Sindh Univ. Res. Jour. (Sci. Ser.) Vol.44 (4) 695-698 (2012)

SINDH UNIVERSITY RESEARCH JOURNAL (SCIENCE SERIES)

Modelling and Simulation of flow around Wind Turbine Blade by using Actuator Surface Model

A. A. MEMON, H.U. ABBASI, A. F. ABBASI*, M. ASAD **

Department of Energy and Environment, Quaid-e-Awam University of Engineering, Science and Technology, Nawbshah

Received 13th February 2012 and Revised 12th August 2012

Abstract: This paper presents the simulation of two dimensional flows around a wind turbine blade by using the improved hybrid model. Modeling of a flow field in wind farm requires prohibitive computational cost due to complex geometry of wind turbine blades. The equivalent simplified models replace the real geometry of wind turbine blades in order to represent flow field with significantly reduced computational cost. The modeling method combines blade element approach for forces calculation and Navier-Stokes equations for the flow field simulation. Blades are replaced with surfaces, where pressure discontinuity is applied which represent normal forces, and source terms are applied in the vicinity of surfaces which represent tangential forces. The improved model results of flow field are then compared with existing models.

Keywords: Wind turbine, CFD, Hybrid model, Actuator surface model.

1. INTRODUCTION

In the wind turbine park, the impact of wind speed fluctuation due to extraction of kinetic energy from the wind has a significant impact on the flow dynamics affecting the subsequent wind turbine machines downstream. Aerodynamic interaction between the wind turbines installed in the wind farm creates momentum deficit which decreases the energy production due to increased mean velocity shear, whereas turbulence increase causes the fluctuations in dynamic loads due to unsteady flow and causing material fatigue on the subsequent wind turbines found in the wake, thus decreasing the blade life. Average power losses due to reduced wind speed are estimated to reach up to 23% of total power output in large offshore wind farms (Vermeer *et al,* 2003, Lange *et al,* 2003), when compared with wind turbines positioned optimally. Thus, aerodynamic interaction of wind turbines installed in a park requires a careful study of loads and flow field for optimum wind energy exploitation. Several factors can affect the optimum performance of wind turbine which includes terrain topography, wind direction and its varying speed etc.

Due to prohibitive computational cost to simulate flow field around a real geometry of a wind turbine, it's convenient to develop a simplified geometry to represent flow around a wind turbine. The models, with simplified geometry, are widely known as hybrid models, i.e. actuator disk model, actuator line model, and actuator surface model, details of each model can respectively be viewed in (Sørensen *et al,* 1992, Watters *et al,* 2009 and Dobrev *et al,* 2007). These hybrid models adopted the blade element method for aerodynamic forces calculation and a Navier-Stokes solver for flow field simulations.

The actuator disc model replaces the wind turbine rotor with a disc, which represents the axisymetric flow, where pressure discontinuity or term sources are applied as stated by (Ammara, *et al,* 2002); this model does not represent the blades individually. To represent each blade correctly (Sørensen *et al,* (2002) proposed a actuator line model, where forces are distributed radially as source terms. But, this model does not correctly represent the blade sections. To represent the wind turbine blade sections correctly, the actuator surface model proposed by (Dobrev *et al,* 2007), where the chord of airfoil varies from root to tip of the blade, which resembles the reality. In previous research work, actuator surface model has been modeled only by representing lift forces as pressure discontinuity; whereas source terms which represent drag forces are neglected with the assumption that these forces are very small and could be ignored. This paper is aimed to improve actuator surface model by taking into account aerodynamic forces of lift and drag, in order to show the influence of drag forces to improve the wind blade representation.

2. METHODOLOGY

Several numerical methods are in practice to calculate aerodynamic loads on the wind turbine blades, (Memon 2012). The blade element method of (Glauert 1935), due to its computational rapidity and simplicity is a popular design code and frequently used for

++Corresponding author: A. A. MEMON,
* Department of Mechanical Engineering Mehran University of Engineering and Technology, Jamshoro
**Center of Excellence in Advanced Studies and Technologies (CESAT), Islamabad

simulations. In this method, the blade is divided into small fractions (elements), and by summing the aerodynamics forces on a blade, the wind turbine behavior can be predicted. The method used two-dimensional tabular data of either detailed complete simulation or results of the experiments. The coefficients of lift and drag are obtained from two dimensional data in order to calculate lift and drag forces:

$$L = \frac{C_l \rho v^2 c}{2} \quad (1)$$

$$D = \frac{C_d \rho v^2 c}{2} \quad (2)$$

C is the cord of airfoil, v is free stream velocity, Cl and Cd are aerodynamic characteristics of section. For development of the hybrid model, blade element method coupled with a Navier-Stokes solver. The flow around the wind turbine is governed by 3D incompressible Navier-Stokes equations:

$$(\vec{V}\nabla)\vec{V} + \frac{\partial \vec{V}}{\partial t} = \vec{f} - \frac{1}{\rho} grad\, p + \nu \Delta \vec{V} \quad (3)$$

