

# Flow Velocity Simulation of Wind Turbines by Computational Fluid Dynamics (CFD)

Hammad ur Rahman<sup>1</sup> , Syed Faisal Shah<sup>2</sup>, Abdullah Jamshaid<sup>3</sup>, Muhammad Usama<sup>4</sup>

<sup>1,2,3</sup>Department of Energy Management & Sustainability (EnMS), (UET Peshawar), Pakistan

<sup>4</sup>Department of Thermal System Engineering, Center for Advanced studies in Energy (UET Peshawar), Pakistan  
hammadrahman85@gmail.com<sup>1</sup>, syedfaisal@yahoo.com<sup>2</sup>, Abdullahjamshaid134@gmail.com<sup>3</sup>, Uet198@gmail.com<sup>4</sup>

Received: 25 January, Revised: 05 February, Accepted: 14 February

**Abstract**— Rapid transition towards renewable, wind is having potential of 14TW. Flow simulations attracted worldwide scholars to optimize wind power production and wind farms. In present work NREL 3MW wind turbine under k-ε RANS model is simulated at two velocities i.e. 10m/s and 15m/s to calculate the flow and pressure distribution over wind turbine. With these variables velocity magnitude, dynamic pressure, wake effect and turbulent dissipation rate results are generated, compared and analyzed. Accurate results are shown in near wake regions. At 10m/s fluctuations in velocity magnitude are recorded less, which leads to less pressure drop and less intensified wake downstream, The distance covered by 2nd wake is recorded more while at 15m/s there are more fluctuations in velocity magnitude this results more pressure drop and provide favorable conditions for turbulent wakes. The distance of 1st and 2nd wake is recorded almost equal while the 1st wake intensity is more. The computational time by k-ε model require less time and provide good results.

**Keywords**— NREL, National Renewable Energy Laboratory, RANS, Reynolds Averaged Navier-Stokes Equation Remote.

## I. INTRODUCTION

Increasing demand of energy in each country leads to global economic development. These economic developments causes major energy shortfall in countries. As world is moving towards renewable and sustainable energy, wind energy has one of the rapid growth rate among renewable energy resources. From 2001, installed capacity of wind energy flew from 24GW to expected 817GW in 2021 [1]. The overall capacity of the wind projects is to be doubled in coming recent years [2]. Till 2035 the share of wind energy will be 35% alone in electricity generation worldwide [3]. Current cumulative installed wind capacity is shown in Fig1. In last decade different innovative techniques are used to optimize the wind power output. These include different analytical and numerical models. Analytical wake models (kinematic wake models) uses velocity deficient profiles, which is obtained from experimental or theoretical work [4]. However the simplicity and efficient computation of this model cannot solve complex problems of fluid mechanics

and aerodynamics of wind turbines [5]. A numerical model is mostly relayed on Computational Fluid Dynamics (CFD) which shows more flexibility and accuracy for different wind velocity and terrain features. Most researchers use CFD models for the designing of wind farm or wind turbine.

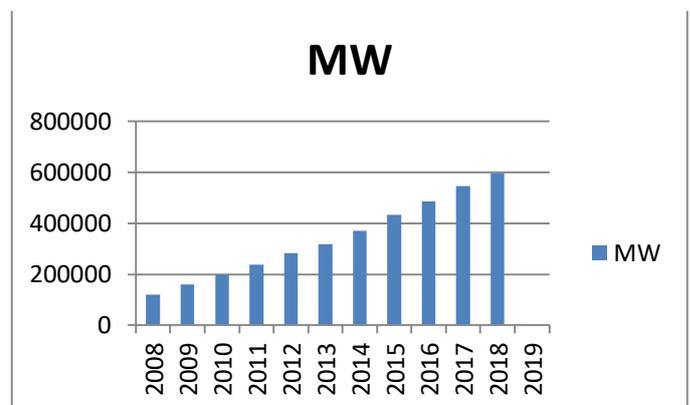


Fig. 1. Cumulative installed wind capacity globally 2001-2018 [6]

Innovations of computer bring Computational fluid dynamics. Complex Navier stroke equation involves fluid flow of viscous compressible, which describe the mechanism around wind turbine blades. Actual fluid flow and field are simulated to obtain realistic results [7],[8] studied wind farms comprising of two three wind turbines. Rotor diameter and wake field is analyzed. [9] Use Ansys software to analyze wake characteristic using over set grid method. The power production curve resembles the practical data.

