IOP Conference Series: Earth and Environmental Science

PAPER • OPEN ACCESS

Numerical investigation of aerodynamic performance of a H-Darrieus wind turbine under turbulent flows

To cite this article: Ashley A. Mason *et al* 2020 *IOP Conf. Ser.: Earth Environ. Sci.* **573** 012045

Numerical investigation of aerodynamic performance of a H-Darrieus wind turbine under turbulent flows

Ashley A. Mason[1], R. Harish[1],M.B. Shyam Kumar[1]
[1]School of Mechanical Engineering, Vellore Institute of Technology, Chennai,

Tamil Nadu 600127, India

*Corresponding author

E-mail address: harish.r@vit.ac.in

Abstract. In the present study, numerical investigation is performed on a H-Darrieus wind turbine under turbulent flow conditions. The turbine contains 3 rotor foils, NACA-0012 with 0° angle of attack, placed at 120° apart from each other, with rotor casing close to the airfoil. The inlet is considered to be velocity inlet and outlet to be pressure outlet while the side walls are given symmetric boundary conditions, as the walls are considered to be infinity. The airfoils are rotating at 150 R.P.M.. The meshing is done using Hexa-Dominant element to capture the flow very precisely and inflations are given wherever necessary. The problem is considered as two-dimensional, and the analysis of the model under the condition of turbulent flow is modeled using S.S.T-k-ω turbulence model. Simulation is performed using finite volume solver Ansys Fluent 2019-R2. The performance of the H-Darrieus wind turbine is analyzed, by varying the wind speed and turbine rotational speed. The further investigations are performed by comparing the flow characteristics between a stationary and rotating turbine. The results are analyzed by plotting the streamlines, velocity and pressure contours. The flow is analyzed at 6m/s, 8m/s, 10m/s as inlet freestream velocity and also by varying the rotor dimension, which was initially made as small as possible. The results indicate that the wind speed and rotational speed significantly influences the significant aero characteristics of the model of the wind turbine. The present results are validated and are in good accord with the benchmark results available in literature.

1. Introduction

Off-Shore wind turbines are a crucial part in harnessing renewable energy for producing electricity by the principle of dynamo. The airfoils play major role as it causes air separation which in turn is responsible for lift force and drag force. The flow is at a very low Reynolds number but still turbulence will occur. Several types of turbines are available like VAWT, HAWT and many researches are still being done to harness the wind energy by affecting the stall angles and vortex generators. Rosario et al. [1] did a study on a 2-D CFD code was used to investigate adverse pressure gradient and performance prediction of the turbine by using URANS model [1,2]. Results were compared to an experiment done in a wind turbine and results were seen to be promising. Marco et al. [3] study was based on a 3 bladed rotor NACA 0021. They investigated quantities like angle of attack, rotor torque, tangential and normal forces, stall angles were put to test [3-4]. Flow inspection and study was majorly near the surface of airfoil, tip precisely, for better understanding of vertical-axis turbines.

Nigam et al. [5] did a 2-D study on HAWT, conducted to visualize lift and drag forces [5] for better prediction in performance using k-w SST model using NACA 63-221 airfoil. The results obtained were compared with experimental results which showed that velocity is more on upper surface of airfoil. Lanzafame et al.[6] considered a 3-Dimensional model of HAWT to investigate the performance and evaluate capabilities (BEM Theory) using 2 equation model, SST k-ω. Ion [7] did research extensively on Vertical axis turbine and different ideas and implementing then to analyze the efficiency of the turbine in different conditions for getting the maximum efficiency possible. Similar tests on three bladed vertical Axis Turbine were reported in literature [8-11]. A 2D model of H-Darrius Wind Turbine seen in figure 1 and 2 was made by using NACA 0015 airfoil, place at 120° apart from each other at 0 AOA. The model was scaled down. The diameter of rotor is14mm and the inlet is taken to be approximately 3 times the dimension of the diameter away from the point of center of rotor, which is considered as the point of reference , while the outlet is relatively placed 8 times the diameter of rotor for capturing the effect of rotor accurately. The height of enclosure is 60 mm and is symmetrical about X-axis.

