Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 51(2) (2019) 85-87
DOI: 10.22060/mej.2017.12740.5420

Turbulent Structures in the Wake of a Wind Turbine Using Large Eddy Simulation

A. Veisi[1], M. H. Shafiei Mayam[2*]

[1]Department of Mechanical Engineering, Sistan and Baluchestan University, Zahedan, Iran
[2]Department of Mechanical Engineering, Bozorgmehr-University of Qaenat, Qaen, Iran

ABSTRACT: In the present work the flow around a horizontal axis wind turbine has been studied using large Eddy simulation at different rotational speeds. The results show increasing rotational speeds causes a higher velocity deficit in the downstream direction. For example, in 1*D* after the wind turbine the minimum velocity is 54% of the initial velocity and reach to the 67% of the initial velocity after wake travel 6*D*. At the rotational speed of λ_3=10 the minimum velocity is 26% of the initial velocity and reach to the 68% of the initial velocity after wake travel 6*D*. The frequency of vortex shedding is increased by increasing the rotational speeds. Shed vortices tend to be extended in the *y* direction and its intensity augmented by increasing the rotational speeds. The strengthen of vortices at higher rotational direction in far wake region not only due to the increased of swirling strength, but it is also due to the collision of vortices and the formation of new vortices. This issue has not been reported in previous works. Also, the increase of turbulence intensity and Reynolds shear stress in the flow direction is due to the severe wind shear and high mechanical production of turbulent kinetic energy.

Review History:

Received: 9 April 2017
Revised: 25 July 2017
Accepted: 8 October 2017
Available Online: 11 October 2017

Keywords:

Wind turbine
Large Eddy simulation
Turbulent flow
Turbulence structures

1- Introduction

The use of renewable energy for energy supplement is inevitable due to the limited recourse of fossil fuels and the problems caused by global warming. Among the renewable energies, wind energy has great importance. Wind energy is widely distributed all over the world and its capacity is larger than the energy consumption [1,2]. A better understanding of wake development is important in the downstream direction in order to minimize the wake interaction and optimize the power efficiency. More studies are limited to measuring the streamwise velocity and lateral velocities, while wind turbines are operated under strong shear force and relatively higher turbulence intensity [3]. The wake region is influenced by shedding vortices and non-homogeneous distribution of turbulence. Limited information about the spatial distribution of turbulence in the wake region and the impact of vortices on the performance of downstream turbine has limited researchers' ability to predict power production and fatigue loads. Hence, a better understanding of the spatial distribution of turbulence structures has great importance to find a regular pattern of these structures in the wake region.

Many researchers have investigated turbulent flow behind the horizontal axis wind turbines by experimental and numerical methods. Porte´-Agel et al. [4] showed Large Eddy Simulation (LES) could be provide spatial distribution of turbulence structures to increase power efficiency and lifetime of the wind turbines. Zhong et al. [5] showed velocity deficit in the blade location is due to the shed vortices. Meyers and Meneveau [14] proposed optimum separation distance is 15*D* for horizontal axis wind turbines by LES, the conventional space was 7*D*.

In the present work large eddy simulation around a single wind turbine at different rotational speeds were investigated. Better understanding of wake development in the wake region is the main objective of the present study. For this purpose, the formation and evolution of vortices are studied. Also, spatial distribution of turbulence parameters is studied, including average velocity, turbulence intensity and shear stress.

2- Numerical Simulation

2- 1- Governing equation

Turbulent flow is three dimensional, time dependent and random. Turbulent flows occur over a wide range of time and length scale. In Reynolds Averaged Navier-Stokes (RANS) all turbulence length scales is time-averaged. In Direct Numerical Simulation (DNS) scales of motion are resolved to the smallest scale which is the Kolmogorov scale. In the LES method energy containing range are solved and smaller scale than l_{EI} are modelled [7].

