A Hybrid CFD/BEM Analysis of Flow Field around Wind Turbines

V. Esfahanian¹*, A. Salavatipour¹†, I. Harsini²‡, A. Haghani³§, R. Pasandeh¹¶, and G. Ahmadi⁴∥

¹ Department of Mechanical Engineering, University of Tehran, Tehran, Iran
² Department of Mechanical Engineering, Karaj Branch, Islamic Azad University, Karaj, Iran
³ Renewable Energy Organization of Iran, Tehran, Iran
⁴ Department of Mechanical and Aeronautical Engineering, Clarkson University, Potsdam, NY, USA

Email: evahid@ut.ac.ir

ABSTRACT

In this study, a mixed CFD (Computational Fluid Dynamics) and BEM (Blade Element Momentum Method) analysis is implemented for simulating the flow field around a wind turbine rotor to predict its aerodynamic performance such as the Power Curve diagram and the forces and moments imposed on the rotor blades that are essential in structure or aeroelastic design. The present approach requires considerable less computational time and memory than three-dimensional simulation of a wind turbine rotor by merely CFD methods. This work consists of two parts: 1- Calculating 2D aerodynamic coefficients of several spanwise sections of the blades by CFD methods, using Fluent commercial software. 2- Simulating 3D flow field through the wind turbine rotor using BEM technique. To validate the current approach, the Combined Experiment Phase II Horizontal Axis Wind Turbine known as NREL Phase II Rotor [1] is used. The numerical results indicate good accuracy with those of experiments.

1 INTRODUCTION

At the present, many countries are increasingly investing in green energy, while the wind is one of the most important sources of the free and clean energies. The kinetic energy of the wind can be absorbed and turned into electrical energy by use of the wind turbines. There are many types of wind turbines but they can be categorized into two general kinds: Horizontal and Vertical Axis Wind Turbines (HAWT and VAWT). However because of the lower efficiency and structural problems, the VAWTs are rarely used for power generation.

For designing a wind turbine, it is of high importance to accurately predict the imposed aerodynamic forces and moments on the structure. These forces are used in aeroelastic simulation and structural design and also in predicting the Power Curve of the wind turbine.

One of the most common ways for predicting these forces is simulating the whole flow field around the turbine by Computational Fluid Dynamics (CFD). Such a CFD simulation is very costly and requires rather a long time. Also there are some engineering methods such as Blade Element Momentum method (BEM) for simulating a rotor that are much faster than CFD methods. But on the other hand, some accuracy is lost due to inviscid approximation used in these methods.

Due to large time requirement and computational cost of simulating a 3D wind turbine by merely CFD methods, a mixed BEM/CFD method is investigated for simulating the flow field around HAWTs which uses CFD for constructing the table of aerodynamic coefficients at different angles of attack and Reynolds numbers. Then by use of the tabulated coefficients, three-dimensional flow field is modeled via BEM method. Then the imposed torque and forces on turbine blades can be calculated.

To validate the implemented method, the NREL Phase II Rotor is chosen for this analysis. This rotor has become a benchmark in wind industry for ver-
ification of different engineering methods. The experimental data came from measurements taken on field installed wind turbine. The joint effort undertaken by several European Union research labs and the United States National Renewable Energy Laboratory (NREL) has documented and made available experimental field data for several wind turbines [1]. Numerical analysts have access to this data, known as IEA Annex XIV by either extracting it from written report or by downloading the electronic version of the report from the internet [2].

2 EXPERIMENTAL TEST CASE

The experimental data used for comparison, is obtained from the NREL Combined Experiments Phase II [3, 4].

This turbine is a fixed pitch, 3 bladed Horizontal Axis Wind Turbine (HAWT) that has a rated electrical power of 19.8 kW. The downwind rotor has a diameter of 10.06 m, hub height of 17.03 m, cut-in wind speed of 6 m/s, zero tilt angle and rotates at constant rotational speed of 71.63 rpm. Blades are untwisted with constant chord length of 0.4572 m and uses S809 airfoil as the cross section along the spanwise direction with some modifications towards the root to blend with the hub spar. Blade set angle (pitch angle) was 12 degrees according to settings of NREL Phase II Rotor experiments.

The present test case is a highly instrumented benchmark. Each blade has flush mounted pressure taps in several radial locations; also total pressure is recorded by pressure probes at four radial stations. The rotor bending loads, torque and angle of attack are also measured in several spanwise stations.

For non-yawed rotor and steady (time averaged) conditions, NREL provides the power curve of the turbine as shown in Figure 1.

3 CALCULATING 2D AERODYNAMIC COEFFICIENTS

Several wind tunnel tests were carried out for S809 airfoil at the Delft University of Technology (DUT), Colorado State University (CSU) and Ohio State University (OSU). However there are some level of discrepancy and difference between the three wind tunnel results [5], especially in post-stall region. Such a discrepancy motivated us to derive the aerodynamic coefficients of the airfoil by CFD simulation.
Enhanced Wall Treatment (EWT).

Two-dimensional computational mesh is a structured multi-blocks C-type grid (figures 2 and 3), that contained totally 23625 cells.

