International Journal of Rotating Machinery
1997, Vol. 3, No. 1, pp. 21–32
Reprints available directly from the publisher
Photocopying permitted by license only

An Aerodynamic Method for the Analysis of Isolated Horizontal-Axis Wind Turbines

CHRISTIAN MASSON*, IDRISS AMMARA and ION PARASCHIVOIU

Bombardier Aeronautical Chair, École Polytechnique, Montréal, CANADA, H3C 3A7

(Received 4 April 1996)

The aerodynamic analysis of a wind turbine represents a very complex task since it involves an unsteady three-dimensional viscous flow. In most existing performance-analysis methods, wind turbines are considered isolated so that interference effects caused by other rotors or by the site topology are neglected. Studying these effects in order to optimize the arrangement and the positioning of Horizontal-Axis Wind Turbines (HAWTs) on a wind farm is one of the research activities of the Bombardier Aeronautical Chair. As a preliminary step in the progress of this project, a method that includes some of the essential ingredients for the analysis of wind farms has been developed and is presented in the paper. In this proposed method, the flow field around isolated HAWTs is predicted by solving the steady-state, incompressible, two-dimensional axisymmetric Navier-Stokes equations. The turbine is represented by a distribution of momentum sources. The resulting governing equations are solved using a Control-Volume Finite Element Method (CVFEM). This axisymmetric implementation efficiently illustrates the applicability and viability of the proposed methodology, by using a formulation that necessitates a minimum of computer resources. The axisymmetric method produces performance predictions for isolated machines with the same level of accuracy than the well-known momentum-strip theory. It can therefore be considered to be a useful tool for the design of HAWTs. Its main advantage, however, is its capacity to predict the flow in the wake which constitutes one of the essential features needed for the performance predictions of wind farms of dense cluster arrangements.

Keywords: Wind Farm, Wind Power, HAWT, Numerical Method, Navier-Stokes Equations

1. INTRODUCTION

After some unsuccessful attempts at constructing and operating very-large-scale isolated wind turbines, the recent tendency is to construct wind farms of medium-size machines (≈500 kW). The strategy currently used during the conception of such wind farms consists in installing the turbines far from each other in order to minimize the interference effects. This practice results in very sparse wind farms where the wind energy potential of a site is inefficiently used. It is justified by the performance losses associated to the

*Corresponding author. Tel.: (514) 340–4582. Fax: (514) 340–5917. E-mail: christian. masson@meca.polymtl.ca.

wake effects, which are significant in dense arrangements. However, a relatively dense but staggered arrangement of the turbines is expected to produce an increase in the performance of the downstream turbines with respect to the isolated-turbine situation. This is due to the beneficial venturi effects that occur between two adjacent turbine wakes. The efficiency of the dense staggered cluster is not expected to be significantly higher than that of the sparse arrangement. However, for a given number of turbines, a dense staggered cluster can be easily conceived that would occupy up to 25% less land than the sparse arrangement. Notwithstanding the farm efficiency increase induced by the venturi effects, a reduction of 25% of the wind farm area can represent a significant economy in operating expenses. Furthermore, some of the construction expenses, such as the grading and electric infrastructure costs, will be reduced.

Various aerodynamic methods, appropriate for the conception of isolated turbines, are available to designers (Gohard [1978], Paraschivoiu [1981], Strickland et al. [1980], Templin [1974], Wilson [1984]). However, efficient and accurate methods for the analysis of a dense cluster where the effects of the three-dimensional turbulent turbine wakes are included are not available. The development of a method that includes the essential ingredients for the successful performance predictions of wind turbines in a dense arrangement is the authors' main objective. In the proposed method, the flow field of the wind farm is predicted by solving the steady-state, incompressible, three-dimensional Navier-Stokes equations. The turbines are represented by distributions of momentum sources, a technique introduced by Rajagopalan [1984] and Rajagopalan and Fanucci [1985]. This is a general formulation which can be applied, in principle, to horizontal-axis and vertical-axis wind turbines and can include the effects of hubs, towers, and local topography. The Navier-Stokes equations are solved using the three-dimensional Control-Volume Finite Element Method (CVFEM) of Saabas and Baliga [1994].

The development of this method is still in its early stage, and the paper is aimed at presenting the progress and at demonstrating the applicability of the proposed method along with its capacity to analyze the performances of wind farms. The mathematical model and numerical method described in the paper are a two-dimensional axisymmetric formulation applicable to isolated HAWTs. This implementation is used in the paper to demonstrate the applicability and viability of the proposed methodology at much lower CPU costs than a fully three-dimensional implementation. Several aspects related to the numerical solution of the mathematical model, such as (*i*) the appropriate extent of the computational domain, (*ii*) the grid spacing needed to obtain accurate performance predictions, and (*iii*) the optimum choice of the various control parameters to obtain converged solutions, can also be studied more efficiently using the axisymmetric formulation. Comparisons between the performance predictions obtained with the proposed formulation and those of the momentum-strip theory are presented to illustrate the accuracy of the proposed methodology. Comparisons with experimental data are also included.

2. GOVERNING EQUATIONS AND ROTOR REPRESENTATION

The flow around an isolated HAWT is governed by the unsteady three-dimensional Navier-Stokes equations which have to be solved in a domain with moving boundaries. Analytical solutions of such a problem are hardly possible while numerical solutions represent a formidable task on today's computers. However, a more tractable model producing meaningful results can be obtained by time-averaging the governing equations and by representing the turbine with momentum sources.

