International Journal of Applied Engineering Research ISSN 0973-4562 Volume 18, Number 2 (2023) pp. 110-114

Aerodynamic Study Of The Rear Spoiler Profile Of A Pick-Up Vehicle

Oscar Tenango-Pirin[1*], Ernesto Atayde[1], Elva Reynoso-Jardón[1], Yuri S. Hernández[2], Yahir Mariaca[1], Raúl Ñeco Caberta[1]

[1]Instituto de Ingeniería y Tecnología, Universidad Autónoma de Ciudad Juárez, Chihuahua, México
[2]Tecnológico Nacional de México/Campus Pachuca, Pachuca de Soto, Hidalgo, México

Abstract
Aerodynamic drag in pick-up type vehicles is one of the main issues concerning their high fuel consumption. External devices attached to their body frame, such as spoilers, have been developed to reduce the drag. In this paper, the effect of the profile shape of a rear spoiler on aerodynamics of a pick-up vehicle was numerically investigated. The NACA 6409, NACA 0015 and NACA 4415 profiles were analyzed and compared with a case where any device was used. Results showed that flow and velocity of air were influenced by the addition and the shape of the spoilers. Low-pressure zone behind the vehicle was also affected due to modification of the boundary layer detachment. Analysis of static pressure on the rear surface showed that it was influenced by using the spoilers. On the other hand, drag and lift coefficients were calculated and evaluated for each case of study. Results allowed to identify the performance of the spoiler profiles on the drag of the vehicle.

Keywords. Pick up vehicle, aerodynamic drag, CFD, rear spoiler

1. INTRODUCTION

Currently, one of the main objectives of the automotive industry is to reduce pollutant emissions. To achieve this, one option is to substitute or to mix fossil fuels with biofuels. For instance, it has been demonstrated that the use of biodiesel blends can reduce the emissions of CO, CO_2 and HC [1]. Reducing fuel consumption by improving combustion is other option. Novel techniques like using ignition catalyst [2] can be mentioned. Other techniques like the reduction of the resistance of the vehicle, the improvement of the engine efficiency, the use of advanced power trains, and the use of an adequate transmission [3] have been developed. When it comes to vehicle resistances, aerodynamic drag is one of the most important given that, while driving in the highway, a vehicle may need approximately 11 % of the fuel to overcome it [4].

The drag resistance is more pronounced in pick-up truck vehicles since their body frame shape promotes the formation of vortices and flow detachment from surfaces. During the movement of the vehicle in the air, forces that influence the performance of the vehicle arise. These are the drag force F_X and the lift force F_L. The first one depends on the air density ρ, the drag coefficient of the vehicle C_D, the frontal area of the vehicle A and the velocity of the vehicle V, as indicated in Equation 1.

$$F_x = \frac{1}{2}\rho C_D A V^2 \qquad (1)$$

On the other hand, the lift force is the vertical component of the resultant force that influences the vehicle stability. This force depends on the parameters indicated in Equation 2, where C_L is the lift coefficient.

$$F_L = \frac{1}{2} p V^2 C_L A \qquad (2)$$

The aerodynamic resistance of the vehicle can be controlled if the values of the drag coefficient of the vehicle change. External devices, such as spoilers and vortex generators (VG's), allow to reduce drag and can be attached to the vehicle without modifying the body frame [5]. However, in pick-up type vehicles the aerodynamic drag is higher due to the shape of the body frame. The formation of low-pressure zones in the box and behind the box drives to a significant increase of drag.

Several studies have focused on the addition of spoilers to reduce drag resistance on this type of vehicles. Sagar et al. [6] found that the addition of a wing or vortex generators in a sedan car led to an increase in static pressure on the rear of the vehicle. Kim et al. [7] showed that the use of a spoiler located on the tailgate of a pickup truck delayed the flow separation at high speeds reducing the drag. However, at low speeds the drag force was increased. Similar results were found by Yakkundi and Mantha [8]. Taniguchi et al. [9] studied the dynamic of the flow between the cabin and the tailgate. They observed that the use of a rear spoiler reduces drag even in crosswinds conditions. Spike et al. [10] concluded that the use of side panels is more effective in reducing drag than a roof spoiler. On the other hand, Luthfie et al. [11] found that at higher angles of the spoiler the drag increases, while lift decreases.

