International Journal of Systems Signal Control and Engineering Application 11 (4-6): 56-63, 2018
ISSN: 1997-5422

Aerodynamic Design and Simulation of a Unipersonal Electric Vehicle Prototype Using Photovoltaic

Cristhian Pineda, Robinson Jimenez Moreno and Olga Lucia Ramos
Faculty of Engineering, Nueva Granada Military University, Bogota, Colombia

Abstract: This study presents the results obtained by design an aerodynamic model of a unipersonal electric vehicle prototype. Vehicle behavior is studied using a Computational Fluid Dynamics (CFD) of a wind tunnel test simulation, showing its aerodynamic coefficient. Flexible photovoltaic panels will be implemented over car body, the same test was applied and being able to see its impact in the aerodynamics behavior from core model. The results shows car-body's aerodynamic design from unipersonal electric vehicle in comparison with aerodynamic changes by adding photovoltaic panels. Likewise, analysis of pressure and temperature are presented in order to visualize possible risks with negatively influence on these devices such as fractures or decrease potential efficiency.

Key words: Aerodynamic, automotive design, CAD technology, computational fluid dynamics, photovoltaic panels, wind tunnel test

INTRODUCTION

The environment and health issues which now a days are presented worldwide are largely due to the high pollution generated by different means of transport that millions of people use every day. Specifically in Bogota, the capital of Colombia an increased number of vehicles powered by combustion engines continues growing. The fleet of city buses and heavy trucks have years of service without proper maintenance, helping to increase the pollution problem (Golay *et al*., 2010).

Therefore, it is important to implement and promote new technologies of mobility an example of this is India where a National Electric Mobility Plan was announced in which the government invest a lot of money in the development and deployment of hybrid and electric vehicles (Saxena *et al*., 2014). Furthermore, the increased use of alternative energy through solar panels is evident in different energy supply applications where high fuel prices, allow electric vehicles with these devices be a viable solution to environmental problems and energy consumption (Bedir *et al*., 2010; Dinis *et al*., 2012).

Additionally, a geometric improvement of the vehicle, namely the aerodynamics associated to the car body, help to reduce the resistive force of the air. For example in the study presented by Ess Corporation (2013) is demonstrated that an enhancement on aerodynamics drastically decreases fuel consumption where a 20% reduction in the drag coefficient could reduce consumption from 7-9% (Zhu and Yang, 2011; Concepcion, 2011). Using an analogy to electric vehicles would reduce energy consumption increasing battery life.

Likewise, good aerodynamic design allows the vehicle to reduce up thrust at high speeds, improving water drainage of rainy days and reduce air resistance (Gomez *et al*., 2011). At low speeds, the forces generated are tiny but as the speed increases can be critical leading to vehicle instability and possible loss of control thereof (Rodriguez *et al*., 2004).

The aerodynamic behavior greatly impacts energy consumption, hence through this study is approached this issue by designing a unipersonal electric vehicle prototype which is provided with solar panels on it so as to provide a possibility of energy self-supply. The aerodynamic analysis with the solar panels inclusion is the main contribution in this research, not found in the review of the state of the art.

The development involves virtual wind tunnel tests, allowing for known the aerodynamic response in general and similarly, flexible photovoltaic panels are installed, allowing to see its impact from variation of the drag coefficient of the car. It is observed the pressure on these latest devices and temperature presented in the areas of installation in order to study fracture risk and the possible decrease in efficiency.

