Copyright IPCO-2014
Vol.2
ISSN : 2356-5608

Conférence Internationale des Energies Renouvelables (CIER'13)
15-17 Décembre 2013 - Sousse, Tunisie

A CFD Method for Aerodynamic Analysis of HAWT Rotors

Arezki Smaïli[#1], Tarek Fengal[#], Christian Masson[*2]

[#]*Laboratoire de Génie Mécanique et Développement, Ecole Nationale Polytechnique*
10 avenue Hassen Badi, P.B. 182 El-Harrach, Algiers Algeria
[1]arezki.smaili@enp.edu.dz

[*]*Département de Génie Mécanique, Ecole de Technologie Supérieure*
1100 Notre-Dame Ouest, Montréal, H3C 1K3, CANADA
[2]christian.masson@etsmtl.ca

***Abstract*— This paper presents a numerical method for investigating aerodynamics of a horizontal axis wind turbine (HAWT). The flowfield around an isolated turbine is described by the Reynolds-averaged Navier-Stokes equations. The turbine is idealized as an actuator disk surface, on which external surficial forces exerted by the turbine blade on the flow are prescribed according to the blade element theory. An in-house code based on an unstructured CVFEM formulation has been employed to solve the resulting mathematical model. The simulations have been carried out through a commercial wind turbine. The mesh dependence study is tackled in order to determine the minimum number grid nodes, as well as the minimum extent of the computational domain. The preliminary results show the potential advantages of the unstructured formulation with respect to structured one; lower number of grid nodes and reduced extent of computational domain up to 29% have been found.**

***Keywords*— Aerodynamics, HAWT, Actuator Disc, Numerical Simulation, Navier-Stokes Equations,**

I. Introduction

Aerodynamic analysis is one of the most critical steps in designing wind turbines. Mainly, three formulations have been used to perform such analysis and are classified as follows. (i) Integral formulations/BEM methods [1], in which the rotor blade is modeled by the actuator disc concept and blade element theory, and the flowfield is described by the integral momentum equation. (iii) Hybrid methods ([2], [3]) in which the flowfield is described either by Navier-Stokes or Euler equations and the rotor blade is modeled by the actuator disc concept. (iii) Full Navier-Stokes methods [4], in which the flowfield is described by Navier-Stokes equations and the rotor blade is introduced by it real geometrical shape using moving reference frame technique. The BEM methods have demonstrated their capabilities for performance predictions, as well as conception and design of wind turbines within a limited range of wind-speeds. There are a number of situations where it is not reasonable to expect BEM methods to offer the desired accuracy, however. In fact, despite the advantage of low computations requirements, these methods cannot describe accurately three dimensional unsteady effects (e.g., turbulence, separated flow, …) and rarely provides detailed aerodynamic information. The complex flow conditions that rotor encounter can be described adequately by the formulations pertaining to the two last classes. The full Navier-Stokes methods are expected to perform more accurate aerodynamic predictions by using modern computational fluid dynamics (CFD) tools. However such methods still require huge computation times. The numerical method presented in this paper belongs to the second class. The flowfield around a turbine is described by the incompressible and axisymetric Reynolds-averaged Navier-Stokes equations. The turbine is idealized as an actuator disk surface, on which external surficial forces exerted by the turbine blade on the flow are prescribed according to the blade element theory [3]. The resulting mathematical model is solved by an in-house code based on an unstructured CVFEM (control volume finite element method) formulation ([5], [6]). The preliminary results including the grid dependence study of power predictions of HAWT-rotors have been presented and discussed.

II. Mathematical Model

A. Rotor Modelling

The rotor modelling approach is based on the actuator disc concept and blade element theory, which are presented as follows.

The actuator-disk concept consists in modelling the rotor as a permeable surface, defined by the rotor-swept area, on which a distribution of forces acts upon the incoming flow at a rate defined by the period-averaged mechanical work that the rotor extracts from the fluid. The rotor's action can be modelled by a distribution of forces, per unit area, on the actuator-disk surface A_R [2]. These forces per unit area of the actuator disk will be referred to as surficial forces in this paper. For HAWT, actuator-disk geometry is defined by the blades' swept area, a circular cone having a base radius represented by $R\cos\gamma$, where R is the blade length and γ is the coning angle of the blades. Fig.1 shows a drawing of a typical actuator disk for HAWT analysis. On this drawing, the blade coning angle has been exaggerated for clarity (typical coning angles are between 0 and 10 degrees). It is assumed that the rotor does not have any spanwise action on the flow; therefore, the surficial force exerted by an elementary actuator-disk surface *dA* may be decomposed into normal and tangential components denoted by f_n and f_t, respectively (see Fig.1).