According to the consulted literature, the use of rear spoilers in pick-ups is a viable option to reduce the drag force. In addition, their shape and position influence the airflow dynamics affecting the low-pressure zones behind the vehicle. Hence, in this work, the effect of the profile shape of a rear spoiler on the aerodynamics of a pick-up vehicle was studied. To evaluate the performance of the spoiler, the drag and lift coefficients were evaluated. The results allowed to identify the influence of the shapes of the profile for the spoiler in the vehicle drag.

2. METHODOLOGY

A CFD study was performed using a CAD model of a generic pick-up vehicle. The shape and dimensions of the vehicle were defined based on others models available in literature. Some of them are generic geometries [7, 12, 13], while others correspond to commercial brands [9, 10]. The body geometry was simplified, and the actual scale of the vehicle was used.

The geometry under study is shown in Figure 1, where main dimensions are shown in m. This model corresponds to the case of reference without spoiler, then the spoilers were added.

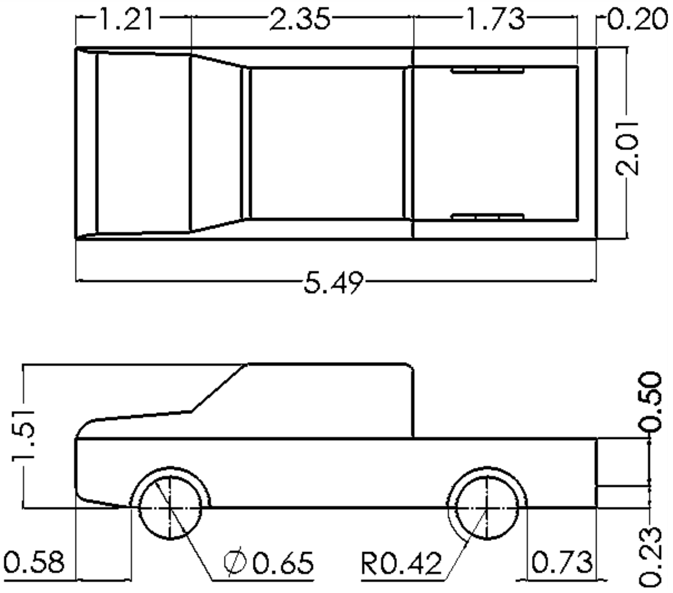

Figure 1: Geometry of the studied vehicle from top and side view.

The vehicle was positioned into a wind tunnel to carry out the tests. Hence, the simulated control volume consists of a wind tunnel, shown in figure 2, where the vehicle was located. The assigned dimensions of the tunnel were 4*L* long, 2.7*L* wide and 2.7*L* high, where *L* corresponds to the length of the vehicle. The flow direction from inlet to outlet is depicted by means of velocity vectors.

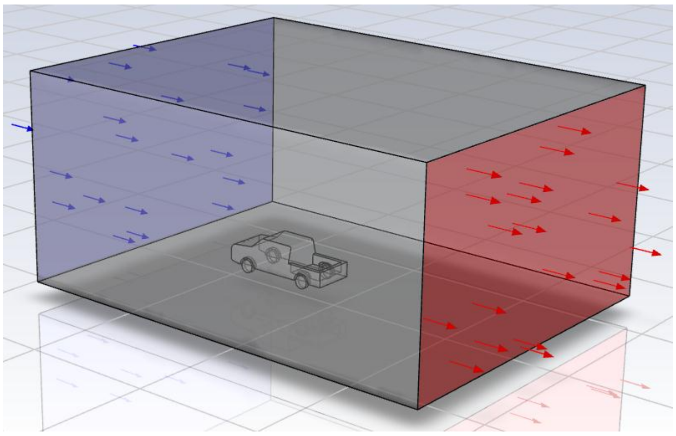

Figure 2: Control volume of air and vectors of wind direction.