MATERIALS AND METHODS

It is performed different models and simulations in order to analyze the features presented in each one to

Corresponding Author: Cristhian Pineda, Faculty of Engineering, Nueva Granada Military University, Bogota, Colombia

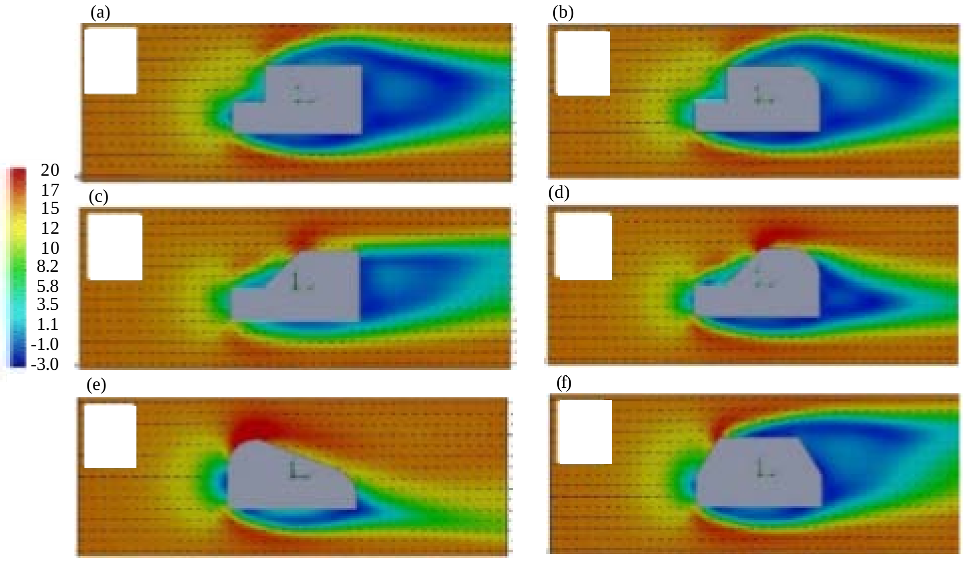

Fig. 1a-f: Airflow distribution at 60 km/h on six different vehicle shapes

establish the most suitable aerodynamic for the vehicle in function to reach a model shape similar to a drop of water (ideal aerodynamics), the problem of this consist in the cockpit is become uncomfortable for the occupants (Gomez *et al*., 2011). This implies the need to get a balance between form and driver comfort.

Considering the above it was performed an air flow behavior study using 5 different vehicle shapes, (Fig. 1). Speed is represented by the color change which has a measuring range from 0-19 m/sec. The tool used for the simulation was SolidWorks ® flow simulation in a computer equipment Hewlett Packard Compaq 6200 PRO SFF which uses Intel Core i5-2500 processor with 8 GB of RAM.

It can be observed that shapes A, B and F, generate excess flow separation, creating slipstreams in the top, bottom and back of the model, affecting the aerodynamic performance of the vehicle, reducing its support and resistance of the rear axles (Perrier *et al*., 2004) (Fig. 1a, b and f). The c and d shapes have a better performance, though presenting a lesser extent of flow separation in the back. These shapes are the most approximate to conventional vehicles nowadays (Fig. 1c and d). Finally, the E shape, closely resembles the shape of a water drop, wide on front and thin at the end, changing proportionally its extent. One can see that this model has the best aerodynamic performance, decreasing almost all flow separation and slipstreams generated (Fig. 1e). According to, the above it is proposed the choice of D shape which allows a good balance between aerodynamic performance and comfort in the interior, making it easy to place the chair between the middle and back inside the cockpit.

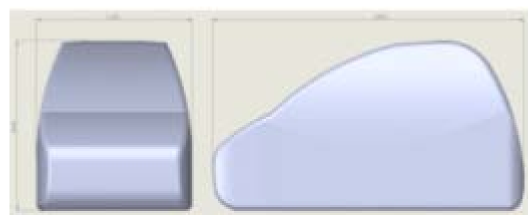

Fig. 2: Front and side design of the vehicle, specifying its measures

To reduce fluid separation and slipstreams generated it is performed some improvements in the edges of the entire vehicle, helping in a small scale with cooling of the panels. This is evidenced by Vinnichenko *et al*. (2014) indicating that a temperature increase by 26.2%, decreases the efficiency of the solar panel by 1.1%. Thus, the prototype vehicle used, seen laterally have a thin design in the front and width design in the back. It is recommended to have a width and length ratio of 1/3 to reduce air resistance (Gomez *et al*., 2011) in this case as the vehicle is unipersonal, designing a long body is not desirable because it is required more material in its manufacture increasing the overall weight of the vehicle and furthermore generating waste of space.