The filtered Continuum and momentum equations are given as follows:

$$\frac{\partial \bar{U}_i}{\partial x_i} = 0 \tag{3}$$

Corresponding author, E-mail: shafiei@buqaen.ac.ir

$$\frac{\partial \bar{U}_j}{\partial t}+\frac{\partial \overline{U_i U_j}}{\partial x_i}=\upsilon\frac{\partial^2 \bar{U}_j}{\partial x_i \partial x_i}-\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_j} \quad (4)$$

where $\bar{U}_i$ is the resolved velocity, $\bar{P}(x,t)$ is the filtered pressure field.

$$\tau_{ij}^R=\overline{U_i U_j}-\bar{U}_i\bar{U}_j \quad (5)$$

The filtered momentum equation can be written as:

$$\frac{\partial \bar{U}_j}{\partial t}+\frac{\partial \bar{U}_i\bar{U}_j}{\partial x_i}=\upsilon\frac{\partial^2 \bar{U}_j}{\partial x_i x_i}-\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_j}-\frac{\partial \tau_{ij}^R}{\partial x_i} \quad (6)$$

Non-isotropic residual stress tensor and residual kinetic energy are defined as follows:

$$\tau_{ij}^r=\tau_{ij}^R-\frac{2}{3}k_r\delta_{ij} \quad (7)$$

$$k_r=\frac{2}{3}\tau_{ii}^R \quad (8)$$

δ_{ij} is the kronecker delta. Isotropic residual stress appeared in the modified filtered pressure field:

$$\bar{p}=\bar{p}+\frac{2}{3}\rho k_r \quad (9)$$

Finally, momentum equation can be rewritten as follows:

$$\frac{\partial \bar{U}_j}{\partial t}+\frac{\partial \bar{U}_i\bar{U}_j}{\partial x_i}=\upsilon\frac{\partial^2 \bar{U}_j}{\partial x_i \partial x_i}-\frac{1}{\rho}\frac{\partial \bar{P}}{\partial x_j}-\frac{\partial \tau_{ij}^r}{\partial x_i} \quad (10)$$

2- 2- Residual-stress model

Closure problem is exist in large eddy simulation like Reynolds equations. To overcome this issue Reynolds stress tensor is obtained by turbulent viscosity model or Reynolds stress models. Closure problem is solved by modelling the residual stress tensor in LES method. The simplest model proposed by Smagorinsky [8]. In this model linear eddy-viscosity model related residual stress tensor to the filtered rate of strain

$$\tau_{ij}^r=-2\upsilon_r\bar{S}_{ij} \quad (11)$$

$$\bar{S}_{ij}=\frac{1}{2}\left(\frac{\partial \bar{U}_i}{\partial x_j}+\frac{\partial \bar{U}_j}{\partial x_i}\right) \quad (12)$$

where $\upsilon(x,t)$ is the eddy viscosity of the residual motions and $\bar{S}_{ij}$ is the filtered rate of strain. The eddy viscosity is modelled by analogy to the mixing-length hypotheses:

$$\upsilon_r=\ell_s^2\bar{S}=\left(C_S\Delta\right)^2\bar{S} \quad (11)$$

where $\bar{S}$ is the characteristic filtered rate of strain, l_s, is the Smagorinsky length scale. In the present study Smagorinsky coefficient is C_S=0.1 Filter with equal to the grid size and is calculated from the following equation:

$$\Delta=\left(\Delta_x\Delta_y\Delta_z\right)^{1/3} \quad (12)$$

2- 3- The horizontal axis wind turbine

The results of the present study are based on the blind test1 workshop [6]. NREL S826 airfoil is used to build the blade along the entire span of the blade.

