Numerical modelling of flow around wind turbines using a hybrid method based on the Navier-Stockes solver and the generalized actuator disc concept

Arezki Smaïli1* and Christian Masson2

1 Laboratoire de Génie Mécanique et Développement
Ecole Nationale Polytechnique, El Harrach, Alger, Algérie
2 Canada Research Chair on the Aerodynamics of Wind Turbines in Nordic Environment
Ecole de Technologie Supérieure, Montréal, Canada

Abstract - This paper presents a numerical method based on Navier-Stokes equations and generalized actuator disc concept to investigate the aerodynamics of horizontal axis wind turbines (HAWT). The rotor modelling is first described, then the formulation to solve the governing equations is outlined. Simulation results on the flow around a typical commercial wind turbine are presented and discussed.

Keywords: Aerodynamics - Actuator disc - HAWT - Navier-Stokes equations - CFD.

1. 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. (ii) Hybrid methods [2], in which the flowfield is described either by Navier-Stokes or Euler equations and the rotor blade is modeled by a generalized actuator disc concept. (iii) Full Navier-Stokes methods [3] in which the flowfield is described by Navier-Stokes equations and the rotor blade is introduced by its 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 (normal flow conditions). 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-Stockes 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 paper is focused on the following points: (i) rotor modeling and the concept of the hybrid method, (ii) modelling of the flow around the turbine, (iii) numerical method and formulation to solve the governing equations and finally, (iv) typical simulation result on rotor-nacelle interactions. A

* Arezki.smaili@enp.edu.dz
typical atmospheric turbulent flow around the rotor and nacelle of an HAWT rated at 750 kW has been investigated. The flow field in the vicinity of the nacelle is extremely complex: it is phase dependent (i.e. varying with the blade’s azimuthal location) as well as time dependent.

In order to propose a practical approach for the solution of this problem, some major simplifying assumptions have been introduced into the analysis. First, the turbulent flow is assumed to be steady and axisymmetric. Consequently, the Reynolds’s averaged incompressible and two dimensional axisymmetric Navier-Stockes equation are considered, in order to describe the flow field around the rotor and nacelle [4].

The standard $k - \varepsilon$ model [5] has been chosen for the description of the fully turbulent flow region, because of the ready availability of $k$ and $\varepsilon$ properties of atmospheric boundary layers in meteorological data. In the lower Reynolds number region, the wall function method is used to treat the near wall region of the nacelle.

2. MATHEMATICAL MODEL

2.1 Rotor modelling

2.1.1 Actuator disk concept

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 HAWTs, 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 $\gamma$ 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 (Fig. 1).

![Fig. 1: Description of the actuator disk](image)
2.1.2 Blade-element theory

The rotor is composed of B blades of length R having a rotational velocity $\Omega$ and adjusted to a tip pitch angle of $\beta_0$. The blade chord $c$ and its twist angle $\beta$ can vary along the blade. Fig. 2a- 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. 2b-. 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 $U_t$:

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

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:

$$\alpha = \arctan \left( \frac{U_n}{U_t} \right) - \beta - \beta_0$$

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:

$$f_n = \frac{B}{2\pi r} \cdot \frac{\rho \cdot V_{rel} \cdot c}{2} \cdot [U_t C_L + U_n C_D]$$
$$f_t = \frac{B}{2\pi r} \cdot \frac{\rho \cdot V_{rel} \cdot c}{2} \cdot [U_n C_L - U_t C_D]$$

$C_L$ and $C_D$ are the dynamic lift and drag coefficients of the blade-defining airfoil, respectively, and are obtained using an appropriate dynamic stall model. 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.
2.2 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

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

\[
 u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ji}}{\partial x_j} + \frac{\partial (f_n + f_t)}{\partial x_i}
\]

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 is called generalised actuator disc concept. \( \tau_{ij} \) is the shear stress tensor given by

\[
\tau_{ij} = (\mu_t + \mu) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

\( \mu \) refers to the molecular viscosity. From the \( k - \epsilon \) turbulence model [5], the turbulent viscosity \( \mu_t \) is given by:

\[
\mu_t = C_\mu \cdot \rho \cdot \frac{k^2}{\epsilon}
\]

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

\[
u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_i} \left( \Gamma_k \cdot \frac{\partial k}{\partial x_i} \right) + P_k - \rho \epsilon
\]

\[
u_j \frac{\partial \epsilon}{\partial x_j} = \frac{\partial}{\partial x_i} \left( \Gamma_\epsilon \cdot \frac{\partial \epsilon}{\partial x_i} \right) + C_{\epsilon 1} \cdot \frac{\epsilon}{k} \cdot P_k - C_{\epsilon 2} \cdot \rho \cdot \frac{\epsilon^2}{k}
\]

Where \( P_k \) is the rate of production of turbulent kinetic energy and is approximated by:

\[
P_k = \mu_t \cdot \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \cdot \frac{\partial u_i}{\partial x_j}
\]

and the diffusion coefficients \( \Gamma_k \) and \( \Gamma_\epsilon \) are given by

\[
\Gamma_k = \mu + \frac{\mu_t}{\sigma_k}
\]

\[
\Gamma_\epsilon = \mu + \frac{\mu_t}{\sigma_\epsilon}
\]

The \( k - \epsilon \) model contains five empirical constants which take on the following values:
The turbulence scalar transport equations are valid only for fully turbulent regions (i.e. large turbulent Reynolds number). An additional model must be introduced to treat the near-wall regions (i.e. lower turbulent Reynolds number). The wall function method [5] is used in the present study to eliminate the large number of grid points needed to resolve the sublayer region. The following relationships are used to bridge the near-wall region:

\[ \tau_w = \rho \cdot U_p \cdot C_\mu^{1/4} \cdot k_p^{1/2} \cdot U^+ \]  