The following measures for the vehicle prototype model are proposed: 1200×2400×1400 mm (Width×Length×Height) (Fig. 2). This choice is based on the dimensional specifications of the commercial vehicle Renault Twizy, since, this model is unipersonal type and this guaranteed the right room for the driver. The mode l is done through CAD Software SolidWorks®,

Table 1: Data entered into the CFD solver SolidWorks®flow simulation

Thermodynamic parameters	Static pressure: 74661.00 Pa Temperature: 293.15 k
Speed parameters	Speed in X direction: 16.66 m/sec Speed in Y direction: 0 m/sec Speed in Z direction: 0 m/sec
Turbulence parameters	Intensity: 0.10%

making multiple extruding and cuts creating smooth contours without sharp corners. From this model the airflow tests are applied to check its aerodynamic behavior.

It is avoided the implementation of cooling air inlets in the first place because the vehicle will not have any radiator installed that required this and second because these inlets greatly affect the drag generated by varying the aerodynamic behavior (Bader *et al*., 2013). Following are two sub-sections in which are illustrated the parameters of the scenario of simulation for wind tunnel virtual test and the results of the simulation

Setting of parameters: To perform wind tunnel testing is required to enter the necessary parameters as environmental conditions and average vehicle speed in the wizard fluid simulation (CFD)-SolidWorks®Flow Simulation (Table 1). Modern wind tunnels allow to modify the intensity of turbulence to very low levels, about 0.05% (Ansys, 2006). This value can be chosen freely and only depend on the characteristics of the tunnel in this case it is considered a value of 0.1%. It's worth making it clear that the CAD Software handles a turbulence model called ETM (Enhance Turbulence Modeling) which combines the classical k-e Model and 2 SWF (Two-Scale Walls Function). More details may be found (Ansys, 2006).

The pressure and temperature selected refers to the average characteristics of the city of Bogota (Colombia). The velocity is 16.66 m/sec (60 km/h) with reference to the maximum allowed speed in this city.

It is necessary to establish a three-dimensional region to develop the study, setting limits with a simulated space where the analysis of the fluid will be made, represented by a prism where the vehicle normally will transit. Additionally, a spatial division (grid) is applied which allow a detailed analysis of the airflow behavior over the vehicle.

It is set boundary limits of the computational domain around the vehicle and the grid over the solid (Ramasamy *et al*., 2010). In Fig. 3 is displayed the boundary conditions represented by a rectangular prism and the direction of the air flow circulation through the longitudinal axis of the model in the opposite of the vehicle). The following measures are proposed for this prism: 3000×3000×8000 mm (Width×Length×Height).

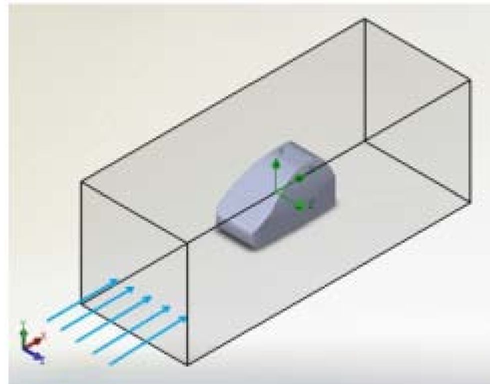

Fig. 3: Rectangular prism which represents the boundary conditions of the CFD analysis and airflow direction

