Small Horizontal Axis Wind Turbine under High Speed Operation: Study of Power Evaluation

Magedi Moh.M. Saad1, Sofian Bin Mohd2, Mohd Fadhli Bin Zulkafli3, Aslam Bin Abdullah4, Mohammad Zulafif Bin Rahim5, Zulkhairi Bin Subari6, Nurhayati Binti Rosly7
1,2,5,6,7Aircraft System and Design Research (ASDR), Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia
3,4Aerodynamic and Propulsion Research (APR), Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia

Abstract. Mechanical energy is produced through the rotation of wind turbine blades by air that convert the mechanical energy into electrical energy. Wind turbines are usually designed to be use for particular applications and design characteristics may vary depending on the area of use. The variety of applications is reflected on the size of turbines and their infrastructures, however, performance enhancement of wind turbine may start by analyzing the small horizontal axis wind turbine (SHAWT) under high wind speed operation. This paper analyzes the implementations of SHAWT turbines and investigates their performance in both simulation and real life. Depending on the real structure of the rotor geometry and aerodynamic test, the power performance of the SHAWT was simulated using ANSYS-FLUENT software at different wind speed up to 33.33 m/s (120km/h) in order to numerically investigate the actual turbine operation. Dynamic mesh and user define function (UDF) was used for revolving the rotor turbine via wind. Simulation results were further validated by experimental data and hence good matching was yielded. And for reducing the energy producing cost, car alternator was formed to be used as a small horizontal wind turbine. As a result, alternator-based turbine system was found to be a low-cost solution for exploitation of wind energy.

1. Introduction
Nowadays, electrical energy from wind turbines is classified as most efficient and worthier than other green energy resources. This remarkable development can be primarily attributed to the development in the large wind turbines technology. However, small wind turbines are still in their developmental stage. In the past decades, the installed power capacity of large wind turbines displayed an annual growth rate of about 30% whereas the small wind turbines revealed only 9% [1]. Hence, there is immediate need to explore the huge potential of small wind turbines to exploit towards generations of electricity with lesser expense as there has been no great concern in power generation sector as small wind turbines are costly.

Generator is the main component of small wind turbines, which converts the mechanical energy into electrical one. Thus, by reducing the cost of generators in such turbines would certainly lower their overall cost, making them cost competitive. A major reason for the lessened utilization of big wind turbines is due to the fact that the current systems make used of high quality generators that are expensive. The earlier study of the energy has successfully yield commercially available small wind turbines which shows that the cost of generated electricity the from small wind turbines are the most cost effective [2].
The objective of this paper is to evaluate the power produced from small wind turbine by using automotive alternator, and determines the feasibility of installing a small horizontal axis wind turbine on top of an automotive car and subsequently validating the results by using computational fluid dynamics (CFD) software. The validation between numerical simulation and experimental test is based on wind speed, in order to achieve good matching between numerical simulation and experimental test which are the main contributions of this paper.

Utilizations of a vehicle alternator as wind turbine generator offers multifarious applications including automobiles, tractors, and other industrial transportation systems with wide operating speed range (400 to 8000 rpm) [5] that are available in many standard output voltages (12, 24, 32 or 48 V). According to Ani et al. [6], the efficiency of the alternator is generally low, where the maximum efficiency (~ 54%) occurs at lower speeds (~1500 rpm). This is advantageous for various low speed applications since the efficiency at higher speed (~2500 rpm) reduces significantly (~45%). Typically, the output power of a vehicle alternator operating at 2500 rpm with 14 V DC is ~ 600 W. The power output achieved at 1000 rpm was found to be 150 W with maximum efficiency (~54%) at 1300 rpm [1, 8].

In this study, a 14 V DC alternator was used as a cheaper alternative. The output voltage is usually maintained at 14 V DC by an internal regulator that samples the voltage and adjusts the field current accordingly. The regulator maintains the desired voltage by varying the duty-cycle of the pulse width modulated (PWM) voltage applied to the field winding. By increasing the electrical load in the vehicle, the output voltage drops as more currents are drawn from the alternator. Whenever the electrical load drops, the regulator detects it and raises the voltage by enhancing the duty cycle to increase the field current and vice versa [9]. The rotor speed measurement was taken using of Tachometer, while the measuring of current output is done after connecting load resistance to the alternator for evaluating power output performance for the system.

