Actuator surface hybrid model

I Dobrev\textsuperscript{1}, F Massouh\textsuperscript{1} and M Rapin\textsuperscript{2}

\textsuperscript{1}ENSAM, 75013, Paris, France
\textsuperscript{2}ONERA, BP 72, 92322 Châtillon Cedex, France

E-mail: ivan.dobrev@paris.ensam.fr

Abstract. A hybrid model has been developed in order to represent the flow downstream a wind turbine rotor. The blades are replaced by their mean surfaces and a “pressure jump” boundary condition is applied here. The hybrid model combines CFD solver with a blade element method (BEM). The solving method is iterative: at the beginning of iteration a BEM determines the pressure discontinuities along the blade span, using the rotor inflow and the aerodynamic properties of blade sections. Then the CFD solver applies this pressure discontinuity in order to model the blade forces and the wake downstream the rotor. At the end of iteration the obtained rotor inflow, the wake development and the residuals are compared with those obtained from previous iteration. If the required precision is attained, the calculation stops. The solution is obtained after thousands of iterations, depending on the number of nodes, residuals etc. The approach of replacing blades by pressure discontinuity surfaces is validated by comparison the wakes in the cases of flat plate and the S809 airfoil. Finally, the proposed hybrid model is used to calculate the performance of NREL Phase VI wind turbine and the results obtained are satisfactory.

1. Introduction

Actually, in order to optimize the wind energy power production, several wind turbines are installed on the same wind farm. However, the proximity between these machines creates problems of aerodynamic interactions. Generally the wind farm development is extremely complex problem and multiplicity of factors comes in play in order to place in position each wind turbine. To optimize the energy production and the operation costs, engineers use software tools developed especially for wind farm design. These software tools take into account wind turbine data, wind speed and direction, site topography, etc. However, in all cases it is needed to avoid the negative effect of aerodynamic interference between the wind turbines. Therefore, it is preferable to investigate the wake development downstream of each wind turbine in the wind farm. The simplified models used for wind farm design are not well adapted and cannot describe correctly all singularities of the wind turbine rotor. They cannot obtain with a sufficient precision the velocity field around the rotor and therefore they are not capable to evaluate the aerodynamic forces applied to blades.

To obtain numerical results with sufficient quality, a CFD simulation with an appropriate fine grid mesh is needed, but the solution is computationally very expensive. As result, modeling a wind turbine farm with more than two machines is impossible in practice. Hence, to overcome this limitation, the rotors must be represented by numerical models that need less grid points.

To simplify the numerical model and reduce the computational cost, it is possible to use an equivalent representation of the rotor blades. This representation must be able to describe the behavior
of the wind turbine rotor without modeling the exact blade geometry in the CFD computations. This kind of numerical modeling belongs to what is called hybrid modeling.

The hybrid models comprise two modules. In the first module a CFD solver computes the velocity field around the wind turbine rotor. Here, the presence of the rotor is modeled with body or surface forces. To obtain these body or surface forces, one second module uses a conventional method based usually on the blade element method (BEM). Thus, there is no need to model airfoil boundary layer and the grid around the blades may be coarsened. As results the need of computer power is reduced.

The reliability of BEM to computes the well-attached flows from known aerodynamic airfoil performances and also of CFD to computes the resulting rotor wake, explains the success of these hybrid models for various problems of interaction in fluids machinery. They are used for numerous propeller-body interactions, problems in marine hydrodynamics, helicopter and wind turbine aerodynamics.

There are numerous applications of hybrid modeling in the field of helicopter aerodynamics. Here, due to the high yaw angle (the angle between the free velocity vector and the rotor axis) and the resulting unsteadiness it is more advantageous to use Generalized Dynamic Wake Theory (GDWT). An interesting three-dimensional unsteady hybrid model that combines GDWT and the thin layer Navier-Stokes code is presented in [1] where two adjacent boundary planes replace the rotor inflow and outflow. The blade rotation is modeled by the rotation of a pressure jump, applied to the plane that represents the rotor outflow. The pressure jump intensity is calculated by GDWT iteratively using the rotor inflow obtained by the CFD. The presented result gives a good approximation compared to the experiment.

In the field of wind turbine aerodynamics, [2] presents a comprehensive review on wake aerodynamics of wind turbines and several hybrid models are discussed. It is shown that many hybrid models use an actuator disk with the application of pressure or volume forces. In these axisymmetric models, the forces vary with the radius but are distributed uniformly along the azimuth direction and as a result the individual presence of blades is lost.