Disadvantages of CFD include long time of circulations, meshing overlapping and sheared topology for high computational accuracy best hardware is required. Time reducing is the main objective in CFD, high knowledge of CFD models and its applications is required to gain desired and accurate results [7]. Ref [10] shows that at specific conditions k-ε model have not mutual agreement with the experimental data under specified conditions some other models are developed by scientist. [7] shows that computation time is large for some

specific geometries, timing of these computation can be reduced by proper meshing and selecting mesh type and size.

In last decade wind turbines are installed in the large wind farms. These wind farms are associated with two major issues; Power production reduction because of wake velocity deficit and Dynamic load on blades of wind turbines because of turbulence increase. These issues are also affected by layout and velocity of air. The production of second row or back turbines are affected by the wakes due to dissipation of energy

from 1st row turbines [11]. The wake dynamics are also associated with the blade length of wind turbine, air velocity and distance between two wind turbines measured in rotor diameters. The problem is effect of velocity on the flow velocity distribution over wind farm, and the effect of pressure distribution over wind turbine [12].

In this study RANS model on ANSYS fluent is used to study velocity and pressure distribution of NREL 3MW wind turbine. k-epsilon model is used to investigate velocity and pressure distribution over 10m/s and 15m/s of air. The results of simulation model are used to show the wake effect of the 1st row turbine downstream.

## II. NUMERICAL MODEL

### A. Incompressible Navier-Stoke Equation:

These are set of complex equations which give a complete set of models for the turbulent flow, but these equations cannot be solved easily. The non-linear convective terms in turbulent flow contributes to a wide range of time and length graph [13]. For example, largest turbulent scales are 1 km, where the smallest scale is 1mm in atmospheric boundary layer (ABL) [14]. Inside the blade boundary layers, the scales are even smaller. The Reynolds number (Re) shows the range of scale, 1 blade and wake encountered large values of Reynolds number calculations leads to computer simulations extremely expensive. Solving all scales in flow, known as direct numerical simulation, is not feasible [15]. The Navier-stokes equation is shown mathematically below.

$$\mathbf{u} = 0 \quad (1)$$

$$\delta \mathbf{u} / \delta t + (\mathbf{u} \cdot \nabla) \mathbf{u} = 1/\rho \nabla p + \nu \nabla^2 \mathbf{u} \quad (2)$$

### B. Reynolds Averaged Navier-Stoke Equation (RANS)

In this set of equation the Navier-stroke equation is decomposed into mean and fluctuating component and solution variables are exact. In the given equations  $\mathbf{u}$  and  $\mathbf{u}0$  present mean and fluctuating components [16].

$$\mathbf{u}(x, t) = \bar{\mathbf{u}}(x) + \acute{\mathbf{u}}(x, t) \quad (3)$$

In momentum and continuity equation if we substitute these condition that taking time mean and making velocity term average ( $\mathbf{u}$ ) this gives the below equation

$$\delta \rho / \delta t + \delta / \delta x_i (\rho u_i) = 0 \quad (4)$$

$$\begin{aligned} \delta / \delta t (\rho u_i) + \delta / \delta x_j (\rho u_i u_j) &= -\delta p / \delta x_i + \delta / \delta x_j \\ [\mu (\delta u_i / \delta x_j + \delta u_j / \delta x_i + 2/3 \zeta_j \delta u_i / \delta x_i)] + \delta / \delta x_j (-\rho u_i u_j) & \end{aligned} \quad (5)$$

The above equation is known as Reynolds averaged Navier-stroke equation because its present average values of velocities and other variables. New terms are added in the equation  $-\rho u_i u_j$  is known as Reynolds stresses and these must be molded to get accurate solution [16]–[18].