The derivation of the governing equations and momentum sources is based on time-averaging techniques and the blade-element theory. The interested reader is referred to the works of Rajagopalan [1984] and Rajagopalan and Fanucci [1985] for a detailed derivation of the source terms of vertical-axis wind turbines. In the case of horizontal-axis wind turbines, of interest here, the derivation is very similar and the reader is referred to Fig. 1 for the definition of the various parameters involved in the evaluation of the momentum sources.

In steady-state, laminar, two-dimensional axisymmetric flow around an isolated HAWT, the mathematical model consists of a set of four differential equations: a continuity equation and three momentum equations. The four dependent variables are $\vec{V}$ (having the components u, v, and w in the x, r, and θ directions) and p. The fluid density is represented by ρ, and its dynamic viscosity by μ. The turbine is composed of B blades having a coning angle γ and a chord c that can vary along the blade. The turbine rotational speed is Ω.

The flow around an HAWT can be represented by the following general formulation:

$$\nabla\cdot(\rho\vec{V}\phi) = \nabla\cdot\Gamma\nabla\phi + (S_p)_\phi + (S_T)_\phi \quad (1)$$

The appropriate governing equations can be obtained from Eq. (1) by defining the dependent variable, ϕ, the diffusion coefficient, Γ, and the volumetric source terms, $(S_p)_\phi$ and $(S_T)_\phi$, according to Table I, where

$$(S_T)_u = K\,(UC_D + WC_L)\cos\gamma \quad (2)$$

$$(S_T)_v = -K\,(UC_D + WC_L)\sin\gamma \quad (3)$$

$$(S_T)_w = K\,(UC_L - WC_D) \quad (4)$$

$$K = -\rho\frac{BV_{rel}c\Delta\ell}{2v_{cv}} \quad (5)$$

$$V_{rel} = \sqrt{U^2 + W^2} \quad (6)$$

$$U = u\cos\gamma - v\sin\gamma \quad (7)$$

$$W = r\Omega - w \quad (8)$$

The source terms $(S_T)_u$, $(S_T)_v$, $(S_T)_w$ are the x-, r-, and θ-component of the mutual time-averaged force, exerted by the fluid and the blades on one another, per unit volume of the fluid. They will be referred to as the momentum-source terms. C_L and C_D are the lift and drag coefficients of the blade-defining airfoil and are, in general, functions of the angle of attack α and the local Reynolds number Re_c. In the proposed model, these coefficients can be taken from either experimental data or numerical results obtained over the appropriate two-dimensional airfoil.

TABLE I Specific Forms of the General Equation

	ϕ	Γ	$(S_p)_\phi$	$(S_T)_\phi$
continuity equation	1	0	0	0
x-momentum equation	u	μ	$-\partial p / \partial x$	$(S_T)_u$
r-momentum equation	v	μ	$-\partial p / \partial r$	$(S_T)_v$
θ-momentum equation	w	μ	0	$(S_T)_w$

In the above equations, v_{cv} is the control volume to which the conservation principles have been applied, and $\Delta\ell$ corresponds to the blades length that crosses this control volume. It is to be noted that $\Delta\ell$ is zero in the region of the flow which is not crossed by the blades. Consequently, the momentum-source terms are non-zero only in the region of the flow crossed by the turbine blades.

The mathematical model described in this section has been obtained by considering laminar flow. This assumption is difficult to justify physically, since it is well known that the wake of a wind turbine is turbulent. Nevertheless, the proposed mathematical model still includes some of the essential elements for accurate performance predictions of wind farms. The major features of the proposed model are (*i*) its capacity to model the details of the wake behind the turbines and (*ii*) its implicit introduction of the interferences between the rotors. The interferences are due to numerous effects such as the pressure variation and the blockage phenomenon.

In this early stage of development, the laminar-flow assumption is used in order to emphasize the various aspects related to the wind turbine modelling without introducing uncertainties inherent to the turbulence models. It is demonstrated in the RESULTS Section that accurate performance predictions can be obtained using the laminar-flow assumption in the case of isolated HAWTs since their performances are not highly influenced by the details of their wakes. However, when a turbine is located behind another, the turbulent wake has to be considered in order to

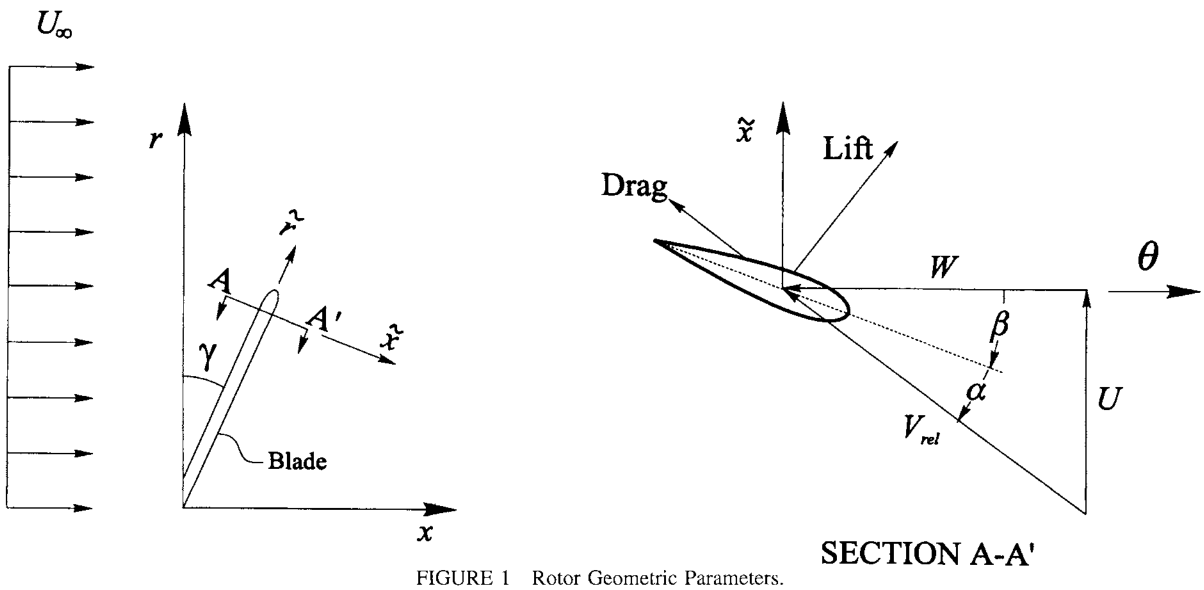

FIGURE 1 Rotor Geometric Parameters.

obtain accurate performance predictions of the downwind turbine.

