Study of Wind Turbine Aerodynamic Performance Using Numerical Methods

S. Derakhshan and A. Tavaziani

Abstract—Design of wind turbine blades strictly depends on high precision, reliable and robust numerical predictions of its performance in all of operation conditions. This paper aims to simulate the flow around horizontal wind turbine blade with Computational Flow Dynamics (CFD) using a validated 3D Navier-Stokes flow solver. The main objectives of this study are investigating of different turbulence models and aerodynamic performance of wind turbine blades. The NREL Phase VI rotor used for CFD simulations and testing. Three different turbulence models included of Spalart- Allmaras, k-epsilon (Launder Sharma) and k-w SST tested and the best model for prediction of wind turbine performance is provided. Since Mach number is less than 0.3, the flow around wind turbine blade is incompressible and precondition used. For all cases the structure grid used for Fluid reticulation grid. For results more accuracy, use of preconditioning is necessary. Outputs of flow solver are t power and pressure coefficients for each section. At the final the k-epsilon with preconditioning code is selected the best model for simulation of flow around wind turbine blades.

Index Terms—Computational fluid dynamics (CFD), horizontal wind turbine (HWT), aerodynamic design, turbulence models.

I. INTRODUCTION

CFD simulations have become an important part of applied projects. In turbo machines, imprecise solving of the flow field around the turbine causes incorrect calculation of the forces acting on the turbine and thus results economic losses.

Many methods proposed and applied for turbulent simulation in analysis of flow around turbo machines in computational flow dynamics. One of the most popular method for this object are Reynolds Averaged Navier- Stokes (RANS).

To simulate various problems, some models have a high degree of accuracy of predictions (e.g. separated flow, separation point, flow reattachment and other characteristics of flow) and one possible way for introducing the appropriate turbulence model is sensitivity analysis.

The main objectives of this study are investigating of different turbulence models and aerodynamic performance of wind turbine blades. The NREL Phase VI rotor is the test case for CFD simulations.

There are many approaches for flow simulations around wind turbine blade. The blade element momentum theory (BEM) is used due to its reliability and fastness in 2D simulations.

Limitation of this approach is that it's weak in modeling of

three dimensional and rotational effects. The other limitation of this method is inability to simulate helical tip vortices and important characteristics of flow.

On the other hand, BEM method for calculate loads on each airfoil need to lift and drag verses angle of attach.

There is another method for analysis of flow used in design of wind turbine blades. Best example of these methods is Computational fluid dynamics (CFD) methods that has most accurate solving.

CFD methods solve the governing fluid dynamics equations of turbomachinery. Computational fluid dynamics (CFD) methods don't need to the data of lift and drag coefficients and able to simulate 3D dimensional flow fields. Major limitation of Computational fluid dynamics (CFD) methods is that computational time is more than the other approaches. CFD simulations studied in some projects.

Duque *et al.* [1] compared the accuracy of three approach include Blade element momentum method, vortex lattice method and Navier- Stokes simulations on phase II wind turbine in their research. Aerodynamic performance of Horizontal Axis Wind Turbine (HAWT) was investigated by Sorensen *et al.* [2] using Ellipsis code.

Zahle *et al.* [3] investigated unsteady effects of tower on blade. This work was done on National Renewable Energy Laboratory (NREL) Phase VI wind turbine and the solver was simulated 3D Navier- Stokes.

Postdam et al. [4] generated difference mesh densities and compared unstructured and structured meshes using 3D Navier- Stokes CFD simulation. In PHD thesis of Lynch [5], many unstructured meshes using CFD simulations were developed. Lanzafame et al. [6] simulated phase VI wind turbine with CFD and Ming-Chen [7] investigated aerodynamic performance of this turbine with finite element approach. In this paper, different models for study of Phase VI Horizontal Axis Wind Turbine (HAWT) aerodynamic performance used and different wind speeds studied. The flow around horizontal wind turbine blade simulated with Computational Flow Dynamics (CFD) using a validated 3D Navier-Stokes flow solver and the results of numerical studies compared with experimental results. In this research, three models included of Spalart- Allmaras, k-E (Launder Sharma), SST k- ω and a precondition approach presented by Hakimi are compared and the best model for simulations is selected.

