AERODYNAMIC ANALYSIS OF AN ELECTRIC VEHICLE EQUIPPED WITH HORIZONTAL AXIS SAVONIUS WIND TURBINES

Blaise Mtopi Fotso¹, Carlos Feudjio Nguefack², Roni-Claudin Talawo³, Médard Fogue⁴

¹,²,³,⁴ Department of Mechanical Engineering and Computed Integrated Manufacturing, FOTSO Victor University Institute of Technology, University of Dschang, Cameroon
¹,²,³,⁴ Industrial Systems and Environmental Engineering Research Unit (UR-ISIE), University of Dschang, Cameroon
¹,²,³,⁴ Mechanics and Modeling of Physical Systems Research Unit (UR-2MSP), University of Dschang, Cameroon

Abstract- This article presents the results of an analysis simulations of the aerodynamic parameters of an electric sedan vehicle fitted with wind turbines. The model consists of an electric sedan vehicle on which are mounted two SAVONIUS horizontal axis turbines, with four gear systems and four generators needed to convert the maximum energy of the wind blown by the displacement of the vehicle of the wind into electrical energy. The whole consists of a single vehicle and two wind turbine models designed by the SOLIDWORKS 2014 software. The analysis of the digital fluid dynamics was performed by using the turbulence model k-ε implemented in SOLIDWORKS FLOW SIMULATION module.

Keywords- Aerodynamic Parameters, Electric sedan Vehicle, SAVONIUS Turbines, digital Fluid Dynamics, Turbulence k-ε, SolidWorks flow simulation

I. INTRODUCTION

The car manufacturers have identified four major areas of development to improve the performance of their vehicles in terms of energy reduction, autonomy and polluting emissions; among other things, the reduction of bearing friction, engine performance, lightening of the vehicle mass and aerodynamics [1]. Studies on aerodynamics can increase the ability of motor vehicles to generate the least resistance possible when moving in the air either to gain speed or to save energy. To achieve this, the manufacturers act on the reduction of the aerodynamic drag or on the reduction of the frontal surface.

Several works have been developed in aerodynamics, especially the study of the numerical simulation of a vehicle equipped with rotating parts in order to reduce its drag coefficient [2]. To do this, they used the k-Omega-SST turbulence model implemented in the OpenFOAM (OF) software and managed using a vane wheel to capture the vortex energy with a power of 16.1 W which forms behind the vehicle and also achieves a total drag reduction of 7.6% [2]. The works of [1] in 2014 carries out an aerodynamic study to control the drag on a body of Ahmed right base. It emerged from this study that the most effective solutions provide drag reductions of the order of 10%. The work of [3] on the aerodynamic analysis of a vehicle frontal baffle was made using digital fluid dynamics with ANSYS CFX software. This work allowed the calculation of the drag coefficient according to the movement of the vehicle for different configurations. In this work, the use of the second deflector resulted in a reduction of 17 860 kg of CO2 emissions [3]. The use of SOLIDWORKS flow simulation for the aerodynamic analysis of a BMW × 4 vehicle has been made. It emerged from this work that at the speed of 20 mph the drag and lift forces were respectively at 15.23 N and 3.19 N and at the speed of 120 mph the drag forces of 546.37 N and lift 116.83 N were found. The second analysis of this work...
shows the impact of the environmental temperature on the drag and lift forces so that at 0°C these forces are 64.10 N and 13.5 N respectively. At 35°C, 56.75 N and 12.12 N were found for the same forces [4].

The use of renewable energies in the traction chain of electric vehicles is of increasing concern to the scientific community. The patented work [5] in 2006 led to the creation of an electric vehicle with a small horizontal axis wind turbine on its roof. Subsequently work [6] in 2011 focused on the establishment of a wind turbine system for vehicles; their model for road and wind tunnel tests resulted in a wind speed of 184 mph, a turbine rotation of 34.27 rpm. In the same way [7] in 2014 perform an assessment of the drag coefficient of a wind turbine system installed on a moving car; they obtain a drag coefficient of the order of 0.39 for the vehicle alone and for the vehicle whose wind turbine is placed at the level of the bumper; they also obtain 0.45 and 0.51 respectively for models with turbine on the hood and on the roof. In 2015, [8] work consisted in designing and installing a three-bladed horizontal wind turbine on a pick-up motor vehicle. Mounted in the rear, their model could generate approximately 200 W of electric power for a vehicle speed of 80 mph. A Preliminary Investigation on Generation of Electricity Using Micro Wind Turbines Placed on a Car was made in 2017 by [9], it emerges from these results of the feasibility of using micro-wind turbines on cars with a reasonable current output that can be used for the auxiliary application inside the car, and obtaining a maximum voltage of 3.5 V and a maximum of 0.8 A of current generated by each micro-wind turbine.