The flow around the wind turbine blade can be considered incompressible as the Mach number does not exceed 0.3. The real geometry is replaced with surfaces in the CFD domain. The mesh is generated by using Gambit 2.4 software, and simulations are carried out using ANSYS Fluent 6.3. The boundary condition of 'fan' to impose pressure discontinuity to represent lift forces is used, and term sources are applied in the vicinity of a surface to represent drag forces. There does not exist the rigid surfaces, as the aerodynamic forces replace them, hence boundary layer need not be modeled. This type of modeling reduces significantly the number of nodes; consequently the computational cost of the flow field around a wind turbine is decreased. In order to assess the efficiency of proposed model, a comparison with existing models vis-à-vis actuator line and actuator surface model is presented.

3. Actuator surface model

In this modeling, surfaces of pressure discontinuity represent the real blades of a wind turbine, and appropriate boundary conditions are applied on those surfaces. As, there does not exist rigid bodies, so computational cost is reduced and boundary layer is not modeled. Normally, the actuator surface is modeled using only the blade normal forces, whereas blade tangential forces are neglected, because of low intensity. In fact, for some angles of attack, tangential forces are cannot be neglected, this research work focuses on the application of tangential forces and these forces are introduced as body forces in the close vicinity of the surface.

This modelling is carried out using CFD solver for flow calculation, and blade element theory for application of aerodynamic forces. The solution is iterative and the data is exchanged between CFD solver and the module of blade element theory. Given initial conditions, blade element theory calculates the pressure distribution on the model, and CFD solver calculates the flow field, having the pressure distribution previously calculated.

In order to show the efficiency of a proposed model, the numerical results are compared with the flow field of actuator line and existing actuator surface, details on each model can be viewed in (Sorensen *et al*, 2002 and Dobrev *et al*, 2007) respectively.

Tangential and normal forces are respectively calculated as follows:

$$dF_t = \frac{1}{2}\rho V_\infty^2 c C_t dr \quad (4)$$

$$dF_n = \frac{1}{2}\rho V_\infty^2 c C_n dr \quad (5)$$

where, V_∞ , c, C_n and C_t are the upstream velocity, cord of the blade section, normal coefficient and tangential coefficients respectively.

In order to compare the efficiency of the proposed model with the existing models, the new actuator surface model is compared with simulated results of actuator line with applications aerodynamic forces as source terms at ¼ of the cord, and existing actuator surface model with application of pressure discontinuity.

4. Simulation of airfoil S809

The airfoil section considered is S809, which is especially designed for wind turbine blades (Sommers, 1994), as shown in Fig 1. This airfoil is selected because of several experimental data availability. The simulation of S809 with complete geometry will serve two purposes: to provide the aerodynamics forces, and to serve the reference to compare it with hybrid models.

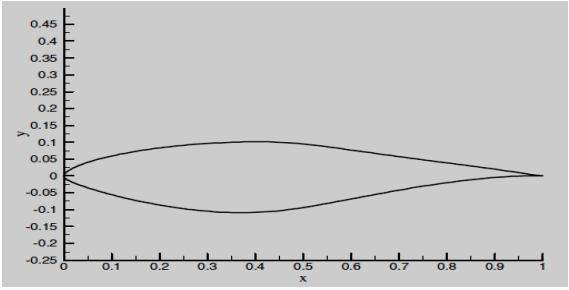


Fig.1 NREL airfoil S809 (Sommers 1994)

The structured H-Type mesh is generated for the complete geometry of an airfoil and equivalent model contains 100 000 grid points. The same mesh is created for the hybrid model in order to compare the flow field results of proposed model with a complete simulation of an airfoil. A non-uniform meshing has been adopted, which is denser or fine near the airfoil surface and coarser away from the airfoil in the flow region as shown in **(Fig.2 and Fig. 3)**.