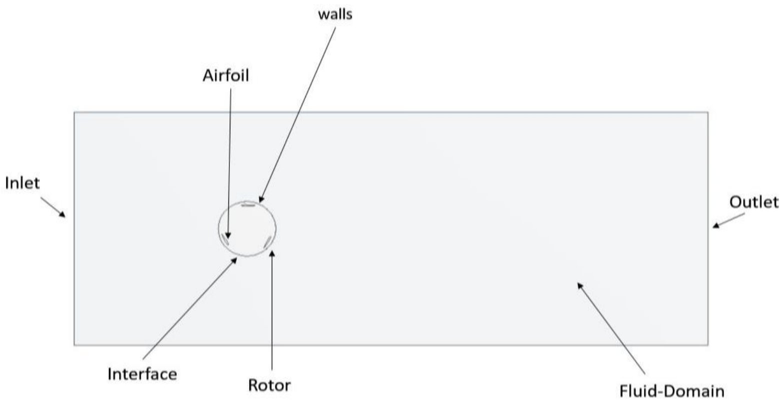

Fig1. Domain of the rotor to be analyzed using Sliding Mesh Technique

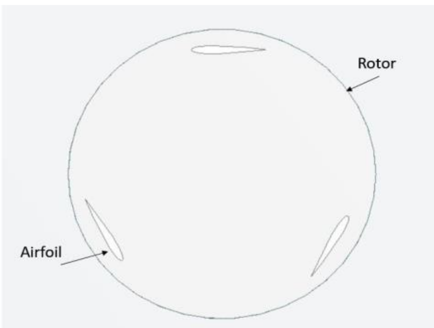

Fig.2 Computational domain with H-Darrieus wind turbine

2. Mathematical modelling-

The following research is carried out by Finite Volume Method (F.V.M). Air is supposed to be flowing with a provided velocity in the simulation, infinitely. The respective study is done by a 2 equation model, which is used to study the variation of pressure and velocity distributions on the airfoil considering turbulent effects. The equations used to solve the model are as follows:

- Continuity Eq.-

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho u}{\partial x}+\frac{\partial \rho v}{\partial y}+\frac{\partial \rho w}{\partial z}=0 \qquad (1)$$

- Momentum Eq.-

X momentum:

$$\frac{\partial \rho u}{\partial t}+\frac{\partial \rho uu}{\partial x}+\frac{\partial \rho uv}{\partial y}+\frac{\partial \rho uw}{\partial z}= -\frac{\partial \rho}{\partial x}+\frac{1}{Re}\left\{\frac{\partial \tau xx}{\partial x}+\frac{\partial \tau xy}{\partial y}+\frac{\partial \tau xz}{\partial z}\right\} \qquad (2)$$

Y momentum:

$$\frac{\partial \rho v}{\partial t}+\frac{\partial \rho vu}{\partial x}+\frac{\partial \rho vv}{\partial y}+\frac{\partial \rho vw}{\partial z}= -\frac{\partial \rho}{\partial y}+\frac{1}{Re}\left\{\frac{\partial \tau yx}{\partial x}+\frac{\partial \tau yy}{\partial y}+\frac{\partial \tau yz}{\partial z}\right\} \qquad (3)$$

Z momentum:

$$\frac{\partial \rho w}{\partial t}+\frac{\partial \rho wu}{\partial x}+\frac{\partial \rho wv}{\partial y}+\frac{\partial \rho ww}{\partial z}= -\frac{\partial \rho}{\partial z}+\frac{1}{Re}\left\{\frac{\partial \tau zx}{\partial x}+\frac{\partial \tau zy}{\partial y}+\frac{\partial \tau zz}{\partial z}\right\} \qquad (4)$$

For Meshing seen in figure 3 and 4, structured grid was used and de-feature sizing was closed. Inflation layer was given near the surface of airfoil for capturing the physics precisely as the rotor rotates. The mesh connections were created manually at the interface so the air passes through the rotor, and doesn't treat it like a solid body. The total mesh was about 2 lakhs with element size as 1mm. Coupled solver was used for convergence study with a time step of 0.01s and the second order technique was used.