The boundary conditions used are also shown in Figure 2. The lateral, upper and lower faces of the tunnel were defined as walls. Faces of the vehicle were also defined as walls. A zero velocity and non-slip condition were assigned to all walls. The front and rear faces were defined as input and output, respectively. A constant airflow velocity of 35 m/s (126 km/h) was employed at inlet based on the literature [14,15,16]. At outlet, atmospheric pressure was assigned. The air was considered as incompressible, continuous, and turbulent with a constant density of 1.2 kg/m^3.

Governing equations of continuity and momentum were solved by means CFD using Ansys commercial software. Hence, steady state Reynolds averaged Navier-Stokes equations (RANS) approach were used. For modeling turbulence, the realizable k-ε model [17] was employed given its good performance on predicting the fluid flow in similar studies from literature [12, 15]. Equations of turbulent viscosity and dissipations rate are given in equations 3 and 4, as follows:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \quad (3)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - \rho C_{2\varepsilon}\frac{\varepsilon^2}{k} + S_\varepsilon \quad (4)$$

where k is the turbulent kinetic energy and ε is its rate of dissipation. μ_t is the turbulent viscosity G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants, σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. Finally, S_k and S_ε are user-defined source terms as described by the Fluent theory guide [18].

A mesh convergence analysis was carried out using the model without spoiler. For all cases, an unstructured mesh with tetrahedral elements was used. Size and number of elements of each mesh are given in Table 1. Also, the calculation of the C_D values for each case can be seen. According to the data obtained, the lowest error value was obtained with Mesh 4, therefore, that element size was used for the rest of the analyzed cases.

Table 1: Mesh convergence analysis.

	Mesh 1	Mesh 2	Mesh 3	Mesh 4
Element size	1100mm	900mm	500mm	350mm
Number of elements	173,226	180,667	265,092	412,841
C_D	0.548	0.537	0.527	0.520
Relative error	N/A	1.99%	1.96%	1.38%

The analyzes comprehend the vehicle with and without rear spoiler, giving rise to four different cases. Three rear spoilers with different profile shape were used, and they were compared against a base case without a spoiler. The profiles studied were: NACA 0015, NACA 4415 and NACA 6409. They were selected due to their performance in reducing drag, as demonstrated in the studies of Grau [19] and Palencia et al. [20]. For the generation of the profiles geometry, the Airfoil tool was used [21]. Figure 3 shows the geometries of the profiles and shows the way in which they were added to the body frame.

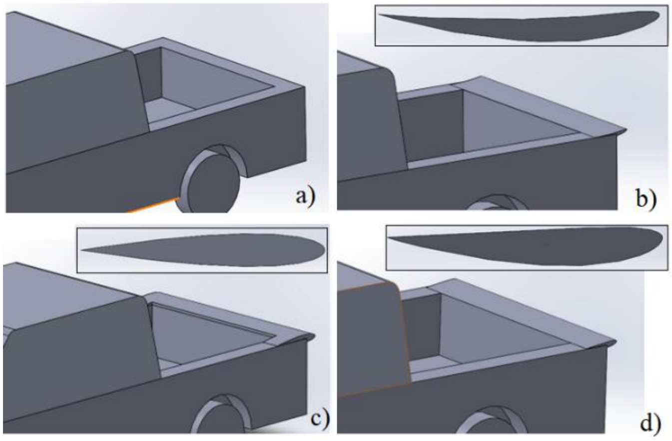

Figure 3: Aerodynamic profiles used for the design of the spoiler: a) case without spoiler, b) NACA 6409, c) NACA 0015 and d) NACA 4415.

3. RESULTS

To analyze the influence of the profile of each spoiler, velocity and static pressure data were analyzed. Figure 4 illustrates the velocity contours in the flow field for each case. It can be seen the formation of a low-velocity zone behind the box due to flow separation at rear part of the vehicle. That zone was reduced when a spoiler is used, since it delays the boundary layer detachment. Then, the airflow is diverted by reducing the distance of the reattachment point of the flow with the ground. The case without a spoiler showed the largest low-velocity zones. Among the cases with a spoiler, the contours of the NACA 4415 profile showed the smallest low-speed zones, such that the flow is closer to the rear surface of the vehicle.

