ANALYSIS OF HORIZONTAL AXIS WIND TURBINES WITH LIFTING LINE THEORY

Daniela Brito Melo
daniela.brito.melo@tecnico.ulisboa.pt
Instituto Superior Técnico, Universidade de Lisboa, Portugal
December, 2016

ABSTRACT

A method to analyze the flow around horizontal axis wind turbines based on the lifting line theory is herewith presented. A source model that simulates drag is also addressed. It is shown that the continuity and the linearized momentum equations naturally lead to these models. The strong relation between the reliability of the results and a proper alignment of the vortex wake is well known and documented. Hence, a non-linear vortex wake alignment scheme is proposed. Results are presented and compared to the NREL turbine experimental data.

KEYWORDS: wind turbine, lifting line theory, wake alignment, source model.

INTRODUCTION

The complexity of the flow around wind turbines makes it difficult to fully understand all the phenomena involved. Its modelling is therefore challenging and simplifications are naturally made. The models are both analytically and numerically complex, even if one reduces the viscosity to zero and assumes a steady state condition.

Being originally introduced by Lanchester-Prandlt, the lifting line theory has been used to model marine propellers and, more recently, wind turbines. A number of researchers have contributed to the expansion of the theory.

The computational science developments allowed a numerical approach to the theory. Kerwin, in 1978 [1], proposed a lifting line code where the induced velocities were calculated by the Biot-Savart law. An application to axial turbines can be found in [2].

The correct modelling of the rotor wake is of the utmost importance for the quality of numerical results. Rigid or semi-empiric wake models are typically used. Ideally, the vorticity should be aligned with the local velocity field. However, it is not feasible to do it on its full extent. A vortex wake alignment scheme is proposed. The wake geometry is forced to be parallel to the flow in multiple sections downstream of the rotor.

The hub is modelled by an infinite cylinder [3]. A source model [4] is coupled with the lifting line theory in order to simulate stall conditions and calculate the drag force. The analytical formulation is based on the continuity and the linearized momentum equations, following Sparenberg’s thoughts [5].

Finally, the numerical results are compared to the NREL turbine experimental data [6].

MATHEMATICAL MODEL

Consider the rotor of a horizontal axis turbine with radius R, diameter D and with Z blades symmetrically placed around a cylindrical hub of radius rh. The rotor is rotating with constant angular velocity \( \omega \) in a uniform inflow moving with velocity \( \bar{U} \). The fluid is assumed to be inviscid and incompressible. We introduce a cartesian coordinate system \((x,y,z)\) and a cylindrical coordinate system \((\rho,r,\theta)\) in a reference frame rotating with the turbine rotor. The flow is steady in the rotating reference frame and the relative velocity field is given by \( \bar{U}_{rel} = \bar{U} - \omega \times \bar{r} \), as illustrated in Fig. 1.
For formulation of the lifting line theory

In the lifting line model each turbine blade is represented by a radial bound vortex, extending from the root to the blade tip and located at \( \theta_k = 2\pi (k-1)/Z \), \( k = 1, \ldots, Z \). The circulation along the lifting lines continuously varies, \( \Gamma_k(r) = -\gamma(r) \hat{e}_r \), where \( \hat{e}_r \) is the radial unit vector. A vortex sheet of strength \( \gamma \) is shed from each lifting line:

\[
\vec{v}_k(r) = \frac{d\Gamma_k(r)}{dr} \hat{e}_t,
\]

where \( \hat{e}_t \) is a unit vector tangent to the vortex sheet and aligned with the local flow velocity field.

The velocity induced by the \( k \)-th lifting line and corresponding vortex sheet, \( \vec{v}_k \), at a field point \( \vec{x} \) is given by the Biot-Savart law:

\[
\vec{v}_k(\vec{x}) = \frac{1}{4\pi} \left( \int_{L_k} \frac{\vec{v} \times \vec{R}}{R^3} dl + \int_{S_k} \frac{\vec{v} \times \vec{R}}{R^3} dS \right),
\]

where \( L_k \) denotes the \( k \)-th lifting line, \( S_k \) the corresponding vortex sheet, \( \vec{R} \) is the vector radius from the integration point to the field point and \( R \) is its module.

