VOL. 11, NO. 18, SEPTEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

AERODYNAMIC CHARACTERISTICS EVALUATION OF WIND TURBINE DUCTING SYSTEM PERFORMANCE FOR PICKUP TRUCK

Sofian Mohd[1], Nik Samsul Bahari Mohamad Zainu[1], Nurhayati Rosly[2] and Aslam Abdullah[2]
[1]Faculty of Mechanical & Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia
[2]Aerodynamics and Propulsion Research Group, Faculty of Mechanical & Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia
E-Mail: sofian@uthm.edu.my

ABSTRACT

Road transportation is one of the biggest sources of pollution contribute to poor air quality and climate change. Through green and clean vehicle invention, air pollution from vehicles can be significantly reduced. In the present study, air flow characteristics in the pick-up truck's wind turbine ducting system were investigated. The wind turbine system was located at the roof of the pickup truck body. The 3-Diamensional wind turbine ducting models were developed using Inventor and analyzed by using ANSYS Fluent software. The air velocity at duct's inlet was set at 16.7 m/s, 25 m/s and 33.3 m/s which are equivalent to pickup truck cruising speed of 60 km/h, 90 km/h and 120 km/h, respectively. At the ducting neck region, the air velocity was slightly increased. The air velocity was significantly increased i.e. three times of inlet velocity with additional of flow guider at the inlet. Wind turbine model can be developed using the optimum blade configuration determined from the obtained simulation results. The improvement of blade design and configuration, and energy conversion system would increase the produced electrical power.

Keywords: wind turbine, pickup truck, aerodynamic performance, simulation, ducting models.

INTRODUCTION

The fuel combustion related with transportation results in emissions of pollutants causes damage to human health, agriculture and global climate change. In solving some of the problems associated with gasoline-powered vehicles, an electric-powered vehicle has been produced but such vehicles are not yet in common use. Today, there are three distinct types of electric vehicles available in market: battery electric vehicles, plug-in hybrids electric vehicles, and hybrids electric vehicles. Recently, many inventions and researches have been conducted regarding to the idea of mounting wind turbine system on electric-powered vehicles.

In 2003, Mitchell came out with the idea to design a power system for powering a vehicle having an airflow channeling device and an electric alternator connected to the fan unit mounted on the vehicle. The objective of the invention is to design an air- or water-power system that can charge or maintain the charge on a vehicle battery while the vehicle is in motion. It can provide electricity to operate an electric motor to run a vehicle. Mitchell stated that, the advantages of his design are, it provides for more aerodynamic and hydrodynamic shape to the power system and able to lower the added drag force of the power system on the vehicle [1].

Deets then suggested a design that is quite similar with what Mitchell has done where a method for charging a vehicle battery using wind power whiles the vehicle is in motion. The system consist of the shrouded enclosure comprising an air intake formed by controllable shrouds, a turbine, an electricity generating device, and a discharge outlet is mounted to a vehicle roof. The advantage of this invention is the ability to utilize the controllable shrouds to both enhance air intake when open and reduce aerodynamic drag when closed. Another advantage of the invention is the ability to increase turbine bearing life through closure of the shrouds, thereby reducing the amount of time the turbine turns [2].

Kousoulis [3] had come out with his invention entitle "Motor Vehicle with Wind Generator Device". The general idea of this invention is to incorporate the wind turbine in producing the electricity using the wind speed created by vehicles. The electricity generated can be connected to the vehicle power accessories and batteries in order to charge them and eliminate the drainage on a main vehicle battery [3]. For this invention, Kousoulis stated that the placement of the supplemental power plant is not significant. The supplemental power plant may be located on any part of a vehicle, including a roof, a side, a front, a rear, and underneath the vehicle.

In this study, the evaluation of the aerodynamic performance of wind turbines that are mounted on the roof of a moving pickup truck is carried out. It involves the modeling and simulation of the wind turbine using ANSYS FLUENT to investigate the performance of the wind turbines based on the velocity of the air flow in the duct of the turbine and the impact on aerodynamic performance pickup truck.

TURBINE DUCTING SYSTEM

Modelling of wind turbine ducting system

One part at turbine duct called a flow guider has been introduced inside of the model [4] to increase the velocity of air and eliminate the moment occurring in an opposite direction of that rotation on the convex blade of the rotor without making any change in the basic structure of the conventional Savonius wind rotor. The lengths and location of the flow guider at neck region have been set according to a research conducted by Phillips which stated

ARPN Journal of Engineering and Applied Sciences

that an augmentation of 1.38 on power coefficient (*Cp*) of Savonius wind rotor is achievable by a diffuser of Exit-Area-Ratio (EAR) of 2.22. An overall length to diameter meter (L/D) of 0.35. These values are used for the diffuser or duct design.

$$EAR = \frac{A_e}{A_i} \qquad (1)$$

Ae = diffuser/duct exit areas,
Ai = diffuser/duct inlet areas

Based on calculation above, the ducting system is having an inlet duct, inlet neck region and outlet widths of 1000 mm, 260 mm and 580 mm respectively. The length of the turbine duct is taken to be 1000 mm for it to cover the whole span of the turbine as shown in Figure-1.