3. NUMERICAL METHOD

The proposed numerical method is a CVFEM based on the primitive-variables, co-located, equal-order formulation of Masson et al. [1994]. Detailed descriptions of this CVFEM, pertaining to the simulation of particulate two-phase flows and internal three-dimensional turbulent flows, are available in the works of Masson [1993] and Saabas [1991]. Therefore, for sake of conciseness, only the aspects relevant to the successful simulation of the flow around a wind turbine are presented in this section.

3.1 Interpolation of ϕ in Convective Term

In the derivation of algebraic approximations to surface integrals of the convective fluxes, two different interpolation schemes for ϕ were presented by Saabas [1991]: the FLow Oriented upwind scheme (FLO); and a MAss Weighted upwind scheme (MAW).

The FLO scheme is based on the earlier work of Baliga and Patankar [1980,1988]. The interpolation function used in this scheme responds appropriately to an element-based Peclet number and to the direction of the element-averaged velocity vector. In planar two-dimensional problems that involve acute-angled triangular elements and relatively low element Peclet numbers, Saabas and Baliga [1994] have proven that the FLO scheme can be quite successful. If high values of the element Peclet number are encountered, however, the FLO scheme can lead to negative coefficients in the algebraic discretised equations (Saabas and Baliga [1994]) and this difficulty is compounded when obtuse-angled triangular elements are used. The donor-cell scheme of Prakash [1987] is one way of ensuring positive coefficients: in this approach, the value of a scalar convected out of a control volume, across its surface, is set equal to the value of the scalar at the node within the control volume. This approach guarantees positive coefficients, but takes little account of the influence of the direction of the flow. Thus it is prone to considerable false diffusion (Prakash [1987]).

The MAW scheme is based on the positive-coefficient schemes of Schneider and Raw [1986]. It ensures, at the element level, that the extent to which the dependent variable at a node exterior to a control volume contributes to the convective outflow is less than or equal to its contribution to the inflow by convection. Thus, it is a sufficient condition to ensure that the algebraic approximations to the convective terms add positively to the discretised equation. It should be noted that the MAW scheme takes better account of the influence of the direction of the flow than the donor-cell scheme of Prakash [1987], so it is less prone to false diffusion. Details of the formulation of the MAW scheme are presented in the work of Saabas [1991].

In problems with acute-angled triangular elements and relatively low element Peclet numbers, the FLO scheme is more accurate than the MAW scheme. As was mentioned earlier, however, when high element Peclet numbers are involved, especially in conjunction with obtuse-angled elements, the FLO scheme produces negative coefficients in the discretised equations. Negative coefficients in the discretized equations can lead to convergence difficulties when iterative solution algorithms, such as SIMPLE or its variants (Patankar [1980]) and CELS (Galpin et al. [1985]), that use segregated or coupled equation line-by-line iterative algorithms to solve the linearized sets of discretised equations, are used.

In the numerical solution of the flow around a wind turbine, a fine grid has to be used in the immediate vicinity of the rotor in order to capture the large variations of the flow properties. The grid size is then increased as the distance to the turbine increases. This coarsening is applied in order to keep a reasonable number of grid points while ensuring the application of the boundary conditions to be far enough from the rotor. The use of such coarse grids results in large Peclet numbers since these are directly proportional to the grid size. Convergence difficulties were encountered during the simulations presented in this paper when the FLO scheme was used. Therefore, the MAW scheme has been used to produce all the results presented in this paper and its application is recommended for the successful solution of the flow around a wind turbine.

3.2 Mass Flow Rate Interpolation

The mass flow rate is calculated using a special treatment borrowed from the works of Prakash and Patankar [1985] and Saabas and Baliga [1994]. This special treatment consists in expressing the mass-flow related velocities (u^m, v^m), as a function of a pseudo-velocity and a pressure gradient term:

$$u^m = \hat{u} + d^u\left(-\frac{\partial p}{\partial x}\right)_{\text{ele}} \quad v^m = \hat{v} + d^v\left(-\frac{\partial p}{\partial r}\right)_{\text{ele}} \tag{9}$$

where $\hat{u}$, and $\hat{v}$ are the pseudo-velocities, and d^u, and d^v are called the pressure-gradient coefficients. These expressions come directly from the discretised momentum equations. However, instead of using the control-volume-averaged pressure gradient, the elemental pressure gradient is used to compute the mass-flow related velocity in each element. This prevents the occurrence of spurious pressure oscillations in the proposed CVFEM.

3.3 Momentum-Source Term Linearization

The momentum-source term, $(S_T)_\phi$, is expressed in the following general form (Patankar [1980]):

$$(S_T)_\phi = (S_T)_C + (S_T)_P\phi \tag{10}$$

In each triangular element, the values of $(S_T)_C$ and $(S_T)_P$ are stored at the vertices, and are assumed to prevail over the corresponding portions of the control volumes within that element. While the volume integration of the momentum-source term is straightforward, its proper linearization is crucial to ensure convergence of the overall algorithm, especially in the context of the segregated iterative solution algorithm used in this work. The linearization consists in the specification of appropriate expressions for $(S_T)_C$ and $(S_T)_P$.