### C. The k-ε turbulence model

In Reynolds averaged Navier-Stokes equations, k-ε turbulence model calculated eddy viscosity by two transport equations for ε (turbulent dissipation rate) and k (turbulent kinetic energy). This model is famous for turbulent flow in industry, heat transfer simulation and fluid flow simulation. Further models includes in eddy viscosity are, Renormalization group (RNG), and realizable k-ε turbulence model. Similar transport equation used for turbulent kinetic energy and turbulent dissipation rate.

Difference among eddy viscosity and, k-ε turbulence model is the turbulent viscosity calculation, in the equation generation and destruction term, and Prandtl number give the turbulent diffusion of k and . The mathematical model is shown below

$$\frac{\delta \rho}{\delta t} + \frac{\delta}{\delta x_i} (\rho u_i) = \frac{\delta}{\delta x_j} \left[ \frac{u_i}{\sigma_t} \frac{\delta k}{\delta x_j} \right] + 2\mu_t E_{ij} E_{ij} - \rho \epsilon \quad (5)$$

$$\begin{aligned} \frac{\delta (\rho \epsilon)}{\delta t} + \frac{\delta (\rho \epsilon u_i)}{\delta x_i} &= \frac{\delta}{\delta x_j} \left[ \frac{\mu_t}{\sigma_\epsilon} \frac{\delta \epsilon}{\delta x_j} \right] + C1 \epsilon \frac{\epsilon}{k} 2\mu_t E_{ij} E_{ij} - \\ C2 \epsilon \rho \frac{\epsilon^2}{k} & \end{aligned} \quad (6)$$

Time scale and turbulent length is shown by k-ε turbulence model, which determines turbulence by solving two transport equations for k and ε. Main assumptions in this model are the turbulent flow and neglecting the effect of molecular viscosity [19], [20]. [21], [22] shows that this model has nearest values to the experimental data however [22] predicts that, as the dissipation proportionally increases the prediction of turbulence become less contracted.

## III. SIMULATION SETUP

A lot of wind turbines and its types are present in literature but we chose the wind turbine, which are mostly use in wind farms, the turbine use in this article is NREL 3MW and its specifications are specified in Table 1.

TABLE I SPECIFICATIONS OF WIND TURBINE [7].

<b>Rotor Diameter</b>	129m
<b>Rated Power</b>	2.9MW
<b>Swept Area</b>	113070m <sup>2</sup>
<b>Orientation</b>	3 blades, Upwind
<b>Blade Length</b>	61m
<b>Hub Height</b>	87m
<b>Inboard Twist</b>	12:3 deg
<b>Hub Diameter</b>	3m
<b>Material</b>	Fiberglass Reinforced With Epoxy Resin
<b>Cut in, Rated, Cut Out</b>	3:5m/s; 14:5m/s; 25m/s

The geometry of wind turbine is drawn in Spice Claim, in which 2D and 3D geometry of the given turbine are formed with proper dimensions according to specification shown in Table 1. Another software known as Design Modeler is used to specify or define area of interest like inputs, outputs, stationary and moving domains. The following Fig2 will show the geometry of wind turbine.

In these simulations analyzing zones are meshed finer to gain computational accuracy. Important zones are wind turbine and the enclosure downstream, which show the turbulence effect and pressure distribution. In simulations the impellers of wind turbine are meshed finer in a cylindrical enclosure with tetrahedral mesh structure and then embed in a whole rectangular domain the Fig3 will show the mesh of simulations. There are about 172,302 nodes and 966,207 elements.

Free flow conditions are adopted at the inlet of enclosure with constant speed of wind chosen 10m/s and 15m/s respectively and outlet of the boundary is at atmospheric conditions. The density of air is taken as 1.225kg/m<sup>3</sup> and density of fiber glass is taken as 2.719kg/m<sup>3</sup>. In mesh Interfaces section, the source is selected as enclosure and wind turbine or propeller is select as target to define the contact regions.

Modeling or setup is the heart of simulation process here we select the model, the selection of model is mostly depend upon type of research and literature study. Every model has its own limitations and advantages; here literature helps in your research. The model here is selected is advanced model of RANS known as k- ε turbulence model in which dissipation, distribution of pressure and aerodynamics are considered. The

software use here is ANSYS 2019 in which Ansysis Fluent solver is used.