Most of the aerodynamics problem involving wind turbine are being tackled by CFD using the Reynolds-averaged Navier–Stokes (RANS) method with different turbulence models [3]. To solve such problems, various numerical schemes are developed including SIMPLE, SIMPLEC, PISO and COUPLED [4]. In this study, the SIMPLE scheme of the ANSYS FLUENT software is used to solve the flow around the SHAWT.

The simulation is evaluated using the following expression for the power:

- **Experimental Power formulas:**
  \[
  \text{Power Output} = (I) \times (V)
  \]
  (1)
  where V is the voltage (Volts), I is the current (Ampere)

- **Calculation Power formulas:**

  \[
  \text{theoretical torque} = 0.5 \times \rho \times A \times V^2
  \]
  (2)

  \[
  \text{actual torque} = 0.5 \times \rho \times A \times (\omega \times r)^2
  \]
  (3)

  \[
  \text{angular velocity} (\omega) = \left(\frac{2\pi}{60}\right) \times \text{rpm}
  \]
  (4)

  \[
  \text{torque coefficient} C_q \equiv \frac{\text{actual torque}}{\text{theoretical torque}}
  \]
  (5)

  \[
  \text{Tip Speed Ratio} (\lambda) = \frac{\text{angular velocity} (\omega) \times r}{V_{in}}
  \]
  (6)

  \[
  \text{power coefficient} C_p = \lambda \times C_q
  \]
  (7)

  \[
  \text{power output} = 0.5 \times \rho \times A \times C_p \times V^3
  \]
  (8)

where \( A \) is the surface area (m^2), \( V \) is the wind speed (m s^{-1}), \( \rho \) is the air density (kg m^{-3}) [10, 11].
2. **Experimental procedure**
In the experiment, the fabricated SHAWT (rotor diameter of 0.36 m, rotor width of 0.04 m, blade span of 0.117 m, and blade chord of 0.035 m) is fixed on a car and positioned to face the incoming wind. Figure 1 illustrates the final position of the entire mode on the car. The experimental data obtained from this experiment is then compared with the simulation results obtained from the CFD simulation.

![Figure 1. Installation whole wind turbine system on car body](image)

3. **CFD analysis**
The aerodynamic behavior of a seven-blade SHAWT is observed by the simulation conducted using ANSYS FLUENT. The wind speed at computational domain entrance was kept at a constant rate. The rotational speed of the turbine (rotating zone) was measured to evaluate the turbine performance. Three dimensional Reynolds averaged Navier-Stokes equations for viscous, incompressible, and continuous flow was employed. In this flow pattern, the temperature variations over the flow field were kept nearly constant. Thus, the energy equation was entirely eliminated [12].

The wind turbine model was developed by using SolidWorks, which comprised of stationary domain (cylinder) for the surrounding air, the distance from the rotor to the inlet, top and outlet was 3 times, 3 times and 7 times the diameter of blade respectively as shown in Figure 2. (b) and a rotor turbine as shown in Figure 2. (a) [13-14].
In this study, a non-uniform, unstructured mesh was used with a considerably medium mesh implemented in regions of high gradients. A tetrahedron element was selected as the mesh for both the stationary and rotating domain.

![Tetrahedral meshes in the model](image.png)

Figure 3. Tetrahedral meshes in the model

Figure 3 depicts the planer cross-sectional view of the mesh layout for the turbine and the cylindrical domain. The total number of mesh elements were 354622 (tetrahedron) and 3979649 (triangular interior faces) respectively. Figure 4 displays the boundary conditions applied to the computational domain. Three types of boundary conditions were used; namely walls, velocity inlet, and pressure outlet [15]. The flow properties at sea level was applied to all regions. Table 1 enlists the simulated feature of flow field.
The rotation of the SHAWT model was made possible by using dynamic mesh and User Defined Function (UDF). The physical properties such as mass properties and rotation axis of the SHAWT are defined in the UDF written in C language.