The momentum-source term can be linearized explicitly in each iteration:

$$(S_T)_C = (S_T)_\phi^* \qquad (S_T)_P = 0 \tag{11}$$

where the superscript * means that the source term has been evaluated using the flow properties obtained at the previous iteration. Implementation of this linearization in the segregated iterative algorithm used in this work resulted in severe convergence problems for highly loaded wind turbines. In such conditions, the value of the momentum-source terms become large, and the resulting momentum equations are source-dominated. This results in very slow convergence rates, and, sometimes the segregated iterative algorithm even diverges. In an effort to improve the robustness of the iterative solution algorithm, the following treatment is proposed:

$$(S_T)_C = 0 \qquad (S_T)_P = \frac{(S_T)^*_\phi}{\phi^*} \tag{12}$$

This linearization has proven to be much less prone to convergence problems than the explicit linearization expressed by Eq. (11).

3.4 Boundary Conditions

The computational domain consists of a simple cylinder that includes the wind turbine. Boundary conditions have to be prescribed on the three faces of this cylindrical domain.

Inlet Boundary: The inlet boundary is a r-θ plane located upstream of the wind turbine. In this plane, the three velocity components are given by the known freestream wind speed while the pressure is calculated from the discretised continuity equations.

Outlet Boundary: The outlet boundary is a r-θ plane located downstream of the wind turbine. In this plane, the pressure is assumed to be uniform and given while the three velocity components are computed from the discretised momentum equations obtained using the outflow treatment of Patankar [1980].

Top Boundary: This is a curved surface located at a radial distance far from the wind turbine blade tip. In this plane, the velocity is set to its freestream value. Pressure is calculated from the discretised continuity equations.

3.5 Overall Solution Algorithm

The discretised equations form a set of coupled nonlinear algebraic equations. In this work, the iterative variable adjustment procedure proposed by Saabas and Baliga [1994] is used to solve the proposed mathematical model. This procedure is akin to SIMPLER (Patankar [1980]) without the pressure correction equation. In order to facilitate implementation and testing of the proposed method, structured grids are used. Thus, a line Gauss-Seidel algorithm based on the tridiagonal matrix algorithm are used to solve the discretised equations for p, u, v, and w.

The momentum-source linearization proposed in this work can lead to large differences between the values of the two pressure-gradient coefficients. In the case of an horizontal-axis wind turbine completely contained within a r-θ plane, for example, d^u is typically much smaller than d^v. This large difference in the values of the pressure-gradient coefficients leads to difficulties in ensuring the overall mass conservation since the streamwise pressure gradient has a negligible effect in the discretised pressure equations. This difficulty is alleviated by prescribing a relatively large number of iterations in the line Gauss-Seidel algorithm used for the solution of the discretised pressure equations.

4. RESULTS

The results presented in this section are aimed at demonstrating the capacity of the proposed methodology to accurately predict the performances of isolated HAWTs, and to analyse wind farms. To this effect, the details of the computed flow field in the vicinity of an isolated HAWT are shown, and comparisons between the predictions of the proposed method, those of the well-known momentum-strip theory, and experimental data when available are presented. Furthermore, the following aspects related to the successful numerical solution of the mathematical model are studied: (i) the determination of the minimum extent of the various boundaries necessary to obtain a computational-domain independent solution, and (*ii*) the sensitivity of the performance predictions with respect to the grid size.

The simulations presented in this section have been realized for two HAWTs: (*i*) the NASA/DOE Mod-0 100-kW Experimental HAWT (Puthoff and Sirocky [1974]) operating at a rotational speed of 40 rpm and a blade pitch angle of 3°, and (*ii*) the INTA Experimental rotor (Hernandez and Crespo [1987]) operating at a rotational speed of 1500 rpm and a blade pitch angle of 9.5°.

4.1 Extent of the Computational Domain

A detailed study of the behaviour of power predictions with respect to the size of the computational domain has been undertaken in order to determine the minimum extent of the computational domain needed to produce relevant performance predictions for HAWTs. In the case of the simulation of the axisymmetric/swirling model, the size of the computational domain is characterized by three length parameters, namely Δx_{UP}, Δx_{DN}, and R_{CD}. These length parameters are presented in Fig. 2 which illustrates the grid topology. It is to be noted that the grids used in the axisymmetric/swirling simulations were much finer than the one shown in Fig. 2.

Figs. 3–5 show the results of the behaviour of the performance prediction with respect to the extent of the computational domain. These graphs present the variation of the difference between the performance prediction at a finite value of one of the length parameters (denoted by either $P(\Delta x_{\mathrm{UP}})$, $P(\Delta x_{\mathrm{DN}})$, $P(R_{\mathrm{CD}})$) and the power predicted for a very large value of the corresponding length parameter (denoted by $P(\infty)$), as a function of a specific length parameter. The length parameters have been nondimensionalized with respect to the rotor diameter D. The grids used were of uniform type with equal grid spacing in the r and x directions. This analysis has been undertaken at the most critical tip speed ratio *TSR* which is believed to be the *TSR* corresponding to the maximum power coefficient C_P. The power coefficient is defined by

$$C_P = \frac{P}{\frac{1}{2}\rho_\infty U_\infty^3 R^2} \tag{13}$$

where P is the mechanical power, U_∞ is the freestream wind velocity, and R is the rotor radius. The operational condition at which the maximum power coefficient occurs is the most critical situation since it corresponds to the regime where a greater portion of the energy available in the flow is ex-