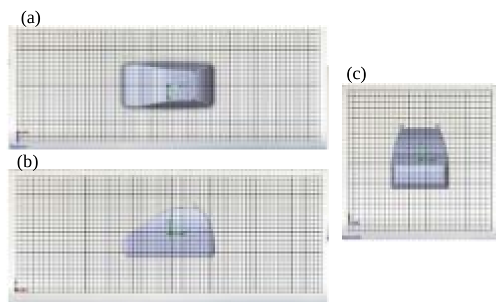

Fig. 4: a-c) Grid for fluid simulation analysis

Fuzzy control section obeys to the need to know the state of charge of the batteries in an intelligent way, since, for the operation of the algorithm it is necessary to have an area of uncertainty that allows to identify the right moment to make the decision to charge besides to know the state of charge of the battery and in this way, carry out the execution of the algorithm of charge optimization.

The regular mesh refinement is done creating ten cells for each meter of the length of each side of the prism, hence, it is obtained by this way a division of 80×30×30 cells along, width and height, respectively, getting a total of 72,000 cells occupying the entire rectangular prism (Fig. 4).

It is applied an irregular refinement in the near spaces to the model edges where sub-divisions are made to achieve a high quality grid (Popiolek and Awruch, 2009). This distribution allows focus on the areas near the vehicle, reducing the amount of memory used and increasing speed in the analysis process (Matsson, 2013).

RESULTS AND DISCUSSION

Following are shown the results of the virtual wind tunnel test. Initially, the test is performed on the general

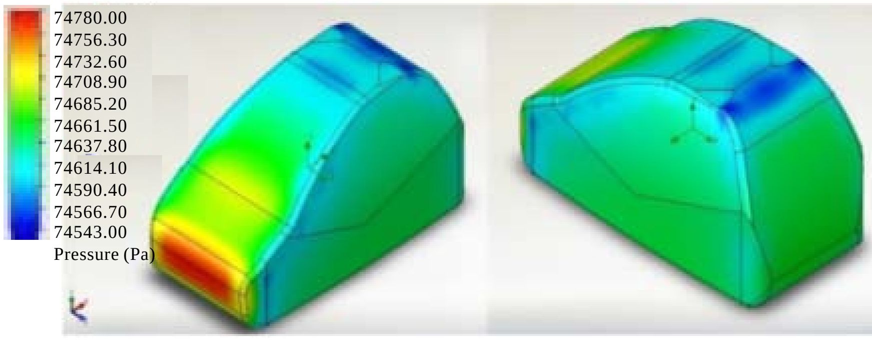

Fig. 5: Pressure distribution on the vehicle. Wind tunnel simulation, speed 16.66 m/sec (60 km/h)

model and then another test is performed with the inclusion of the photovoltaic panels. It is observed three important parameters: pressure, temperature and drag coefficient. The first two factors could affect the integrity of the photovoltaic panels due to an excessive force could break them and an increased temperature would decrease the efficiency thereof.

The simulation is performed with the model without panels in order to permit a subsequent analysis with the inclusion of these devices on car body in terms of pressure and then in terms of aerodynamic drag. Following are presented each of these including an analysis of the temperature.

Pressure analysis on the car body without panels: A pressure increase generated on the bumper and at the start of the windshield is observed, due to the change in curvature. Low pressure zones can also be observed in the curves of the side windows, the roof and the rear of the vehicle (Fig. 5).

By taking a specific measurement of the pressure generated by the longitudinal section of the vehicle (Fig. 6) can be seen peaks in the graph which represent increases pressure on the areas of the bumper and intersecting the vehicle windscreen. As we reach to the final length of the vehicle, the pressure tends to be equal to the inlet pressure but at the end of this model, pressure dropped more than the normal atmosphere, creating a small vacuum.

In general, the air flow on the vehicle tends to be smooth and continuous in front but in the rear occurs a slight suction and turbulence due to the vacuum created, Fig. 7. Being a unipersonal vehicle, space required is low, so, the length change to reduce this problem is limited.