II. MODEL DESCRIPTION AND FORMATION

The phase VI rotor is designed by Giguère and Selig [8] from 988 through 1999. This turbine power control is stall-regulated type.

All sections of this turbine have the S809 airfoil in

Manuscript received January 31, 2014; revised April 30, 2014.

The authors are with Iran University of Science and Technology, Tehran, Irhm2431 Iran (e-mail: shderakhshan@iust.ac.ir, ali_tavaziani@mecheng.iust.ac.ir).

different scales and angles. Phase VI wind turbine is a horizontal axis wind turbine which is upwind with tapered and non-linear twisted. The twist distribution is shown in picture 1 that indicates in 70% span and twist is zero (Fig. 1). In test of wind turbine a two blade configuration of this type with 5.029 m radiuses that operating with 72 rpm as radial speed was used [9].





Number of blades	2 blade	
Rotor Diameter	10.58 m	
Hub Height	12.192 m	
Rotational Speed	71.63 rpm	
Cone angle	0 degrees	
Power Regulation	Stall	
Blade tip pitch angle	3 degrees	
Rotor location	Upwind	
Rotational Direction	CCW	
Twist angle	Non-linear twist along	
Airfoils	S809 in different scales and angles	

TABLE I: THE ARRANGEMENT OF CHANNELS

Unsteady Aerodynamics Experiments was done in the 80 by 120 foot wind tunnel by National Renewable Energy Laboratory (NREI) in NASA Ames Research center.

For create geometry with these characteristics, first, control points at different sections were created then this control points were connected by B-spline carves. Comprehensive information about wind turbine geometry Phase 6 in [9] can be found. In Fig. 2 and Fig. 3 the designed blade is shown in different in various shots.

III. MESH GENERATION

A structured mesh around one blade and the surrounding air of turbine was generated and for other blade periodically repeated. With the aim of generating a structured hexahedral grid, the solution domain was divided into 16 component parts. As is shown in Fig. 7, the computational domain is six times the blade length in radial direction and 15 times the blade length in axial direction.





Fig. 3. Twist in blade.

Mesh is generated with structured grid of 2697136 nodes that 1527115 node of this is on blade and 1169621node is around blade. The thickness of the first cell is 1×10^{-5} m. y^+ value in this case is 0.2 to 1.1 and due to the low Reynolds models used in this study, this range is suitable. Mesh contour is shown in Fig. 4 [10].



Mesh in blade-to-blade view, on blade and 3D mesh generation around blade is shown in Fig. 5, Fig. 6, and Fig. 7. The independence of wind turbines power from grid number was checked for blade geometry simulation as depicted in Fig. 8. It was found that power varies by less than 0.05% for the blade when grid numbers are more than 1.2 million respectively.





IV. NUMERICAL SIMULATION

For reducing the computation, and because yaw system is not used in the wind turbine and due to the symmetry flow field, the grid generation was imposed on one blade but flow computation was solved on whole rotor with two blades.

To simulate and solve equations of flow around the blade, computational Flow Dynamics (CFD) using a validated 3D Navier–Stokes flow solver is used. It's based on finite volume discretization. The physical model used in solver is the Reynolds- Averaged Navier–Stokes equations in rotating frames of reference coupled with various turbulence models. Solves were done until convergence criterion satisfied and global residual decreased to less than 10e⁻⁵ for discretized equations.

A. Investigation Reynolds Number

Reynolds number for tip and root is shown in Table II. From this table, at all wind speeds, flow is turbulent.