The rotor is composed of *B* blades of length *R* having a rotational velocity Ω and adjusted to a tip pitch angle of β_o. The blade chord *c* and its twist angle β can vary along the blade. Fig.2*a* shows a representation of the rotor for a given azimuth position. The forces due to lift and drag over a blade section at a given radial position are presented in Fig.2*b*. The fluid velocity relative to the blade V_{rel} is decomposed, in the plane of the section, into a normal component U_n and a tangential component U_t:

$$V_{rel} = \sqrt{U_n^2 + U_t^2}$$
$$U_n = -u_i n_i \qquad (1)$$
$$U_t = r\Omega - u_i t_i$$

where u_i is the i^{th} fluid velocity component and n_i and t_i are the appropriate cosine directors of the unit vectors ***n*** and ***t*** respectively. It is also convenient to define the geometric angle of attack according to the relationship:

$$\alpha = \arctan\left(\frac{U_n}{U_t}\right) - \beta - \beta_o \qquad (2)$$

Blade-element theory implies that the local forces exerted on the blades by the flow are dependent only on airfoil aerodynamic properties and relative fluid velocity. Decomposing these forces onto the ***n*** and ***t*** axes and time-averaging the forces exerted by the blades on the flow during one period of rotation yields the following expressions for the normal and tangential components of the surficial forces exerted by the rotor on the flow:

$$f_n = \frac{B}{2\pi r}\frac{\rho V_{rel} c}{2}\left[U_t C_L + U_n C_D\right]$$
$$f_t = \frac{B}{2\pi r}\frac{\rho V_{rel} c}{2}\left[U_n C_L - U_t C_D\right] \qquad (3)$$

C_L and C_D are the lift and drag coefficients of the blade-defining airfoil, respectively. Furthermore, to take into account the effects of the blade tip vortices, the lift of the two-dimensional airfoil has to be corrected using the Prandtl tip-loss factor.

B. Governing Equations

For steady and incompressible flow conditions around the wind turbine, the time-averaged continuity and the Navier-Stokes equations are written in a Cartesian tensor form

$$\frac{\partial u_i}{\partial xi} = 0 \qquad (4)$$

$$\rho u_j \frac{\partial u_i}{\partial xj} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ji}}{\partial x_j} + \frac{\partial (f_n + f_t)}{\partial x_i} \qquad (5)$$

where u_i is the i^{th} flow velocity component. The introduction of the rotor's normal and tangential forces (Eq.3) in Navier-Stokes equation is called a hybrid method. τ_{ij} is the shear stress tensor given by

$$\tau_{ij} = (\mu_t + \mu)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \qquad (6)$$

μ refers to the molecular viscosity. From the $k-\varepsilon$ turbulence model [7], the turbulent viscosity μ_t is given by

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \qquad (7)$$

where *k* is the turbulent kinetic energy and ε is the turbulent energy dissipation. The turbulence model is composed of two equations, one for *k* and another for ε presented as follows:

$$\rho u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_i}\left(\Gamma_k \frac{\partial k}{\partial x_i}\right) + P_k - \rho\varepsilon \qquad (8)$$

$$\rho u_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_i}\left(\Gamma_\varepsilon \frac{\partial \varepsilon}{\partial x_i}\right) + C_{\varepsilon 1}\frac{\varepsilon}{k}P - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \qquad (9)$$

where, P_k is the rate of production of turbulent kinetic energy and is approximated by

$$P_k = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} \qquad (10)$$

and the diffusion coefficients Γ_k and Γ_ε are given by

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \qquad (11)$$

$$\Gamma_\varepsilon = \mu + \frac{\mu_t}{\sigma_\varepsilon} \qquad (12)$$

The $k-\varepsilon$ model contains five empirical constants which take on the following values:
$C_\mu = 0.033,\ C_{\varepsilon 1} = 1.176,\ C_{\varepsilon 2} = 1.92,\ \sigma_k = 1.0,\ \sigma_\varepsilon = 1.3$