To overcome this limitation and to represent more realistically the flow field downstream the rotor, a three-dimensional representation of the rotor blades is developed in [3]. In this model, the geometry of real blades is replaced in CFD by body forces radially distributed along lines. Similarly to actuator disk model, which uses one disk to represent the rotor, the authors in [3] name their model “actuator line”. Here, the blade forces are determined by means of two dimensional airfoil corrected data and the results of CFD computations are used for the reference velocity. Compared to actuator disk, this model permits to represent individually each blade with its tip and root vortices and thus to improve rotor wake representation. The comparison of the actuator line with experimental data reveals the effectiveness of this proposed model. More recently, [4] applies the same approach. Several simulations are presented for a wind turbine in the case of normal and yawed operations. The comparison with experiments revealed that for higher yaw angles, the actuator line model gives better results than the actuator disk model.

Compared to actuator line model the model proposed in the present work goes further. Here, each blade is replaced by a surface of pressure discontinuity. By this way the difficulties of body force distribution around the actuator line are overcome. Compared to preceding works presented in [1], the blades are not modeled as rotating pressure jump but with their mean surfaces. Also the pressure distribution is not constant along the chord and varies. This chord distribution approximates the thin pressure plate distribution without leading edge singularity. The aim is to improve the blade representation and therefore the initial conditions of wake development.

2. Actuator surface model

Actuator blade model combines a BEM with a CFD solver. In the CFD domain, the rotor geometry is simplified and the blades are replaced by surfaces with boundary condition “pressure discontinuity”. Hence, the surface forces replace the rigid blade wall and the number of nodes is significantly reduced, as there is no need to model the boundary layer.
Starting from initial approach for the upstream flow, the blade geometry and the airfoil data, the BEM module calculates the pressure jump distribution on the surface replacing the blade. Then the CFD module computes the flow velocity field, using as boundary condition, the pressure distribution previously obtained from the BEM module. The solution is carried out iteratively, exchanging data between the BEM and CFD modules; it stops after convergence is reached.

The calculation of pressure discontinuity is based on the blade element approach. At the blade radius \( r \), the elementary forces acting in the normal and tangential directions on a blade element with span \( dr \) and chord \( c \) are:

\[
dF_n = \frac{1}{2} \rho W_R^2 c C_n(\alpha) dr
\]

and

\[
dF_t = \frac{1}{2} \rho W_R^2 c C_t(\alpha) dr
\]

In above formulas the force coefficients \( C_n \) and \( C_t \) are determined using the aerodynamic blade sections performances \( C_n=C_n(\alpha) \) and \( C_t=C_t(\alpha) \). The angle of attack \( \alpha \) is

\[
\alpha(r) = \phi(r) - \beta(r),
\]

where \( \beta \) is the blade section pitch angle and \( \phi \) is the angle between the plane of rotation and the reference relative velocity \( W_R \). In the vortex line methods or BEM the induced velocity can be evaluated explicitly, hence it is easy to calculate the flow angle \( \phi \):

\[
\phi(r) = \tan^{-1}\left( \frac{V_0 - w_{ia}(r)}{\Omega r + w_{ig}(r)} \right)
\]

However, in actuator surface model the angle of attack \( \alpha \) cannot be calculated without some assumptions concerning the flow around the rotor, because when using CFD there is no means to separate the induced velocity in equation (4) from the rest of velocity field.

**Figure 1.** Shape of chordwise pressure distribution.  
**Figure 2** Scheme of coupling of CFD and BEM modules

For our approach, it is helpful to present the flow around the wind turbine as the sum of a non-perturbed flow and another flow induced by the wind turbine blades. The induced velocity field by the wind turbine may be separated into two components:

- Local induced flow, created by the presence of the blade airfoils.
Global induced flow, due to the presence of the rotor like the actuator that extracts kinetic energy from the wind and that slows down the mass of air, which passes through the disk. In the vicinity of actuator disk, the globally induced velocities are slightly axially changed. Upstream of the blades sections, in the distance of some chord lengths, the flow is not perturbed by the presence of local blade section. Therefore for the reference plane, where the velocity vector must be determined, it is acceptable to use one plane placed slightly upstream, where the flow is not influenced locally by the blade sections, but sufficiently close to the rotor plane of rotation to have the same global induced field.

Obviously, if a plane such as the proposed one is used as the reference plane, the standard airfoil performance must be corrected. The reference plane is closer to the airfoil compared to the appropriate plane usually specified in “infinity”. Therefore, for the same airfoil force coefficients the reference angles of attack are different. This approach is very advantageous when airfoil performances are known from experiment [5] or from numerical simulations. Usually they are determined close to the blade and in the proposed model they may be used immediately, without any correction.