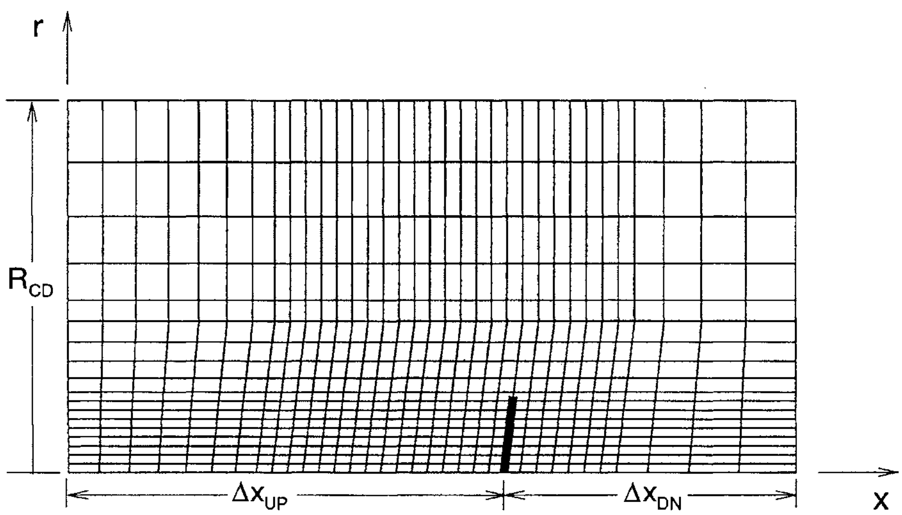

FIGURE 2 Computational Domain and Grid Topology.

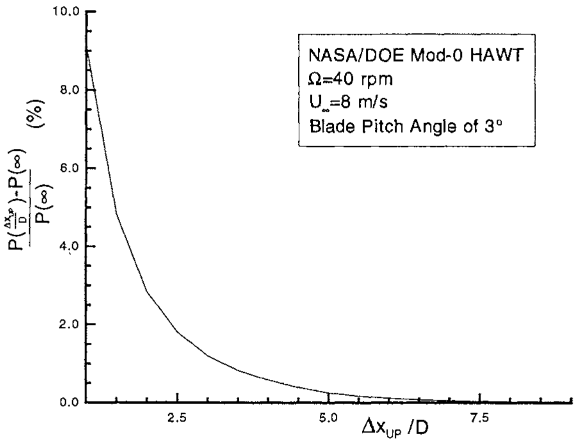

FIGURE 3 Variation of the Performance Prediction with Respect to Δx_{UP}.

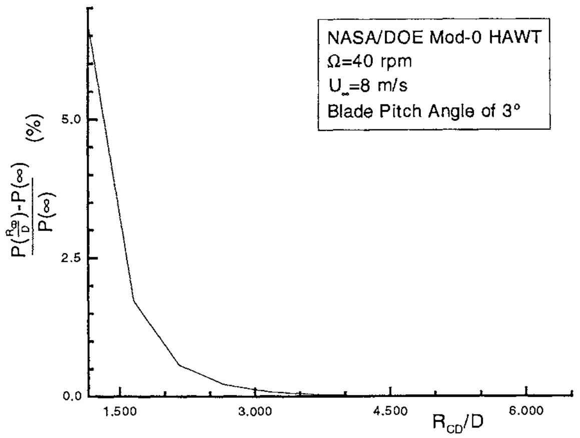

FIGURE 5 Variation of the Performance Prediction with Respect to R_{CD}.

tracted by the rotor. For a blade pitch angle of 3° and a rotational speed of 40 rpm, the freestream wind speed at which the maximum C_P is reached for the NASA/DOE Mod-0 rotor is near 8 m/s. This study has revealed that the power predictions of an isolated rotor are significantly influenced by the length of the computational domain upstream of the rotor (i.e. Δx_{UP}) and the position of the constant-radius boundary (i.e. R_{CD}). Based on the results presented in Figs. 3 and 5, $\Delta x_{\mathrm{UP}}/D = 7.5$ and $R_{\mathrm{CD}}/D = 4.0$ seems to be large enough to produce performance predictions independent of the extent of the domain. For the downstream extent of the computational domain, a much smaller value can be used. Fig. 4 suggests that $\Delta x_{\mathrm{DN}}/D = 4.5$ is more than sufficient. These values of the extent of the computational domain have been used to produce the results presented in the remainder of this section.

4.2 Grid Dependence Study

Fig. 6 presents the difference between the performance prediction obtained at a given number of grid points N, $P(N)$, and the grid-independent power prediction, $P(\infty)$. This grid dependence study has been

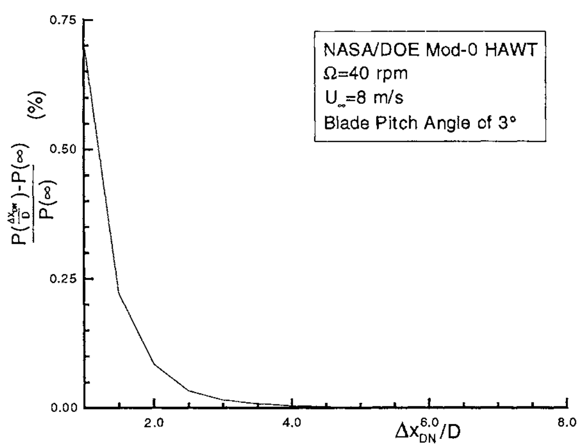

FIGURE 4 Variation of the Performance Prediction with Respect to Δx_{DN}.