Wind speed (m/s)	Tip	Root
5	937834.8	314063.2
7	945496.8	397878.6
10	961576	533956.6
13	982911.5	675770.5
15	999892.7	771945.4
20	105192	1015385

B. The Governing Equations

For Newtonian fluids in Cartesian coordinate, conservation equations of mass, momentum and energy which refer to (1) where u_i is velocity in direction of x_i . P is static pressure, H is total enthalpy, ρ is density, τ_{ij} is viscous stress tensor, q_j is heat flux term and S is additional term added to source terms. H is total enthalpy and h is static enthalpy.

$$\begin{cases} \frac{\partial \rho}{\partial t} + \frac{\partial \left(\rho u_{j}\right)}{\partial x_{j}} = 0 \\ \frac{\partial \left(\rho u_{i}\right)}{\partial t} + \frac{\partial \left(\rho u_{j} u_{i}\right)}{\partial x_{j}} = -\frac{\partial P}{\partial x_{i}} + \frac{\partial \tau_{ij}}{\partial x_{j}} + S_{ui} \end{cases}$$
(1)
$$\frac{\partial \left(\rho H\right)}{\partial t} - \frac{\partial P}{\partial t} + \frac{\partial}{\partial x_{j}} \left(\rho u_{j} H\right) = -\frac{\partial q_{j}}{\partial x_{j}} + \frac{\partial \left(u_{i} \tau_{ij}\right)}{\partial x_{j}} + S_{E} \end{cases}$$

Molecular flux τ_{ij} and q_j are referred to (2) and (3) that

 μ , λ , $\Gamma_k \cdot h_k$ and Y_k respectively are dynamic viscosity, conduct coefficient, diffusion, static enthalpy and component mass. Second term in equation (3) denotes molecular distribution of energy.

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \frac{2}{3} \mu \frac{\partial u_l}{\partial x_l} \delta_{ij}$$
(2)

$$q_{j} = -\lambda \frac{\partial T}{\partial x_{j}} - \sum_{k}^{n} \Gamma_{k} h_{k} \frac{\partial Y_{k}}{\partial x_{j}}$$
(3)

Considering the turbulence models and writing State variables into a mean value and a fluctuating quantity, flow equations referred to (4) where μ_{eff} and H are referred to (5) and (6).

$$\begin{cases} \frac{\partial \rho}{\partial t} + \frac{\partial \left(\rho u_{j}\right)}{\partial x_{j}} = 0 \\ \frac{\partial \left(\rho u_{i}\right)}{\partial t} + \frac{\partial \left(\rho u_{j}u_{i}\right)}{\partial x_{j}} = -\frac{\partial P}{\partial x_{i}} + \frac{\partial}{\partial x_{j}} \left\{ \mu_{eff} \left(\frac{\partial ui}{\partial x_{j}} + \frac{\partial ui}{\partial x_{j}} \right) \right\} + S_{ui} \\ \frac{\partial \left(\rho H\right)}{\partial t} - \frac{\partial P}{\partial t} + \frac{\partial \left(\rho u_{j}H\right)}{\partial x_{j}} = \frac{\partial}{\partial x_{j}} \left(\lambda \frac{\partial T}{\partial x_{i}} + \frac{\mu_{t}}{Pr_{t}} \frac{\partial h}{\partial x_{j}} \right) \\ + S_{E} \frac{\partial}{\partial x_{j}} \left\{ u_{i} \left[\mu_{eff} \left(\frac{\partial ui}{\partial x_{j}} + \frac{\partial ui}{\partial x_{j}} \right) - \frac{2}{3} \mu_{eff} \frac{\partial u_{I}}{\partial x_{I}} \delta_{ij} \right] + \mu \frac{\partial k}{\partial x_{j}} \right\}$$
(4)

$$H = h + \frac{1}{2}u_i u_j + k \tag{5}$$

$$\mu_{eff} = \mu + \mu_t \tag{6}$$

Three turbulence models were used in this study. First model is one equation model called Spalart- Allmaras.

The Spalrt- Allmaras model uses only one additional equation to model turbulence viscosity transport.

This model is suitable for low Reynolds numbers and is useful for its robustness and treat complex flow characteristics. This model proposed for aerospace applications and is a general model for turbomachinery.

The SST k- ω model provided for combination of k- ω formula in wall reigns with k- ε model in regions away from the wall so has all the advantages of the two models and is used in a wide range of problems.