The total induced velocity is obtained by the sum of the blades contributions:

\[
\vec{v}(\vec{x}) = \sum_{k=1}^{Z} \vec{v}_k(\vec{x}).
\]

From symmetry considerations, the velocity induced by the sum of the \( Z \) lifting lines on the lines themselves is zero. As a result, the induced velocity field reduces to:

\[
\vec{v}(\vec{x})_{LL} = \frac{1}{4\pi} \sum_{k=1}^{Z} \frac{\vec{v} \times \vec{R}}{R^3} dS. \tag{4}
\]

At this point, the induced velocity field is function of the radial distribution of circulation and of the vortex sheet geometry.

The total velocity field, \( \vec{V} = \vec{U}_\infty + \vec{v} \), is related to the lift force by the Kutta-Joukowski theorem:

\[
\vec{L} = \rho \vec{V} \times \vec{\Gamma}, \tag{5}
\]

where \( \rho \) is the density of the fluid and \( \vec{L} \) the lift force per unit span.

The blade section lift coefficient, \( C_L \), is defined by:

\[
C_L = \frac{L}{\frac{1}{2} \rho V^2 c} \tag{6}
\]

where \( L = |\vec{L}| \), \( V \) is the module of the projected velocity on the section plane and \( c \) is the airfoil chord.

Linear analysis of the flow induced by a blade

Consider an inertial reference frame \( (x', y', z') \), fixed with the uniform inflow \( \vec{U} \), and a cylindrical coordinate system \( (x', \rho', \theta') \).

Imagine a finite impermeable surface applying a force \( \vec{F} \) on the fluid and hence disturbing the incoming flow. Let \( \vec{v} \) be the induced velocity field which tends to zero at infinity.

In inviscid flow, the equation of motion reduces to the Euler equation. We consider the linearized form of the equation assuming \( \vec{v} \), grad \( \vec{v} \), grad \( p \) and \( \vec{F} \) of \( O(\epsilon) \), where \( \epsilon \) is a small parameter with respect to which we linearize the equation and \( p \) is the pressure. By neglecting quantities of \( O(\epsilon^2) \), we obtain the following expression for the incompressible Euler equation:

\[
\frac{\partial \vec{v}}{\partial t} = -\frac{1}{\rho} \text{grad} \ p + \frac{1}{\rho} \vec{F}, \tag{7}
\]

where \( \vec{F} \) is the body force per unit volume and \( t \) is time. It is assumed that \( \vec{F} \) comes into action at \( t = t_0 \) and that for \( t < t_0 \) the fluid is at rest.
A vector field that tends to zero at infinity is totally defined by its divergence and rotation. The continuity equation for incompressible fluid sets the divergence to zero. An expression for the rotation is obtained by applying the operator rot to both sides of equation (7) and integrating with respect to time:

$$\text{rot} \vec{v} = \int_{t_0}^{t} \frac{1}{\rho} \text{rot} \vec{F} \, dt, \quad (8)$$

where the variable $\tau$ defines the integration in time.

Using the mathematical identity:

$$\nabla^2 \vec{v} = \text{grad} \text{div} \vec{v} - \text{rot} \text{rot} \vec{v}, \quad (9)$$

we explicitly define the velocity field, $\vec{v}$, as a function of the force field, $\vec{F}$.

Let each turbine blade be represented by a lifting surface, $\tilde{B}$. The surface moves within the fluid in the negative direction of $x'x''$, with velocity $U$, and rotates with velocity $\omega$ around $x'x''$, in the negative direction.

Consider an infinite helicoidal surface $H$, with pitch $b = 2\pi U/\omega$, which moves with the flow without disturbing it. Let $\tilde{B}$ be the projection of $B$ on the helicoidal reference surface as illustrated in Fig.2. If $\tilde{B}$ moves in the close neighbourhood of $H$, the force field exerted on $\tilde{B}$ can be considered on $B$. This approximation introduces errors of $O(\varepsilon^2)$, neglected in the linearized theory.

Following the simplifying assumptions of the lifting line theory, the force field acting on $B$ is concentrated on a line $L_b$. At $t = t_0$ the blade starts to move and $L_b$ begins to glide along $H$ with velocity $\vec{U}_\infty(r')$, symmetric of $\vec{U}_\infty$. Let $S(t)$ be the area of $H$ swept by $L_b$ from $t_0$ to a given instant $t$ (Fig.3).