In this study, the Savonius blade model of the vertical axis wind turbine (VAWT) type will be used with respect to the air flow at the roof of the pickup truck body and axis of the wind turbine rotor. The whole body size of the system is set to be 0.1m in height and 1m width (0.1 m^2 front areas). Air velocity of turbine duct inlet for CFD simulation of flow on the turbine model will set at 16.7 m/s, 25 m/s and 33.3 m/s which are equivalent to pick-up truck cruising speed of 60 km/h, 90 km/h and 120 km/h, respectively. Velocity of air before hitting the turbine blade will be evaluated at the neck region of turbine duct.

CFD method and setup

A basis on previous study by Golecha et al. [5] is used as the standard SST-omega model was selected as the turbulence model for numerical simulation.

Table-1, Table-2, Table-3 and Table-4 shows the solver setup, viscous model and Turbulence model controls for present simulation respectively. A few assumptions have been made in the present simulation. The assumptions are air flow was steady state condition with constant velocity at inlet, constant pressure outlet, no slip wall boundary conditions at the around inside turbine duct, flow guider and turbine blade.

Table-1. Solver setting.

CFD Simulation	3ddp (3-D Double Precision)
Solver	
Solver	Pressure Based
Space	3D
Time	Steady
Velocity Formulation	Absolute

Table-2. Viscous model and turbulence model settings.

Turbulence Model	SST k-ε
k-epsilon Model	Standard
Operating Conditions	Ambient

Table-3. Boundary condition settings.

Boundary Conditions		
Velocity Inlet	Magnitude (Measured normal to Boundary)	16.7 m/s, 25 m/s and 33.33 m/s (constant)
Turbulence Specification Method	Intensity and Viscosity Ratio	
Turbulence Intensity	1.00%	
Turbulence Viscosity Ratio	20	
Pressure Outlet	Gauge Pressure magnitude	0 Pascal
Gauge Pressure direction	normal to boundary	
Turbulence Specification Method	Intensity and Viscosity Ratio	
Backflow Turbulence Intensity	10%	
Backflow Turbulent Viscosity Ratio	10	
Wall Zones	- Duct surface-no slip wall B/c - Blade surface- no slip wall B/C -Flow Guider surfaces – no slip wall B/C	
Fluid Properties	Fluid Type	Air
Density	ρ = 1.175 (kg/m^3)	
Kinematic viscosity	ν = 1.7894×10^(-5) (kg/(m·s))	

Table-4. Solution controls.

Equations	Flow and Turbulence
Discretization	• Pressure: Standard • Momentum: Second Order Upwind • Turbulence Kinetic Energy: Second Order Upwind • Turbulence Dissipation Rate: Second Order Upwind
Monitor	Residuals & Drag Coefficient
Convergence Criterion	- Continuity = 0.001 - X-Velocity = 0.001 - Y-Velocity = 0.001 - k = 0.001 - epsilon = 0.001

VOL. 11, NO. 18, SEPTEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

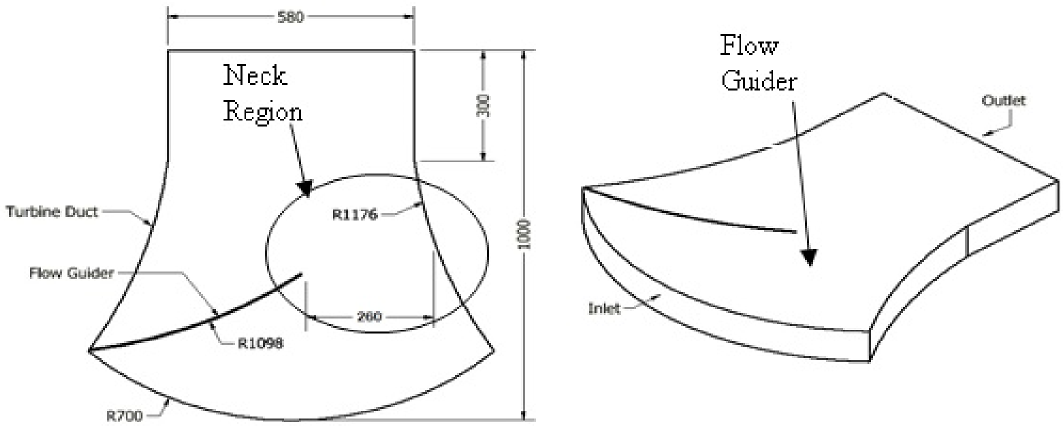


Figure-1. Turbine duct system present model.