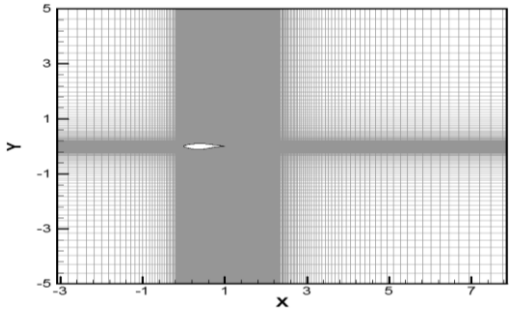


Fig.2 Non-uniform mesh of S809 airfoil

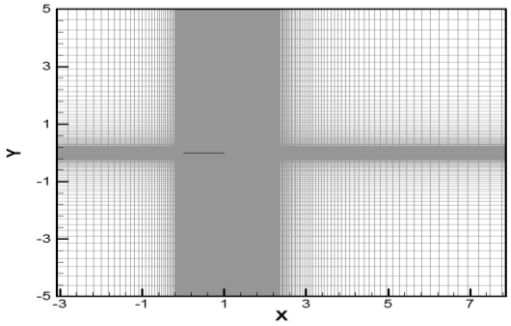


Fig. 3 Non-uniform mesh of hybrid model

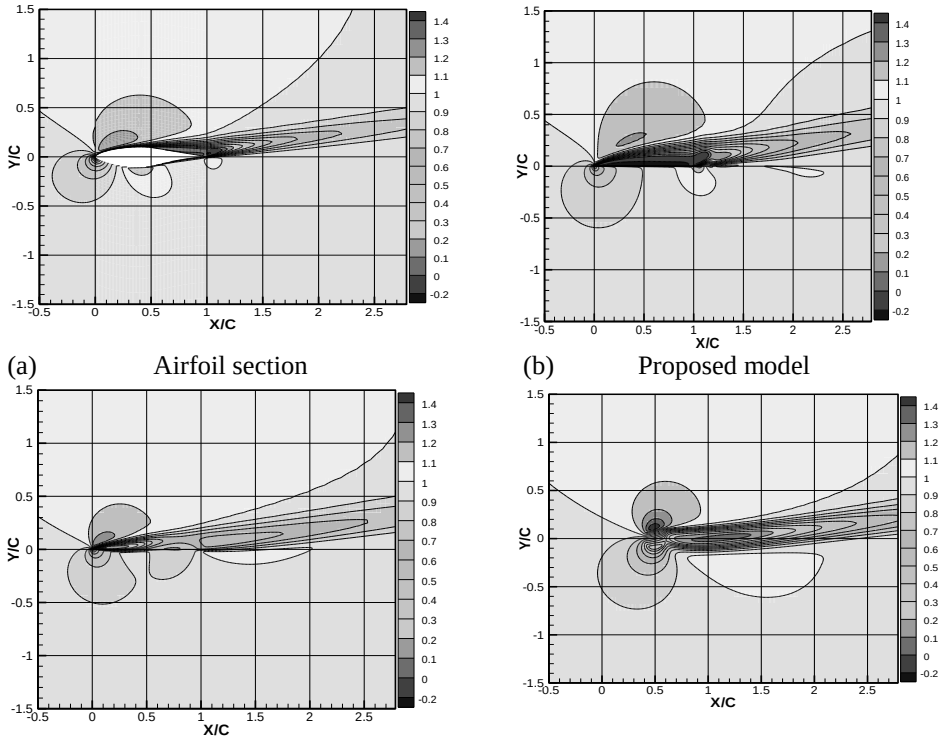
The inlet velocity is fixed to be 10 m/s, and turbulence model used is k-ω of which intensity is fixed 0.5%. This model is mostly used in wind turbine flow applications (Sanderse *et al,* 2011); secondly this turbulence model is well adapted for RANS (Reynolds's average Navier-Stokes) calculations.

5. Validation of Numerical simulation of S809

The computational results of lift coefficients of airfoil S809 are compared with the experimental results of Delft University wind tunnel data in order to validate the simulation results.

6. RESULTS AND DISCUSSIONS

To validate the proposed model of actuator surface, the simulation results are compared with the full geometry of airfoil section as shown in **(Fig.4 (a),** proposed model **(Fig.4(b),** existing actuator surface model **(Fig.4 (c),** and actuator line model **(Fig.4 (d).** The proposed model is shown for three angles of attack, including angle of attacks beyond the stall angle.