In the CFD the obtained normal force from equation (1) is applied, like a pressure discontinuity along the chord, which replaces the airfoil. In order to make the velocity induced by this discontinuity more adequate, it is preferable to use the chordwise distribution shown in figure 1. This pressure distribution shape is close to the thin flat plate pressure distribution. At leading edge the pressure have no singularity, also the moment of pressure forces with respect to the point at 1 / 4 of the chord is equal to zero.

The hybrid model proposed, figure 2, is based on the CFD code Fluent and the solution is obtained iteratively. In the CFD model, which represents flow field around the wind turbine, the blades are replaced by surfaces (with respect to the) defined as “fan” boundary condition. This boundary condition corresponds to an imposed pressure difference between adjacent cells, located at the opposite sides of the boundary. Once at the beginning of the current iteration the CFD code executes a user-defined function UDF in C language. This function plays the role of the BEM solver and calculates the pressure distribution from blade geometry, rotor inflow and aerodynamic data of blade section.

Figure 2. Velocity field around airfoil S809

Figure 3. Velocity field around a line with prescribed pressure discontinuity

The UDF function has access to all grid and flow variables, which are needed to calculate the relative velocity vector along one reference line. Consequently, the normal and tangential force coefficients for all blade section along the span are calculated from the blade geometry and airfoil data. Then the resulting pressures are imposed on the blade equivalent surfaces using the simplified
pressure distribution presented in figure 1. The solution is obtained after thousands of iterations depending on the number of nodes, demanded residuals, etc.

3. Numerical results

3.1. Hybrid model in the case of airfoils
To validate our approach of actuator surface method the velocity field around the S809 airfoil is compared with that created by a pressure discontinuity distributed along the chord line for same condition. The comparison is carried out in the case of $14^\circ$ angle of attack. The pressure distributed along the pressure discontinuity line produces the same normal force as the presented airfoil. The shape of its pressure distribution is similar to figure 1. The CFD solver is Fluent 6.2 and the turbulence model is k-ω SST. The Reynolds number of flow corresponds to $1.10^6$ and the turbulence intensity of upstream flow is 1%. For all simulations the grid is the same of 240 nodes uniformly distributed along the airfoil surface. The C-mesh around the airfoil is constituted by 24 cell layers. The initial size of the cell in a normal direction represents $1.10^{-3}$ of the chord length and the growth factor is 1.15. In the case of pressure discontinuity line calculation, the airfoil interior is meshed, the airfoil surface is defined as permeable and along the chord line the boundary condition of type “fan” is applied.

The results of calculation are presented as normalized velocity in figure 3 and figure 4. At a distance of one chord upstream the airfoil, the differences between the velocity fields are negligible (less than 1%). Downstream the airfoil the differences between the flow velocities could reach 10%. Similar differences are obtained for all angles of attack between $6^\circ$ and $24^\circ$. The results are satisfying, but it must be noted that it is impossible to model the airfoil shear layers and the turbulent wake with a pressure discontinuity boundary condition, without a boundary condition “wall”.

3.2. Hybrid model in the case of wind turbine
The studied wind turbine is the NREL Phase VI case. The turbine has two-bladed rotor of 10-meter diameter with blade sections of S809 airfoil, [5]. The choice of NREL wind turbine was made because of its large experimental database. It is useful to validate the proposed model using the available inflow measurements at five blade radii for different upstream velocities. At the same time, for these blade sections, there are also measurements for the chordwise pressure distribution. From these measurements, it is possible to create a data matrix for the normal and tangential force coefficients versus angle of incidence along the blade span. Indeed, using techniques of interpolation a two-dimensional function $C_n = C_n(\alpha, r/R)$ depending on the radius can be created, figure 5.

![Figure 5. Normal force coefficient distribution](image)

![Figure 6. Simplified blade model](image)
During calculation in order to obtain the pressure distribution, for each node of pressure discontinuity surface the velocity, the angle of attack $\alpha$, the distance $s/c$ from the leading edge and the relative radius $r/R$ are determined, figure 6. Using the $r/R$ and angle of attack $\alpha$, the force coefficients can be obtained from the experimental data, figure 5. Then using the distance $s/c$ and the shape of chordwise pressure distribution, shown on figure 1, it is possible to calculate the value of pressure discontinuity that Fluent will apply as “fan” boundary condition.