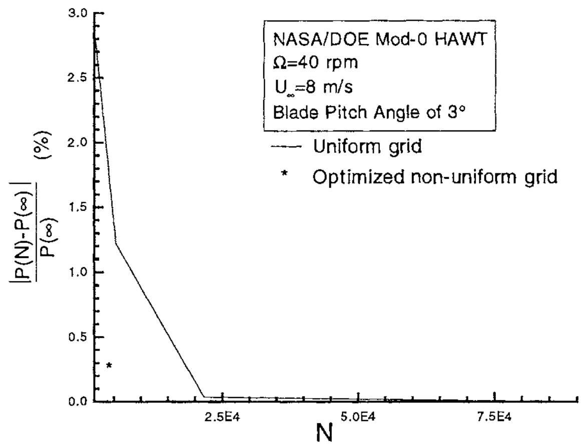

FIGURE 6 Variation of the Performance Prediction with Respect to N.

realized on uniform grids with equal grid spacing in the r and x directions in order to facilitate the determination of the grid-independent power prediction. Fig. 6 shows that the grid-independent solution is reached near N = 25 000. This very large number of grid points needed to obtain the grid-independent power prediction is due to the use of uniform grids. However, for practical calculations, nonuniform grids should be used in order to minimize the number of grid points needed. Using a uniform and fine grid in the vicinity of the rotor along with an expanding grid in the rest of the computational domain, it has been possible to reduce the number of points needed to obtain a solution close to the grid-independent one to 3 000 (see Fig. 6).

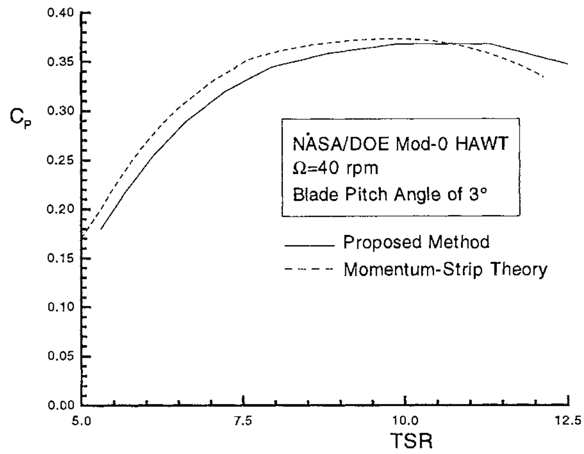

FIGURE 8 Power Coefficient Predictions for the NASA/DOE Mod-0 HAWT.

4.3 Comparisons with the Momentum Strip Theory and Experimental Data

Figs. 7 and 8 show comparisons between the performance predictions of the NASA/DOE Mod-0 100-kW Experimental HAWT produced by the proposed methodology and the results of the momentum-strip theory. The agreement between the two methods is good. This was to be expected since the factor limiting the accuracy of the results is the use of static two-dimensional lift and drag coefficients, on which both methods are based. As stated before, the main motivation in developing the proposed method resides in its inherent modelling of the rotor/rotor, rotor/ground, and rotor/tower interactions and its capacity to produce the details of the flow field around the turbines.

Figs. 9 and 10 present similar comparisons for the case of the INTA Experimental HAWT. In this case, the agreement is very good between the two methods. Fig. 10 also shows some experimental data. The performance predictions produced by the two methods are in relatively good agreement with the measured performances.

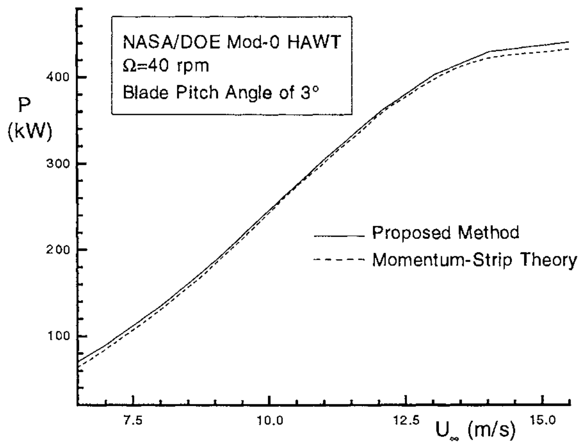

FIGURE 7 Power Predictions for the NASA/DOE Mod-0 HAWT.

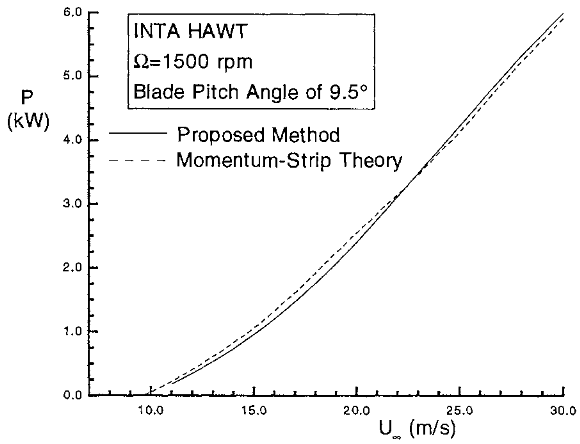

FIGURE 9 Power Predictions for the INTA HAWT.