2- 4- Boundary conditions and computational mesh

Wind velocity is 10 m/s at the inlet of the wind tunnel. Turbulence intensity is %0.3 at the inlet of the wind tunnel. Non-slip condition is applied on all the walls. It should be noted the boundary condition of the present work are corresponding to the Blind test1 workshop. Turbulence intensity defined as the ratio of standard deviation to the reference velocity:

$$TI=\frac{\left\langle u^2\right\rangle^{1/2}}{U_{ref}} \quad (1)$$

For the outlet where the flow leaves the domain, the ambient pressure (zero pressure) was applied. Tip speed ratio defined as:

$$\lambda=\Omega R/U_{ref} \quad (2)$$

Ω is rotational velocity, R is rotor radius and U_{ref} is velocity reference. Computational domain consists of two parts: moving part and stationary part. Due to use of the sliding mesh technique for moving zone, unlike the Multi-Reference Frame (MRF) model, the current model consists of a two-layer interface. Grid near the blade, hub and tower are considered very fine in order to solve severe velocity gradient in the vicinity of the rotor. The wind tunnel in this study has a total length, cross section and height, 11.4 m, 2.7 m, 1.8 m, respectively. The wind turbine is located 5*D* rotor diameters downstream of the inlet section of the wind tunnel. *D* is rotor diameter. The hub height of the wind turbine from the floor is 0.817 mm.

In this study finite volume method is used to discretize the equations. Pressure term is achieved through the SIMPLE algorithm. The convective terms were discretized using Bounded Central Differencing scheme. The total number of cells 2.7×10^6 is chosen in the present study. The computational grid 7.2 million to study selected. Time step size is 0.000969 s.

3- Results and Discussions

3- 1- Validation

In this section, the results of LES in a single turbine are compared with the experimental data and two other numerical simulations at different rotational speeds. The results of the present work are consistent with the experimental data. The average difference from the experimental data is about 3%, except on occasion in 5*D* where the difference is approximately 8%. This discrepancy is also seen in the results of Hansen and Kono [6].

3- 2- Turbulence structures

Many researchers emphasized vorticity is not a suitable tool to reveal vortices because it could not differentiate between pure shearing motions and the actual swirling motions. Various tools and algorithms have been developed and proposed by many investigators for the identification of vortices [9-16]. In this study, the λ_{ci} method is used to extract the vortices.

Shed vortices are closer together by increasing the rotational speeds. Tip vortices tend to be extended in the *y* direction. Tip vortices are transported longer distances than the root vortices and later dissipated. In the upper part of wind tunnel, the velocity vectors are in the flow direction in the vicinity of tip vortices (dark patch). The velocity vectors are in the opposite direction of flow in the vicinity of root vortices (light patch). Swirling strength is increased by increasing the rotational speeds.

3- 3- Velocity field

Wake region is extended and more area by increasing the rotational speeds. Wake is recovered as moving in the downstream direction. Velocity distribution of streamwise velocity shows the asymmetry in the wake region, especially in λ_1=3, which is due to the effect of tower and floor of the wind tunnel.

3- 4- Turbulence intensity

Turbulence intensity is defined as a measure of fatigue loads on different parts of a wind turbine. Turbulence intensity in the streamwise direction (I_z) has been maximum just behind the swept area. However, the effect of turbulence intensity is quickly dissipated at this region, but turbulence intensity is augmented at tower location by moving in downstream direction.

3- 5- Reynolds stress

Reynolds shear stress ($-\overline{u'w'}$) has a high level of intensity behind the wind turbine. The distribution of Reynolds shear stress $-\overline{u'w'}$ is similar to the distribution of turbulence intensity I_z. This is due to the severe wind shear and high mechanical production of turbulent kinetic energy.

4- Conclusions

In the present work flow field around a horizontal axis wind turbine at different rotational speeds have been studied by the Large Eddy Simulation method. The evaluation of turbulent flow characteristics in the wake region is the main objective of this study. As the rotational speeds are increased the vortices are closer to each other and form a continuous path. Since, kinetic energy is harvested by wind turbine streamwise velocity is decreased and lateral velocities are increased. Velocity distribution in the streamwise direction shows a different pattern for λ_1=3 and λ_2=6. Velocity deficit is maximum in the root of blade for λ_1=3, while for λ_2=6 the maximum velocity deficit happened at the tip of the blade. This issue shows by increasing the rotational speed, energy extraction is increased on the tip location. A severe increase of turbulence intensity and Reynolds shear stress in the near wake region is due to the severe wind shear and high mechanical production of turbulent kinetic energy.