In this article, ours presents works deals mainly with the numerical analysis of the aerodynamic parameters of a passenger vehicle on which SAVONIUS horizontal axis wind turbines are mounted on the roof. It will be a question for us initially to model in 3D three models (vehicle alone, vehicle equipped with wind turbines with right generators and vehicle equipped with wind turbines with side generators). Then, in a second step, we will perform an aerodynamic analysis of each of the models using the computational fluid dynamics method using the SOLIDWORKS FLOW SIMULATION 2014 software. This analysis aims to obtain and compare for each model the velocity, pressure, fluid flow lines and convergence curves and drag coefficient values.

II. METHODOLOGY

The 3D models of the sedan vehicle alone, SAVONIUS turbines and their assembly are realized using the geometric module of the SOLIDWORKS 2014 software. Then the numerical simulations are carried out in the SOLIDWORKS FLOW SIMULATION 2014 module and finally an analysis and comparative study of the different results is presented.

2.1. Modeling of the vehicle and wind turbines

On this point, we highlight the different models (the model of car alone, and two models of car equipped with wind turbines placed on the roof with different positions of the generators).
2.2. Numerical simulation

2.2.1. Pre-treatment

Pre-treatment includes cleaning or recognizing the geometric outlines of models in SOLIDWORKS Flow Simulation, generating the compute domain and the mesh.

2.2.2. Domain of calculation

The calculation domain is a rectangular prism in which the calculation is performed. The planes at the boundaries of the domain are orthogonal to the axes of the Cartesian coordinate system. In external analysis as is the case in this article, the domain of calculation covers the surrounding space of the model.
The mesh technology of SOLIDWORKS Flow Simulation is based on the use of Cartesian meshes; which results in fully localized cells in solid bodies (solid cells), in the fluid (fluid cells) and in cells that intersect the solid-fluid boundary called "partial cells". Such a mesh is defined as a set of rectangular parallelepipeds (rectangular cells) adjacent to each other and to the outer limit of the computational domain, oriented according to the Cartesian coordinates. The rectangular parallelepipeds cut by the surface are treated specifically according to the boundary conditions defined on the surface [11].

2.2.4. Physical model

2.2.4.1. Physical models

In fluid regions, SOLIDWORKS Flow Simulation solves the Navier-Stokes equations, which are formulations of the laws relating to the conservation of mass, momentum, and energy [11]. We obtain the following equations:
\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \quad (1) \]

\[ \frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} + \frac{\partial P}{\partial x_i} = \frac{\partial}{\partial x_j} (\tau_{ij} + \tau_{ij}^R) + S_i \quad (2) \]

\[ \frac{\partial \rho H}{\partial t} + \frac{\partial (\rho u_i H)}{\partial x_i} = \frac{\partial}{\partial x_i} (\rho u_i) + \frac{\partial P}{\partial t} - \tau_{ij} \frac{\partial u_i}{\partial x_j} + \rho \varepsilon + S_i u_i + Q_H \quad (3) \]

With: \( H = h + \frac{u^2}{2} \)

Where: \( \rho \) is density of air, \( u_i \) is velocity vector, \( \rho u_i u_j \) is Reynolds stress, \( P \) is static pressure, \( \tau_{ij} \) is shear stress, \( S_i \) is strain tensor, \( H \) is total energy, \( q_i \) is heat flow density, \( \varepsilon \) is dissipation, \( h \) is internal energy.

The modified k-\( \varepsilon \) turbulence model with damping functions proposed by Lam and Bremhorst (1981) describes laminar, turbulent, and transitional flows of homogeneous fluids consisting of the following turbulence conservation law [11]:

\[ \frac{\partial k}{\partial t} + \frac{\partial (k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu \frac{\partial k}{\partial x_i} \right) + \tau_{ij} \frac{\partial u_i}{\partial x_j} - \rho \varepsilon + \mu_t P_B \quad (4) \]

\[ \frac{\partial \varepsilon}{\partial t} + \frac{\partial (\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu \frac{\partial \varepsilon}{\partial x_i} \right) + C_{\varepsilon 1} \frac{\varepsilon}{k} \left( f_1 \tau_{ij} \frac{\partial u_i}{\partial x_j} + C_B \mu_t P_B \right) - \frac{f_2 C_{\varepsilon 1} \rho \varepsilon^2}{k} \quad (5) \]

Où: \( C_{\mu} = 0.09 \); \( C_{\varepsilon 1} = 1.44 \); \( C_{\varepsilon 2} = 1.92 \); \( \sigma_k = 1 \); \( \sigma_{\varepsilon} = 1.3 \); \( \sigma_B = 0.9 \)

\( C_B = 1 \) si \( P_B > 0 \), \( C_B = 0 \) si \( P_B < 0 \), the turbulent viscosity is determined from:

\[ \mu_t = f_\mu \frac{C_{\mu} \rho k^2}{\varepsilon} \quad (6) \]

Lam and Bremhorst’s damping function \( f_\mu \) is determined from:

\[ f_\mu = \left( 1 - e^{-0.025 R_y} \right)^2 \left( 1 + \frac{0.5}{R_t} \right) \quad (7) \]

Where: \( R_y = \frac{\rho \sqrt{k}}{\mu} \), \( R_t = \frac{\rho k^2}{\mu \varepsilon} \) and \( y \) is the distance between the point and the wall.