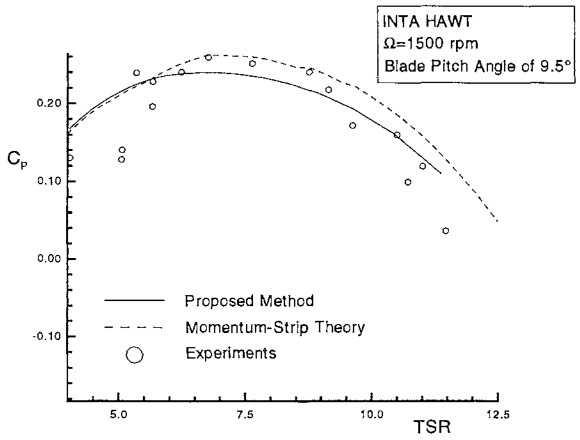

FIGURE 10 Power Coefficient Predictions for the INTA HAWT.

4.4 Predicted Flow Field

Using the nonuniform 3 000-point grid, the flow field around the NASA/DOE Mod-0 HAWT has been computed. The resulting pressure and velocity fields in the vicinity of the rotor are presented in Figs. 11 and 12. These results illustrate the capacity of the proposed methodology to produce the details of the flow behind a wind turbine, which constitutes one of the essential features for the successful analysis of wind farms.

5. CONCLUSION

The idea of using the venturi effects in wind farms is a new concept. More fundamental research is required to determine its applicability. For instance, the sensitivity of the cluster efficiency to the wind orientation is a crucial aspect related to the viability of this concept. The development of a method appropriate for the analysis of dense cluster arrangements corresponds to the first step in this feasibility study. Such a method can allow the determination of the optimum cluster arrangement, which is expected to be highly related to a specific site (wind speed and orientation variations, topography, etc.). Furthermore, the digital version of such a method represents a powerful tool for the designers of wind farm projects.

The method presented in this paper includes some of the important ingredients needed for the successful analysis of dense cluster arrangements. The implementation of this method has been realized under the assumption of axisymmetric swirling flow. This implementation has allowed to efficiently illustrate the applicability and viability of the proposed methodology by using a formulation that necessitates a minimum of computer resources. Additional features have to be included in this method before its application to

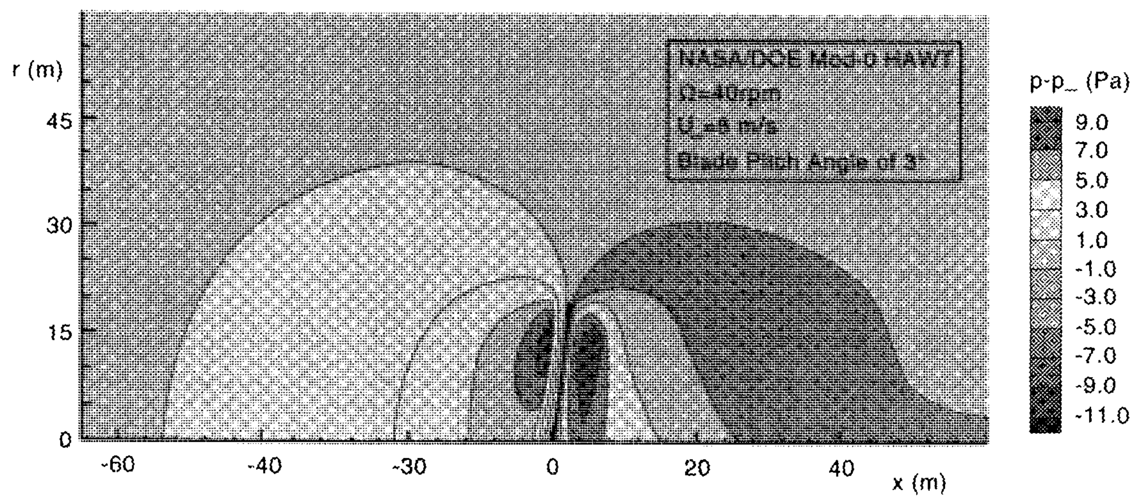

FIGURE 11 Computed Pressure Field for the NASA/DOE Mod-0 HAWT.

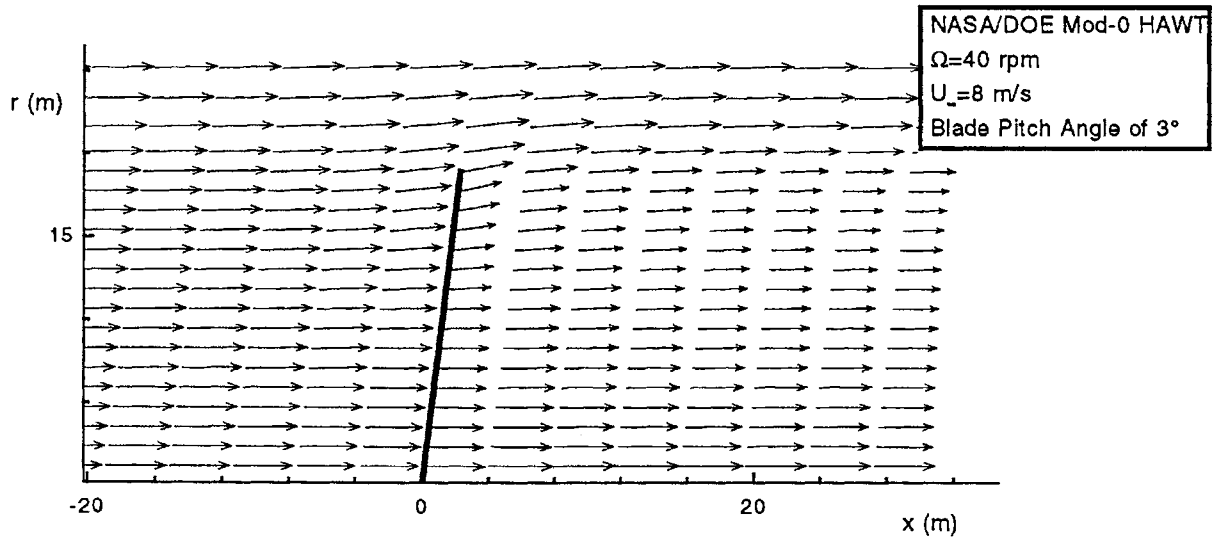

FIGURE 12 Computed Velocity Field for the NASA/DOE Mod-0 HAWT.

the performance predictions of wind farms. The fully three-dimensional formulation has to be implemented. Furthermore, an appropriate turbulence model should be selected and implemented in order to obtain accurate performance predictions for turbines located behind others. Nevertheless, the axisymmetric formulation produces performance predictions for isolated HAWTs with the same level of accuracy than the well-known momentum-strip theory. It can be considered to be a useful tool for the design of horizontal-axis wind turbines.