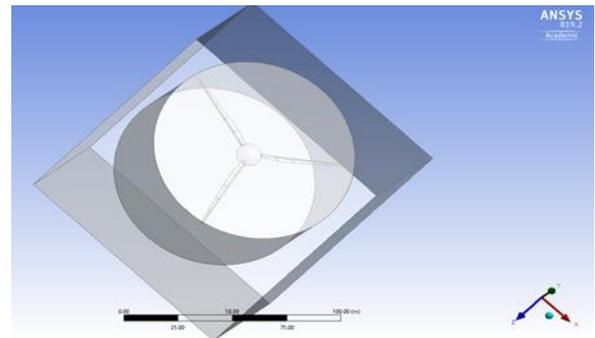


Fig. 2. Geometry of the project

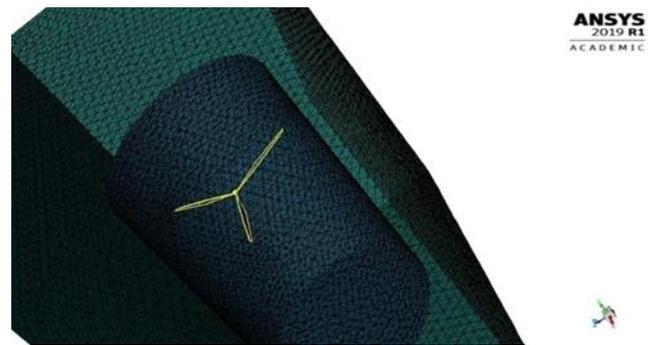


Fig. 3. Mesh of the project

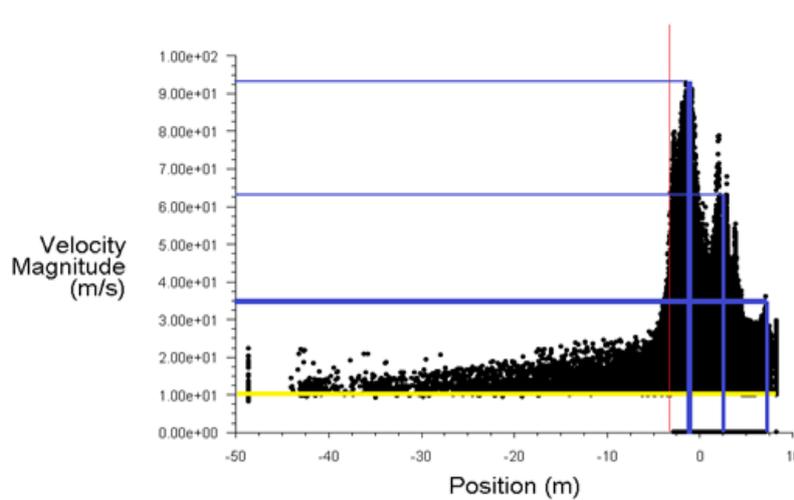


Fig. 4. Velocity Distribution at 10m/s

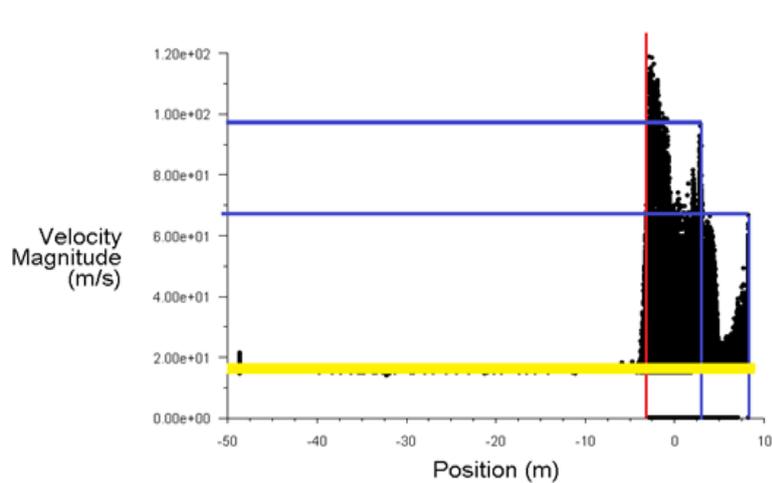


Fig. 5. Velocity Distribution at 15m/s

#### IV. RESULTS AND DISCUSSIONS

In this study, velocity distributions of wind are studied over wind turbine with respect to 10m/s and 15m/s of wind speed. The wake dynamics, Dynamic pressure and turbulent dissipation are compared.