The studied case is for upwind rotor with blade tip pitch of $3^\circ$ and average section Reynolds number about $1.10^6$. A cylinder that has a radius of $3.316R$ represents the flow field around the wind turbine rotor. It has also the length of $6R$ upstream and of $25R$ downstream the rotor plane. Here the value $3.316$ is used because it leads to the same ratio as between the wind tunnel cross-section and the area of the wind turbine rotor. Hence, experiments and numerical simulations have the same coefficient of blockage. The surface replacing the blade is represented by 4 000 nodes from the 500 000 ones used for the whole model. This surface is divided in the chordwise direction into 40 intervals, which are refined near the leading edge, where the pressure discontinuity gradient is strongest. In spanwise direction the blade is divided into 100 intervals equally spaced. The initial cells size in normal direction is 0.01 chords and the growth factor of 1.5 is used. To improve the wake calculation the mesh enclosed in cylinder with length of $10R$ and diameter of $1.25R$ downstream the rotor is refined. The applied interpolation function, figure 5, for the normal force coefficients is based on the inflow measurements at five spanwise stations ($r/R = 0.30, 0.47, 0.63, 0.80, 0.95$) obtained for the case “H” [5]. To apply these data without any correction, it is used a reference line that passes through the geometrical points where experimental coefficients are obtained.

Calculation is carried out iteratively. After numerous iterations, according to the number of the nodes and the value of the residuals required, the convergence process is achieved. Usually, the computed rotor power reaches a constant value quickly, but additional iterations are needed to obtain the wake development.

The comparison between wind turbine performances obtained experimentally and numerically by means of actuator surface method is presented in Fig. 7. The disagreement in case of high velocities is due to the fact that the effect of “centrifugal pumping” is not modelled, because of lack of radial forces. Some incertitude is involved because it is difficult to interpolate the force coefficients near tip and root regions. To overcome this problem, more experimental or additional CFD results are needed. However, it is useful to employ a correction for the circulation distribution near the blade tip region and also for the induced velocity angles. In vicinity of the blade ends the flow is highly three-
dimensional; therefore the hybrid model is non-adequate, similarly of all models that use the airfoil performances.

4. Conclusion
In the present work, an actuator surface model is proposed for the calculation of the flow around wind turbine rotors. The objective is to validate the feasibility of this model that couples a BEM with a CFD solver. To represent the rotor, the blades are replaced by thin surfaces constituted by the blade mean surfaces. On these surfaces, a pressure discontinuity is applied. This pressure discontinuity is calculated from aerodynamic properties of the blade section airfoils and the rotor inflow. In order to improve the initial conditions of the wind turbine wake development, the surface that replaces blade has the same pitch angle as the original and its chordwise pressure is variable. Therefore the actuator surface model is different from all existing hybrid models that use the actuator lines or that modeled blade as rotating pressure jump.

The model uses a modified thin flat plate chordwise pressure distribution, without singularity in the leading edge point. This permits to create a velocity field similar to that around an airfoil, for the same normal force coefficient. The numerical calculations carried out in case of S809 airfoil proved this approach. It is to be noted that for a distance of one chord upstream the leading edge, the difference is near one degree for flow angles and less than 1% for velocity magnitudes. Downstream the airfoil the difference is less than 10% for velocity magnitude.

In the case of wind turbine simulation the actuator surface model is capable to reproduce rotor mechanical power performances, but for high wind velocity some disagreement with experimental results is revealed. This is due to fact that in case of high wind velocity the flow is detached. Therefore the flow is highly three-dimensional and the actuator surface model is non-adequate.

The suggested model here has two significant advantages compared to the other hybrid models. The first advantage is the possibility of using the 3-D airfoil data without applying any correction. The second advantage is that the velocity field downstream the blades are closer to reality compared to the methods that use actuator disk or actuator line. Compared to the CFD methods, which use the complete three-dimensional rotor geometry, this model have the advantage of using a limited number of nodes. Hence, the size of the model is now suitable for studying wind farm design.

References
[1] Boyd D Jr, Barnwell R and Gorton S 2000 A Computational Model for Rotor-Fuselage Interactional Aerodynamics, 38th AIAA Aero. Sci. Meet. and Ex. Reno AIAA 2000-0256
[2] Vermeer L J, Sorensen, J N and Crespo A 2003 Wind turbine wake aerodynamics Progress in Aerospace Sciences 39 467–51
[3] Sørensen J and Shen W 2002 Numerical modeling of wind turbine wakes Journal of Fluids Engineering 124 393-399
[4] Mikkelsen R 2003 Actuator Disc Methods Applied to Wind Turbines Ph.D. Thesis Technical University of Denmark
[5] Hand M, Simms D, Fingersh L J et al. 2001 Unsteady Aerodynamics Experiment Phase VI: Wind Tunnel Test Configurations and Available Data Campaigns NREL/TP-500-29955