Pressure analysis on the installed photovoltaic material: Making use of flexible solar panels on the car body

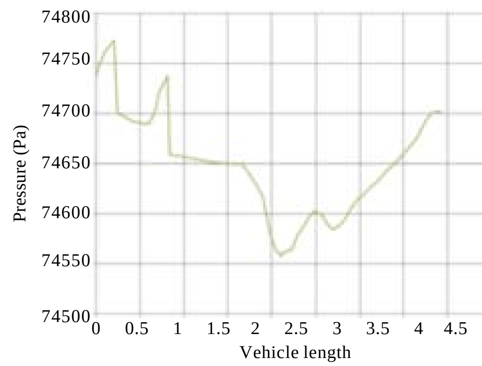

Fig. 6: Pressure distribution through the longitudinal section of the vehicle

(hood and roof) a similar behavior is observed in the areas of inclusion compared to the previous model (Fig. 8).

On the front panel is generated a pressure increase about 70 Pa which rise as they approach the windshield because of this and compared to the pressures generated on the body in the previous subsection it is proposed to increase the coverage area of the front panel by selecting a longer panel and move the panel to the center of the hood (away from the windshield) where less pressure on the hood is presented. On the roof panel a slightly lower pressure than the atmosphere is presented, about 40 Pa.

Solar panels are normally protected by tempered glass which has a strength of 100 MPa surface pressure (19), therefore, there is no danger that could affect the integrity of such devices in any of the two positions.

Temperature analysis: Discussed above, long fluid separation regions were generated in higher temperature zones and observing Fig. 9 it is evidenced the temperature increase in 0.15°K on the rear of the vehicle where it is

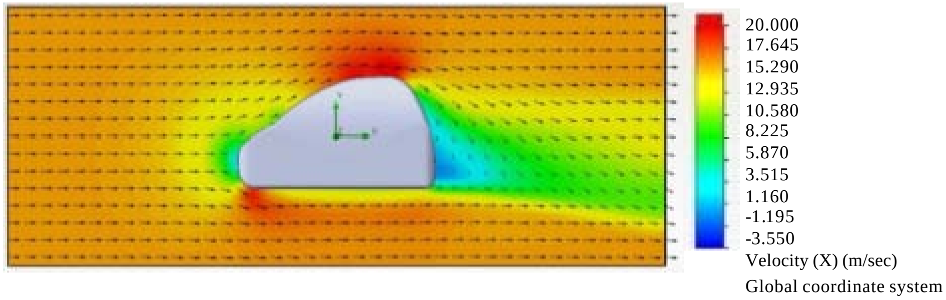

Fig. 7: Air flow the speed, direction and turbulence are shown

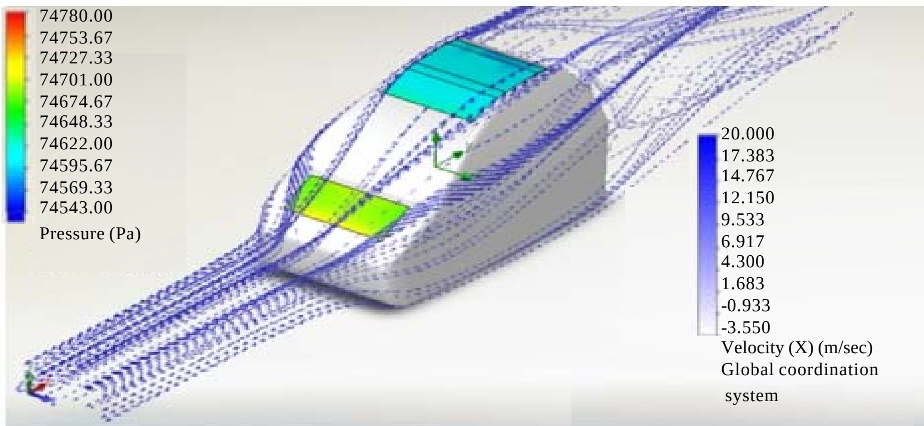

Fig. 8: Pressure on photovoltaic panels installed on the vehicle, generated during the wind tunnel test