The concentrated force $\vec{F}$, on $L_b$, is defined by:

$$\vec{F}(x', t) = \int (x') \delta(r'\phi(t)) \delta(x' - \chi(t)) \quad (10)$$

where $\delta( )$ is the Dirac delta function, $\phi(t) = -\omega t$ the angular position of the line, $\chi(t) = b\phi(t)/2\pi$ the axial position and $\int$ the force vector.

The velocity field induced by $\vec{F}$ is equivalent to the flow generated by a layer of dipoles over $S(t)$. It is, in fact, a layer of divergenceless dipoles, which has zero divergence everywhere [5]. Consequently, there is a mass transport from the sinks to the sources of the dipoles (Fig.4). The velocity field is also characterized by concentrated vorticity on the dipole layer. The dipoles are aligned with $\vec{f}$ and its strength by unit area is given by $\int (r')/\rho U_\infty(r')$.

As previously stated, the divergence is zero everywhere in the fluid domain and the vorticity is concentrated at the dipoles layer. It is shown in [5] that it is possible to reproduce the flow induced by the force $\vec{F}$ by means of a vorticity consideration.
Consider an infinitesimal area \( dA \) on the dipole layer. In that area we can define a singular dipole with strength \( f(l) \). A vortex of strength \( \Gamma = f(l) dA \) may be defined along the boundary \( \partial A_n \) of an area \( dA_n \), perpendicular to the dipole axis, as illustrated in Fig.5.

![Dipole and corresponding concentrated vortex](adapted from [5]).

When the force vector \( f \) is perpendicular to its velocity, the disturbance induced by the dipole layer is given by a vortex sheet that extends over \( S(t) \), similarly to the lifting line theory.

When the force vector \( f \) is tangent to its velocity, we can also give an impression of the flow induced by \( f \) by means of vortex tubes along \( S(t) \).

If \( f \) is tangent to its velocity and does not depend on the time, the layer of divergenceless dipoles reduces to a source line in the instantaneous location of \( L_b \) and a sink line in the place where \( L_b \) started to move. There is a fluid transport from the sink line to the source line which sets divergence to zero everywhere.

**Source model**

The drag force acting on the blades can be modeled by source lines, coincident with the lifting lines, extending from the root to the blade tip and located at \( \theta_k = 2\pi(k-1)/Z \), \( k = 1, ..., Z \). The source flow rate per unit span continuously varies along the line and is represented by \( m(r) \). For each source line we consider a downstream sink line at \( \theta_k \), with radial coordinate \( r \). A vortex tube with intensity \( Y(r) \) can be considered on that point. If we denote the vortex tube cross section area by \( hdr \), its intensity can be related to \( m \) by the expression:

\[
Y(r) = m(r)/h. \tag{13}
\]

As proposed in [4], \( h \) is defined as the perpendicular distance between \( J_k \) and the stream function of the adjacent source line, \( G_{k+1} \):

\[
h = \frac{2\pi}{Z} \sin\beta_i. \tag{14}
\]
where $\beta_i$ is the hydrodynamic pitch angle.

To compute the induced velocity at the source line one has to take into consideration the $Z$ source lines and the vortex tube:

$$\bar{u}(x)|_{SL} = -\frac{m(r_p)}{2h} \delta_t + \sum_{k=1}^{Z} \int_{\ell_k \cap \phi} \frac{m(l) \cdot \vec{R}}{4\pi R^3} dl \quad (15)$$

where the first term denotes the velocity induced by the vortex tube, calculated with the law of Biot-Savart, and the second term accounts for the velocity induced by all the source lines, excluding the calculation point.

The total velocity, $\bar{V} = \bar{U}_m + \bar{u}$, can be related to the drag force by the Blasius theorem:

$$\bar{D} = \rho \bar{V} m, \quad (16)$$

where $\bar{D}$ is the drag force per unit span.

The blade section drag coefficient, $C_D$, is defined by:

$$C_D = \frac{D}{\frac{1}{2} \rho \bar{V}^2 c} \quad (17)$$

where $D = |\bar{D}|$.

**The combined lifting line and source model**

The total velocity field $\bar{V}$ is defined by the sum of the undisturbed velocity field, $\bar{U}_\infty$, with the velocity induced by the vortex system, $\bar{v}$, and the velocity induced by the source lines and corresponding stream surfaces, $\bar{u}$:

$$\bar{V} = (U + v_x + u_x, v_z + u_r, \omega r + v_\theta + u_\theta), \quad (18)$$

where

$$v_r = v_y \cos \theta + v_z \sin \theta, \quad (19.a)$$

$$v_\theta = -v_y \sin \theta + v_z \cos \theta, \quad (19.b)$$

and $u_r$ and $u_\theta$ are computed in an analogous way.