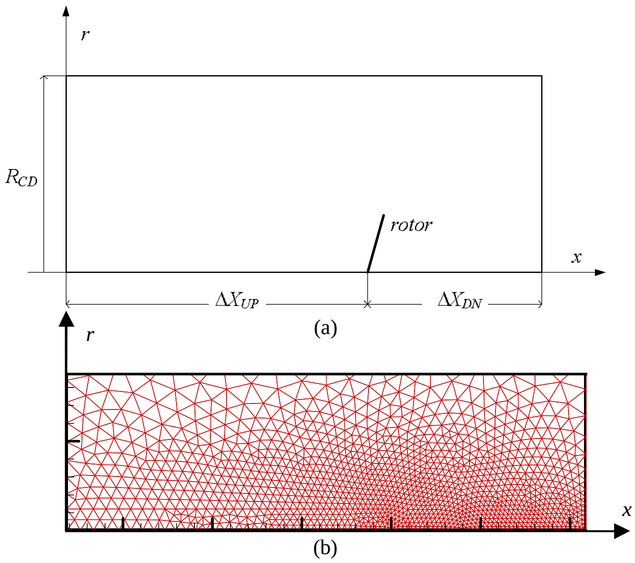

Fig. 1 (a) Computational domain and (b) Grid topology

III. NUMERICAL METHOD

To solve the resulting governing equations, an unstructured Control-Volume Finite Element Method (CVFEM), described in a previous authors work [6], has been used. In this section, the computational domain and the boundary conditions are presented.

A. Computational Domain

The flow field in the vicinity of the turbine (i.e. ignoring the effects of the tower, nacelle and the ground) immersed in a uniform incoming flow parallel to the turbine's axis of rotation is axisymmetric. Thus, the computational domain consists of a cylinder including the rotor. Fig.1 shows a (x, r) section of the domain and grid topology. The size of the domain is characterized by three length parameters, namely, ΔX_{UP}, ΔX_{DN}, and R_{CD} [8]. As presented below, in section IV, the mesh dependence study is tackled in order to determine the minimum number grid nodes for the discretization of the domain, as well as the minimum extent of the computational domain.

B. Boundary Conditions

The following three boundaries considered are :

Inlet boundary: The inlet boundary is an $r-\theta$ plane located upstream of the wind turbine. In this plane, the distributions for the velocity components as well as the turbulence properties are assumed to be uniform with values corresponding to the neutral planetary boundary layers properties at hub height [9].

Outlet boundary: The outlet boundary is an $r-\theta$ plane located downstream of the wind turbine. Here, velocity field and turbulence properties are calculated using the outflow treatment of Pantankar [10], while the pressure is specified and assumed to be uniform over the entire plane.

Top boundary: This lateral surface is located at radial distance far from the axis of turbine. As was the case for the inlet conditions, in this boundary undisturbed neutral flow conditions are prescribed for three velocity components and for turbulence properties, while pressure is calculated from the continuity equations.

IV. RESULTS AND DISCUSSIONS

Previously Masson et al. [8] have proposed a structured CVFEM formulation and demonstrated its capability for aerodynamic analysis of HAWT. In this paper, an unstructured version is presented and investigated, which would guarantee more mesh flexibility and would require less grid-nodes. For this purpose, the mesh dependence of the rotor performance predictions has been investigated: (i) to determine the minimum number of grid-nodes required for the discretization of the computational domain, and (ii) to determine the minimum extent of the domain, that would lead to grid independent numerical solution. The simulations have been carried out through a 750 kW commercial machine.