WIND TURBINE ROTOR

Location of wind turbine

Based on Sofian *et al.* [4], the best position to install the wind turbine system is at the top roof of vehicle because of the highest velocity of air around it [6-7]. Wind turbine model was created and positioned on the top of pickup truck body in a way to investigate the power of output system. Figure-2 and Figure-3 shows the wind turbine install on the top roof of the pickup truck. The size of the wind turbine and pickup truck models are similar to the size used in the first case. The inlet area for the wind turbine system is set to be 1000 x 100 millimeter square.

Figure-2. Isometric view of the wind turbine on top of a pickup truck model.

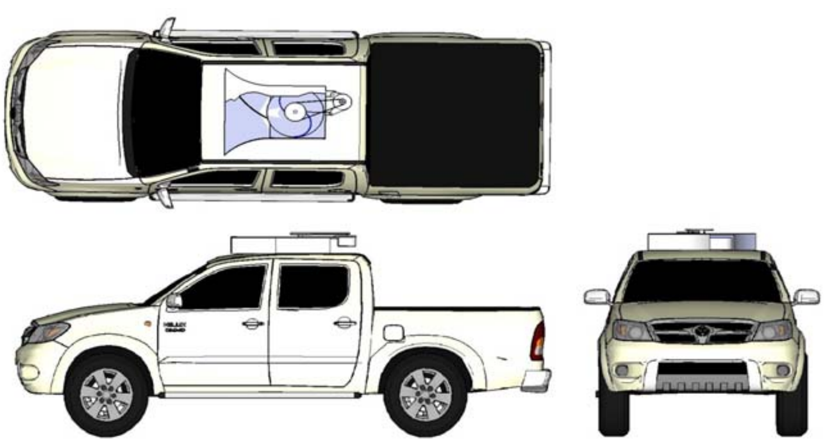


Figure-3. Orthographic view of the wind turbine on a model of pickup truck.

VOL. 11, NO. 18, SEPTEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Analysis of aerodynamic performance of a ducting system

Performance of ducting system between proposed model by Mohd Sofian et al. with the present model was evaluated to determine the effect of turbine ducting design. The evaluation of this study based on the characteristic of the air flow inside the turbine's duct. Figure-4 shows the characteristic of the velocity of air flow inside both model. During the CFD simulation, inlet velocity both of model was set to be 25 m/s which assume a pickup truck is travelling at 90 km/h. Result of the simulation shown that the velocity of air flows inside the present model before it hits the turbine blade is approximately 78.5 - 85 m/s and velocity value for Rexca model is only approximately 63.3 - 69.7 m/s. It is obvious that the velocity of air flow at neck region for this study is higher and better then previous study by Mohd Sofian et al.

It can be seen from the figure that air velocity inside turbine duct with flow guider before it hits the turbine's rotor was significantly increased three times (25 m/s at inlet and 78 m/s at neck region) with additional of flow guider at the inlet of turbine duct. The ducting design will be fabricated to accommodate the wind turbine rotor system.

