surface (bearing velocity and pressure discontinuities), in a regular, control-volume based CFD code, with the purpose of modeling the wake induction on blade aerodynamics of a wind turbine. As a first step, the case of the tapered wing in uniform translation has been analyzed. Results of the method applied to the case of the taper wing have been qualitatively satisfying in term of induced angle of attack, but predictions of induced drag are generally overestimated for aspect ratios of the wing higher than 4 and taper ratios lower than 0.7. Improvement in accuracy has been obtained through implementation of a newly developed second order convection scheme (based on the MAW scheme) which is more appropriate for modeling rapid variations in flow velocities, as is the case around the taper wing. Regarding discretization of the domain solution, grid-independency has been studied and shows that 1000 nodes is a minimum number to discretize the surface carrying discontinuities. The proposed method has shown to be a good tool to evaluate the relationship between inflow angle measured at some location upstream, and local angle of attack.

The study of the rotating wing, i.e. the blade of a wind turbine has been initiated. The method has been used with simplifications to the case of the NREL Phase VI rotor. A comparison with results obtained by an actuator disk method (without tip loss correction), shows that in ranges where the blade sections exhibits high lift characteristics, lower angles of attack are predicted using the present method rather than the actuator disk. This intermediate result is thought as a first proof that the proposed method is able to model the influence of the wind turbine vortical structure on the flow. Future work will hopefully present numerous insights into the method application to the case of wind turbine rotors.
Acknowledgments

This study received support from the Canada Research Chairs Program. Support from the Natural Sciences and Engineering Research Council of Canada (NSERC) in the form of research grants is gratefully acknowledged.

References

[1] Schreck S, Hand M, Fingersh L, Robinson M and Simms D 2001 Proceedings of the European Wind Energy Conference (Copenhagen, Denmark) pp 35-47
[2] Leishman J G 2002 AIAA Paper 2002-0037
[3] Van Kuik, G A M 2003 AIAA Paper 2003-0356
[4] Shen W Z, Mikkelsen R and Sorensen J N 2005 Wind Energy 8 457-475
[5] Massou, F, Dobrev I and Rapin, M 2006 AIAA paper 2006-0782
[6] Leclerc C and Masson C 2004 Proceedings of the Conference The Science of Making Torque from Wind (Delft, Netherlands)
[7] Leclerc C and Masson C 2004 Proceedings of the ASME Wind Energy Symposium (Reno, NV) 394-402
[8] Rajagopalan R G and Fanucci J B 1985 Journal of Propulsion and Power 1 432-36
[9] Masson C, Ammara I and Paraschivoiu I 1997 International Journal of Rotating Machinery 321-32
[10] Sorensen J N and Myken A 1992 Journal of Wind Engineering and Industrial Aerodynamics 39 139-49
[11] Hansen A C and Butterfield C P 1993 Annual Review of Fluid Mechanics 25 115-49
[12] Saabas, H J 1991 Control Volume Finite Element Method for Three-Dimensional, Incompressible, Viscous Fluid Flow (Montreal, Canada: McGill University)
[13] Tran L D, Masson C and Small A 2006 International Journal for Numerical Methods in Fluids 51 749-771
[14] Leclerc C and Masson C 2005 ASME Journal of Solar Energy Engineering 127 200-8
[15] Butterfield C P, Musial W P, Scott G N and Simms, D A 1992 NREL combined Experiment Final Report Phase II, NREL/TP-442-4802, (Golden, CO: NREL)
[16] Mc Cormick B W 1979 Aerodynamics, Aeronautics and Flight Mechanics (New York: John Wiley & Sons)
[17] Anderson J D 1991 Fundamentals of Aerodynamics (Mc Graw Hill)
[18] Whale J, Fisichella C J and Selig M S 2000 Journal of Solar Energy Engineering 122 196-202
[19] Sediben, S and Baliga B R 1996 Porous Flow Heat Transfer 331 81-93
[20] Hand M M , Simms D A, Fingersh L J , Jager D W , Cotrell J R, Schreck S and Larwood S M 2001 NREL Report TP-500-29955