##### A. Velocity Distribution

The wake field calculations are obtained from the velocity distribution graphs. In which the velocity of air changing with its position (gradient) is shown. The velocity distribution is shown in the following Fig 4, 5.

The red line shows the position of wind turbine in Fig4 and 5, yellow line shows the input velocity of wind. The peaks shows wake effect, as one wake is going to dismiss formation of next wake starts subsequently due to pressure drop. Downstream of wind turbine the pressure-drop gradually dismisses and the wake dynamics is calculated. As we see the distance covered by 1st wake in Fig4 is 3.75m while in Fig5 its about 7m. The distribution of 1st wake in 15m/s is about 47 percent more. There is sudden velocity increase and drop which leads to more pressure drop. In 15m/s the top speed attain is about 115m/s and slowest is about 68m/s. 10m/s shows a relative good result and smooth pressure drop the maximum and minimum velocities attained in 1st wake is 93m/s and 47m/s respectively.

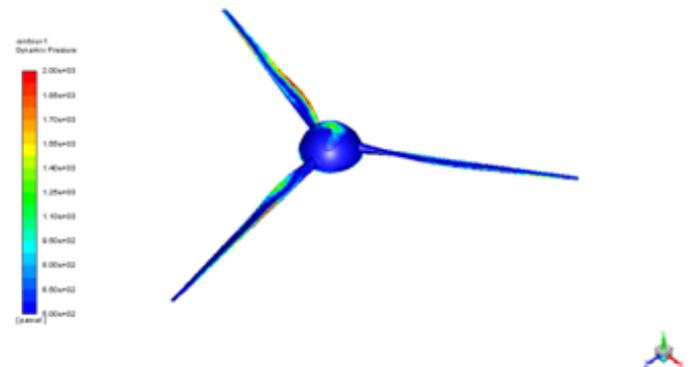


Fig. 6. Pressure Distribution at 10m/s

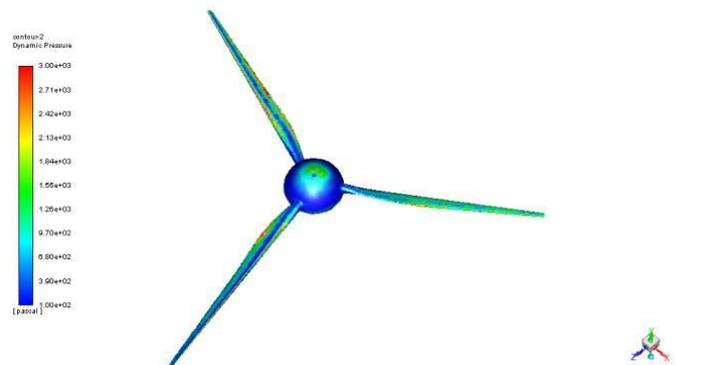


Fig. 7. Pressure Distribution at 15m/s

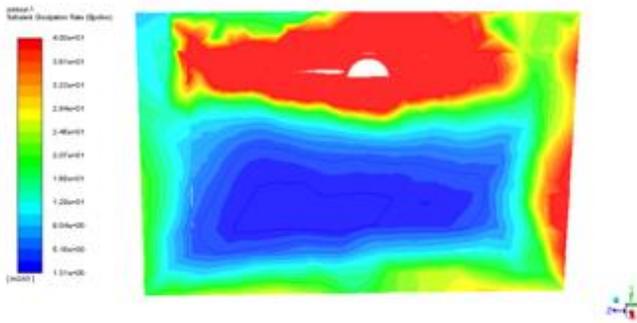


Fig. 8. Turbulent Dissipation Rate ( )

In 2nd wake the distance covered by 10m/s is about 4.7m, in 15m/s it is about 5m/s the percent increase is almost same but the difference here is intensity of the wind and pressure drop which leads to fatigue load on downstream turbine. The maximum velocities at 10 and 15m/s attain in second wake is 63m/s and 98m/s respectively. The minimum velocities are 30m/s and 27m/s respectively. This huge pressure drop at 15m/s show the maximum intensity of the wind where instillation of turbine will cause serious fatigue load.