The flow of air around the moving vehicle gives rise to aerodynamic forces which can be very important especially at high speeds. The drag force is defined as the projection of the resultant of the aerodynamic torsor on the axis of the main direction of the flow. The drag coefficient is the ratio of this force on the dynamic pressure relative to the reference speed and the projected area in the main direction of the flow [1]. It is defined as:

\[ C_z = \frac{F_z}{\frac{1}{2} \rho U_\infty^2 S_z} \quad (8) \]

With \( F_z \) the drag force, \( \rho \) the density of air, \( U_\infty \) the reference speed and \( S_z \) the projected area in the main direction of the flow.
2.2.4.2. Boundary conditions

### Tableau 1. Boundary conditions

| N°   | Parameters                        | Values                      |
|------|-----------------------------------|-----------------------------|
| 1    | Velocity inlet                    | 27.78 m/s                   |
| 2    | pressure outlet                   | 101325 Pa                   |
| 3    | Air density                       | 1.225 kg/m³                 |
| 4    | dynamic viscosity                 | $1.7965 \times 10^{-5}$ kg/(m.s) |
| 5    | Turbulence Specification method   | k-\epsilon turbulence model |
| 6    | Turbulence Kinetic Energy         | 1                           |
| 7    | Turbulence intensity              | 0.1%                        |

### III. RESULTS AND DISCUSSIONS

The results presented below are, firstly, the surface distribution of the pressure on the different models, the evolution of the flow of the wind speed, the fluid flow lines in a second time and finally the convergence curves of the drag coefficient aerodynamic.

**Figure 8. Pressure distribution on the car body alone**

Figure 8 shows a maximum overpressure at the front bumper and mirrors of the order of 101848.92 Pa, this excess pressure decreases at the base of the windshield until reaching the average value of 101469.02 Pa. A slight depression of the order of 101469.02 Pa is observed on the roof.

**Figure 9. Pressure distribution on the car with straight generators mounted on the roof**

**Figure 10. Pressure distribution on the car with side generators mounted on the roof**

We note here because of the presence of wind turbines on the roof for figure 9 and 10, a maximum overpressure at the front of the vehicle, on the base of the windshield and mirrors (of the
order of $10^{1721}$ Pa in average is almost identical to that of the vehicle model alone). The same value is on the blades and the front faces of the generators.

We note for Figure 11, that the air particles that move far away begin to be braked gradually to the front of the vehicle before being accelerated to the roof level until reaching the average speeds of 31 m/s. Then again idle or almost stopped at the rear of the vehicle. The maximum wind speed is observed on the roof which makes the use of wind turbines at this level favourable.

Figures 12 and 13 clearly show significant changes in the air distribution around the car and also its aerodynamic performance. High speeds of the order of 25 m/s on average on the front faces of the turbines which is favourable for their rotation and the production of electricity.

Figures 11 to 14.
The Velocity streamlines observed for figures 14, 15 and 16 show that before the obstacle the flow is laminar and that at the outer edges of the vehicle, the air particles are accelerated with virtually zero velocity at the rear of the vehicle. More particularly for the models carrying the wind turbines, one observes an average speed of approximately 27 m/s on the front faces of the wind turbines and speed which is null on their backsides.

The curves of Figs. 17, 18 and 19 below represent the convergences of the drag coefficient (Cz) of the different models.
From these results, we can see that the drag coefficient of our model vehicle alone is \(0.3148705\) which corresponds to that of sedan vehicle as shown in the work of [10]. The presence of wind turbines on the roof increases the drag coefficient of the models. so, we have \(0.4613440\) for the model with straight generators that is to say an increase of \(46.51\%\) compared to the vehicle alone and \(0.4779705\) for the model with side generators so an increase of \(51.17\%\). The aerodynamic parameters of two models show us that the most optimal is that equipped with the turbine with straight generators.

IV. CONCLUSION

Aerodynamics studies the phenomena that accompany any movement between a body and the air around it. The parameters of the aerodynamic drag (the drag coefficient and the frontal area of the vehicle) are the points on which the car manufacturers act to reduce the driving resistance of the vehicles. The study proposed in this paper was to numerically evaluate the aerodynamic parameters of a sedan vehicle equipped wind turbines. The vehicle and the turbines were first modeled by the software SolidWorks 2014, then SolidWorks flow simulation 2014 allowed us to carry out the analysis of the distribution of speed and pressure on the models and also the flow of the air around these. The convergence curves of the drag coefficients have finally been exposed. On-roof turbine models have the highest drag coefficients of \(0.4613440\) for the straight generators model and \(0.4779705\) for the side-generator model, while the vehicle alone gives a coefficient of drag of \(0.3148705\). However, in order to further enrich this work, a comparative study could be made using the ANSYS FLUENT software and a determination of the electrical power delivered by the wind turbines.