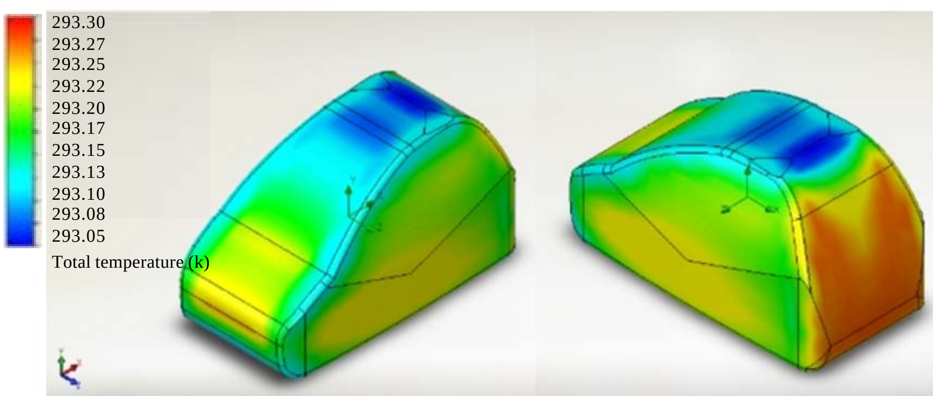

Fig. 9: Temperature distribution on the vehicle. Wind tunnel simulation, speed 16.66 m/sec (60 km/h)

produced a slight separation of the fluid. Therefore, this small increase in temperature would not reach to adversely affect the performance of a solar panel, allowing be sure to install photovoltaic device on any location.

The best location for the panels is the roof because high air flow at this site allows keep them cool with a slight decrease in temperature of only 0.10°K, followed by the hood and the doors which are maintained with the room temperature.

Aerodynamic coefficient (without photovoltaic panels): The drag force may be expressed by the following Eq. 1 (Rodriguez *et al*., 2004):

$$F = \frac{1}{2}\rho C_x A_f V^2 \tag{1}$$

Where:

ρ = Air density

C_x = Vehicle aerodynamic coefficient

A_f = Front area of the vehicle
V = Speed of circulation

In this particular case, the fluid (air) is analyzed its density in the city of Bogota: 0.879038421 kg/m^3; this data was obtained using the formulation of the air density (1981/91), published by National Institute of Standards and technology (Davis, 1992) where are entered pressure values (560 mmHG), temperature (20°C) and relative humidity (80%), characteristic values of Bogota (NIH., 2000).

The velocity is 16.66 m/sec (60 km/h) with reference to the maximum allowable speed within the city of Bogota. The frontal area of the vehicle is 143235.11 cm^2 (1.43235111 m^2) data obtained by performing a frontal projection (shade) of the model, calculated by the CAD Software. The model mass is 250.10 kg, using materials such as fiber glass type E this value can be observed through the physical properties calculated by the CAD Software. This type of fiber is chosen because of its high tensile strength, high stiffness, non-flammable and inexpensive.

The drag coefficient of the vehicle obtained by the solver SolidWorks ® Flow Simulation was 0.2933160 and the average force was 51.5914 N generated by the air on vehicle movement (Table 2). To obtain these results it is necessary to introduce the drag coefficient Eq. 2:

$$C_x = \frac{F}{\frac{1}{2}\rho A_f V^2} \qquad (2)$$

Equation 2 is programmed in the simulator where is referenced the force generated on the vehicle, the program analyzes in real time both variables, delivering stable values at the end.

The drag coefficient for conventional vehicles is set in a range between 0.29 and 0.33. The submitted value of this test can be compared to the table presented by Marchese and Golato (2011) where the researchers present a list of vehicles with their data, matching this model with the Fiat Uno and the Alfa Romeo 155.