Many numerical analyses has been conducted on S809 airfoil with different numerical approach and different turbulence modeling, i.e. Wolfe and Ochs [6], Zhang et al [9] and Bertagnolio et al [10]. The most important point that can be found in these analyses is the failure or partial failure of the simulations that neglected the phenomenon of transition, in prediction of aerodynamic coefficients (specially the drag coefficient). Therefore, the most essential point in 2D-aerodynamic simulation of each sections of the blade is the determination of the location of transition for each side of blade section.

According to experiments performed in Delft University of Technology, the chord wise location that transition occurs has been determined as a function of angle of attack, in different Reynolds numbers [12].

As it is illustrated in figures 4 and 5, the transition on lower side of the airfoil happens in about 52% of the chord length from the leading edge in a wide range of angles of attack. Similarly on the upper surface, when the AOA ≤ 5 degrees, natural transition occurs in 56% of the chord length from the leading edge. But as the AOA increases and the separated region on the upper side expands, transition occurs with separation and moves towards the leading edge of the airfoil, where it asymptotically approaches to averagely 4% of the chord length after the leading edge.

In order to model the laminar zone before the transition
and the turbulent zone after that, the computational domain is decomposed into three regions by three normal lines to the airfoil surface according to the locations that transition occurs in each side of the blade section. These lines are set as interior lines in Gambit. The three separated zones are illustrated in Figure 6. In upper side of the airfoil, it is assumed that the Laminar/Turbulent zone is laminar when $\text{AOA} \leq 7^\circ$, then it becomes turbulent in higher angles of attack.

3.2 Two-Dimensional Numerical Approach

Two-dimensional numerical simulations are performed by use of Fluent commercial software that uses Finite Volume Method (FVM) in CFD calculations. A density based, time dependent solver with implicit formulation and Roe-FDS type of flux calculation is chosen.

The $\text{SST} k - \omega$ turbulence of Menter is used in turbulent zones due to good prediction in separated flow simulation. The model uses the standard $k - \omega$ model near the wall, but switches to a $k - \varepsilon$ model away from the wall [11].

Second order upwind discretization is chosen for momentum equations and turbulence equations including specific dissipation rate and turbulent kinetic energy equations are discretized using first order upwind scheme.

Due to expansion of the wake region and increase in vortex shedding as the angle of attack increases, time dependency and unsteadiness of the numerical solution will also tends to increase. This phenomenon causes the aerodynamic coefficients to oscillate in time. Therefore, the time average of the oscillating quantities are determined, in order to have specific and constant lift and drag coefficients at each Reynolds number and angle of attack.

3.3 Two-Dimensional Results

Two-dimensional numerical simulations are performed in 4 different Reynolds number ($\text{Re} = 0.25 \times 10^6, 0.5 \times 10^6, 1.0 \times 10^6$ and $1.5 \times 10^6$) in a wide range of angles of attack ($-5^\circ \leq \text{AOA} \leq 40^\circ$) in steps of one degree. These simulations are used to construct the lift and drag polars for S809 airfoil that are illustrated in figures 7 and 8.

Also 4 other numerical simulations are investigated at Reynolds number of $2 \times 10^6$, and in four angles of attack: $-0.01^\circ$, $1.02^\circ$, $5.13^\circ$ and $9.22^\circ$, in order to verify the calculated pressure distribution around the airfoil with DUT low-speed wind tunnel experiments [12] (figures 13 to 16).

Two-dimensional numerical results are validated against wind tunnel tests at Delft, Ohio State and Colorado State Universities (figures 9 to 16).

As illustrated in figures 9 to 12, the numerically calculated aerodynamic coefficients stand in good agreement with experimental wind tunnel results. Although as shown in figures 9 and 10, the OSU and DUT experiments are inconsistent especially in $\text{AOA} \geq 15^\circ$. Also the numerically calculated pressure coefficients show great accordance with DUT experiments.
Figure 8: Numerical lift polar for S809 airfoil

Figure 11: Numerical lift curve in comparison with CSU wind tunnel tests at $Re = 0.5 \times 10^6$

Figure 9: Numerical lift curve in comparison with DUT and OSU wind tunnel tests at $Re = 1.0 \times 10^6$

Figure 12: Numerical pressure drag curve in comparison with CSU wind tunnel tests at $Re = 0.5 \times 10^6$

Figure 10: Numerical drag curve in comparison with DUT and OSU wind tunnel tests at $Re = 1.0 \times 10^6$

Figure 13: Pressure distribution around S809 airfoil at AOA $= -0.01^\circ$, $Re = 2 \times 10^6$
4 Three-Dimensional Simulation of the Flow Field

A numerical code is developed for converting the 2D calculated and tabulated aerodynamic coefficients into three-dimensional results, in order to simulate the flow field through the rotor. This code uses the classical Blade Element Momentum (BEM) method with several corrections and modifications.