Let $v_a(r)$, $v_t(r)$, $v_\alpha(r)$ be the module of the axial, spanwise and tangential components of $\bar{v}$ at the lifting line $k = 1$, and, similarly, $u_a(r)$, $u_z(r)$, $u_\alpha(r)$ the module of the axial, spanwise and tangential components of $\bar{u}$ at the same lifting line.

From symmetry, the velocity induced on any other lifting line is identical.

The module of the projected velocity on the section plane, $V$, is defined as:

$$V = \sqrt{(U - v_a - u_a)^2 + (\omega r + v_t - u_t)^2}. \quad (20)$$

The velocity vector and the aerodynamic forces applied on each blade section are illustrated on Fig.7.

![Fig. 7- Velocity triangle and forces on a blade section.](image)

The thrust, $T$, and the torque, $Q$, are obtained by integration along the radius and by the sum of each blade's contribution. The axial force and power coefficients are defined by:

$$C_T = \frac{T}{\frac{1}{2} \rho \bar{U}^2 \pi R^2} \quad (21)$$

$$C_P = \frac{\omega Q}{\frac{1}{2} \rho \bar{U}^3 \pi R^2}. \quad (22)$$

Henceforth, all variables are presented in their dimensionless form. The rotor radius $R$ is the reference length and the uniform inflow velocity $U$ is the reference velocity.

The hydrodynamic pitch angle $\beta_i$ is easily related to the induced velocities (see Fig.7):

$$\tan \beta_i = \frac{1 - v_a - u_a}{\lambda r + v_t - u_t} \quad (23)$$

where $\lambda = \omega R / U$ is the tip speed ratio (TSR).

$\beta_i$ is also related to the section pitch angle, $\psi$:

$$\beta_i = \psi + \alpha, \quad (24)$$

where $\alpha$ is the angle of attack.
Finally, the force and power coefficients may be computed by the following expressions:

$$C_T = \frac{2Z}{\pi} \int_{r_h}^{1} (\lambda r + v_t - u_a) \Gamma \left( 1 + \frac{C_D}{C_L} \tan \beta_i \right) dr,$$  \hspace{1cm} (25)$$

$$C_P = \frac{2Z \lambda}{\pi} \int_{r_h}^{1} (1 - v_a - u_a) \Gamma \left( 1 - \frac{C_D}{C_L} \cot \beta_i \right) r dr,$$  \hspace{1cm} (26)$$

where $C_D/C_L = m/\Gamma$, according to equations (5), (6), (16) and (17).

**NUMERICAL METHOD**

In the vortex lattice model the lifting lines and the shed vortex sheets are discretized by a lattice of concentrated vortices [1]. The lifting lines are discretized in $M$ elements along the radius. On each element, the circulation is assumed to be constant. Then, each element is represented by $\Gamma_i = -\Gamma_i \delta r$, with $\Gamma_i = |\Gamma_i|$, $i = 1, ..., M$. At the elements corner points the circulation is discontinuous and a concentrated vortex is shed with strength $\gamma_j$:

$$\gamma_j = \Gamma_j - \Gamma_{j-1}, \quad j = 2, ..., M$$

$$\gamma_1 = \Gamma_1, \quad \gamma_{M+1} = -\Gamma_M.$$  \hspace{1cm} (27)$$

Similarly, the source lines are discretized in $M$ elements of constant strength, $m_j$.

A uniform distribution defines elements of the same size. A cosine distribution allows a successful numerical convergence [2]:

$$\tilde{r}_i = \frac{1}{2} (1 + r_h) - \frac{1}{2} (1 - r_h) \cos \left( \frac{\pi (i - \frac{1}{2})}{M} \right)$$  \hspace{1cm} (28)$$

$$r_j = \frac{1}{2} (1 + r_h) - \frac{1}{2} (1 - r_h) \cos \left( \frac{\pi (j - \frac{1}{2})}{M} \right).$$  \hspace{1cm} (29)$$

where $\tilde{r}_i$ denotes the dimensionless radial coordinate of the control points, $i = 1, ..., M$, and $r_j$ is the dimensionless radial coordinate of the elements corner, $j = 1, ..., M + 1$.