REFERENCES

I. Y. Eulalie, Étude aérodynamique et contrôle de la traînée sur un corps de Ahmed culot droit, Thèse de doctorat, Université de Bordeaux, Décembre 2014.

II. L. Gagnon, M. J. Richard, and B. Levesque, "simulation of a rotating device that reduces the aerodynamic drag of an automobile," Canadian Society for Mechanical Engineering, Vol. 35, No. 2, pp. 229-249, 2011.

III. C. Malca, N. Alves, and A. Mateus, "Aerodynamic Analysis of a Frontal Deflector for Vehicles," International Journal of Mechanical and Mechatronics Engineering, Vol. 9, No. 1, pp. 60-64, 2015.
IV. A. shrivastava, D. K. singh and P. M. mishra "aerodynamic analysis of the vehicle using the solidwork flow simulation," International Journal of Recent Trends in Engineering & Research (IJRTER), Vol. 04, Issue 03, March-2018.

V. E. Deets, Wind driven generator for powered vehicles, Patent 7135786 B1, 2006.

VI. S. Tickoo, "wind turbine for automobiles," American Society for Engineering Education, AC 2011-2461, 2011.

VII. S. Mohd, R. Nurhayati, A. Rexca, S. Syariful, and A. Aslam, "An evaluation of drag coefficient of wind turbine system installed on moving car," Applied Mechanics and Materials, ISSN: 1662-7482, Vol. 660, pp 689-693, October, 2014.

VIII. M. R. Awal, M. Jusoh, M. N. Sakib, F. S. Hossain, M. R. C. Beson, and S. A. Aljunid, "Design and implementation of vehicle mounted wind turbine," ARPN Journal of Engineering and Applied Sciences, Vol. 10, No. 19, pp. 8699-8706, October, 2015.

IX. V. K. T. Bangi, Y. Chaudhary, R. K. Guduru, K. T. Aung and G. N. Reddy, "Preliminary investigation on generation of electricity using micro wind turbines placed on a car," International Journal of Renewable Energy Development, ISSN: 2252-4940, Vol. 6.1, pp.75-81, February 15, 2017.

X. B. Bijlanil, P. P. Rathod, and A. S. Sorthiya, "Experimental and computational drag analysis of sedan and square-back car," International Journal of Advanced Engineering Technology, Vol. IV, pp 63-65, April-June, 2013.

XI. A. Sobachkin, G. Dumnov, "Base numérique de la CFD intégrée à la CAO, livre blanc, DASSAULT SYSTEMES," NAFEMS World Congress, 2014.

XII. P. Adarsh, A. G. Cherian, K. S. Arun, D. R. Kumar, "Numerical investigation of drag on a trailing aerodynamic sedan vehicle," International Journal of Mechanical And Production Engineering, Vol. 2, No. 4, April-2014.

XIII. A. Shrivastava, D. kumar, and P. Mohan, "Aerodynamic analysis of the vehicle using the SolidWork flow simulation," International Journal of Recent Trends in Engineering and Research, Vol. 04, Issue 03, March 2018.

XIV. A. Kacem, "External Flow Analysis over Tow-box form (TOYOTA-Land Cruiser model) Using CFD," IOSR Journal of Mechanical and Civil Engineering, Vol. 14, Issue 4, Ver I, PP 10-15, Jul. – Aug. 2017.

XV. B. Harshavardhan, and J. Govardhan, "Aerodynamic analysis of a BMW car body," International Journal of Modern Trends in Engineering and Research, Vol. 03, Issue 04, pp. 377-380, April 2016.

XVI. A. Kazi, P. Acharya, A. Patil and A. Noraje, "Effect of spoiler design on hatchback car," International Journal of Modern Trends in Engineering and Research, Vol. 03, Issue 09, pp. 192-200, September 2016.

XVII. M. Huang, Z., Gu, C., Feng, "Coupled Analysis of Unsteady Aerodynamics and Vehicle Motion of a Passenger Car in Crosswind Condition," Journal of Applied Fluid Mechanics, Vol. 10, No. 2, pp. 625-637, 2017.

XVIII. G. Dias, N. R. Tiwari, J. J. Varghese, G. Koyeerath, "Aerodynamic Analysis of a Car for Reducing Drag Force," IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), Vol. 13, Issue 3, Ver. I, PP 114-118, May-Jun. 2016.