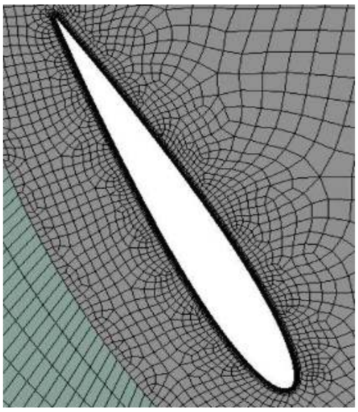

Figure 3 Inflations around the airfoil

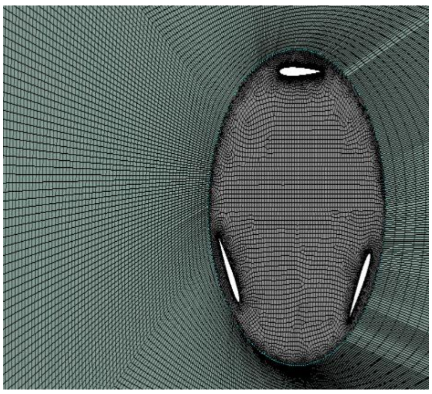

Figure 4 Structured Meshing

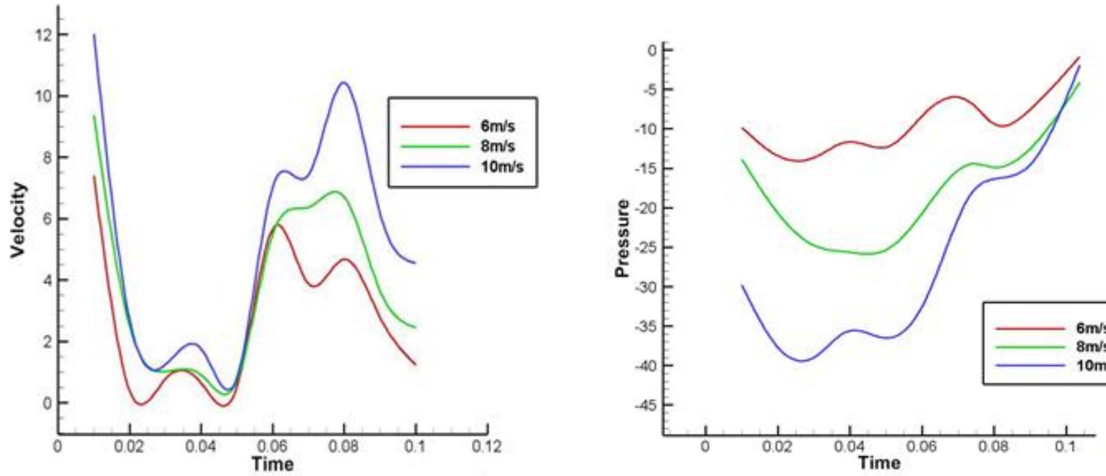

Fig. 5 Pressure vs Time and Velocity vs Time

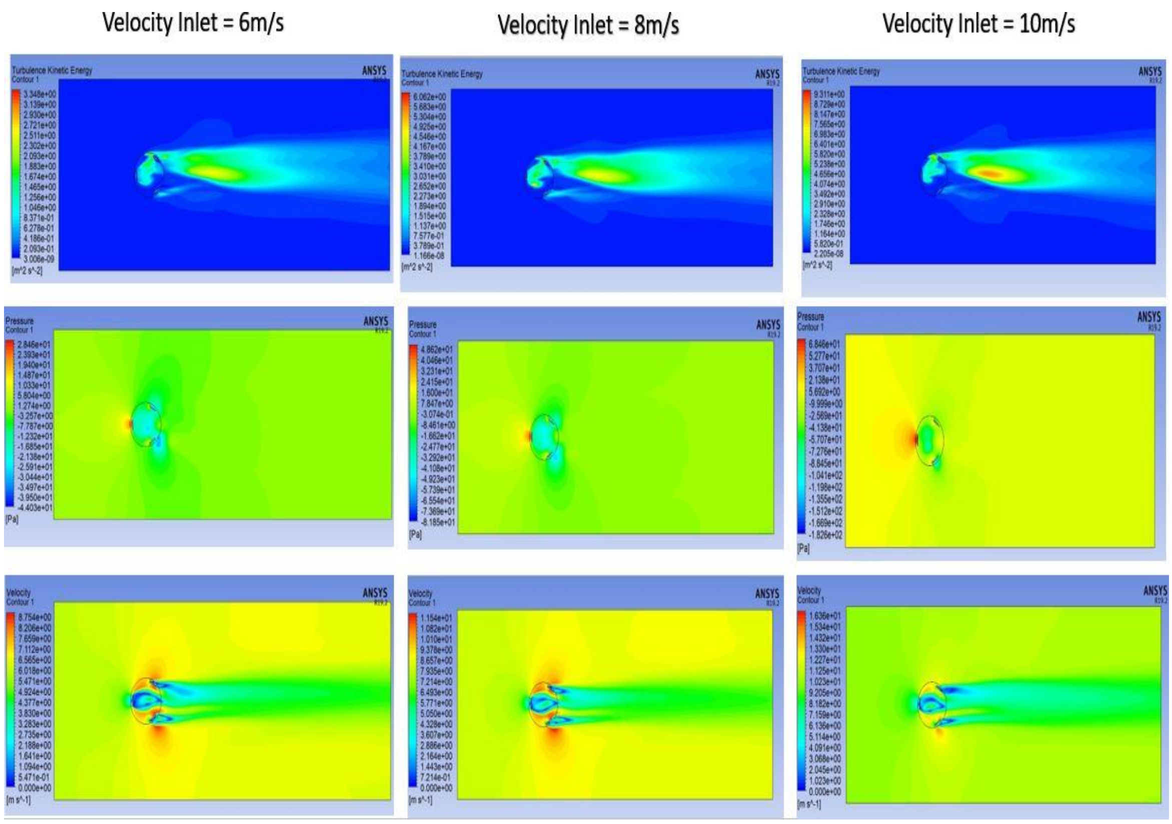

Fig. 6 Contour Plots of Pressure, Turbulent K.E. and Velocity w.r.t different inlet velocities streams

3. Results and discussions

Figure 5 indicates the variations of pressure, turbulent kinetic energy and velocity distributions in the upstream as well as in the downstream side of the modeled turbine for three different inlet velocity values (6/8/10 m/s) . The complete analysis shown is for a 360° revolution of airfoil take the leftmost point of the rotor as the point of analysis i.e. (0.007, 0, 0) where all dimensions are in meter. The comparison of turbulent K.E. in the above figure indicates that the intensity of turbulent K.E. significantly increases with an increase in inlet velocity at the downstream side of the wind turbine. This is due to the increase in downstream wake generated due to the flow separation. It is also observed that the pressure distribution intensity is highest for an inlet flow velocity of 10 m/s when compared to 6,8 (m/s) velocity . The velocity contours indicates the vortex shedding pattern at the downstream side of the modeled turbine and the velocity distributions increases with increase in inlet flow velocity. Moreover, for a high velocity of 10 m/s the flow patterns are attached along the surface of the wind turbine and this is due to the highly turbulent flows generated at high flow velocities. Figure 6 indicates the variations of pressure and velocity for the transient time considered in the present study. The analysis was setup for a time period of 1 sec, by analyzing the pressure and velocity of each airfoil by changing the angle by 2.7° consistently. It is observed that the magnitudes of pressure and velocity values increases with increase in inlet flow velocities. The trend also indicates that the turbulent flow is highly non-linear and the average pressure distribution near the wind turbine is increased by 48% and 63% for inlet flow velocities of 8,10 (m/s) in comparison to 6 m/s. Similarly with the increase in the inlet velocity increases the velocity distributions near the wind turbine by 42% and 57% for inlet flow velocities of 8 m/s and 10 m/s.