### B. Pressure Distribution

The pressure inserted by a fluid in motion is known as dynamic pressure. The k-ε model provides a reasonable result in near wake region. The following Fig 6, 7 shows the counter of pressure distribution on the blades of wind turbine. The figures show that the dynamic pressure at thrust section is observed. Pressure distribution in both cases is concentrated at tips and thrust sections of the wind turbine blades. Pressure at front side of wind turbine is observed as compared to the back side. As we can see there more pressure distribution is at 15m/s than 10m/s.

### C. Turbulent Dissipation (ε)

Turbulent dissipation rate is the conversion of turbulence energy into heat energy due to eddies. The contour of the turbulence dissipation is shown in the Fig8, is rated higher near wind turbine causing turbulence and eddies, these eddies are converted into heat energy. Turbulent dissipation rate is rated more in 15m/s as higher pressure drop and turbulence is formed. This dissipation leads to formation of steam on the surface of blades which produce noise and additional load on the blades.

### CONCLUSIONS

In this study, simulations of velocity distribution, pressure distribution and turbulent dissipation of NREL 3MW wind turbine are simulated over k-ε model. These variables are conducted in two different velocities of air i.e. 10m/s and 15m/s. The results obtained are compared for wake calculation downstream and dynamic load. The both results show accurate distribution in near wake region. The result of velocity distributions lead to wake formation. At 15m/s there is significant pressure drop in 2nd wake and distance covered by 2nd wake is almost equal to 1st wake which shows the pressure drop downstream but intensity of the 1st wake is higher. Fluctuations are recorded more and velocity becomes stabilize

more than three wakes downstream. While at 10m/s there is also same pattern but there is intensified and smaller distance covered by first wake as compared to second wake the pressure drop is recorded smaller and velocity is moving towards inlet conditions. The fluctuations of velocity become stable downstream after second wake.

The pressure distribution is calculated by near flow regions of k-ε model which is useful in aerodynamics of wind turbine and optimal wind farm design. The results suggest that dynamic pressure is generated downstream more in 15m/s. The tips and thrust section of wind blades observe more pressure and the profile is almost same in both conditions. In transport equation all three flows are integrated i.e. heat, mass and momentum transfer. Turbulent dissipation rate is useful in calculations of heat transfer but it has effect on the flow. Heat generation leads to velocity and pressure increase and this effect is mostly recorded at near wind turbine regions. The intensity is recorded more in 15m/s which leads to more pressure drop and attain fatigue load on the blades of wind turbine.