4.1 Classical BEM Method

BEM method couples the one-dimensional momentum analysis for an ideal wind turbine with the geometrical and aerodynamic parameters of the blades. The ideal wind turbine is assumed to have a permeable disk instead of rotor that is frictionless and implies no rotational velocity component in the wake [8]. These assumptions are then corrected with some modifications to this theory.

In BEM method the blade is divided into some radial stations, and then all the computations are carried at each section separately. In this study each blade is decomposed to 39 radial sections by the following cosine distribution:

\[ r_i = \frac{R}{2} \left( 1 + \cos \gamma_i \right) \]  

In the last expression, \( \gamma_i \) varies from 0° (Tip of the blade) to 152° (15% of the blade inboard where it connects to its shaft) in steps of 4 degrees.

According to BEM theory the angle between the plane of rotation and the relative velocity (Figure 17) at each section is computed in order to find the local effective angle of attack as follows:

\[ \tan(\phi) = \frac{(1-a) V_0}{(1+a') r \omega} \]  

\[ \alpha = \phi - \theta \]

Then the normal and tangential force coefficients (normalized with respect to \( 1/2 \rho V_{rel}^2 c \)) are found by projecting the lift and drag into the normal and parallel directions to the rotor plane:

\[ C_n = C_l \cdot \cos \phi + C_d \cdot \sin \phi \]  

\[ C_t = C_l \cdot \sin \phi - C_d \cdot \cos \phi \]
By use of the computed coefficients, the axial and tangential induction factors can be found as follows:

\[ a = \frac{1}{4\sin^2\phi/(\sigma C_n) + 1} \]  

(6)

\[ a' = \frac{1}{4\sin\phi\cos\phi/(\sigma C_t) - 1} \]  

(7)

where, \( \sigma \) is the solidity defined as the fraction of the annular area in the control volume which is covered by blades and \( N \) is the number of the blades.

\[ \sigma = \frac{c(r) \cdot N}{2\pi r} \]  

(8)

Again the effective angle of attack can be computed by these induction factors and the procedure follows this iterative trend until reaching a certain limit of numerical error.

### 4.2 Modifications to BEM Method

The BEM method that is used here is subjected to three different corrections including:

1- Prandtl’s tip loss factor that corrects the assumption of infinite number of the blades in a permeable disk and considers the influence of the vortices shed from the blade tips into the wake [8]. Prandtl introduced a correction factor to equations (6) and (7) as follows:

\[ a = \frac{1}{4F\sin^2\phi/(\sigma C_n) + 1} \]  

(9)

\[ a' = \frac{1}{4F\sin\phi\cos\phi/(\sigma C_t) - 1} \]  

(10)

where, \( F \) is defined in the following form:

\[ F = \frac{2}{\pi} \cos^{-1}(e') \]  

(11)

2- Second modification is the Spera’s correction for large values of axial induction factors [13], where the simple momentum theory breaks down. Spera presented the following formulation for calculating the thrust coefficient:

\[ C_T = \begin{cases} 
\frac{4a(1-a)F}{a^2 + a(1 - 2a_c)} & a \leq a_c \\
\frac{4F}{a^2 + a(1 - 2a_c)} & a > a_c 
\end{cases} \]  

(13)

In the above expression, \( a_c \) is approximately 0.2.

By equating the theoretical thrust coefficient with the Spera’s empirical formula, the axial induction factor for \( a > a_c \) will be corrected as follows:

\[ a = 1 + \frac{K}{2} (1 - 2a_c) - \sqrt{\frac{K(1 - 2a_c) + 2}{4(Ka_c^2 - 1)}} \]  

(14)

where, \( K = \frac{4F\sin^2\phi}{(\sigma C_n)} \).

3- When the flow begins to separate on a rotating wing; the Coriolis force in the spanwise direction becomes significant. So it tends to postpone the stall occurrence. To take into account the blade stall behavior, the numerical BEM code utilized the Du-Selig stall delay model. Du and Selig [15] developed a stall delay model based on the 3D integral boundary-layer analysis to determine the effects of rotation in boundary layer separation. This model is described in Ref. [15].

### 4.3 Three-Dimensional Results

The power curve of the test turbine is obtained numerically using the described method with and without stall delay model. These results as illustrated in Figure 18 are compared against the NREL phase II experimental power curve.

BEM results without stall delay model show considerable difference with experimental power curve as the rotor approaches stall speeds. However, when the Du-Selig stall delay model is enabled, the numerical power curve becomes very close to the experimental NREL power curve. Du-Selig stall delay model slightly over predicts the power of the rotor at wind speeds below 17 m/sec and under predicts the power at higher free stream wind speeds. But this deviation is less than 9% at worst condition. Also it must be mentioned that the presented method is highly dependent on the tabulated 2D aerodynamic coefficients.
This method requires reasonably less computational resources and time while presents good accuracy in comparison with full three-dimensional CFD modeling.

Figure 18: Power curve of the NREL Phase II rotor

ACKNOWLEDGEMENTS

The authors would like to thank Vehicle, Fuel and Environment Research Institute (VFERI) of university of Tehran and Iranian Organization of Renewable Energy (SANA) for the support of the present work.

REFERENCES