TABLE I
SIZE OF COMPUTATIONAL DOMAIN [8]

$\Delta X_{UP}/D$	$\Delta X_{DN}/D$	R_{CD}/D
7.5	4.5	4.0

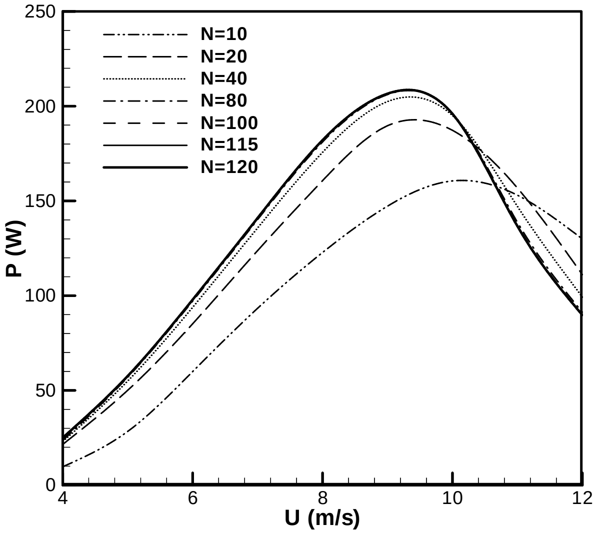

Fig. 2 Grid dependence study: Power curve dependence on the number of rotor nodes

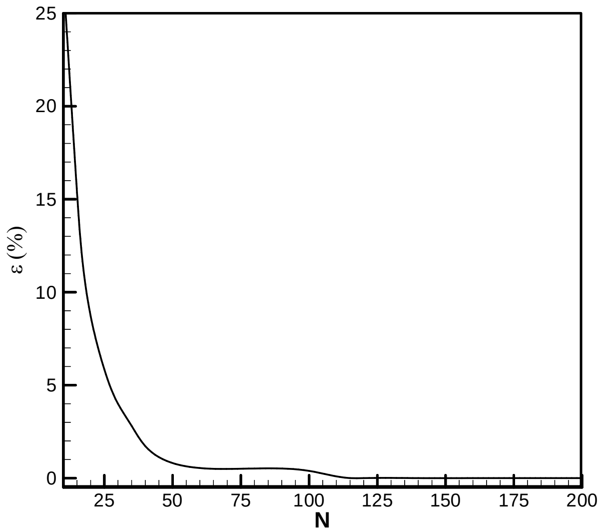

Fig. 3 Grid dependence study: Error parameter given by Eq. 13 as a function of number of rotor nodes N

To determine the appropriate grid that would produce rotor-performance predictions independent of grid dimensions, simulations have been performed for typical wind speeds varying from 4 m/s to 12 m/s, by considering different grid dimensions characterized by the number of nodes N that discretize the rotor varying in the range 5-120; while the three length parameters: ΔX_{UP}, ΔX_{DN}, and R_{CD}, assume the values suggested elsewhere [8] which are listed in Table 1. Fig.2 shows the evolution of the turbine power as function of wind speed obtained at different rotor grid nodes N. As it can be seen for N>40, all the power curves appear to be independent of grid dimensions; differences up to 3% have been noted. To figure out the minimum number N_{min}, the following error parameter has been introduced:

$$\varepsilon = \frac{|P(N) - P(400)|}{P(400)} \quad (13)$$

where, $P(N)$ is the power obtained at a given number of grid nodes N; $P(400)$ is the power obtained at fine mesh corresponding to the grid-independent numerical solutions. Fig.3 shows the evolution of ε as function grid nodes N, for a given wind speed. These results clearly indicate that the minimum number of rotor nodes is $N_{min} = 115$.

Now, let see whether the domain extent might be reduced again, and thus lowering again the total number of grid nodes required to discretize the computational domain. For this purpose, the following error parameter has been introduced:

$$\varepsilon = \frac{|P(\Lambda) - P(\Lambda_{ref})|}{P(\Lambda_{ref})} \quad (14)$$

Where, $P(\Lambda)$ is the power prediction obtained at given length ratio Λ (namely $\Lambda = X_{UP}/D, X_{DN}/D, R_{CD}/D$; where D refers to rotor diameter), $P(\Lambda_{ref})$ is power prediction obtained at the reference length ratio Λ_{ref} given by Table 1.