The k- ϵ model is semi-empirical model that used widely in industrial problem Due to its economical and reasonable accuracy.

k- ε Launder Sharma is a low Reynolds model. It's a good method for flows that standard model hasn't suitable accuracy. In this study y^+ is between 0.2 to 1.1, so k- ε launder Sharma turbulence model is used.

In the low Mach number regimes, algorithms designed for compressible flows have low efficiency. The difficulty is in applying compressible codes for low Mach number. When flow velocity is low in comparison with acoustic speeds, the codes converge slowly.

Low Mach numbers, the need for development of

preconditions for more accurate solves and fast convergence increases.

In this study, Hakimi precondition for this purpose has been used [11].

C. Flow Simulation

Three dimensional Navier- Stokes and RANS equations solved on the blade. Three turbulence models used for comparing that includes Spalart- Allmaras, k-E (Launder Sharma) and SST k-w. Results selected from [12]. Results of simulation with this models, accuracy and comparison between experimental and three turbulence models are shown in Fig. 9 and value of errors are in Table III.



Fig. 9. Comparison of experimental power and computed power.

TABLE III: COMPUTATION POWER WITH MODELS AND CORRESPONDING ERRORS

Wind speed (m/s)	SA power	SA Error	SST K-ω power	SST k-ω Error
5	2.38	3.92%	2.46	7.32%
7	5.58	7.54%	6.82	12.95%
10	10.93	8.19%	11.85	17.31%
13	8.81	10.81%	10.93	10.67%
15	7.66	14.23%	6.46	27.62%
20	7.54	9.22%	6.06	26.04%

Wind speed (m/s)	k-ε Launder Sharma power	k-ε Launder Sharma Error	EXP data power (kw)
5	2.22	3.16%	2.29
7	5.82	3.61%	6.04
10	10.54	4.34%	10.10
13	9.21	6.74%	9.88
15	8.13	8.83%	8.92
20	7.62	7.07%	8.2

If U_{∞} is wind speed in terms of m/s, r is radius of section from center of hub in terms of m, Ω is the rotational speed in terms of rad/s and p_{∞} is density of free stream (wind speed) in terms of kg/m^3 so pressure coefficient is calculated from:

$$C_{p} = \frac{(p - p_{\infty})}{0.5p_{\infty}(U_{\infty}^{2} + (\Omega r)^{2})}$$
(7)

In Fig. 10, Fig. 11, and Fig. 12, pressure coefficients from numerical calculations with different turbulence models are compared. Pressure coefficients are computed with CFD using three turbulence models and compared with experimental data at 3 wind speed; include 7, 10, and 15 m/s in 5 sections of 3D blade, 30 %, 47 %, 63 %, 80 %, and 95 %.







Fig. 10. Comparison pressure coefficient between experimental and numerical in 7 m/s wind speed.

Experimental results have been extracted from [12].

ŝ



Fig. 11. Comparison pressure coefficient between experimental and numerical in 10 m/s wind speed.



Fig. 12. Comparison pressure coefficient between experimental and numerical in 15 m/s wind speed.

In Fig. 13, Fig. 14, and Fig. 15, pressure contours in three speeds 7, 10, and 15 on suction and pressure sides are shown:





According to pressure contours a considerable variation from leading edge to trailing edge and from root to tip in both suction and pressure sides are occurs.

In Fig. 16, Fig. 17, and Fig. 18 related speed contours in investigated sections, in different speeds and for k- ϵ turbulence model are shown. The other models not showed in the results because of not to make the figures crowded.



Fig. 16. Contours of relative velocity at 7 m/s for k- ε model.

Based on Fig. 9 at 5 to 10 wind speeds (low speeds) three turbulence models have similar predictions in power.

At higher speeds (10 to 20) k- ε (Launder Sharma) turbulence model predicted the actual results with more accuracy and has best agreements with experimental so it's best between three models.





Fig. 17. Contours of relative velocity at 10 m/for k- ϵ model.