The velocity induced by the discrete vortex system is given by the discrete form of equations (3) and (4). The trailing vortices are discretized in straight-line segments, as explained in the next section. Hence, the induced velocities are computed with the Biot-Savart law.

The axial and tangential induced velocities calculated at the control points, $\tilde{r}_i$, can be represented by the following expression:

$$v_{a,t} = \sum_{j=1}^{M} C_{a,t,i,j} \Gamma_j$$  \hspace{1cm} (30)$$

Where $C_{a,t}$ are the axial and tangential vortex induced velocity matrices.

The velocity induced by the source lines and corresponding stream surfaces are given by the discrete form of equation (15). The resultant expression can be easily computed analytically. The corresponding axial and tangential induced velocities are expressed by:

$$u_{a,t} = \sum_{j=1}^{M} S_{a,t,i,j} m_j.$$  \hspace{1cm} (31)$$

where $S_{a,t}$ are the axial and tangential source induced velocity matrices.

**Wake model**

The turbine wake model considers a free vortex wake aligned in multiple sections, similarly to the work of Baltazar [7]. The trailing vortices are discretized in $N$ straight-line segments with coordinates:

$$x_2 = x_1 + b_1 \left( 1 + \frac{v_\theta}{\lambda r} \right) \frac{2}{1 + N_\theta}$$  \hspace{1cm} (32)$$

$$\theta_2 = \theta_1 + \left( 1 + \frac{v_\theta}{\lambda r} \right) \frac{2\pi}{1 + N_\theta}.$$  \hspace{1cm} (33)$$

where $N_\theta$ is the number of increments in which we divide a full rotation of the blades and $b = \pi r \tan \beta_i$ the dimensionless wake pitch. The 1 and 2 indexes refer, respectively, to the upstream and downstream edges of each straight-line segment.

The wake expansion is neglected. Hence the radial coordinate of the trailing vortices remain unchanged.
The wake is extended downstream and truncated at a station with axial coordinate \( x_{ew} \). The vortex wake is divided into two parts:

a) From the lifting line to a far wake section, \( x_{fw} \), a transition wake region where variations of pitch and tangential velocities take place;

b) From \( x_{fw} \) to \( x_{ew} \), an ultimate region where the wake parameters remain constant and assume the values of the far wake station.

In the transition wake, \( n \) sections of alignment, characterized by an axial coordinate \( x_a, a = 1, ..., n \), are defined. \( x_{fw} \) is given by the last section of alignment, \( x_n \).

We introduce an index \( \xi, \xi = 1, ..., N + 1 \), that identifies, for each trailing vortex, the consecutive straight-line edges. The axial and tangential coordinates of each edge can be defined as \( x(\xi, r_j) \) and \( \theta(\xi, r_j) \). At each alignment section, the induced velocities at M control points are calculated. The radial coordinates are given by \( \bar{r}_i \). The axial and tangential coordinates of the alignment control points are radially interpolated from \( x(\xi_a, r_j) \) and \( \theta(\xi_a, r_j) \), where \( \xi_a \) is the index that identifies the straight-line edges with an axial coordinate immediately bellow the value of \( x_a \) (Fig. 8).

The induced velocities are computed in the \( n \times M \) alignment points. \( \tan \beta_i \) and \( b \) are easily obtained and interpolated/extrapolated to the radial coordinate of the trailing vortex.

The new wake geometry is generated from equations (32) and (33), considering \( b \) and \( v_\theta \) interpolated from the values obtained in the alignment sections. In the ultimate region, \( b \) and \( v_\theta \) are assumed constant and identical to those obtained at the far wake station.

The turbine hub is modeled by an infinite cylinder [3]. For this purpose an image vortex system is established as proposed in [8]. It is constituted by straight-line vortices with equal axial and tangential coordinates as the trailing vortex and radial coordinates, \( r_v \), given by:

\[
r_v = \frac{r_h^2}{r_j},
\]

***Iterative solution of vortex and source strengths***

It is possible to establish the following system of equations, according to equation (23):

\[
\sum_{j=1}^{M} \Gamma_j \left( C_{aij} + \tan \beta_i C_{\theta ij} + \left( \frac{C_p}{C_i} \right)_i \left[ S_{aij} - \tan \beta_i r_j \right] \right) = 1 - \lambda \bar{r}_i \tan \beta_i \Gamma' \quad i = 1, ..., M.
\]

(35)

The wake geometry, \( \tan \beta_i \) and \( C_p/C_i \) must be prescribed in order to start the iterative process.