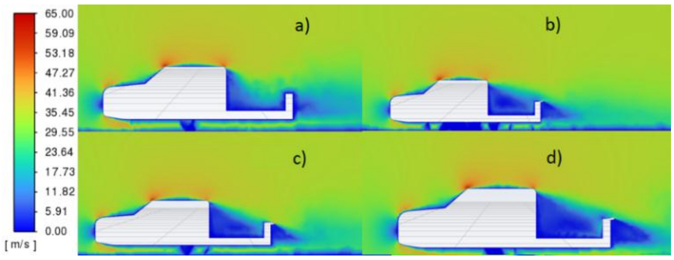

Figure 4: Velocity contours: a) Without aileron, b) NACA 6409, c) NACA 0015 and d) NACA 4415.

For a better understanding, the streamlines velocity for each case were analyzed. The flow dynamics was similar for the cases using a spoiler, where the formation of two vortices was observed at the rear of the box. As an example, Figure 5 shows the streamlines corresponding to the case of the NACA 0015 profile. In this case, it was observed that the main flow coming from the cabin intercepts on the upper surface of the spoiler. Then, the flow is diverted generating a separation while slightly increasing its speed. This last behavior was only observed in this case. On the remaining cases, the flow velocity reduced while leaving the spoiler surface.

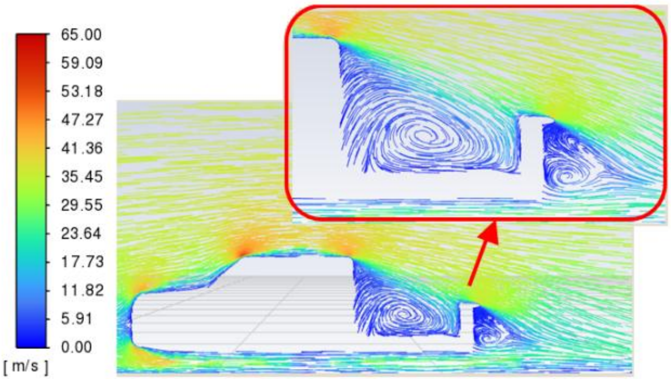

Figure 5: Flow streamlines for the case with NACA 0015 profile.

The flow velocity variations influenced the static pressure distribution. Figure 6 shows the static pressure contours on the rear surface of the vehicle for each case of study. It can be seen that the use of a spoiler results in an increase in static pressure in this area. In addition, an increase in static pressure was observed on the upper surface of the ailerons. This confirms that, when a spoiler is added, the more airflow is attached to the back surface resulting in a low-pressure zone reduction. In the case of the NACA 0015 profile, a low-pressure area was generated in the central area of the trailing edge of the spoiler, as can be seen in Figure 6 c). This is attributed to the flow behavior previously described in Figure 5.

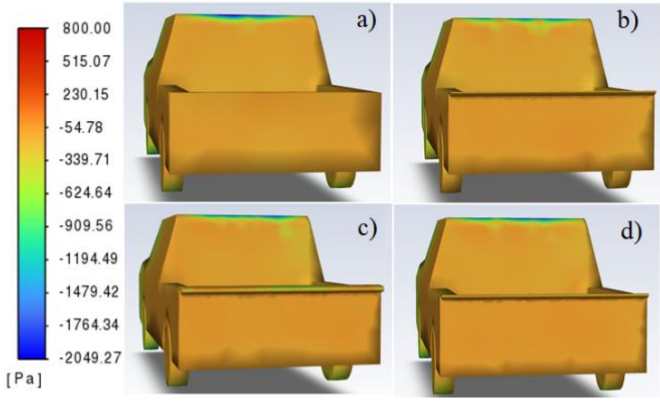

Figure 6: Static pressure contours: a) Without spoiler, b) NACA 6409, c) NACA 0015 and d) NACA 4415.

Data of static pressure at the rear surface of the vehicle was obtained. Figure 7 shows the curves of the data of dimensionless vertical distribution obtained at the center of the surface. It can be observed that the use of some spoiler increases the static pressure compared to the case without a spoiler. Among the studied profiles, the NACA 4415 demonstrated the best performance by having the highest pressure values. This allowed the drag to be further reduced by reducing the low-pressure zone. In addition, drag coefficients were calculated for each case.