Being a unipersonal vehicle its length is much lower compared with a conventional one which normally tends to increase the drag coefficient (Hui *et al*., 2012) but in this case the construction significantly decreased the value being an asset to overall behavior of the vehicle, hence, it is possible to obtain an excellent drag coefficient ignoring the length-width ratio.

Table 2: Of wind test results (whitout photovoltaic panels)

Mass	Units	Values
X forces	N	51.252
Aerodynamic coeficient		0.3191576

Table 3: Of wind test results (whitout photovoltaic panels)

Mass	Units	Values
X forces	N	51.252
Aerodynamic coeficient		0.2933160

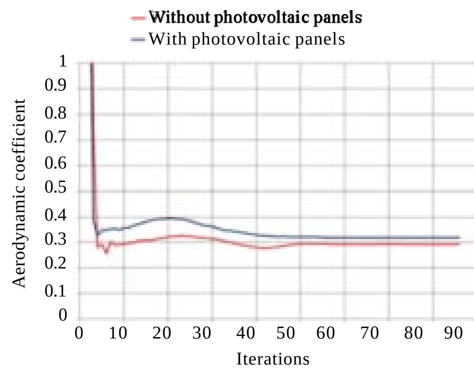

Fig. 10: Comparison of aerodynamic coefficients obtained in test

Aerodynamic coefficient (with photovoltaic panels): Using the same configuration will proceed to re-run the wind tunnel test this time with photovoltaic panels installed on the vehicle. The drag coefficient of the vehicle obtained this time was 0.3191576 with an average force generated on the vehicle of 52.5090 N (Table 3).

Comparing the results of the previous test (Fig. 10) it is shown the aerodynamic coefficient is increased by 8.81% with the implementation of photovoltaic panels on the vehicle. This negatively acts on car because as mentioned in the introduction, impact directly in energy consumption. Therefore, it is necessary that the energy supplied by these devices further satisfy this increased energy.

CONCLUSION

The unipersonal electric vehicle was designed successfully, from the selection of the most balanced shape in relation to the aerodynamic performance and comfort. All edges and surfaces were improved, obtaining an increase in the aerodynamic performance through flow separation reduction and voids generated. It is achieved a good drag coefficient within the normal range of conventional vehicles (0.29).

It is evidenced that including photovoltaic panels on the body affects the aerodynamic behavior increasing the drag coefficient. The consequence of this results in increased energy consumption, although, this analysis was not performed in this study it is compared with previous research by other academic peers.

A great advantage of working with CFD Software is that it is a quick and inexpensive process compared to building a real wind tunnel. Through this test it was possible to observe the behavior of pressure and temperature presented on the body and solar panels this allow to view graphically areas of the car body where it is better to place the solar panels and watch the potential of affecting the integrity of these devices. In this case, none of these variables affect negatively to such devices.

These results are the basis for sizing the required power to start-up the vehicle which will be performed in future research.

ACKNOWLEDGEMENTS

This research is supported financially by the Vice-rector of investigations of the Universidad Militar Nueva Granada, through the ING 1535, developed within the research group GAV of Mechatronics Engineering program.

REFERENCES

Ansys, 2006. 7.2.2 Determining turbulence parameters. Ansys, Canonsburg, Pennsylvania, USA. https:// www.sharcnet.ca/Software/Fluent6/html/ug/node217.htm.

Bader, D., T. Indinger, N.A. Adams, P. Unterlechner and G. Wickern, 2013. Interference effects of cooling airflows on a generic car body. J. Wind Eng. Ind. Aerodyn., 119: 146-157.

Bedir, A., B. Ozpineci and J.E. Christian, 2010. The impact of plug-in hybrid electric vehicle interaction with energy storage and solar panels on the grid for a zero energy house. Proceedings of the 2010 IEEE PES International Conference on Transmission and Distribution and Exposition, April 19-22, 2010, IEEE, New Orleans, Louisiana, ISBN: 978-1-4244-6546-0, pp: 1-6.