4. Results and discussions

4.1. Simulation Results

Figure 5 illustrates the relationship between the wind speed and rotor rpm at different time step for two cases; incoming wind speed of 33.33 m/s and 22.22 m/s. For both cases in the simulation, at 0.002 sec the rotor started to rotate and the rpm kept on increasing as the time step increases.

The blades are specially curved-shape, similar to the airfoil wings on a plane. When the wind blows the blades, it moves them upward by the force produced due to pressure difference, leading the turbine blades to start spinning. Figure 6 illustrates the different pressure between wind speed 22.22 m/s and 33.33 m/s when the rotor started to rotate, in Figure 6 (a) the maximum pressure at the front rotor hub in black circle around 1100 Pa and around -4000 Pa at tip blade as clearly in black circle while in Figure 6 (b) in black circle the maximum pressure at the front rotor hub around 590 pa and around -2000 pa at tip blade in circle.

Concentration effect can be clearly seen by wind speed increasing due to flow separation at the sharp corners of the blades when flow passes through the rotor. Axial direction wind speed increases due to curvature blade then decreases slowly in the downstream due to the blockage effect by blade walls as clearly illustrated in first rotation in black circle in Figure 7 (a). After the rotor started to rotate, the vortices started to develop in the rotor downstream at its second and third rotation as shown in Figure 7 (b) and (c). These vortices affected the aerodynamic performance, and hence turbine power output.
Figure 5. RPM at different time step

Figure 6. Pressure contour in the Y-Z plane at (a) Wind speed 33.33 m/s (b) Wind speed 22.22 m/s

Figure 7. Wind speed contour and streamline in the Y-Z plane at different time step
4.2. Experimental Results

Experimental data from small axis wind turbine was used to validate the numerical results. This experiment was conducted using a car driven on Johor Bharu (Malaysia) highway, about 12 km from a start point to an end, the car speed was consistently set similar to numerical simulation speeds. Figure 8 exhibited the whole turbine system installed on the car’s body for the test.

![Installed turbine during test](image)

Figure 8. Installed turbine during test

In wind turbine, the rotor efficiency according to the number of blade rotation was achieved at this speed, and in addition to that, drag, torque and the power output heightens with the increasing wind speed. The maximum drag was produced numerically at maximum speed (33.33 m/s) is 59 N, whereas this value of drag did not greatly affecting the performance of the car. Figure 9 demonstrates the wind speed dependent variation in the effective rpm and torque that obtained in experiment test.

![RPM and Torque of the turbine rotor](image)

Figure 9. RPM and Torque of the turbine rotor
Figure 10 compares the values of power output obtained from both the numerical simulation and experiments as a function of wind speed. As the wind speed increase, the power output was also increase as illustrated. Both numerical simulation and experimental test show a similar trend in this Figure. The maximum power produced by CFD was discerned to be 194.84W at the maximum speed of 33.33 m/s, while the maximum power produced by the system without the alternator was 183.674W at same speed depending on the measurement of rpm, whereas, the maximum power produced by the entire system (fan connected with automotive alternator) was 107.46 W. The observed difference of the experimental results from the numerical simulation was mainly attributed to the following: (i) design of automotive alternator for high rpm application; (ii) wire resistance and internal resistance for automotive alternator; (iii) In automotive alternator, a belt system is used to increase the performance, whereas this model was made without any belt system due to lack of equipment; (iv) regulator and rectifier inside the automotive alternator prevent the increase in the voltage [1, 6]. From the experimental results, the 24 V DC alternator is more efficient to be utilize as generator for small wind turbine applications to fit the high-power output produced.