References

[1] Chen, D., Zhang, W., 2007. “Exploitation and research on wind energy”. *Energy Conservation Technology*, 4(1), pp. 339–343.

[2] Li, Y., Paik, KJ., Xing, T., Carrica, PM., 2012. “Dynamic overset CFD simulations of wind turbine aerodynamics”. *Renewable Energy*, 37(1), pp. 285–298.

[3] Zhang, W., Markfort, CD., Porté-Agel, F., 2012. “Near-wake flow structure downwind of a wind turbine in a turbulent boundary layer”. *Experiments in Fluids*, 52(5), pp. 1219–1235.

[4] Porté-Agel, F., Wu, YT., Lu, H., Conzemius, RJ., 2011. “Large-eddy simulation of atmospheric boundary layer flow through wind turbines and wind farms”. *J Wind Eng Ind Aerodyn*, 99(1), pp.154–68.

[5] Zhong, H., Du, P., Tang, F., Wang, L., 2015. “Lagrangian dynamic large-eddy simulation of wind turbine near wakes combined with an actuator line method”. *Appl Energy*, 144(1), pp.224–33.

[6] Krogstad, PÅ., Eriksen, PE., 2013. “ ‘Blind test’ calculations of the performance and wake development for a model wind turbine”. *Renewable Energy*, 50(1), pp.325–333.

[7] Pope, SB., Turbulent Flows, pp. 351-558, *Cambridge University Press*, 2000.

[8] Smagorinsky, J., 1963. “General Circulation Experiments With the Primitive Equations”. *Mon. Weather Rev*, 91(1), pp.99–64.

[9] Jeong, J., Hussai, F., 2006. “On the identification of a vortex”. *J. of Fluid Mechanics*, 285(1), pp. 69-94.

[10] Perry, A.E., Chong, M.E., Cantwell, B.J., 1990. “A general classification of three-dimensional flow fields”. *J. Physics of Fluids*, 2(1), pp.765-777.

[11] Zhou, J., Adrian, RJ., Balachandar, S., Kendall, TM., 1999. “Mechanism for generating coherent packets of hairpin vortices in channel flow”. *Journal of Fluid Mechanics*, 387(1), pp. 353-94396.

[12] Maciel,Y., Shafiei Mayam, MH,. Hairpin structures in a turbulent boundary layer under stalled-airfoil-type flow conditions , Progress in Turbulence III,*Proceedings of the iTi Conference in Turbulence*, 2008.

[13] Shafiei Mayam, M.H., Maciel, Y., Hairpin structures in a turbulent boundary layer with strong adverse pressure gradient, *International Symposium on turbulence and shear flow phenomena*, 2007.

[14] Shafiei Mayam M.H., Maciel Y., Coherent structures in a turbulent boundary layer in stalled-airfoil0type flow conditions, 2007.

[15] Maciel, Y., Shafiei Mayam, M. H., Hairpin structures in a turbulent boundary layer under stalled-airfoil-type flow conditions. *Progress in Turbulence III*, 2008.

[16] Veisi, AA., Shafiei Mayam, MH., 2017. “Large Eddy Simulation of flow around a single and two in-line horizontal-axis wind turbines”.*Energy*, 121(1), pp.533-544.

Please cite this article using:

A. Veisi, M. H. Shafiei Mayam, Turbulent Structures in the Wake of a Wind Turbine Using Large Eddy Simulation,

Amirkabir J. Mech. Eng., 51(2) (2019) 85-87.

DOI: 10.22060/mej.2017.12740.5420