Fig. 18. Contours of relative velocity at 15 m/s for k-ɛ model.

According to results and comparisons, k- ϵ (Launder Sharma) is suitable for calculation of horizontal axis wind turbine (HAWT) power in low, medium and high speeds. According to studies it was predicted that the SST k- ω has better accuracy in prediction of performance of wind turbine rotor.

For more accuracy in two equation turbulence models like k- ω SST, the grid must arrangement on normal direction. Reference [13] studies on different turbulence models. It shows high sensitivity of k- ε to y^+ while SST k- ω and Spalart- Allmaras have less sensitivity. But this study was done on a flat plate.

Comparisons between two turbulence models include Spalart- Allmaras and SST $k-\omega$ investigated in [14] and the results showed the superior results of Spalart- Allmaras method.

In other study in [15] some models including two equation turbulent models which is investigated in this paper are compared. This study was on a blade of National Renewable Energy Laboratory (NREL) phase III that hasn't twist. Two turbulence models was k- ϵ and SST k- ω . The results showed that k- ϵ has best prediction similar to this paper.

Reference [6] blade of Phase VI wind turbine is simulated using FLUENT software and the turbulence model was SST k- ω . The results of this study shows inability of this turbulence model in prediction of wind turbine flow characteristics. Also according to the section IV-A and low Reynolds on turbine blades, excellence of k-e Launder Sharma is predictable. The figures of pressure coefficients show considerable agreement with experimental data. May noticed that some of the figures show large differences between results of numerical and experimental. By investigated the velocity contours and pressure coefficients at that section and speed, a separation occur. The separation causes large vorticities and it's hard for the turbulence models that investigated in this paper to capture those. In the velocity contours, the blue regains after airfoils show separation and vorticites.

Velocity in sections of blade is calculated by $v = \sqrt{U^2 + (\Omega r)}$ that U is wind speed, Ω is rotational speed and r is radius of section.

For 7 m/s, velocity at 30% span is 13.36vm/s and related Mach number is 0.0389. At 95% span, velocity and Mach number are 36.04 m/s and 0.105. For wind speed of 15 m/s, velocity at 30% span is equal to 18.826 m/s and Mach number is 0.056. At 95% span velocity and Mach number are 39.021 m/s and 0.114.

According to Mach number of each section and comparison of numerical results of Hakimi approach and experimental results, use of Hakimi approach increase accuracy of predictions specially in low speeds. The reason is application of precondition approach for low Mach number so since Mach number of root is less than tip, effect of precondition approach is major.

V. CPU COST

The corresponding computation time has been presented in Table IV the processes were done using a PC computer with 4 GB of RAM memory, Core 2 and 2.5 GHZ of speed.

Process	CPU time for	Number	Total CPU
	one process	Of process	time
Grid independency	8 hour	8	64 hour
SA model	10 hour	3	30 hour
K-ε model	15 hour	3	45 hour
SST k-ω model	20 hour	3	60 hour
Hakimi approach with	13 hour	3	39 hour
k-ε			
Total	238 hour		

TABLE IV: REYNOLDS NUMBER FOR TIP AND ROOT

VI. CONCLUSION

Aerodynamic performance of Phase VI wind turbine was investigated.

Flow around wind turbine was simulated with Navier-Stokes equations using three difference turbulence models and results compared with experimental data.

To simplify the problem and due to the symmetry flow field, mesh was generated on one blade and similar mesh on other blades extended.

To increase the mesh density in important fields, computation domain was divided to 16 component parts.

For simulation of flow around wind turbine blade, Navier-Stokes equation used with different turbulence models include Spalart-Allmaras, k- ϵ and SST k- ω .

According to numerical results, at 5 to 10 wind speeds (low speeds) three turbulence models have similar predictions in power but at higher wind speeds k- ϵ has predicted with more accuracy, so it's best between three

models.

Some of the figures showed large differences in prediction and experiment results. The reason is the separation causes large vorticities and it's hard for the turbulence models that investigated in this paper to capture those.