Concepcion, M., 2011. [Advanced Hybrid Systems]. Create Space Independent Publishing, USA., ISBN-13: 9781463575717, Pages: 62 (In Spanish).

Davis, R.S., 1992. Equation for the determination of the density of moist air. Metrologia, 29: 67-70.

Dinis, C., A. Roque, D .M. Sousa and V.F. Pires, 2012. Computational application to study the performance of an electric vehicle with photovoltaic panels. Proceedings of the 2012 International Conference on Renewable Energy Research and Applications (ICRERA), November 11-14, 2012, IEEE, Nagasaki, Japan, ISBN: 978-1-4673-2328-4, pp: 1-5.

Ess Corporation, 2013. Enhanced turbulence modeling in solidworks flow simulation. Ess Corporation, Mississauga, Ontario.

Golay, F., J.P. Robra, N.Y. Rojasy and A. Clappier, 2010. An emissions inventory of air pollutants for the city of Bogota, Colombia. Master Thesis, Ecole Polytechnique Fédérale de Lausanne, Lausanne, Switzerland.

Gomez, T.E., J.A. Navarro and J. Garcia, 2011. [Structures of the Vehicle]. 2nd Edn., Ediciones Paraninfo S.A., Madrid, Spain, (In Spanish).

Hui, Z., W. Jia and Y. Zhigang, 2012. Numerical analysis on effect of vehicle length on automotive aerodynamic drag. Proceedings of the IET International Conference on Information Science and Control Engineering, December 7-9, 2012, IET, Shenzhen, China, ISBN:978-1-84919-641-3, pp: 1-17.

Marchese, R.A. and M.A. Golato, 2011. [Fuel consumption and energy in transport (In Spanish)]. Rev. CET. UNTucuman, 33: 1-9.

Matsson, J.E., 2013. An Introduction to SolidWorks Flow Simulation. SDC Publications, Mission, Kansas,.

NIH., 2000. [Meteorology and environmental studies of Colombia]. National Institute of Hydrology, Rockville, Maryland, USA. (In Spanish)

Perrier, J.M., E. Bonviny and E. Ibarrola, 2004. [Verification of the aerodynamic performance of a land vehicle using calculation and experimental techniques (In Spanish)]. Comput. Mech., 23: 1-21.

Popiolek, T.L. and A.M. Awruch, 2009. An adaptive mesh strategy for transient flows simulations. Proceedings of the 3rd International Southern Conference on Computational Modeling (MCSUL), November 23-25, 2009, IEEE, Rio Grande, Brazil, ISBN:978-1-4244-5980-3, pp: 71-76.

Ramasamy, D., K. Kadirgama, A.K. Amirruddin and M.Y. Taib, 2010. A vehicle body drag analysis using computational fluid dynamics. Proceedings of the 2010 1st International Conference on Mechanical Engineering Research and Postgraduate Students, May 26-27, 2010, Universiti Malaysia Pahang, Gambang, Malaysia, ISBN: 78-967-5080-9501, pp: 342-353.

Rodriguez, P.L., A.M. Daniel and V. Carlos, 2004. [Automotive Engineering: Systems and Dynamic Behavior]. Ediciones Paraninfo S.A., Madrid, Spain, (In Spanish).

Saxena, S., A. Gopal and A. Phadke, 2014. Electrical consumption of two-, three-and four-wheel light-duty electric vehicles in India. Appl. Energy, 115: 582-590.

Vinnichenko, N.A., A.V. Uvarov, I.A. Znamenskaya, H. Ay and T.H. Wang, 2014. Solar car aerodynamic design for optimal cooling and high efficiency. Solar Energy, 103: 183-190.

Zhu, H. and Z. Yang, 2011. Simulation of the aerodynamic interaction of two generic sedans moving very closely. Proceedings of the 2011 International Conference on Electric Information and Control Engineering (ICEICE), April 15-17, 2011, IEEE, Wuhan, China, ISBN: 978-1-4244-8036-4, pp: 2595-2600.