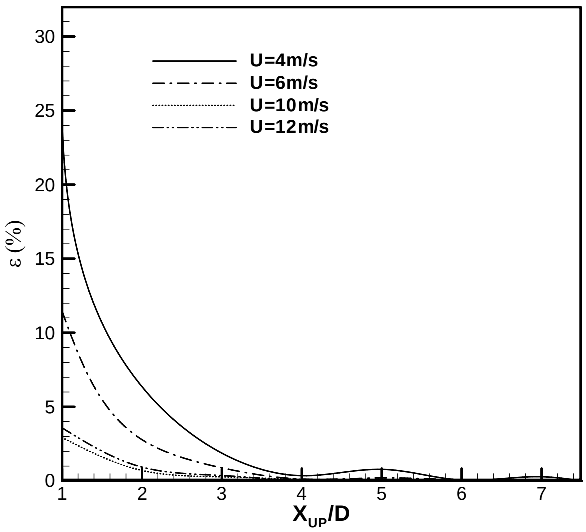

Fig. 4 Grid dependence study: Error parameter given by Eq. (14) as a function of length ratio $\Lambda = X_{UP}/D$

Fig.4 shows the results obtained for $\Lambda = X_{UP}/D$. As it can be noted, lower values of ε (less than 0.3%) are found around $\Lambda = 4$. While for the other ratios, $\Lambda = X_{DN}/D$ and R_{CD}/D, (results not presented here) still remain equal to that of reference values given in Table 1.

Due to this new value of length ratio Λ, $X_{UP}/D = 4$, the extent of domain can be reduced by about 29%, and thus lower number of total grid nodes can be used.

V. CONCLUSION

An unstructured CVFEM formulation is proposed for the aerodynamic analysis of axisymmetric flowfields around an HAWT rotor. The preliminary results presented in this paper show the potential advantages of such numerical method with respect to structured formulation; lower number of grid nodes and reduced extent of computational domain up to 29% have been found.

Finally, future works will be mainly focused on detailed optimisation studies of grid and extent of computational domain of more complete physical problems, such as the effects of atmospheric turbulence, static and dynamic stalls, and 3D flowfields around HAWT. Also, detailed comparison between the unstructured and structured formulations will be undertaken forthcoming.

ACKNOWLEDGMENT

The support from General Directorate for Scientific Research and Technological Development (DG-RSDT), as well as from Ministère de l'enseignement supérieur et la recherche scientifique of Algerian government in the form of research grand to Prof. A. Smaïli is gratefully acknowledged.

REFERENCES

[1] Wilson, R. E. and Lissaman, P.B.S. Applied Aerodynamics of Wind Power Machines, Oregon State University, NTIS PB 238594, 1974
[2] Madsen, H. A. The Actuator Cylinder a Flow Model for Vertical Axis Wind Turbines, *Aalborg University Centre*, Denmark, 1992
[3] Masson, C., Smaïli, A., Leclerc, C. (2001) Aerodynamic Analysis of HAWTs Operating in Unsteady Conditions, WIND ENERGY, Vol.4, pp. 1-22
[4] SØrensen N. N., Michelsen J. A., and Schreck S. (2002) Navier-Stokes Predictions of the NREL Phase VI Rotor in the NASA Ames 80ftx120ft Wind Tunnel, Wind energy, 5: pp.151-169.
[5] Masson, C., Saabas, H. J., Baliga, B. R., "Co-Located Equal-Order Control-Volume Finite-Element Method for Two Dimensional Axisymmetric Incompressible Flow", International Journal for Numerical Methods in Fluids, Vol. 18, pp. 1-26, 1994
[6] Tran, L.D., C. Masson, A. Smaïli "A Stable Second-Order Mass-Weighted Upwind Scheme for Unstructured Meshes" International Journal for Numerical Methods in Fluids, Vol 51, pp 749-771, 2006
[7] Launder, B.E. and Spalding, D.B. "*The Numerical Computation of Turbulent Flows*", Computer Methods in Applied Mechanics and Engineering, Vol.3, No.2, pp. 269-289, 1974
[8] Masson, C. , I. Ammara, I. Paraschivoiu "An Aerodynamic Method for the Analysis of Isolated Horizontal-Axis Wind Turbines" Int. Journal of Rotating Machinery, Vol.3, No.1, pp. 21-32, 1997
[9] Panofsky, H.A., Dutton, J.A. "*Atmospheric Turbulence: Models and Methods for Engineering Applications*", John Wiley & Sons, 1984
[10] Patankar, S.V. "*Numerical Heat Transfer and Fluid Flow*", McGraw-Hill, 1980.