(a) Airfoil section (b) Proposed model

(c) existing model (d) Actuator line model

Fig. 4 Flow field at 10° angle of attack

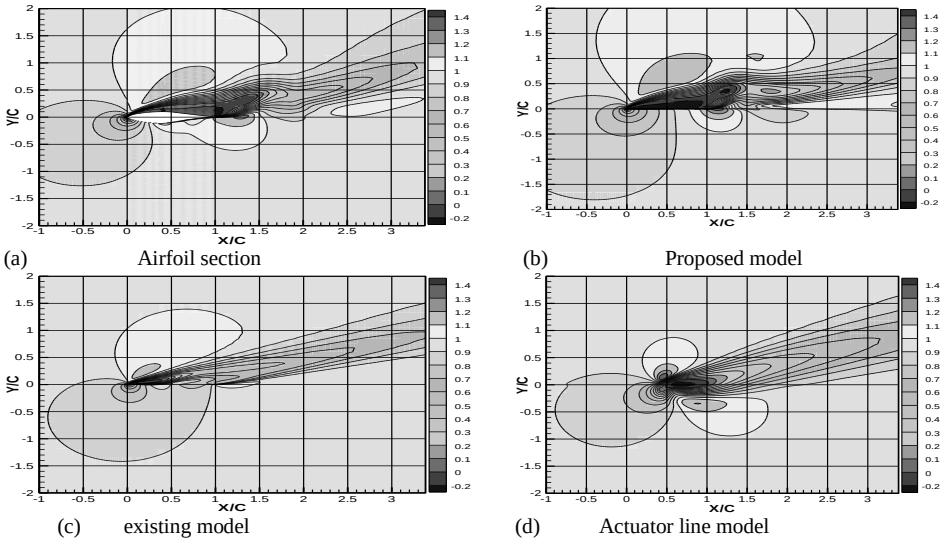


Fig. 5 Flow field at 30° angle of attack

Globally, the results show the best reproduction of flow field with proposed actuator surface model for all simulated angles of attack. Fig. 4 shows the comparative simulated results, where the flow velocity is higher on suction side, and lower on pressure side in the case of full geometry. The velocity gradients resemble more accurately with a proposed model, in this work. Therefore, higher angle of attack of 30° **(Fig. 5 (ab)** and comparable with full geometry simulation **(Fig. 5 (cd)**

7. CONCLUSIONS

New technique of modeling actuator surface is proposed for wind farm simulation. The proposed model has been used to demonstrate the efficiency to reproduce the flow field around an airfoil section of S809, which is especially designed for wind turbine applications. This applied research work will help to simulate easily wind farms in order to contribute towards environmental protection from pollutant exhaust gases from conventional sources of energy extraction processes.

The proposed approach replaces the wind blades with surfaces of pressure discontinuity in addition to body forces to represent respectively normal and tangential forces acting on the wind turbine blades. The computation cost is significantly reduced as the real geometry is replaced with its equivalent forces on flow; therefore boundary layer is not modeled. The proposed model is tested for several and even higher angles of attack, which is crucial for wind turbine applications. The results are quite satisfactory with proposed approach, where as the existing actuator models are unable to demonstrate the capabilities to represent the flow field of a real wind turbine.

REFERENCES:

Ammara, I., and C. Leclerc, (2002) "A Viscous Three-Dimensional Differential/Actuator-Disk Method for the Aerodynamic Analysis of Wind Farms", Journal of Solar Energy Engineering , **(124)**: 345-356.

Dobrev, I., F. Massouh, and M. Rapin (2007) "Actuator Surface Hybrid Model", Journal of Physics Conference Series, Vol. **(75)**: 77-82.

Glauert, H., (1935) "Aerodynamic theory", Vol. **(4):** Berlin, Germany, Julius Springer, 169–360.

Lange, B., H. P. Waldl, and A. Guerrero, (2003) "Modelling of Offshore Wind Turbine Wakes with the Wind Farm Program FLaP". Wind Energy 6 **(1)**: 87–104.

Memon, A.A, (2012) "Développement d'un modèle de surface active pour améliorerla représentation" des charges aérodynamiques sur une pale éolienne, 165Pp.

Sommers D., (1994) "Design and Experimental Results for S809 Airfoil", NREL/SR-440-6918.

Sørensen J.N. (1992) "Unsteady actuator disc model for horizontal axis wind turbines". J of Wind Engineering & Industrial Aerodynamics, **(39):** 139-49.

Sørensen J. N. and W. Z. Shen (2002) "Numerical modeling of wind turbine wakes", Journal of Fluids Eng. (Transactions of the ASME), 124 **(2)**: 393–399.

Sanderse B.S.P., and B. Koren (2011) "Review of computational fluid dynamics for wind turbine wake aerodynamic", J of Wind Energy**,** Vol. 14 **(7)**: 799–819.

Watters C. S. and C. Masson (2009) "Modelling of lifting-device aerodynamics using the actuator surface concept", International journal for numerical methods in fluids, 62 **(11):** 1264-1298.