(13)

Where

\[ U^+ = \frac{1}{\kappa} \ln (E \cdot y^+) \]  

(14)

\[ y^+ = \frac{\rho \cdot C_\mu^{1/4} \cdot k_p^{1/2} \cdot y_P}{\mu} \]  

(15)

The boundary conditions for \( k \) and \( \varepsilon \) are respectively given by:

\[ \frac{\partial k}{\partial y} = 0 \]  

(16)

\[ \varepsilon_p = \frac{C_\mu^{3/4} \cdot k_p^{3/2}}{\kappa \cdot y_P} \]  

(17)

Here, \( y \) is the local coordinate normal to the wall, \( \tau_w \) is the wall shear stress, \( \kappa \) is Karman's constant (\( = 0.40 \)), and \( E \) is an empirical constant (\( = 9.8 \)). The subscript \( P \) refers to the grid node next to the wall.

When the mesh is such that \( y^+ < 11 \) at node \( P \), the law of the wall is represented by:

\[ U^+ = y^+ \]  

(18)

3. NUMERICAL METHOD

To solve the resulting governing equations, an unstructured Control-Volume Finite Element Method (CVFEM), described in a previous author’s work [4], has been considered.

3.1 Computational domain

The flow field in the vicinity of the turbine and nacelle (i.e. ignoring the effects of the tower 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 and nacelle.

Fig. 3 shows a \((x, r)\) section of the domain. This domain is discretized into unstructured meshes composed of triangular elements.
Fig. 3: Computational domain

The complete set of fluid equations, expressed in the axisymmetrical coordinate system, consists of the continuity equation, three momentum equations for transport of velocity, and two equations for modelling turbulent kinetic energy and turbulent energy dissipation. Axisymmetric turbulence terms have been introduced as elsewhere [6].

3.2 Boundary conditions

The four boundaries considered are:

**Inlet boundary**- The inlet boundary is an $\theta - r$ plane located upstream of the wind turbine. In this plane, the distributions for the velocity components as well as the $\varepsilon - k$ properties are assumed to be uniform with values corresponding to the neutral planetary boundary layers properties at hub height [8].

**Outlet boundary**- The outlet boundary is an $\theta - r$ plane located downstream of the wind turbine. Here, velocity field and $\varepsilon - k$ properties are calculated using the outflow treatment of Pantankar [9], 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.

**Wall boundary**- The presence of the nacelle in the computational domain is represented by the wall region, where turbulence properties and velocity fields are prescribed using the wall function method.

4. RESULTS AND DISCUSSION

As reported in previous authors’ work [10], the computations presented here have been carried out for a variable speed stall-controlled three-bladed wind turbine rated at 750 kW and having a rotor diameter of 48m. Mainly, this paper is focused on the investigation of the impact of the turbine's rotating blades on the flow field over the nacelle, i.e. rotor-nacelle interaction.

By keeping the rotor blades stationary, the flow field over the nacelle has been computed for wind speed varying from 4 m/s to 13 m/s. Figure 4 shows the simulation
results for velocity and pressure fields around the nacelle for $U_0 = 13$ m/s. Note here the significant speedup region (i.e. low pressure region). As discussed below, it is often recommended that the anemometer be located in this speedup region. The behaviour of simulation results seems to be quite reasonable.

Furthermore, a comparison between the numerical results (viscous and turbulent) and those obtained analytically (by considering an inviscid fluid flow) is presented in order to estimate the importance of the effects of viscosity and turbulence where nacelle anemometers are typically installed. Figure 5 shows the simulation results for velocity and pressure fields around the nacelle obtained in the presence of upstream rotating rotor blades, for $U_0 = 13$ m/s. The velocity and pressure decrease (i.e. the rotor wake) downstream from the rotor, away from the nacelle.

Similarly to the flow over the nacelle alone, the speedup region seems to be unaffected by the rotor wake. To quantify the impact of the turbine rotor on the flow field over the nacelle and thus assess a suitable position for the anemometer, the distributions of velocity and turbulence properties within the speedup region (i.e. anemometer position) have been investigated.

Figure 6 shows the vertical distributions of the velocity (i.e. velocity profiles), namely, axial velocity (i.e. $u_x$) and radial velocity (i.e. $u_r$), obtained at an axial position, $x = 4.8$ m, within the speedup region, for $U_0 = 13$ m/s. A comparison between the rotor-nacelle results and those without the rotor effect is presented.

In the vicinity of the nacelle (i.e. around $r = 2.4$ m), the velocity profiles exhibit pronounced peaks, which then decrease to reach the free stream condition. In the region
where $r < 3\, \text{m}$, the presence of an upstream rotating rotor seems to increase the velocity magnitudes slightly, particularly the axial velocity profile (Fig. 6a). However, in the outer region (i.e. around $r > 3\, \text{m}$) a significant decrease of the velocity magnitudes is noted, which is quite reasonable behaviour due to the rotor wake.

5. CONCLUSION

A hybrid method based on generalised actuator disc concept and Navier-Stokes equations has been described. As typical example, the investigation of the impact of rotor-nacelle interactions is presented and discussed.

The proposed method has demonstrated its ability to predict turbulent flow over a nacelle by reproducing the expected physical behaviour of the flow. The numerical method might be a useful tool for locating nacelle anemometers.

The proposed approach only models the flow field properties averaged over a finite number of blade rotations. To model the instantaneous flow field around rotors and nacelles, the phase dependency of the flow field has to be taken into account.

This can be done by introducing a near wake model [7] or by using the actuator-line approach [11] for the rotor representation.

REFERENCES