4. Conclusions

The numerical study is carried out on a H-Darrieus wind turbine under turbulent inlet flow conditions of 6,8,10 m/s. The turbine is designed with three rotor foils of NACA-0012 with 0° angle of attack placed at 120° apart from each other, with rotor casing close to the airfoil. The inlet is considered to be velocity inlet and outlet to be pressure outlet while the side walls are given symmetric boundary conditions, as the walls are considered to be infinity. The airfoils were considered rotating at 150 R.P.M. The problem is formulated as two-dimensional flow and the turbulent flow is modeled using S.S.T. k-ω turbulence model. The aero performance of H-Darrieus wind turbine is investigated by varying the wind speed and turbine rotational speed. The comparison of turbulent kinetic energy indicates that the intensity of turbulent kinetic energy increases with increase in inlet velocity at the downstream side of the wind turbine. This is due to the increase in downstream wake generated due to the flow separation. It is also observed that the pressure distribution intensity is highest for an inlet flow velocity of 10 m/s when compared to 6 and 8 (m/s). Moreover, for a high velocity of 10 m/s the flow patterns are attached along the surface of the wind turbine and this is due to the highly turbulent flows generated at high flow velocities. The results indicate that the turbulent flow is highly non-linear and the average pressure distribution near the wind turbine is increased by 48% and 63% for inlet flow velocities of 8 m/s and 10 m/s in comparison to 6 m/s. Similarly the increase in inlet velocity increases the velocity distributions near the wind turbine by 42% and 57% for the inlet flow velocities of 8 m/s and 10 m/s. The results from the present study will be useful in designing H-Darrieus wind turbine with improved aerodynamic performance suitable for renewable energy applications.

References

1. Rosario L., Stefano Ma., Michele M., "2D CFD Modelling of H-Darrieus Wind Turbines using a Transition Turbulence Model", *Energy Procedia* Vol **45**, 2014, pp.131 – 140

2. Niels N. S., "CFD Modelling of Laminar-turbulent Transition for Aerofoils and Rotors Using the g –γ Req Model", Wind Energy,Vol **12**:,2009, pp. 715–733

3. Marco R., Alessandro E., Ernesto E., "The Darrieus wind turbine: Proposal for a new performance prediction model based on CFD", *Energy* **30**, 2011, pp.4919-4934

4. Bravo R., Tullis S., Ziada S. , "Performance Testing of a small Vertical-Axis Wind Turbine, Mechanical Enginnering Department" , McMaster University, https://www.academia.edu/download/54779218/Performance_Testing_of_a_Small_Vertical-Axis_Wind_Turbine.pdf (2007)

5. Nigam P.K., Tenguria N., Pradhan M.K., "Analysis of horizontal axis wind turbine blade using CFD", *International Journal of Engineering, Science and Technology*, Vol. **9**, No. 2, , 2017,pg. 46-60

6. Lanzafame R., Mauro S., Messina M., "Wind turbine CFD modelling using Correlation based transitional model", *Renewable Energy* **52**, 2013 pp. 31-39

7. Ion.P, "Double-Multiple Streamtube Model for Studying Vertical-Axis Wind Turbines", J Propulsion https://doi.org/10.2514/3.23076 (2012)

8. Biswas A., Gupta R., "Computational fluid dynamics analysis of a twisted three-bladed H-Darrieus rotor", *Journal of Renewable and Sustainable Energy* **2**,2010,pp. **04311**1..

9. Ziada S, Tullis S., Bravo R.,. "Performance Testing of a Small Vertical-Axis Wind Turbine". Mechanical Engineering Department, McMaster University.

10. Sayed M, Lutz Th, Kramer E, Shayegan Sh, Wuchner R, Aeroelastic analysis of 10 MW wind turbine using CFD–CSD explicit FSI-coupling approach, Journal of Fluids and Structures, 87, 2019, pp.354-377.

11. Kozmar H, Laschka B, Wind-tunnel modeling of wind loads on structures using truncated vortex generators, Journal of Fluids and Structures, 87,2019,pp.334-353.