5. Conclusion
This paper evaluated the power output of a SHAWT under high wind speed operation. An automotive alternator was used as generator for SHAWT application. Such turbine was constructed and installed on a top of a car for further experimentation. The maximum power produced experimental was 183.67 W at wind speed 33.33m/s, For the every characteristic of the used rotor geometry and wind speed in the validation, simulations was conducted numerically and the maximum power output was found to be 194.48 W, hence noting that the numerical simulation results were found satisfactory and matching the practical outcomes. Therefore, it is noteworthy to deduce that the simulation environments are sufficiently accurate to fit our work. The idea of evaluating a SHAWT at high speed is demonstrated to be beneficial in terms of low cost wind energy generation, devoid of skilled manpower and easy maintenance. It is established that despite its inherent low efficiency, the vehicle alternator system can be a viable alternative for high wind speed application.
Acknowledgement
The present work has been supported by the Ministry of Education Malaysia under Exploratory Research Grant Scheme (Vote No. E034).

References
[1] Ani, Samuel Ofordile. Low cost small wind turbine generators for developing countries. TU Delft, Delft University of Technology, 2013
[2] Goh, Seach Chyr, Sethu Raman Boopathy, Chidambaresan Krishnaswami, and Jörg Uwe Schlüter. "Tow testing of Savonius wind turbine above a bluff body complemented by CFD simulation." Renewable Energy 87 (2016): 332-345.
[3] Gebreslassie, Mutalem G., Stephanie O. Sanchez, Gavin R. Tabor, Michael R. Belmont, Tom Bruce, Grégoire S. Payne, and Ian Moon. "Experimental and CFD analysis of the wake characteristics of tidal turbines." International Journal of Marine Energy 16 (2016): 209-219.
[4] Cao, Han. Aerodynamics analysis of small horizontal axis wind turbine blades by using 2D and 3D CFD modelling. Diss. University of Central Lancashire, Preston, 2011
[5] Fernández, H., A. Martínez, V. Guzmán, and M. Giménez. "Wind turbine generation system implemented with a car alternator for use in isolated locations." In International conference on renewable energies & power quality, ICREPQ, Barcelona, p. 6. 2004.
[6] Ani, Samuel Ofordile, Henk Polinder, and Jan Abraham Ferreira. "Small wind power generation using automotive alternator." Renewable Energy 66 (2014): 185-195.
[7] Arafat, Md Yeasin, Mahtab Murshed, Md Mahamudul Hasan, and M. Abdur Razzak. "Design aspects and performance analysis of inner and outer rotor permanent magnet alternator for direct driven low-speed wind turbine." In Advances in Electrical, Electronics, Information, Communication and Bio-Informatics (AEEICB), 2016 2nd International Conference on, pp. 604-609. IEEE, 2016.
[8] Boldea I. Variable speed generators. Boca Raton, FL: CRC Press; 2006.
[9] Kassakian IG, Wolf H-C, Miller JM, Hurton CJ. Automotive electrical systems circa 2005. IEEE Spectrum August, 1996:22e7.
[10] Manwell, James F., Jon G. McGowan, and Anthony L. Rogers. Wind energy explained: theory, design and application. John Wiley & Sons, 2010.
[11] Magedi. Saad,. Sofian Bin Mohd, Mohd Fadhli. “A Survey on the Use of Ram Air Turbine in Aircraft”. Presented on 7th international conference on mechanical manufacturing engineering, Yogyakarta 01-3 August 2016.
[12] Sanderse, Benjamin, S. P. Pijl, and B. Koren. "Review of computational fluid dynamics for wind turbine wake aerodynamics." Wind Energy 14.7 (2011): 799-819.
[13] Lee, Nak Joong, et al. "Performance study on a counter-rotating tidal current turbine by CFD and model experimentation." Renewable Energy 79 (2015): 122-126.
[14] Park, Sewan, Sunho Park, and Shin Hyung Rhee. "Influence of blade deformation and yawed inflow on performance of a horizontal axis tidal stream turbine." Renewable Energy 92 (2016): 321-332.
[15] Hartwanger, David, and Andrej Horvat. "3D modelling of a wind turbine using CFD." NAFEMS Conference, United Kingdom. 2008.