Mach number for the wind turbine was low so for more accuracy, use of precondition was seemed helpful. In this paper Hakimi precondition was used and results showed good agreement with actual data. Finally for prediction of performance of horizontal axis wind turbine, k- ε Launder Sharma turbulence model using Hakimi precondition is suggested.

REFERENCES

- E. Duque, C. V. Dam, and S. Hughes, "Navier-stokes simulations of the NREL combined experiment phase II rotor," presented at the 37th Aerospace Sciences Meeting and Exhibit, American Institute of Aeronautics and Astronautics, Reno, NV,U.S.A. January 1999.
- [2] N. N. Sørensen, "CFD modelling of laminar-turbulent transition for airfoils and rotors using the γ – model," *Wind Energy*, vol. 12, no. 8, pp. 715-733, 2009.
- [3] F. Zahle, N. N. Sørensen, and J. Johansen, "Wind turbine rotor-tower interaction using an incompressible overset grid method," *Wind Energy*, vol. 12, no. 6, pp. 594-619, 2009.
- [4] M. A. Potsdam *et al.*, "Unstructured mesh CFD aerodynamic analysis of the NREL Phase VI rotor," presented at the 47th AIAA Aerospace Sciences Meeting, Orlando, Florida, U.S.A. January 2009.
- [5] C. E. Lynch, Advanced CFD Methods for Wind Turbine Analysis, 2011.
- [6] R. Lanzafame *et al.*, "Wind turbine CFD modeling using a correlationbased transitional model," *Renewable Energy*, vol. 52, pp. 31-39, 2013.
- [7] M. C. Hsu, I. Akkerman, and Y. Bazilevs, "Finite element simulation of wind turbine aerodynamics: validation study using NREL Phase VI experiment," *Wind Energy*, 2013.
- [8] P. Giguere *et al.*, "Design of a tapered and twisted blade for the NREL combined experiment rotor," *NREL/SR*, vol. 500, pp. 26173, 1999.
- [9] M. M. Hand *et al.*, "Unsteady aerodynamics experiment phase VI: wind tunnel test configurations and available data campaigns," *NREL/TP*, vol. 500, pp. 29955, 2001.
- [10] User Manua, NUMECA, 2013.
- [11] N. Hakimi, Preconditioning Methods for Time Dependent Navier Stoks Equations, 1997.
- [12] M. M. Yelmule and E. A. Vsj, "CFD predictions of NREL Phase VI rotor experiments in NASA/AMES wind tunnel," *International Journal of Renewable Energy Research*, vol. 3, no. 2, pp. 261-269, 2013.
- [13] P. G. Huang, J. E. Bardina, and T. J. Coakley, "Turbulence Modeling Validation, Testing, and Development," NASA Technical Memorandum, 1997.
- [14] S. Benjanirat, "Computational studies of the horizontal axis wind turbines in high wind speed condition using advanced turbulence models," Ph.D. dissertation, School of Aerospace Engineering. Georgia Institute of Technology, 2006.
- [15] A. Gupta, "Prediction of aerodynamic forces on wind turbine blades using computational fluid dynamics," Ph.D. dissertation, Applied Science in Industrial Systems Engineering. University of Regina, 2007.



S. Derakhshan was born in Shahrekord, Iran on Sep. 11, 1978. He received PhD degree in mechanical engineering from University of Tehran, Tehran, Iran, 2008.

He is working as an assistant professor in School of Mechanical Engineering, Iran University of Science & Technology (IUST), since 2010, January. He was also an invited lecturer at University of Tehran and University of Power and Technology, Tehran, Iran in

2009. In addition, he was a visiting professor in Jiangsu University, September 2010. He has taught some courses in the field of Fluid Mechanics, CFD, Renewable Energy and Hydraulic Turbomachinery.

He has published more than 20 papers in peer review journals and presented more than 10 papers in international conferences. He also published one book entitle" Renewable Energy" in Persian.

Dr. Derakhshan has awarded 22th Khwarizmi International Award (2008, Tehran, Iran).