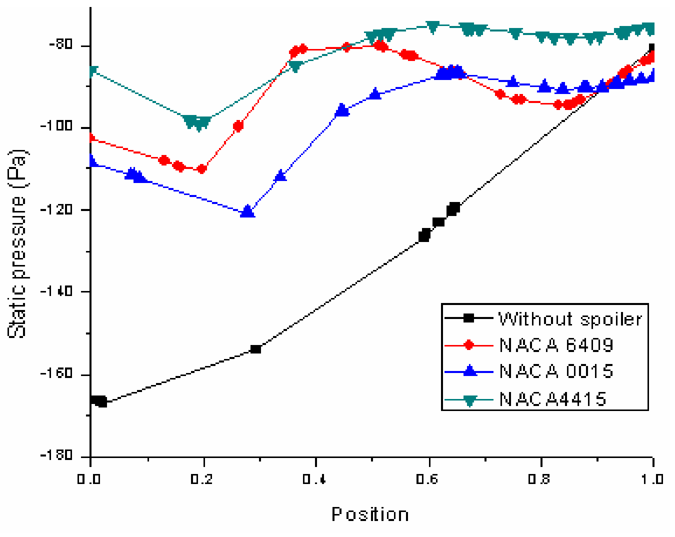

Figure 7: Static pressure distribution of the studied cases.

Table 2 shows the calculated values for the drag coefficient C_D and the lift coefficient C_L. The value of the drag coefficient for the case without spoiler was 0.508, which is the same to that reported in a similar study [13]. The drag coefficient was reduced with the use of the spoilers compared to the case without the device. The lowest value was obtained with the NACA 4415 profile, which agrees with the previously described velocity and static pressure results. On the other hand, the C_L values were barely affected by using spoilers, except for the case with the NACA 6409 profile. In this case, the C_L value was increased, which is not desirable since it could affect vehicle stability.

Table 2: Drag and lift coefficients for each case study.

Coefficient	Without spoiler	NACA 6409	NACA 0015	NACA 4415
C_D	0.508	0.460	0.455	0.438
C_L	0.123	0.260	0.126	0.124

4. CONCLUSIONS

In this work, the performance of various profiles of a rear spoiler for a pick-up type vehicle was studied. The NACA 6409, NACA 0015 and NACA 4415 spoiler were studied and compared with a model without spoiler. The velocity results indicated that the addition of the device reduced the low-pressure zone at the rear of the vehicle. An effect on flow dynamics was also observed in this area. When analyzing the distribution of static pressure on the rear surface of the box, it was observed that it increased with the use of the spoilers. In addition, curves of data of static pressure on this surface showed that effect on the vehicle. Drag and lift coefficients were also calculated. These coefficients indicated that the NACA 4415 profile had the best performance in reducing drag, which was also observed with the static pressure and velocity results. With the use of this profile for the spoiler, it was possible to reduce the value of the drag coefficient by approximately 13.7%. On the other hand, the NACA 6409 revealed an increase in lift coefficient.

5. REFERENCES

[1] ND, M. M., Pani, S., & Navindgi, M. C., 2022, “Effect of Biofuel-Nanoparticle Blend on the Performance and Emission Characteristics of Compression Ignition Engine”, International Journal of Applied Engineering Research, 17(1), 54-65.

[2] Vivekananthan, R. & Anil, K.B., 2021, “Experimental Investigation of the Effect of Oxyhydrogen on Spark Ignition Engines”, International Journal of Applied Engineering Research, 16(5), 340-345.

[3] Ehsani, M., Gao, Y., & Emadi, A., 2010, “Modern electric, hybrid electric, and fuel cell vehicles, fundamentals, theory and design”, 2nd edition, CRC press.

[4] Das, R. C., & Riyad, M., 2017, “CFD analysis of passenger vehicleat various angle of rear end spoiler”, Procedia engineering, 194, 160-165, http://dx.doi.org/10.1016/j.proeng.2017.08.130

[5] Hernández, Q., D., “Tecnologías que contribuyen al ahorro de combustible en vehículos ligeros,” Dir. Movil. y Transp., pp. 1–12, 2020.