Acknowledgement

This work is supported by the "Ministère de l'Énergie et des Ressources du Québec" through the "Programme d'aide au développement des technologies de l'énergie". The authors would like to express their gratitude to R. F. Legault and J. E. Wade, from Kenetech Windpower, for the numerous helpful discussions they have had with them. The results of the momentum-strip theory have been obtained using the computer code developed by Tayeb Brahimi and Dan Chocron, of the Bombardier Aeronautical Chair.

References

Baliga, B.R., Patankar, S.V., 1980. A New Finite-Element Formulation for Convection-Diffusion Problems, *Numerical Heat Transfer*, Vol. 3, pp. 393–409.

Baliga, B.R., Patankar, S.V., 1988. Elliptic Systems: Finite Element Method II, in Minkowycz, W.J. et al. (Eds.), *Handbook of Numerical Heat Transfer*, Wiley, New York, Chap. 11, pp. 421–461.

Galpin, P.F., Van Doormaal, J.P., Raithby, G.D., 1985. Solution of the Incompressible Mass and Momentum Equations by Application of a Coupled Equation Line Solver, *Int. J. for Num. Meth. in Fluids*, Vol. 5, pp. 615–625.

Gohard, J.D., 1978. *Free Wake Analysis of Wind Turbine Aerodynamics*, TR 184-14, Massachusetts Institute of Technology, Cambridge.

Hernandez, J., Crespo, A., 1987. Aerodynamic Calculation of the Performance of Horizontal Axis Wind Turbines and Comparison with Experimental Results, *Wind Engineering*, Vol. 11, pp. 177–187.

Masson, C., 1993. *Numerical Prediction of Gas-Solid Particle Flows over a Wide Range of Concentration in Irregular Geometries*, Ph.D. Thesis, Dept. of Mech. Eng., McGill University, Montreal, Canada.

Masson, C., Saabas, H.J. and Baliga, B.R., 1994. Co-Located Equal-Order Control-Volume Finite-Element Method for Two-Dimensional Axisymmetric Incompressible Fluid Flow, *International Journal for Numerical Methods in Fluids*, Vol. 18, pp. 1–26.

Paraschivoiu, I., 1981. Double-Multiple Streamtube Model for Darrieus Wind Turbines, *Second DOE/NASA Wind Turbines Dynamics Workshop*, NASA CP-2185, pp. 19–25.

Patankar, S.V., 1980. *Numerical Heat Transfer and Fluid Flow*, McGraw-Hill, New-York.

Prakash, C., Patankar, S.V., 1985 A Control Volume-Based Finite-Element Method for Solving the Navier-Stokes Equations Using Equal-Order Velocity-Pressure Interpolation, *Numerical Heat Transfer*, Vol. 8, pp. 259–280.

Prakash, C., 1987. Examination of the Upwind (Donor-Cell) Formulation in Control Volume Finite-Element Methods for Fluid Flow and Heat Transfer, *Numerical Heat Transfer*, Vol. 11, pp. 401–416.

Puthoff, R.L., Sirocky, P.J., 1974 *Preliminary Design of a 100-kW Wind Turbine Generator*, NASA TMX-71585.

Rajagopalan, R.G., 1984. *Inviscid Upwind Finite Difference Model for Two-Dimensional Vertical Axis Wind Turbines*, Ph.D. Thesis, West Virginia University, Morgantown.

Rajagopalan, R.G., Fanucci, J.B., 1985. Finite Difference Model for the Vertical Axis Wind Turbines, *Journal of Propulsion and Power*, Vol. 1, pp. 432–436.

Saabas, H.J., 1991. *A Control Volume Finite Element Method for Three-Dimensional, Incompressible, Viscous Fluid Flow*, Ph.D. Thesis, Dept. of Mech. Eng., McGill University, Montreal, Canada.

Saabas, H.J. and Baliga, B.R., 1994. Co-Located Equal-Order Control-Volume Finite-Element Method for Multidimensional, Incompressible, Fluid Flow—Part I, *Numerical Heat Transfer*, Vol. 26B, pp. 381–407.

Schneider, G.E., Raw, M.J., 1986. A Skewed Positive Influence Coefficient Upwinding Procedure for Control Volume Based Finite Element Convection Diffusion Computation, *Numerical Heat Transfer*, Vol. 9, pp. 1–26.

Strickland, J.H., Webster, B.T., and Nguyen, T., 1980. *A Vortex Model of the Darrieus Turbine: An Analytical and Experimental Study*, SAND79-7058, Sandia National Laboratories, Albuquerque.

Templin, R.S., 1974. *Aerodynamic Performance Theory for the NRC Vertical-Axis Wind Turbine*, LTR-160, National Research Council of Canada, Ottawa.

Wilson, R.E., Walker, S.N., 1984. *Performance Analysis of Horizontal Axis Wind Turbines*, Oregon State University.