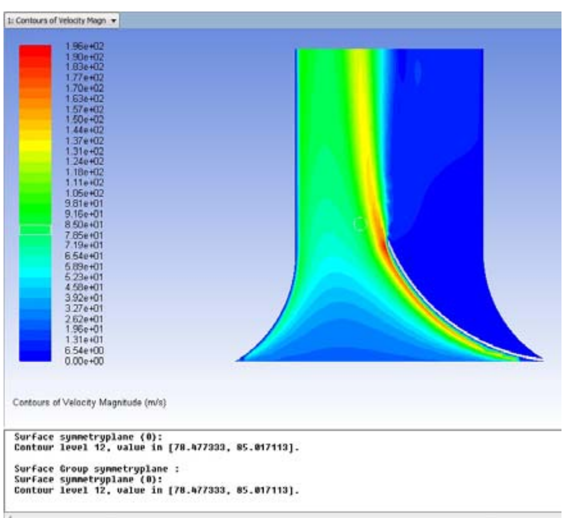
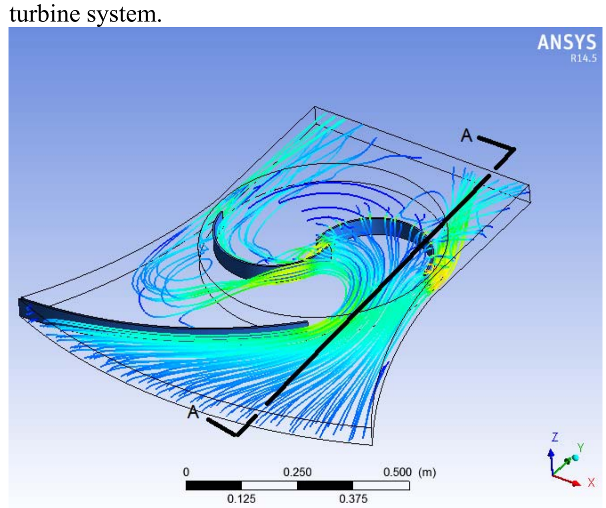
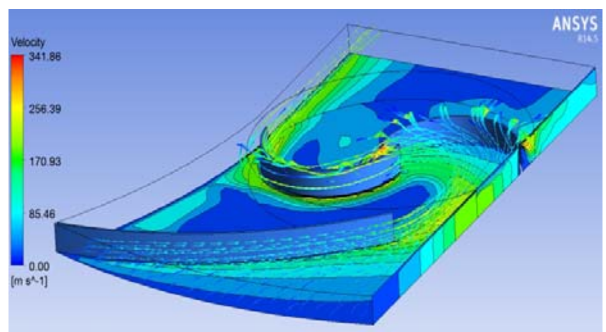


Figure-4. Characteristic of the velocity of air flow inside.

Figure-5 shows the simulation result of the Savonius wind turbine rotor system. The purpose of this investigation is to evaluate the effect of the flow guider plate at duct and to identify the speed of the air flow inside the ducting system with turbine rotor. For this simulation, the inlet velocity has been set at 16.7 m/s, 25 m/s and 33.3 m/s. The result shows that the velocity of air at the neck of duct before it hits the turbine rotor is increase around three times for each speed at inlet. The figure also show that air velocity inside the duct system before it hit the turbine blades decreased about 10-15 % because the blade was acts as a barrier that prevents air flow to enable it to rotate and generate power for wind turbine system.



(a)



(b)

Figure-5. a) Streamline velocity vectors air flow, b) section A-A at neck region.

CONCLUSIONS

It is clear that the wind turbine is a machine that is able to convert kinetic energy into electrical energy in producing electricity. The air velocity characteristics inside the wind turbine system and output power of this system were numerically and experimental investigated. To study the air flow characteristic inside the turbine duct at a certain speed before fabrication stage, the wind turbine model with different ducting profile were simulated by ANSYS Fluent software.

The air velocity at inlet of the system was set at 16.7 m/s, 25 m/s and 33.3 m/s which are equivalent to pickup truck cruising speed of 60 km/h, 90 km/h and 120 km/h, respectively. At the ducting neck region, the air velocity was slightly increased. The air velocity was significantly increased i.e. three times of inlet velocity with additional of flow guider at the inlet.

In this study, outlet area of present model was designed to be smaller compare with benchmarks model by Mohd Sofian *et al.* Result of modification created greater velocity of air going through neck region of wind turbine and it does not affect the aerodynamic performance of the wind turbine significantly.

ACKNOWLEDGEMENTS

The present work has been supported by Ministry of Education Malaysia under Exploratory Research Grant Scheme (Vote No. E034).

REFERENCES

[1] Mitchell W. 2003. U.S. Patent Application No. 10/706: 592.

[2] Deets E. 2006. U.S. Patent No. 7,135,786. Washington, DC: U.S. Patent and Trademark Office.

[3] Kousoulis T. 2003. U.S. Patent Application No. 10/650: 888.

[4] Sofian M., Nurhayati R., Rexca A., Syariful S. S., and Aslam A. 2014. An evaluation of drag coefficient of wind turbine system installed on moving car. In Applied Mechanics and Materials 660: 689-693. Trans Tech Publications.

[5] Kailash G., Eldho T. I., and Prabhu S. V. 2012. Performance study of modified Savonius water turbine with two deflector plates. International Journal of Rotating Machinery, 2012.

[6] Phillips D. G. 2003. An investigation on diffuser augmented wind turbine design (Doctoral dissertation, Research Space, Auckland).

[7] Yang Z., and Khalighi B. 2005. CFD simulations for flow over pickup trucks (No. 2005-01-0547). SAE Technical Paper.

[8] Hu X. X., and Wong T. T. 2011. A numerical study on rear-spoiler of passenger vehicle. World Academy of Science, Engineering and Technology. 57: 636-641.