The source strengths are then related to \( \Gamma \) by the expression \( C_p/C_i \rightarrow m/\Gamma \).

Having the values of \( \Gamma \) and \( m \) one can compute the induced velocities and a new wake geometry is established.

**RESULTS AND DISCUSSION**

The numerical analysis conditions were set by three upwind NREL/NASA Ames wind tunnel tests. The blades geometry can be found in [9]. The convergence analysis and the wake alignment was performed for a 5.41 TSR and did not take into consideration the velocities induced by the source lines.

**Convergence**

A wake with constant pitch and aligned on the lifting line is considered. The lines are discretized in 20 elements. The power coefficient evolution is shown in Fig. 9, with respect to some values of \( N_\theta \) and \( x_{ew} \). The numerical predictions converge to a certain figure, close to the induction factor method result.
The variation of the power coefficient with the number of elements on the lifting line, $M$, is plotted in Fig. 10, for a uniform and a cosine distribution. The latter required fewer elements to converge. $N_\theta$ and $x_{ew}$ were set as 200 and 10, respectively.

Wake alignment

The wake alignment model was tested for three and two sections. The results are presented for the last ($n = 2$), where the vortices were lined up at $x/R = 0$ (lifting line) and $x/R = 1$.

It was considered $M = 30$, $N_\theta = 200$ and $x_{ew} = 10$.

The radial distributions of the dimensionless pitch and tangential induced velocity on the alignment sections are presented in Fig. 11. It is possible to conclude that the aligned wake and a constant pitch wake, aligned at the lifting line ($n = 1$), are significantly different.

The effect of the alignment on the circulation is shown in Fig. 12. The power coefficient increases as the circulation considerably rises (Tab. 1).

The $n = 3$ wake, aligned at $x/R = 0$, $x/R = 1$ and $x/R = 2$, and the $n = 2$ wake produced similar results. The radial distribution of pitch and tangential induced velocity on $x/R = 2$ were, in fact, very close to the distributions registered on section $x/R = 1$. The power coefficient decreased 0.007% with the extra alignment section.

<table>
<thead>
<tr>
<th>Wake</th>
<th>$C_p$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant pitch ($n = 1$)</td>
<td>0.2790</td>
</tr>
<tr>
<td>Aligned on $x/R = 0$ and $x/R = 1$ ($n = 2$)</td>
<td>0.3332</td>
</tr>
</tbody>
</table>

Tab. 1 – Power coefficient, $C_p$, for a wake of constant pitch ($n = 1$) and a wake aligned in two sections ($n = 2$).
Source model

The source model was studied for three different values of TSR = \{2.92, 3.8, 5.41\}. The vortex induced velocities were computed with the induction factor method and the hub effect was not considered. The lifting line was once again discretized in M = 30 elements.

The results for TSR = 5.41 did not change significantly with the source model.

As expected, the circulation decreases for lower values of TSR. This behaviour is related to an increase of the angle of attack (Fig.13). For TSR = 2.92, the predicted 2D flow is stalled from the root to the half span.

Some oscillations are noticeable between 50% and 70% of the blades span. It happens for a range of angles of attack close to the onset of stall.

Comparison with experimental data

The numerical predictions of the power coefficient are compared with the experimental measurements presented in [6]. The experimental data are plotted in Fig.14 with the respective standard deviation. For TSR = 5.41 the result for the two sections wake alignment (n = 2) is also shown.

The power coefficient is strongly correlated to the experimental data, for the three tested TSRs. For an aligned wake (TSR = 5.41) it is contained within the experimental uncertainty interval.

CONCLUSIONS

A method to analyze the flow around horizontal axis wind turbines is presented.

The lift and drag forces are simulated by the means of an inviscid fluid model that couples the
lifting line theory with a source model. The continuity and the linearized momentum equations are the foundation of an analytical development that naturally leads to these models.

A non-linear vortex wake alignment scheme is also proposed. The results tend to a certain value when increasing the wake and the lifting line discretization. The power coefficient increases when one aligns the vortex wake in downstream sections.

The numerical results are compared to the NREL turbine experimental data. The lifting line theory coupled with the non-linear vortex wake alignment scheme successfully predicted the power coefficient. The source model allows the analysis in stall conditions and effectively calculated the power coefficient decreasing trend.

REFERENCES