### REFERENCES

- [1] "Global wind report – annual market update," GWEC., Tech. Rep., 2016, 2017. [Online]. Available: <http://files.gwec.net/files/GWR2016.pdf>
- [2] E. G. Antonini, D. A. Romero, and C. H. Amon, "Improving cfd wind farm simulations incorporating wind direction uncertainty," *Renewable Energy*, vol. 133, pp. 1011–1023, 2019.
- [3] I. E. A., "World energy outlook," International Energy Agency., Tech. Rep., 2012.
- [4] A. Crespo, J. Hernandez, and S. Frandsen, "Survey of modelling methods for wind turbine wakes and wind farms," *Wind Energy: An International Journal for Progress and Applications in Wind Power Conversion Technology*, vol. 2, no. 1, pp. 1–24, 1999.
- [5] S. McTavish, S. Rodrigue, D. Feszty, and F. Nitzsche, "An investigation of in-field blockage effects in closely spaced lateral wind farm configurations," *Wind Energy*, vol. 18, no. 11, pp. 1989–2011, 2015.
- [6] M. Ghasemian, Z. N. Ashrafi, and A. Sedaghat, "A review on computational fluid dynamic simulation techniques for darrieus vertical axis wind turbines," *Energy Conversion and Management*, vol. 149, pp. 87–100, 2017.
- [7] D. W. R. Zhang, "Aerodynamic analysis of the wind turbine by two different numerical methods," *ICCM2019*, pp. 381–390, Jul. 2019.
- [8] e. a. Choi, Nak, "'cfd study on aerodynamic power output changes with inter-turbine spacing variationfor a 6 mw offshore wind farm.," *Energies*, pp. 7483–7498., Nov. 2014.
- [9] H. Dong, S. Cao, T. Takemi, and Y. Ge, "Wrf simulation of surface wind in high latitudes," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 179, pp. 287–296, 2018.
- [10] K. R. P. C. J.M. Prospathopoulos, E.S. Politis, "Evaluationof the effects of turbulence model enhancements on wind turbine wake predictions we.419," *Wind Energy* 14 (2), pp. 285–300, Feb. 2011.
- [11] R. J. Barthelmie, S. C. Pryor, S. T. Frandsen, K. S. Hansen, J. Schepers, Rados, W. Schlez, A. Neubert, L. Jensen, and S. Neckelmann, "Quantifying the impact of wind turbine wakes on power output at offshore wind farms," *Journal of Atmospheric and Oceanic Technology*, vol. 27, no. 8, pp. 1302–1317, 2010.
- [12] L. Vermeer, J. N. Sørensen, and A. Crespo, "Wind turbine wake aerodynamics," *Progress in aerospace sciences*, vol. 39, no. 6-7, pp. 467–510, 2003.
- [13] K. Grogg, "Harvesting the wind: the physics of wind turbines," *Physics and Astronomy Comps Papers*, vol. 7, 2005.
- [14] "Renewables 2019 global status report," REN21, Tech. Rep., 2019.
- [15] R. J. Barthelmie, S. T. Frandsen, M. Nielsen, S. Pryor, P.-E. Rethore, and H. E. Jørgensen, "Modelling and measurements of power losses and

turbulence intensity in wind turbine wakes at middelgrunden offshore wind farm,” *Wind Energy: An International Journal for Progress and Applications in Wind Power Conversion Technology*, vol. 10, no. 6, pp. 517–528, 2007.

- [16] A. Inc, “Ansys fluent theory guide,” 2013.
- [17] T. Ahmad, “Wind farm coordinated control and optimisation,” Ph.D. dissertation, Durham University, 2017.
- [18] M. H. Baloch, G. S. Kaloi, and Z. A. Memon, “Current scenario of the wind energy in pakistan challenges and future perspectives: A case study,” *Energy Reports*, vol. 2, pp. 201–210, 2016.
- [19] R. Barthelmie, S. Frandsen, K. Hansen, J. Schepers, K. Rados, Schlez, A. Neubert, L. Jensen, and S. Neckelmann, “Modelling the impact of wakes on power output at nysted and horns rev,” in *European wind energy conference*, 2009.
- [20] B. Sanderse, S. Van der Pijl, and B. Koren, “Review of computational fluid dynamics for wind turbine wake aerodynamics,” *Wind energy*, vol. 14, no. 7, pp. 799–819, 2011.
- [21] M. I. C. Cabezon´ D, Sanz J, “A. cfd modeling of the interaction between the surface boundary layer and rotor wake. comparison of results obtained with different turbulence models and mesh strategies,” *European Wind Energy Conference and Exhibition, Marseille*, 2009.
- [22] S.N.P.E.C.Z Radoos KG, Prospathopoulous,JMM,”A cfd modeling is issues of wind turbine waless under stable atmospheric conditions,” *European Wind Energy Conference and Exhibition Marsile* 2008.

#### How to cite this article:

Hammad ur Rahman , Syed Faisal Shah, Abdullah Jamshaid, Muhammad Usama “Flow Velocity Simulation of Wind Turbines by Computational Fluid Dynamics (CFD)”, *International Journal of Engineering Works*, Vol. 8, Issue 02, PP. 73-78, February 2021, <https://doi.org/10.34259/ijew.21.8027378>.