[6] Sagar, D., Paul, A. R., Upadhyay, R. R., & Jain, A., 2010, “Aerodynamic effects of rear spoiler and vortex generators on passenger cars”, Proceedings of 5th International Conference on Theoretical, Applied, Computational and Experimental Mechanics (ICTACEM-2010), No. 311, pp. 760-762.

[7] Kim, J., Pattermann, J., Menon, V., & Mokhtar, W., 2011, “A CFD Study of Pickup Truck Aerodynamics”, Proceedings of the 2011 ASEE North Central & Illinois-Indiana Section Conference.

[8] Yakkundi, V., and Mantha, S., 2018, “Effect of Spoilers on Aerodynamic Properties of Car Effect of Spoilers on aerodynamic properties of a car”, vol. 7, no. 3, pp. 271–280.

[9] Taniguchi, K., Shibata, A., Murakami, M., & Oshima, M., 2017, “A study of drag reduction devices for production pick-up trucks”, No. 2017-01-1531, SAE Technical Paper, doi: 10.4271/2017-01-1531.

[10] Spike, C. G., Finn, T. J., Dubreuil, E. M., Wessner, A. K., & Lee, S. J., 2016, “Reduction of Aerodynamic Drag on a Commercial Pickup Truck via External Flow Devices”, In 54th AIAA Aerospace Sciences Meeting, p. 1107, doi: 10.2514/6.2016-1107.

[11] Luthfie, A. A., Romahadi, D., Ghufron, H., & Murtyas, S. D., 2020, “Numerical simulation on rear spoiler angle of mini mpv car for conducting stability and safety”, Sinergi, 24(1), 23-28, doi: 10.22441/sinergi.2020.1.004.

[12] Mukda, P., 2016, “Effect from Accessories on Pickup Aerodynamics by Computational Fluid Dynamics”, In 6th Int'l Conference on Advances in Engineering Sciences and Applied Mathematics (ICAESAM'2016), pp. 59-63.

[13] Ha, J., Jeong, S., & Obayashi, S., 2011, “Drag reduction of a pickup truck by a rear downward flap”, International

Journal of Automotive Technology, 12, 369-374. doi: 10.1007/s12239.

[14] Musa, J. K., & Patel, P. D., 2016, “Aerodynamic Drag Reduction of A Passenger Vehicle by Controlling the Wake using CFD Analysis”, vol. 2, no. 11, pp. 282–288.

[15] Hortelano-Capetillo, J. G., Vázquez, J. M., & Rodríguez-Ortíz, G., 2020, “Análisis aerodinámico en CFD de los alerones en automóviles convencionales Aerodynamic analysis in CFD of spoilers in conventional cars” Revista de Ingeniería, 4(11), 1-12. doi: 10.35429/jie.2020.11.4.1.12.

[16] Elsayed, O., Omar, A. A., Jeddi, A., Elhessni, S., & Hachimy, F. Z., 2021, “Drag Reduction by Application of Different Shape Designs in a Sport Utility Vehicle”, International Journal of Automotive and Mechanical Engineering, 18(3), 8870-8881. doi: 10.15282/ijame.18.3.2021.03.0680.

[17] Launder, B.E., & Spalding, D.B., 1974, “The numerical computation of turbulent flows”, Computer Methods in Applied Mechanics and Engineering, 3, 269-289.

[18] Ansys Fluent., 2018, “Fluent 19.1 Theory guide”, Release 19.1.

[19] Grau, E., 2019, “Análisis aerodinámico de las principales configuraciones de parte trasera de un vehículo convencional,” Grau en Eng. en Tecnol., Ind. Análisis, pp. 2–105, 2018.

[20] Palencia, J., García, N., and Pinto, M., 2015, “Estudio Del Comportamiento Aerodinamico De Un Aleron Trasero En El Diseño De Vehiculo Tipo Formula SAE,” no. October 2015, pp. 1–10, 2008.

[21] AirfoilTools.com, “NACA 4 digit airfoil generator,” consulted in June of 2022. http://airfoiltools.com/airfoil/naca4digit